Chapter 1

Adding Constraints to Sketches

Learning Objectives

After completing this chapter, you will be able to:

- *Add geometric constraints to sketches*
- *Control the display of constraints*
- *Apply constraints automatically*
- *Apply dimensional constraints to sketches*
- *Convert a dimensional constraint into an annotational constraint*
- *Create expressions*

Key Terms

- *Horizontal*
- *Vertical*
- *Coincident*
- *Fix*
- *Collinear*
- *Parallel*
- *Concentric*
- *Tangent*
- *Symmetric*
- *Equal*
- *Smooth*
- *Infer Constraints*
- *Dimensional Constraints*
- *Annotational Constraints*
- *Dynamic Constraint Mode*
- *Annotational Constraint Mode*
- *Fully-Defined Sketch*
- *Under-Defined Sketch*
- *Over-Defined Sketch*
- *Parameters Manager*

INTRODUCTION

In AutoCAD, a wide set of tools are available that help you create parametric drawings. These tools are available in the **Parametric** tab of the **Ribbon** as well as in the **Parametric** toolbar. The parametric nature of a drawing implies that the shape and size of a geometry can be defined using standard properties or parameters. The main function of parametric property is to drive a new size or shape to the selected geometry without considering its original dimensions. Also, you can modify the shape and size of any entity of a sketch at any stage of the design process. This property makes the designing process very easy.

To make a drawing parametric, you need to apply constraints. When you apply constraints to sketches, it restricts their degrees of freedom, thereby making them stable. The stability ensures that the size, shape, and location of sketches do not change unexpectedly with respect to surroundings. You can apply two types of constraints to sketches, geometric and dimensional. Geometric constraints are applied between two entities in a sketch and they control the shape of a drawing, whereas dimensional constraints are applied to a single entity and they control the size of the object in a drawing. Remember that dimensions applied and dimensional constraints applied are two different things. You will learn about the difference in this chapter.

If you need to draw a sketch of parts such as bolt, nut, valves, and so on in which the size of one entity is driven by one basic dimension, then in AutoCAD, you can draw these parts quickly and easily using the **Parameters Manager**.

ADDING GEOMETRIC CONSTRAINTS

AutoCAD provides you with twelve types of geometric constraints. Usually, these constraints are applied between two entities in a sketch and control the shape of a drawing. However, there are constraints, such as Fix, that are applied to a single entity. Whenever you apply a geometric constraint between two entities, the entity that is selected last is affected. For example, if you apply the parallel constraint between two lines, the line selected second will orient toward the first line. It is recommended to first apply geometric constraints so that the shape of a sketched entity does not change on applying dimensional constraints. If the **Infer Constraints** button is chosen in the Status Bar, some of the geometric constraints are applied automatically while sketching or editing. For example, if you draw a horizontal line then the Horizontal constraint will be applied automatically and the corresponding constraint bar will be displayed above the line. In the following section, different geometric constraints and the procedure to apply them manually are discussed.

Applying the Horizontal Constraint

Ribbon: Parametric > Geometric > Horizontal **Command:** GEOMCONSTRAINT > H
Toolbar: Geometric Constraint >Horizontal or Parametric > Constraint drop-down > Horizontal
Menu Bar: Parametric > Geometric Constraints > Horizontal



The Horizontal constraint forces a selected line, polyline, or two points to become horizontal. To apply this constraint, choose the **Horizontal** tool from the **Geometric** panel of the **Parametric** tab of the **Ribbon**. The prompt sequence after choosing this tool is given next.

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Select an object or [2Points] <2Points>: *Select a line or polyline.*

Remember that if you apply this constraint to a line on which no other constraint is applied, then the line will orient such that the endpoint nearer to the location you clicked on will become the fixed point and the other endpoint will become horizontal, thereby making the line horizontal.

If you need to make two points horizontal, press Enter when the **Select an object or [2Points] <2Points>** prompt is displayed. The following prompt sequence will be displayed:

Select an object or [2Points] <2Points>: Enter.
Select first point: *Select the first point.*
Select second point: *Select the second point.*



Note

*You can also apply constraints by entering **GEOMCONSTRAINT** at the command prompt.*

Applying the Vertical Constraint

Ribbon: Parametric > Geometric > Vertical **Command:** GEOMCONSTRAINT > V
Toolbar: Geometric Constraint > Vertical or Parametric > Constraint drop-down > Vertical
Menu Bar: Parametric > Geometric Constraints > Vertical



The Vertical constraint forces a selected line, polyline, or two points to become vertical. To apply this constraint, choose the **Vertical** tool from the **Geometric** panel of the **Parametric** tab. The prompt sequence after choosing this tool is given next.

Select an object or [2Points] <2Points>: *Select a line or polyline.*

Remember that if you apply this constraint to a line on which no other constraint is applied, then the line will orient such that the endpoint nearer to the location you clicked on will become the fixed point and the other endpoint will become vertical, thereby making the line vertical.

If you need to make two points vertical, press Enter when the **Select an object or [2Points] <2Points>** prompt is displayed. The following prompt sequence will be displayed:

Select an object or [2Points] <2Points>: Enter.
Select first point: *Select the first point.*
Select second point: *Select the second point.*



Note

On applying a constraint to an entity, the corresponding symbol is displayed near the entity, as shown in Figure 1-1. If you move the cursor over the symbol, the symbol will be highlighted and a cross mark (X) will be displayed. Click on the cross mark to hide the symbol. However, if you place the cursor on the entity on which a constraint has already been applied, a glyph will be displayed stating that a constraint is applied.

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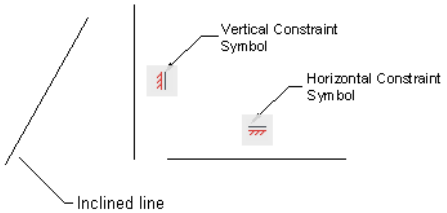


Figure 1-1 *The Vertical and Horizontal Constraint symbols*

Applying the Coincident Constraint

Ribbon: Parametric > Geometric > Coincident **Command:** GEOMCONSTRAINT > C
Toolbar: Geometric Constraint > Coincident or
Parametric > Constraint drop-down > Coincident
Menu Bar: Parametric > Geometric Constraints > Coincident

The Coincident constraint forces a selected point or keypoint of an entity to be coincident with the point or keypoint of another entity. The entity selected can be a line, polyline, circle, an arc or ellipse. Similarly, the selected point can be a point, or keypoints such as an endpoint or midpoint of a line, an arc or a polyline, or the centerpoint of an arc, ellipse, or a circle. To apply this constraint, choose the **Coincident** tool from the **Geometric** panel of the **Parametric** tab; the cursor will be replaced by a selection box along with the coincident symbol. Also, you will be prompted to select first point. Move the cursor over an entity; the keypoint that is close to the cursor will be highlighted in red. Next, click to select it; you will be prompted to select the second point to be coincident with the first object. Select the keypoint of the second entity; the keypoint of the second entity will move and coincide with that of the first entity.

Even if you have drawn the complete sketch by snapping keypoints, the coincident constraint will not be applied automatically. To apply the coincident constraints at all intersections at the same time, invoke the **Coincident** tool. The command prompt sequence to apply the **Autoconstrain** is given next.

Select first point or [Object/Autoconstrain] <Object>: *A* Enter.
Select objects: *Select objects individually or by using the selection window:*
Select objects: *Select more objects or* Enter.

Note

The options to change the default settings for the ***Autoconstrain*** *option will be discussed later in this chapter.*

Applying the Fix Constraint

Ribbon: Parametric > Geometric > Fix **Command:** GEOMCONSTRAINT > F
Toolbar: Geometric Constraint > Fix or Parametric > Constraint drop-down > Fix
Menu Bar: Parametric > Geometric Constraints > Fix

The Fix constraint forces the selected entity to become fixed at a position. If you apply this constraint to a line or to an arc, its location will be fixed with respect to the selected

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keypoint. But you can change the size of the entity by dragging its endpoints. To apply the Fix constraint, choose the **Fix** tool from the **Geometric** panel of the **Parametric** tab; you will be prompted to select a point or an object. Next, move the cursor over the entity; the point closer to the cursor will be highlighted. Click to fix the highlighted keypoint. Now, you cannot move the fixed keypoint. However, you can drag and relocate other keypoints.

To fix all keypoints of an object, invoke the Fix constraint and press Enter when the **Select point or [Object] <Object>** command prompt is displayed; you will be prompted to select an object. Select the object; all its keypoints will be fixed.

Applying the Perpendicular Constraint

Ribbon: Parametric > Geometric > Perpendicular **Command:** GEOMCONSTRAINT > P
Toolbar: Geometric Constraint > Perpendicular or
Parametric > Constraint drop-down > Perpendicular
Menu Bar: Parametric > Geometric Constraints > Perpendicular



The Perpendicular constraint forces the selected lines to become perpendicular to each other. To apply this constraint, choose the **Perpendicular** tool from the **Geometric** panel; you will be prompted to select the first object. Select a line or a polyline; you will be prompted to select the second object. Select another line or a polyline. The object selected second will become perpendicular to the object selected first. Figure 1-2 shows two lines before and after applying the Perpendicular constraint.

Applying the Parallel Constraint

Ribbon: Parametric > Geometric > Parallel **Command:** GEOMCONSTRAINT > PA
Toolbar: Geometric Constraint > Parallel or Parametric > Constraint drop-down > Parallel
Menu Bar: Parametric > Geometric Constraints > Parallel



The Parallel constraint forces the selected lines to become parallel to each other. To apply this constraint, choose the **Parallel** tool from the **Geometric** panel; you will be prompted to select the first object. Select a line or a polyline; you will be prompted to select the second object. Select another line or a polyline; the object selected second will become parallel to the object selected first. Figure 1-3 shows two lines before and after applying the Parallel constraint.

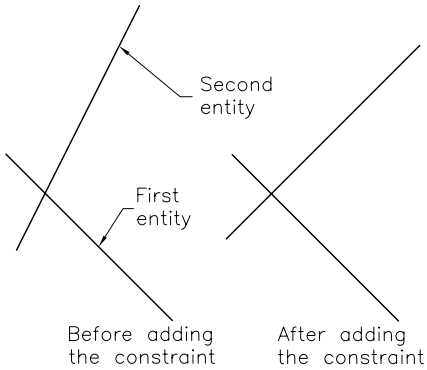


Figure 1-2 *Entities before and after applying the Perpendicular constraint*

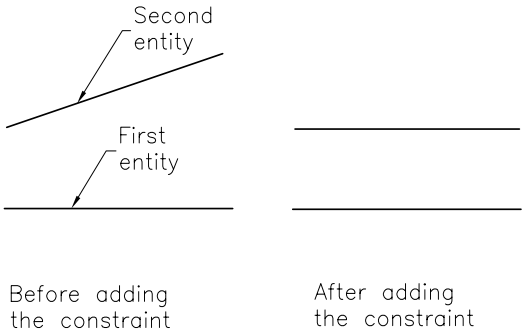


Figure 1-3 *Entities before and after applying the Parallel constraint*

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Applying the Collinear Constraint

Ribbon: Parametric > Geometric > Collinear **Command:** GEOMCONSTRAINT > COL
Toolbar: Geometric Constraint > Collinear or Parametric > Constraint drop-down > Collinear
Menu Bar: Parametric > Geometric Constraints > Collinear



The Collinear constraint forces the selected lines to lie on the same infinite line. To apply this constraint, choose the **Collinear** tool from the **Geometric** panel; you will be prompted to select the first object. Select a line; you will be prompted to select the second object. Select another line; the line selected later will become collinear with the first line.

To apply the Collinear constraint to multiple lines, choose the **Collinear** tool; the **Select first object or [Multiple]** prompt will be displayed at the command prompt. Enter **M** and press Enter; you will be prompted to select the first object. Select a line; you will be prompted to select the second object. Select the lines that need to be made collinear with the line selected first and then press Enter; the selected lines will become collinear to the line selected first.

Applying the Concentric Constraint

Ribbon: Parametric > Geometric > Concentric **Command:** GEOMCONSTRAINT > CON
Toolbar: Geometric Constraint > Concentric or
Parametric > Constraint drop-down > Concentric
Menu Bar: Parametric > Geometric Constraints > Concentric



The Concentric constraint forces the selected arc or circle to share the center point of another arc, circle, or ellipse. To apply this constraint, choose the **Concentric** tool from the **Geometric** panel; you will be prompted to select the first object. Select an arc, ellipse, or a circle; you will be prompted to select the second object. Select another arc, ellipse, or circle; the object selected second will become concentric to the object selected first.

Applying the Tangent Constraint

Ribbon: Parametric > Geometric > Tangent **Command:** GEOMCONSTRAINT > T
Toolbar: Geometric Constraint > Tangent or Parametric > Constraint drop-down > Tangent
Menu Bar: Parametric > Geometric Constraints > Tangent



The Tangent constraint forces the selected arc, circle, spline, or ellipse to become tangent to another arc, circle, spline, ellipse, or line. To apply this constraint, choose the **Tangent** tool from the **Geometric** panel; you will be prompted to select the first object. Select an arc, circle, spline, line, or an ellipse; you will be prompted to select the second object. Select another arc, circle, spline, ellipse, or line; the object selected second will become tangent to the object selected first. Figure 1-4 shows a line and a circle before and after applying the Tangent constraint. Figure 1-5 shows two arcs before and after applying the Tangent constraint.

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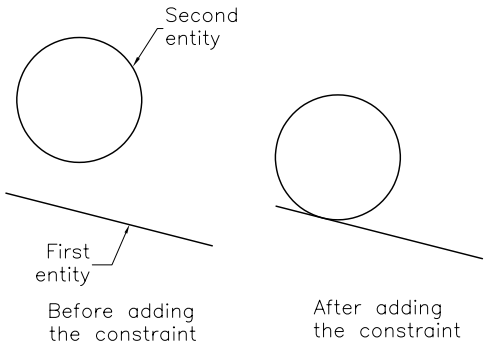


Figure 1-4 A line and a circle before and after applying the Tangent constraint

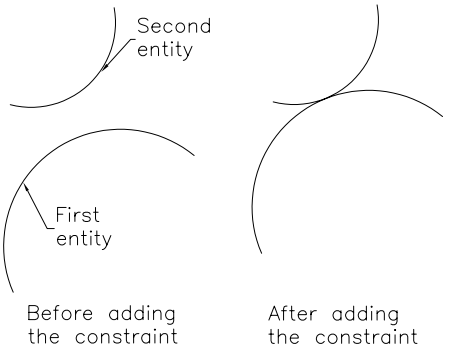


Figure 1-5 Two arcs before and after applying the Tangent constraint

Applying the Symmetric Constraint

Ribbon: Parametric > Geometric > Symmetric **Command:** GEOMCONSTRAINT > S
Toolbar: Geometric Constraint > Symmetric or
Parametric > Constraint drop-down > Symmetric
Menu Bar: Parametric > Geometric Constraints > Symmetric

The Symmetric constraint forces two selected lines, arcs, circles, or ellipses to remain equidistant from the centerline. To apply this constraint, choose the **Symmetric** tool from the **Geometric** panel; you will be prompted to select the first object or two points. Select an arc, an ellipse, a line, or a circle; you will be prompted to select the second object. Select another arc, ellipse, circle, or line; you will be prompted to select the symmetry line. Select a line; the selected objects will become symmetric with respect to the selected line. You can also make two points symmetric. To do so, press Enter at the **Select first object or (2 points)** command prompt. Next, select two points and the symmetric line; the selected points will become symmetric to the selected line.

Applying the Equal Constraint

Ribbon: Parametric > Geometric > Equal **Command:** GEOMCONSTRAINT > E
Toolbar: Geometric Constraint > Equal or Parametric > Constraint drop-down > Equal
Menu Bar: Parametric > Geometric Constraints > Equal

The Equal constraint forces the selected lines or polylines to have equal length, or the selected circles, arcs, or an arc and a circle to have equal radii. To apply this constraint, choose the **Equal** tool from the **Geometric** panel; you will be prompted to select an object. Select the first object; you will be prompted to select the second object. Select another object; the size of the second object will become equal to that of the object selected first.

If you need to apply the equal constraint to multiple objects, choose the **Equal** tool from the **Geometric** panel; the Select first object or [Multiple] prompt will be displayed at the command prompt. Enter **M** and press Enter; you will be prompted to select the first object. Select an object; you will be prompted to select other objects. Select the other objects that need to be equal and press Enter; the size of the objects selected later will become equal to that of the object selected first.

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Applying the Smooth Constraint

Ribbon: Parametric > Geometric > Smooth **Command:** GEOMCONSTRAINT > SM
Toolbar: Geometric Constraint > Smooth or Parametric > Constraint drop-down > Smooth
Menu Bar: Parametric > Geometric Constraints > Smooth



The Smooth constraint is used to apply curvature continuity between a spline and an entity connected together. The entities that can be connected to a spline include a line, spline, polyline, or an arc. To apply this constraint, choose the **Smooth** tool from the **Geometric** panel; you will be prompted to select the first object. Select a spline as the first object; you will be prompted to select the second object. Select another object; the smooth constraint will be applied between the selected objects.

CONTROLLING THE DISPLAY OF CONSTRAINTS

To control the display of the constraints, click on the inclined arrow in the **Geometric** panel of the **Parametric** tab; the **Constraint Settings** dialog box will be displayed, as shown in Figure 1-6. In this dialog box, the **Geometric** tab will be chosen and the check boxes of all constraints will be selected. Also, the **Show constraint bars after applying constraints to selected objects** check box will be selected by default. Therefore, on applying a constraint, a constraint bar will be displayed above the sketch and the symbol of that particular constraint will be displayed in the constraint bar by default. If you do not need any particular constraint symbol to be displayed then in the **Constraint bar display settings** area, clear the check box of the constraints that you do not want to be displayed. If you do not need the constraint bar to be displayed on applying the constraints, then clear the **Show constraint bars after applying constraints to selected objects** check box. However, if the **Show constraint bars when objects are selected** check box is selected, then the constraint bars will be displayed when you select an object. You can control the transparency of the constraint bars by setting the value in the **Constraint bar transparency** area of the **Constraint Settings** dialog box. In AutoCAD, you can apply the constraints while sketching or editing. To do so, select the **Infer geometric constraints** check box. You can also choose the **Infer Constraints** button from the Status Bar to apply constraints while sketching or editing. After specifying the parameters, choose the **OK** button to exit the dialog box.

On moving the cursor over a constraint bar, the constraint symbol and entities on which the constraint was applied will be highlighted. If multiple constraints are displayed in a sketch, it causes confusion. Therefore, you need to hide or display the constraint symbols. To hide a particular constraint, move the cursor over that constraint; the constraint bar will be highlighted and a cross-mark will be displayed on it. Click on the cross-mark to hide that constraint. To hide all constraints, choose the **Hide All** button in the **Geometric** panel. To display all hidden constraints, choose the **Show All** button from the **Geometric** panel. If you need to display constraints of a particular entity, choose the **Show/Hide** button from the **Geometric** panel; you will be prompted to select an entity. Select the entity from the drawing area and press Enter; a menu will be displayed and you will be prompted to select an option. Select the **Show** option; the constraints of that particular entity will be displayed. Remember that hiding a constraint does not delete that constraint. To delete a constraint, move the cursor over the constraint bar and press the Delete key. If you need to delete multiple applied constraints, choose the **Delete Constraints** button in the **Manage** panel of the **Parametric** tab of the **Ribbon**; you will be prompted to select entities. Select the entities individually or by using the selection box, and then press Enter; the constraints applied on the selected entities will be deleted.

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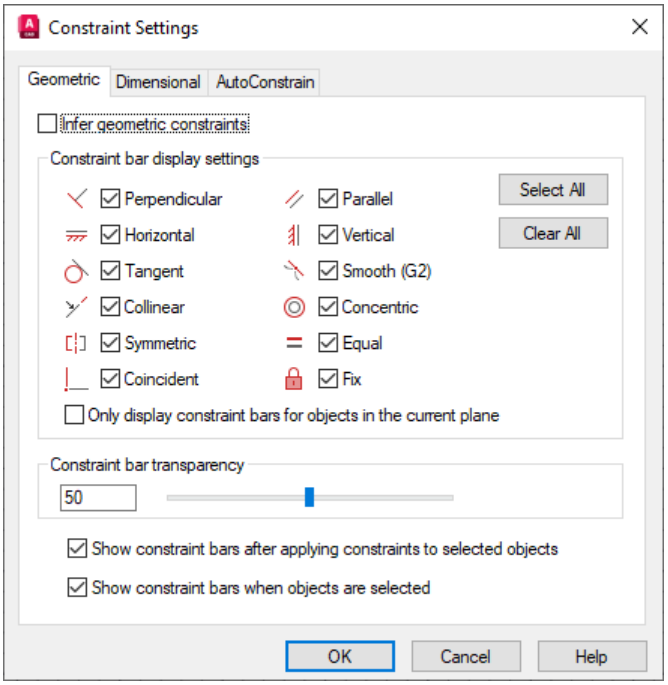


Figure 1-6 *The* ***Constraint Settings*** *dialog box with the* ***Geometric*** *tab chosen*



Tip

If you move the cursor over a constraint bar and right-click, a shortcut menu will be displayed. Choose the corresponding option to delete or hide a constraint. Also, you can choose the ***Constraint Bar Settings*** *option to display the* ***Constraint Settings*** *dialog box.*

Example 1

Draw the sketch shown in Figure 1-7 and apply appropriate constraints to it. Do not dimension the sketch.

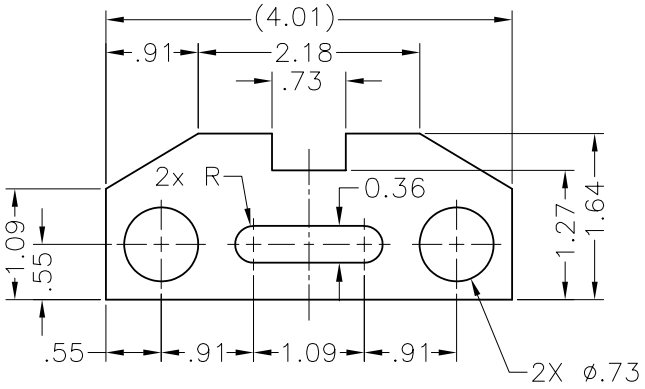


Figure 1-7 *Sketch for Example 1*

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In this example, you will select the **Infer geometric constraints** check box and draw the sketch. Therefore, some of the constraints will be applied while sketching. Next, you will apply the constraints that are not applied automatically while sketching. First, you will create the outer loop of the sketch using the **Line** tool.

1. Start a new file with the *acad.dwt* template in the **Drafting & Annotation** workspace.

2. Ensure that the **Infer Constraints** button is chosen in the Status Bar. By using the **Line** tool, draw the outer profile similar to that shown in Figure 1-7. Make sure that the endpoints are snapped so that they form a closed sketch, as shown in Figure 1-8. Note that in your drawing, the constraints applied by default may be different from that shown in Figure 1-8, as it depends upon the start point and the procedure used to draw the lines.

   Next, you need to apply the constraints that are not applied by default. First, you will apply the Equal constraint.

3. Choose the **Equal** tool from the **Geometric** panel in the **Parametric** tab; the cursor changes to a selection box with the Equal constraint's symbol attached to it. Also, you are prompted to select the first object.

4. Select the vertical line on the left as the first object; you are prompted to select the second object.

5. Select the vertical line on the right as the second object; the equal constraint is applied and the constraint bar is displayed.

6. Similarly, apply the equal constraint between the two inclined lines, two small vertical lines, and two small horizontal lines. The sketch after applying all the constraints is shown in Figure 1-9.

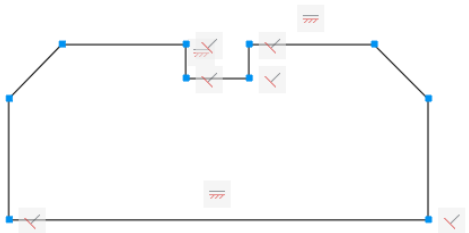


Figure 1-8 *Outer loop of the sketch*

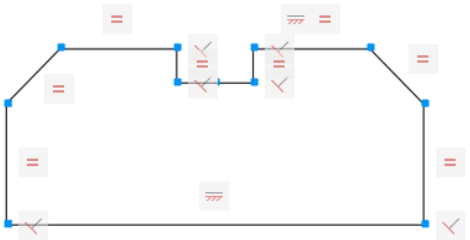


Figure 1-9 *Sketch after applying all constraints*

Next, you need to apply the symmetric constraint between entities on either side of the centerline.

7. Choose the **Infer Constraints** button from the Status Bar to turn off the infer constraints.

8. Draw a vertical centerline that passes through the midpoint of the horizontal line at the bottom, as shown in Figure 1-10.

9. Choose the **Symmetric** tool from the **Geometric** panel; the cursor changes to a selection box with symmetric symbol attached to it. Also, you are prompted to select the first object.

10. Select the vertical line on the left as the first object; you are prompted to select the second object.

11. Select the vertical line on the right as the second object; you are prompted to select the symmetry line.

12. Select the centerline as the symmetry line; the vertical lines become symmetric to the centerline and the symmetric symbol is displayed, as shown in Figure 1-11. Note that in this figure, other constraints are hidden for clarity.

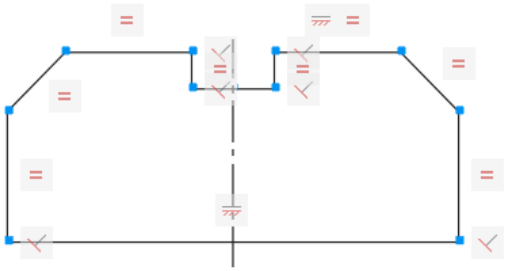


Figure 1-10 *Sketch after drawing the centerline*

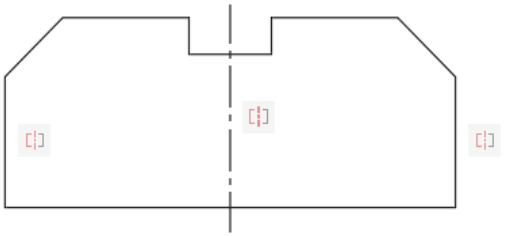


Figure 1-11 *The Symmetric constraint symbol*

13. Similarly, apply the symmetric constraints between the two inclined lines and then between the two small horizontal lines. The three symmetric constraints are applied as shown in Figure 1-12.

14. Choose the **Hide All** button from the **Parametric** tab of the **Geometric** panel to hide all constraint bars. Figure 1-13 shows the sketch after hiding all constraint bars.

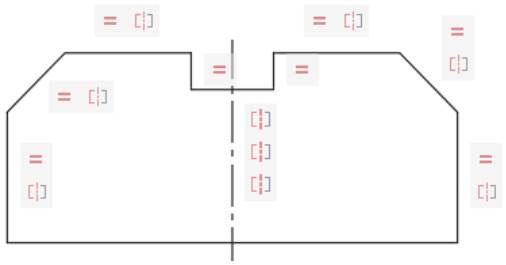


Figure 1-12 *Sketch with symmetric constraints*

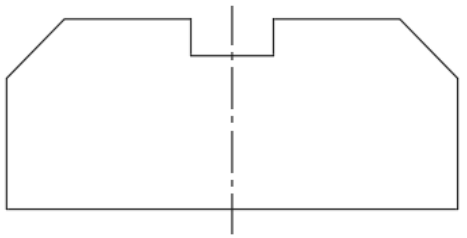


Figure 1-13 *Sketch after hiding all constraints*

Next, you need to create the inner loop using the **Arc**, **Circle**, and **Line** tools at an arbitrary location. But the profile should be similar to that given in Figure 1-7. Next, you need to apply the constraints.

15. Draw two circles of any radius inside the outer loop.

16. Choose the **Infer Constraints** button from the Status Bar to turn on the infer constraints.

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17. Draw a slot using the **Center**, **Start**, **End** arc and **Line** tools. The sketch after drawing the circles and the slot is shown in Figure 1-14. You can also draw the profile of the slot using the **Polyline** tool. However, the sketch in this example has been drawn using the **Center**, **Start**, **End** arc and **Line** tools.

18. Apply the equal constraint first between two circles and then between two arcs.

19. Apply the symmetric constraint between the circles as well as between the arcs.

20. Choose the **Tangent** tool from the **Geometric** panel; the cursor changes to a selection box with the tangent symbol attached to it. Also, you are prompted to select the first object.

21. Select an arc; you are prompted to select the second object.

22. Select a line; the tangent constraint will be applied at one end of the line and the tangent symbol will be displayed.

23. Similarly, apply the tangent constraint at all other places in the slot, as shown in Figure 1-15. You need to choose the **Show/Hide** button to display all constraints.

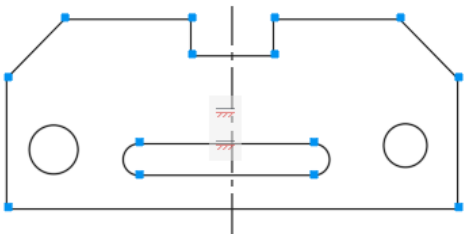


Figure 1-14 *Sketch after applying the Coincident constraint*

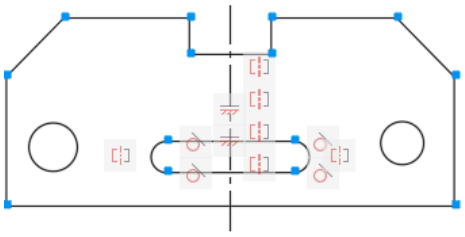


Figure 1-15 *Sketch after applying all Tangent constraints*

Next, you need to make the centerpoint of the circles and the slot to be in a straight line.

24. Choose the **Horizontal** tool from the **Geometric** panel; the prompt sequence that follows is given next.

Select an object or [2Points] <2Points>: Enter.
Select first point: *Select the circle on the left side* Enter.
Select second point: *cen* Enter.
of: *Select the center of the arc on the left side* Enter.

The sketch after applying the Horizontal constraint is shown in Figure 1-16.

25. Similarly, apply the Horizontal constraint to the circle and arc on the right side.

26. Choose the **Show/Hide** button from the **Parametric** tab and hide the display of all constraint bars and save the drawing. The sketch after applying and hiding all constraints is shown in Figure 1-17.

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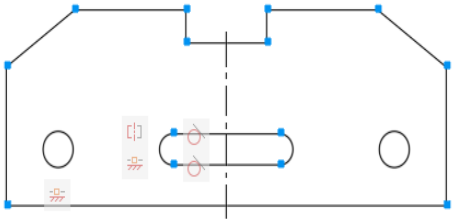


Figure 1-16 Sketch after applying the Horizontal constraint between a circle and an arc

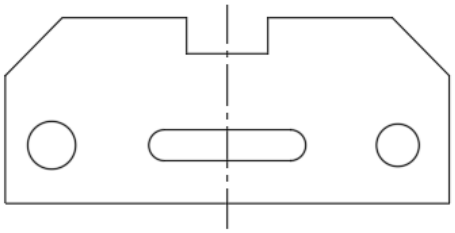


Figure 1-17 Sketch after applying and hiding all constraints

APPLYING CONSTRAINTS AUTOMATICALLY

In the previous section, twelve types of constraints are discussed. Nine of these constraints can be applied automatically, provided you have a closed sketch. To apply constraints automatically, choose the **AutoConstrain** tool in the **Geometric** panel of the **Parametric** tab of the **Ribbon**; you will be prompted to select an object. Select the entities individually or by using the selection box, and then press Enter; all possible constraints will be applied automatically.

You can also specify the settings of the constraints that need to be applied automatically. To do so, choose the **AutoConstrain** button; the **Select objects or [Settings]** prompt will be displayed at the command prompt. Enter **S** at the command prompt and press the Enter key; the **Constraint Settings** dialog box will be displayed, as shown in Figure 1-18.

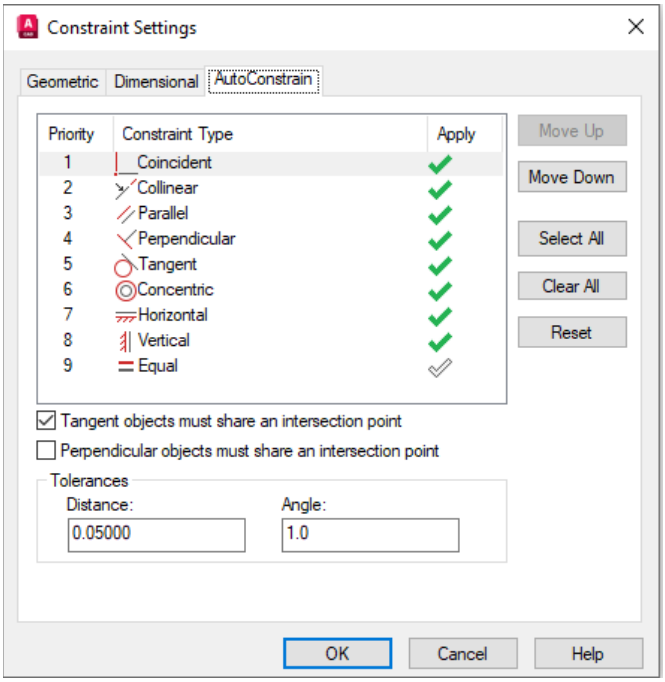


*Figure 1-18 The **Constraint Settings** dialog box with the **AutoConstrain** tab chosen*

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In this dialog box, the **AutoConstrain** tab is chosen by default and the nine constraints that can be applied automatically are displayed. You can rearrange these constraints according to your priorities by choosing the **Move Up** or **Move Down** button. If you do not need a particular constraint to be applied automatically to the sketch, click on the green tick mark corresponding to the constraint; the tick mark will turn white indicating that the corresponding constraints will not be applied automatically. If the **Tangent objects must share an intersection point** check box is selected, the tangent constraint will be applied, only if the two entities intersect at a point. If you clear this check box, the tangent constraint will be applied, only if the distance between the endpoints satisfies the value specified in the **Tolerances** area. Similarly, if the **Perpendicular objects must share an intersection point** check box is selected, the Perpendicular constraint will be applied, only if the two entities intersect at a point. If you clear this check box, the Perpendicular constraint will be applied, only if the distance between the endpoints satisfies the tolerances value specified in the **Tolerances** area.

Exercise 1

Draw the sketch shown in Figure 1-19 and apply appropriate constraints to it. Do not dimension the sketch.

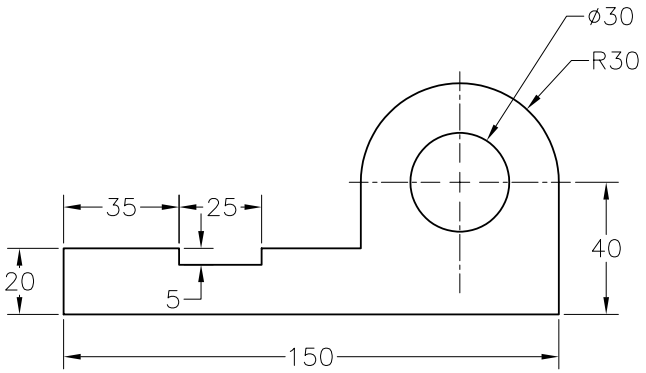


Figure 1-19 *Sketch for Exercise 1*

APPLYING DIMENSIONAL CONSTRAINTS

The older versions of AutoCAD were mainly used during the documentation stage of a design. However, the introduction of dimensional constraint has made AutoCAD a complete design package. In AutoCAD, you can apply dimensional constraint to an entity and change the size of the entity with ease. Therefore, now you can use AutoCAD at the initial design stage itself. Remember that the dimensions, which you apply using the tools in the **Dimensions** panel of the **Annotate** tab of the **Ribbon** do not change the size of the entity. However, because of their associative nature, if you change the size of an entity by dragging grips, the dimension will get changed.

You can apply the horizontal, vertical, aligned, angular, radius, and diameter dimensional constraint to a sketch. The procedure to apply a dimensional constraint to an entity is similar to that of applying dimension to an entity. For example, to apply the horizontal dimensional constraint, invoke the **Linear** tool from **Parametric > Dimensional > Dimensional Constraints** drop-down; you will be prompted to select the first constraint point. Move the cursor near one of

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the keypoints; it will be highlighted. Click to select the keypoint; you will be prompted to select the second constraint point. Move the cursor near the other keypoint and click, when it gets highlighted; the dimension will be attached to the cursor and you will be prompted to place the dimension. On placing the dimension, the dimension value will be displayed in a textbox. Press Enter to accept the value. If you need to change the value, enter a new value in the dimension box and press Enter. A lock symbol will be displayed next to the dimension value indicating that it is a dimensional constraint. In this way, you can apply other dimensional constraints as well.

Instead of selecting keypoints to apply the dimensional constraint, you can also select the entity to be dimensioned. To do so, invoke the **Linear** tool from **Parametric > Dimensional > Dimensional Constraints** drop-down and press Enter when the **Specify first constraint point or [Object] <Object>** prompt is displayed; you will be prompted to select an object. Select the object; the dimension will be attached to the cursor. Place it at the required position.



Tip

*You can also enter the arithmetic functions directly in the text box that is displayed on placing a dimensional constraint. For example, if you have an arithmetic function such as (22 *5.5)-30+5, which is equal to 96, you do not need to calculate this function. Just enter the function in the text box and press Enter; AutoCAD will automatically solve the mathematical expression. However, only the mathematical expression will be displayed in the text box and not the result.*

CONVERTING A DIMENSIONAL CONSTRAINT INTO AN ANNOTATIONAL CONSTRAINT

As discussed earlier, the dimensional constraint applied and the dimensions applied are different. The dimensional constraints are applied by using the tools available in the **Dimensional** panel of the **Parametric** tab. Whereas, dimensions are applied using the tools in the **Dimensions** panel of the **Annotate** tab. The dimensions that are applied using the tools from the **Dimensions** panel are called annotational constraints as they are used only for annotational purpose and cannot control the size of an entity.

If you have applied a dimension (annotational constraint) to an entity, you can convert it into a dimensional constraint. To do so, choose the **Convert** tool from the **Dimensional** panel of the **Parametric** tab; you will be prompted to select a dimension. Select the dimension and press Enter; the dimension (annotational constraint) will be converted into the dimensional constraint.

You can also convert a dimensional constraint into an annotational constraint. To do so, select a dimensional constraint and invoke the **PROPERTIES** palette. Select **Annotational** from the **Constraint Form** field in the **Constraint** rollout; the dimensional constraint will change to the annotational constraint and the options in the **Properties** rollout will change accordingly.

A dimensional constraint will not be plotted when you plot the drawing using the **Plot** tool. Therefore, you need to convert a dimensional constraint into an annotational constraint, as discussed above. You can also place the dimensional constraint in the annotational constraint mode. To do so, expand the **Dimensional** panel in the **Parametric** tab. You will notice that the **Dynamic Constraint Mode** is chosen by default. Choose the **Annotational Constraint Mode**. Now, if you place the dimensional constraint, you can plot them.

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You can also make a dimensional constraint as a reference constraint. A reference constraint is displayed in brackets and is driven by a dimensional or annotational constraint. To make a dimensional constraint as reference, invoke the **PROPERTIES** palette of a dimensional constraint and select **Yes** in the **Reference** field of the **Constraint** rollout. If you have changed a dimensional constraint to an annotational constraint, you can make this annotational constraint as a reference constraint using the **PROPERTIES** palette.

CONCEPT OF A FULLY-DEFINED SKETCH

It is important for you to understand the concept of fully-defined sketches. If you have created a sketch in the design stage, you need to add the required geometrical and dimensional constraints to the sketch to constrain its degrees of freedom with respect to the surrounding. After applying the required constraints, the sketch may exist in any of the following three states:

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

Under-defined

An under-defined sketch is the one in which some of the geometrical constraints or dimensional constraints are not defined. As a result, the degrees of freedom of the sketch are not fully constrained, and therefore, the entities may move or change their size unexpectedly. So, you need to add some geometrical constraints or dimensional constraints to it to make it fully-defined.

Fully-defined

A fully-defined sketch is the one in which all entities and their positions are fully-defined by geometrical constraints, dimensional constraints, or both. In a fully-defined sketch, all degrees of freedom of a sketch are constrained, and therefore, the sketched entities cannot move or change their size and location unexpectedly.

Over-defined

An over-defined sketch is the one in which some of the geometrical constraints, dimensional constraints, or both are conflicting. Also, if the geometrical constraints or the dimensional constraints in a sketch exceed the required number, the sketch will be over-defined. If you try to apply more geometrical constraints or dimensional constraints to a fully-defined entity, a message box will be displayed with the message that applying this constraint will over define the entity, as shown in Figure 1-20 and Figure 1-21.



Note

*If you apply dimensional constraint to a fully-defined entity, the **Dimensional Constraints** message box will be displayed, refer to Figure 1-21. Choose the **Create a reference dimension** option from the message box to convert that particular dimension into a reference dimension.*

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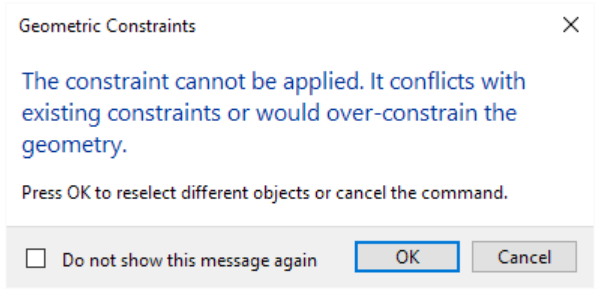


Figure 1-20 *The* ***Geometric Constraints*** *message box displayed on applying more than the required geometric constraints*

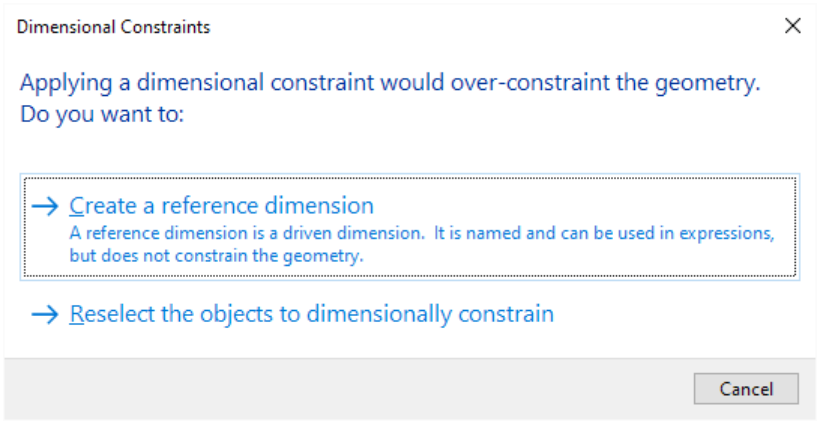


Figure 1-21 *The* ***Dimensional Constraints*** *message box displayed on applying more than the suggested dimensional constraints*

CONTROLLING THE DISPLAY OF THE DIMENSIONAL CONSTRAINT

On applying a dimensional constraint to an entity, a value will be displayed along with a name. If you have also applied annotational constraints to the sketch, then the sketch will be cluttered with dimensions. Therefore, you need to hide these constraints. To do so, choose the **Hide All** button from the **Dimensional** panel of the **Parametric** tab.

You can hide the selected dimensional constraints of an object by using the **Show/Hide** button. To do so, choose the **Show/Hide** button and select the respective constraint and press Enter. Next, right-click in the drawing area; a shortcut menu will be displayed. Choose the **Hide** option from it; the selected constraint will be hidden.

To display all dimensional constraints in the drawing, choose the **Show All** button from the **Dimensional** panel of the **Parametric** tab. To hide the annotational constraint, turn off the corresponding layer from the **LAYER PROPERTIES MANAGER**.

When you apply a dimensional constraint to an entity, a default name and a lock symbol will be displayed along with the value. If you need to display only the value, click on the inclined arrow

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on the **Dimensional** panel of the **Parametric** tab; the **Constraint Settings** dialog box will be displayed with the **Dimensional** tab chosen, as shown in Figure 1-22. Specify the format of the dimensional constraint to be displayed in the **Dimension name format** drop-down list available in the **Dimensional constraint format** area of this dialog box.

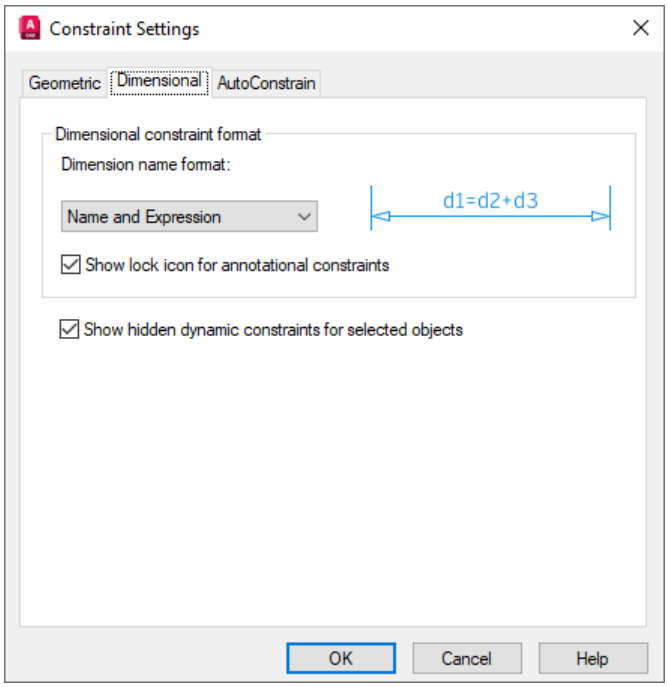


Figure 1-22 *The* ***Constraint Settings*** *dialog box with the* ***Dimensional*** *tab chosen*

If you convert a dimensional constraint into an annotational constraint, a lock symbol will be displayed. If you do not need the lock symbol to be displayed for an annotational constraint, clear the **Show lock icon for annotational constraints** check box from the **Constraint Settings** dialog box. In case, you want to view the dynamic constraints of an entity whose dimensional constraints are hidden by choosing the **Show lock icon for annotational constraints** button, select the **Show hidden dynamic constraints for selected objects** check box in the **Constraint Settings** dialog box. Now, if you select an entity whose dynamic constraints are hidden, the dynamic constraints will be displayed.

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Example 2

In this example, you will draw the sketch shown in Figure 1-23, and then apply the geometric and dimensional constraints to it.

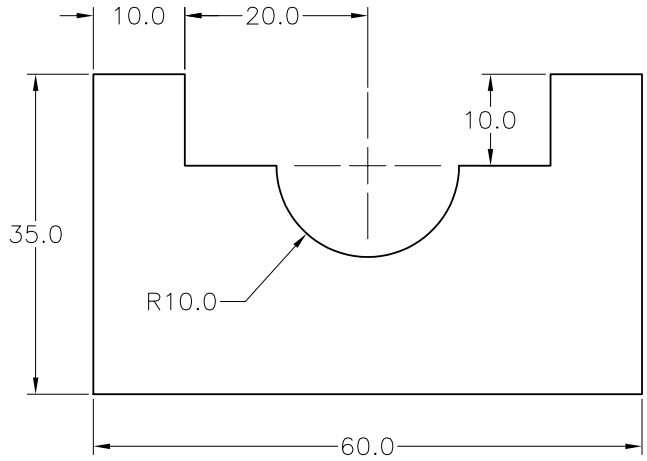


Figure 1-23 *Sketch for Example 2*

First, you will draw the profile and apply the geometrical constraints.

1. Start a new file with the *acad.dwt* template in the **Drafting & Annotation** workspace and draw the outer profile with dimensions close to that given in Figure 1-23. Make sure that the endpoints are snapped so that they form a closed sketch, as shown in Figure 1-24.

2. Choose the **AutoConstrain** button from the **Geometric** panel of the **Parametric** tab, enter **S** at the command prompt, and press Enter; the **Constraint Settings** dialog box is displayed.

3. Make sure that all constraints are selected. As there is no need to apply the concentric or tangent constraints in this sketch, select the **Concentric** and **Tangent** constraints individually and move them down using the **Move Down** button.

4. Choose the **OK** button; you are prompted to select objects. Select all entities of the sketch by dragging a selection box. Next, press Enter; all possible constraints are applied to the sketch, as shown in Figure 1-25.

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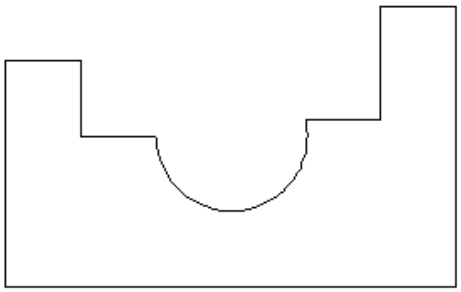


Figure 1-24 *Closed sketch drawn*

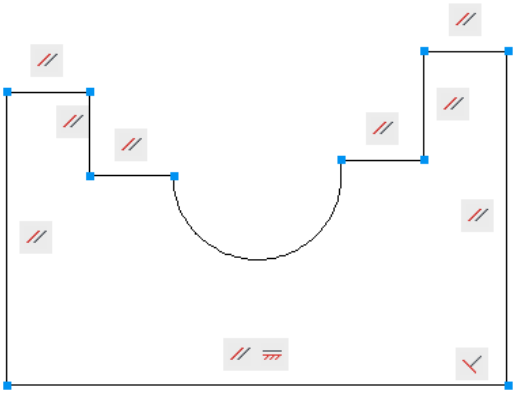


Figure 1-25 *Constraints applied automatically*

Note

The constraints applied using the ***AutoConstrain*** *tool may vary depending upon the profile. In your case, if the constraints are different from that shown in Figure 1-25, apply the missing constraints. Also, choose the* ***Show All*** *button from the* ***Geometric*** *panel if the constraints are not visible by default.*

5. Now, you need to apply the equal constraint. To do so, choose the **Equal** tool from the **Geometric** panel of the **Parametric** tab and apply the Equal constraint between the lines that are supposed to have equal dimensions. For dimensions, refer to Figure 1-23. The equal constraints need to be applied to the two pairs of horizontal and two pairs of vertical lines in the sketch. The sketch after applying the Equal constraint is shown in Figure 1-26.

Note

If you do not want to apply the Equal constraint to vertical lines, you need to apply the Collinear constraint to the two pairs of horizontal lines. But after applying the Equal constraint if you apply the Collinear constraint, a message box will be displayed stating that this constraint cannot be applied and conflicts the existing constraints or is over-constraining the geometry.

Next, you need to make the centerpoint of the circle and the horizontal line to be in a line.

6. Choose the **Horizontal** tool from the **Geometric** panel of the **Parametric** tab; the prompt sequence that will follow is given next.

Select an object or [2Points] <2Points>: Enter.
Select first point: *Move the cursor over the horizontal line and click when the endpoint near the arc is highlighted.*
Select second point: *cen* Enter *and select the arc on the left side.*

The sketch after applying the **Horizontal** constraint is shown in Figure 1-27.

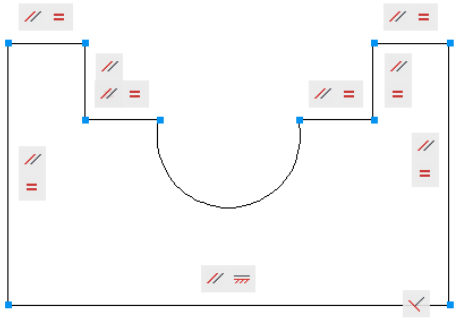


Figure 1-26 *Sketch after applying the Equal constraint*

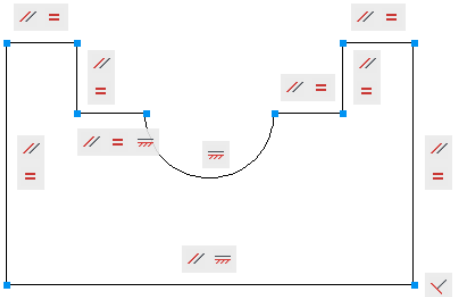


Figure 1-27 *Sketch after applying the Horizontal constraint*

7. Choose the **Hide All** button from the **Geometric** panel in the **Parametric** tab; all constraint bars disappear.

Now, you need to apply dimensional constraints. Before that, expand the **Dimensional** panel and ensure that the **Dynamic Constraint Mode** option is chosen.

8. Choose the **Linear** tool from **Parametric > Dimensional > Dimensional Constraint** drop-down and follow the prompt sequence given next.

   Specify first constraint point or [Object] <Object>: Enter.
   Select object: *Select the horizontal line at the bottom.*
   Specify dimension line location: *Move the cursor down and place the horizontal dimension at the required location. Enter **60** in the text box and press Enter.*

9. Press the Enter key again or choose the **Linear** tool from the **Dimensional** panel of the **Parametric** tab and follow the prompt sequence given next.

   Specify first constraint point or [Object] <Object>: Enter.
   Select object: *Select the vertical line on the left.*
   Specify dimension line location: *Move the cursor toward the left and place the vertical dimension at the required location. Enter **35** in the textbox.*

10. Similarly, apply the dimension to the horizontal and vertical lines, refer to Figure 1-28 and change the value to **10**.

11. Choose the **Radius** tool from the **Dimensional** panel of the **Parametric** tab; you are prompted to select the arc or the circle.

12. Select the arc and enter **10** as radius.

Now, you need to apply the dimension between the center of the arc and the vertical line on the left, refer to Figure 1-23.

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13. Invoke the **Linear** tool from the **Parametric > Dimensional > Dimensional Constraints** drop-down and follow the prompt sequence given next.

Specify first constraint point or [Object] <Object>: *Select the keypoint at the top of the smaller vertical line on the left.*
Specify second constraint point: *cen* Enter *and select the arc.*
Specify dimension line location: *Place the dimension at the required location and enter* ***20****; the* ***Dimensional Constraints*** *message box is displayed.*

14. Choose the **Create a reference dimension** option from the message box. The value **20** is displayed in parenthesis stating that it is a reference dimension. The sketch after applying all dimensional constraints is shown in Figure 1-28.

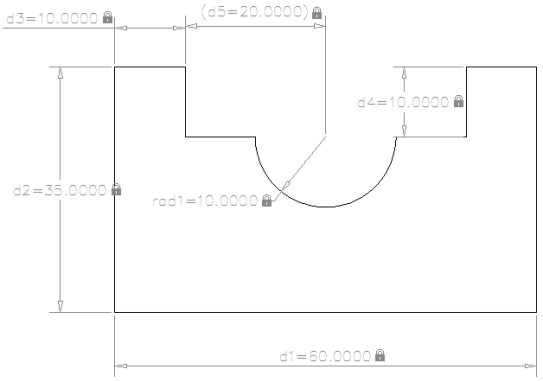


Figure 1-28 *Sketch with all dimensional constraints*

WORKING WITH EQUATIONS

Equations are mathematical relations between dimensions. Consider the sketch of a bolt. All the dimensions of a bolt are driven by the base diameter. In AutoCAD, if you need to draw the sketch of an object whose dimensions are driven by a base dimension, you can draw it easily by using equations in which all dimensions are equated with base dimensions. If you want to relate one or two dimensions with base dimension, you can do so by applying a dimensional constraint. But if most of the dimensions need to be represented as equations, then you need to use the **Parameters Manager**. Both these methods are discussed next.

Adding Equations while Applying Dimensional Constraints

When you apply the dimensional constraint using the tools from the **Dimensional** panel, the dimensions are displayed along with a default name. For the next dimensional constraint to be placed, you can represent it as an equation with the name of the first constraint. For example, to create a rectangle of length *l* units and width *w* units, which is half the length, you need to follow the steps given below.

1. Draw a rectangle and apply the required geometric constraints to it.
2. Choose the **Linear** tool from the **Dimensional** panel and select one of the lines representing the length and place the dimension; a text box will be displayed.
3. Enter **10** in the text box and press Enter; a new dimension will be displayed with the default name d1=10.0000.

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4. Choose the **Linear** tool from the **Dimensional** panel and select one of the lines representing the width and place the dimension; a text box will be displayed.
5. Enter **0.5*d1** in the text box and press Enter; the rectangle will resize and a new dimension will be displayed with the default name fx: d2=0.5*d1.
6. Now, if you change the dimension of the length, the width will also change automatically.

Note that, if you change the default name, d1, the name in the expression d2 will also change.

Adding Equations Using the Parameters Manager

Adding equations to dimensions while applying constraints may look simpler, but in actual practice, when there are more dimensions, it is quite difficult. This problem can be solved by applying equations to dimensions using the **PARAMETERS MANAGER**. After applying dimensional constraints, choose the **Parameters Manager** button from the **Manage** panel of the **Parametric** tab; the **PARAMETERS MANAGER** palette will be displayed with all dimensions and expressions, as shown in Figure 1-29. The names of the dimensional constraints applied will be listed under the **Name** column in the **Dimensional Constraint Parameters** node. Also, the expression of the corresponding constraints will be listed in the **Expression** column.

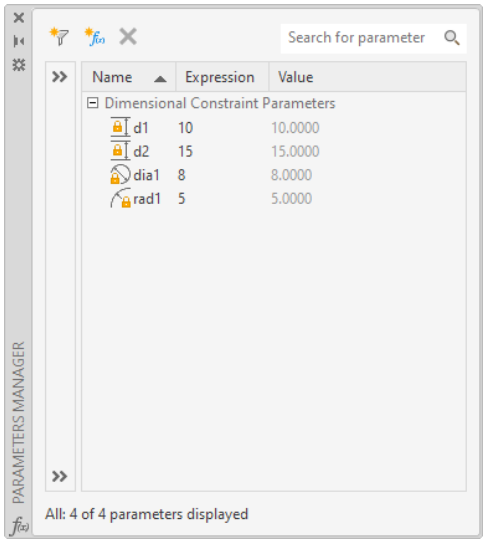


***Figure 1-29** The **PARAMETERS MANAGER** palette*

Now, you can change the existing expressions or add new expressions. To do so, double-click in the corresponding **Expression** field and add a new expression. If you want to add a list of new user-defined expressions, choose the **Creates a new user parameter** button from the **PARAMETERS MANAGER** palette; a new field will be added under the **User Parameters** node.

Example 3

In this example, you will open the sketch created in Exercise 1 of this chapter. You had applied geometrical constraints to this sketch. Now, you will apply dimensional constraints and then equate all the dimensions except the dimension 20 mm, with respect to the base length, 150 mm. For the dimension 20 mm, you will apply user-defined expressions as d150 = 20, d75 = 10, d300 = 40, and d100 = 13.3.

1. Choose the **Open** button from the **Quick Access Toolbar** and open the drawing created in Exercise 1 of this chapter.

2. Make sure that you have applied all the required geometrical constraints, as shown in Figure 1-30.

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Note

If you have not applied all constraints to the sketch, it may get deformed while applying dimensional constraints.

3. Invoke the **Linear** tool from **Parametric > Dimensional > Dimensional Constraints** drop-down and follow the prompt sequence given next.

   Specify first constraint point or [Object] <Object>: Enter.
   Select object: *Select the horizontal line at the bottom.*
   Specify dimension line location: *Move the cursor down and place the horizontal dimension at the required location. Enter **150** in the text box and press Enter.*

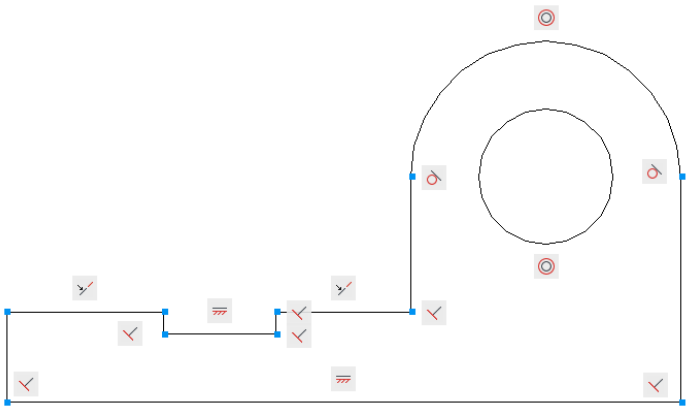


Figure 1-30 *Sketch with all constraints applied*

4. Press Enter again or choose the **Linear** tool from the **Dimensional** panel and follow the prompt sequence given next.

   Specify first constraint point or [Object] <Object>: Enter.
   Select object: *Select the vertical line on the left.*
   Specify dimension line location: *Move the cursor to left side and place the vertical dimension at the required location. Enter **20** in the text box and press Enter.*

5. Similarly, apply dimensions to other horizontal and vertical lines.

6. Choose the **Radius** tool from the **Dimensional** panel of the **Parametric** tab; you are prompted to select an arc or a circle.

7. Select the arc and enter **30** as the value of radius.

8. Choose the **Diameter** tool from the **Dimensional** panel of the **Parametric** tab; you are prompted to select the circle.

9. Select the circle and enter **30** as diameter.

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10. Choose the **Parameters Manager** button from the **Manage** panel; the **PARAMETERS MANAGER** palette will be displayed with the names of dimensional constraints and their values, as shown in Figure 1-31.

11. Double-click on **40** in the **Expression** column of the **PARAMETERS MANAGER** palette and enter **d1/3.75**.

12. Resize the palette. Double-click in the **Description** column corresponding to the value **40** and enter **Hole Location**.

13. Similarly, add the expression and description to all dimensions, except the vertical dimension **20**, as given next.

Value to be selected from the Expression column	**Expression to be added**	**Description to be added**
35	d1/30*7	Slot Location
25	d1/6	Slot Width
5	d1/30	Slot Depth
30 (Diameter value)	d1/5	Hole Diameter
30 (Radius value)	d1/5	Radius

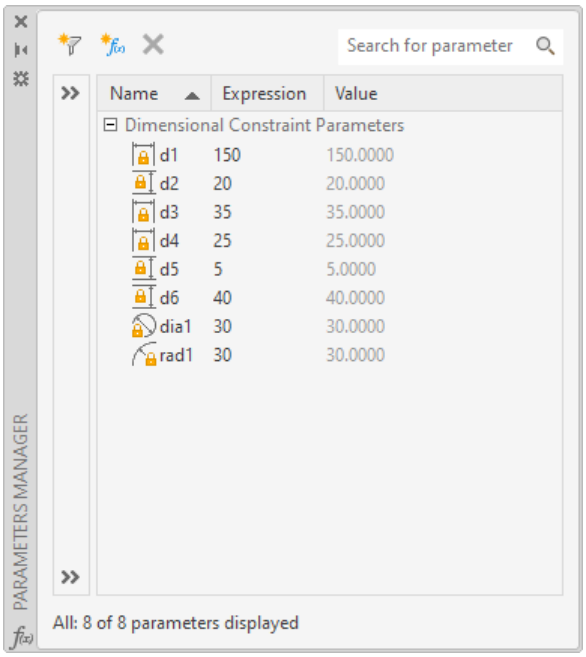


Figure 1-31 *The* ***PARAMETERS MANAGER*** *palette after applying all dimensions*

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Note

*If the **Type** and **Description** columns are not added by default, right-click on the title of a column and choose appropriate options.*

The **PARAMETERS MANAGER** palette after you have changed all the expressions, is shown in Figure 1-32.

For the value 20, you need to create a user-defined expression as follows:

14. Choose the **Creates a new user parameter** button in the **PARAMETERS MANAGER** palette; the **User Parameters** node is added with the default name, **user1**.

15. Change the name user1 to **d150**, the corresponding expression to **20**, and the description to **If d1 = 150**.

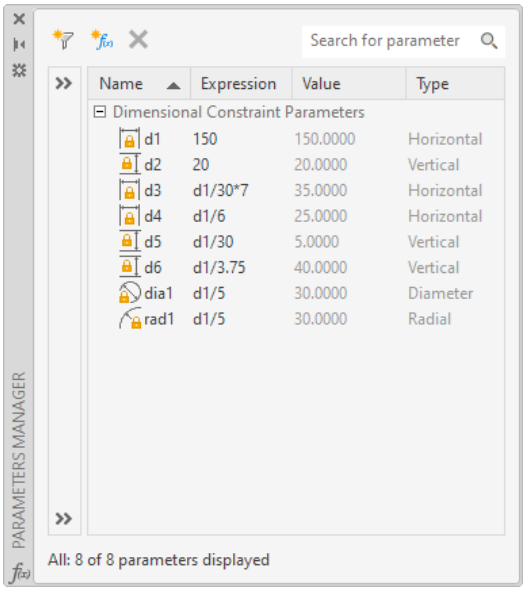


***Figure 1-32** The **PARAMETERS MANAGER** palette after adding all expressions*

16. Similarly, add three more user parameters and their corresponding expressions and descriptions as given below.

User parameter to be added	**Expression to be added**	**Description to be added**
d75	10	If d1 = 75
d300	40	If d1 = 300
d100	13.3	If d1 = 100

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Note

*You can also write a suitable expression for the dimension **20** in the **Dimensional Constraints** node. For illustration, the expression for the dimension **20** is given as a user variable.*

17. If the dimensional constraints are displayed, choose the **Show/Hide** button from the **Dimensional** panel of the **Parametric** tab to hide the dimensional constraints.

18. Now, change the dimensional constraints **d1** and **d2**, as specified below in the **PARAMETERS MANAGER** palette. Observe the change in the sketch in the drawing area and the dimension value in the **Value** column of the **PARAMETERS MANAGER** palette.

d1 = 300, d2 = d300
d1 = 100, d2 = d100
d1 = 150, d2 = 20

Self-Evaluation Test

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

1. Which of the following tools should be chosen in the **Dimensional** panel to apply both horizontal and linear dimensional constraints?

(a) **Aligned** (b) **Linear**
(c) **Lateral** (d) None of these

2. What will be the state of the sketch which does not have all its constraints defined?

(a) Under-defined (b) Fully-defined
(c) Over-defined (d) All of these

3. AutoCAD provides you _____ types of geometric constraints.

4. The _____ constraints are applied between two entities in a sketch and they control the shape of the drawing.

5. The _____ constraint forces the selected lines to lie on the same infinite line.

6. Invoke the _____ palette to add equations to dimensional constraints.

7. Whenever you apply a geometric constraint between two entities, the entity selected first will be affected. (T/F)

8. You cannot apply the coincident constraint to all the keypoints at a time. (T/F)

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9. It is always recommended to first apply dimensional constraints so that the shape of the sketched entity does not change on applying geometric constraints. (T/F)

10. You can add equations while applying dimensional constraints. (T/F)

Review Questions

Answer the following questions:

1. How many geometric constraints can be applied automatically?

   (a) Nine (b) Ten
   (c) Eight (d) Twelve

2. Which of the following constraints forces two lines or polylines to have equal length and the selected circles, arcs, or an arc and circle to have equal radii?

   (a) **Concentric** (b) **Equal**
   (c) **Collinear** (d) **Lock**

3. The _____ makes you to change or modify the shape and size of an entity of a sketch at any stage of the design process.

4. The _____ constraint is used to apply curvature continuity between a spline and an entity connected together.

5. A _____ dimensional constraint is displayed in brackets and is driven by a dimensional or annotational constraint.

6. The dimensions that you apply using the tools in the _____ panel do not change the size of an entity.

7. The symmetric constraint forces two selected lines, arcs, circles, or splines to remain equidistant from the centerline. (T/F)

8. Using the fix constraint, you can only fix the points. (T/F)

9. You can also enter arithmetic symbols directly in the text box that is displayed on placing a dimensional constraint. (T/F)

10. If you hide the display of a constraint, the constraint applied gets deleted. (T/F)

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Exercise 1

Draw the sketches shown in Figure 1-33 and Figure 1-34. Next, add geometric and dimensional constraints to them.

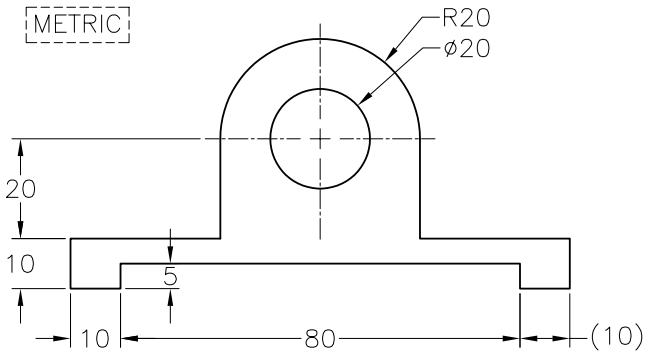


Figure 1-33 *Sketch for Exercise 2*

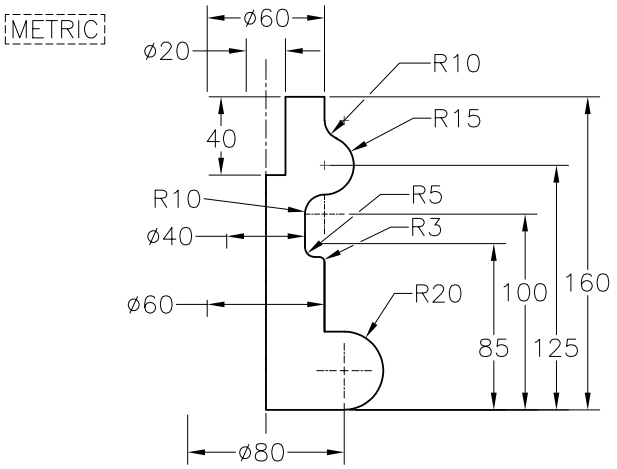


Figure 1-34 *Sketch for Exercise 2*

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Answers to Self-Evaluation Test

1. b, **2.** a, **3.** twelve, **4. Geometric**, **5. Collinear**, **6. PARAMETERS MANAGER**, **7.** F, **8.** F, **9.** F, **10.** T

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