Chapter 3

Getting Started with Advanced Sketching

Learning Objectives

After completing this chapter, you will be able to:

- *Draw arcs using various options*
- *Draw rectangles, ellipses, elliptical arcs, and polygons*
- *Draw polylines and donuts*
- *Place points and change their style and size*
- *Draw an infinite line*
- *Write simple text*
- *Use different templates*

Key Terms

- *Arc*
- *Rectangles*
- *Explode*
- *Ellipse*
- *Elliptical Arc*
- *Polygon*
- *Polylines*
- *Donut*
- *Points*
- *Construction Line*
- *Text*
- *Ray*
- *FILLMODE*
- *PDMODE*
- *DDPTYPE*

DRAWING ARCS

Ribbon: Home > Draw > Arc drop-down **Toolbar:** Draw > Arc
Menu Bar: Draw > Arc **Command:** ARC/A

An arc is defined as a segment of a circle. In AutoCAD, an arc is drawn by using the tools available in the **Arc** drop-down of the **Draw** panel in the **Ribbon**, refer to Figure 3-1. You can choose a tool on the basis of known parameters. Remember that the tool that was used last to create an arc will be displayed in the **Draw** panel. The methods to draw an arc are discussed next.

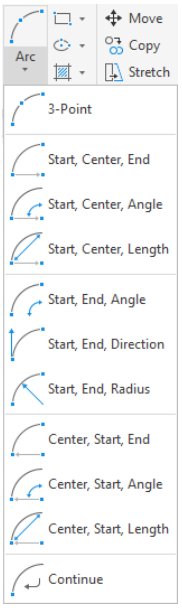


Figure 3-1 *The tools in the* ***Arc*** *drop-down*

Drawing an Arc by Specifying Three Points

To draw an arc by specifying the start point, endpoint, and another point on its periphery, choose the **3-Point** tool from the **Arc** drop-down in the **Draw** panel, refer to Figure 3-1. On doing so, you will be prompted to specify the start point. Specify the first point or specify coordinates for the first point. Then, specify the second point and endpoint of the arc, refer to Figure 3-2.

Following is the prompt sequence to draw an arc by specifying three points (You can also specify the points by using the mouse).

Choose the ***3-Point*** *tool from the* ***Arc*** *drop-down in the* ***Draw*** *panel*
(Ensure that dynamic input is off)
Specify start point of arc or [Center]: **2,2** Enter
Specify second point of arc or [Center/End]: **3,3** Enter
Specify end point of arc: **3,4** Enter

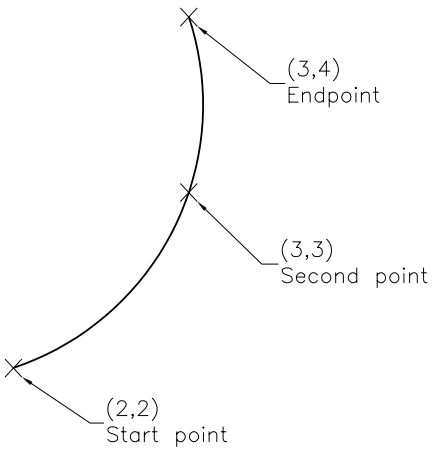


Figure 3-2 *Drawing an arc using the* ***3-Point*** *tool*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Exercise 1 *3-Point*

Draw several arcs by using the **3-Point** tool. The points can be selected by entering coordinates or by specifying points on the screen. Also, try to create a circle by drawing two separate arcs and a single arc and notice the limitations of the **Arc** tools.

Drawing an Arc by Specifying its Start Point, Center Point, and Endpoint

If you know the start point, endpoint, and center point of an arc, choose the **Start, Center, End** tool from the **Arc** drop-down in the **Draw** panel and then specify the start, center, and end points in succession; the arc will be drawn. The radius of the arc is determined by the distance between the center point and the start point. Therefore, the endpoint is used to calculate the angle at which the arc ends. Note that in this case, the arc will be drawn in a counterclockwise direction from the start point to the endpoint around the specified center, as shown in Figure 3-3.

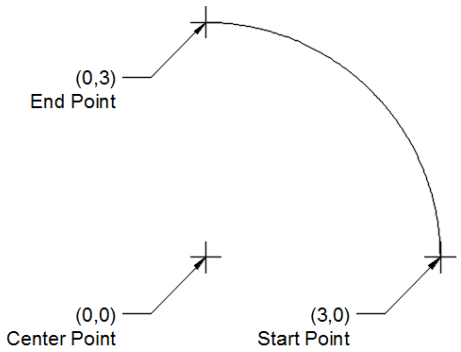


Figure 3-3 *Drawing an arc using the* ***Start, Center, End*** *tool*

Drawing an Arc by Specifying its Start Point, Center Point, and Included Angle

Included angle is the angle between the start and end points of an arc about the specified center. If you know the location of the start point, center point, and included angle of an arc, choose the **Start, Center, Angle** tool from the **Arc** drop-down in the **Draw** panel and specify the start point, center point, and included angle; the arc will be drawn in a counterclockwise direction with respect to the specified center and start point, refer to Figure 3-4.

If you enter a negative angle value, the arc will be drawn in a clockwise direction, refer to Figure 3-5. Following is the prompt sequence to draw an arc by specifying the center point at (2,2), start point at (3,2), and an included angle of -60 degrees:

Choose the ***Start, Center, Angle*** *tool from the* ***Draw*** *panel (Ensure that dynamic input is off)*
Specify start point of arc or [Center]: **3,2** Enter

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Specify center point of arc: **2,2** Enter
Specify included angle (hold Ctrl to switch direction): **-60** Enter, refer to Figure 3-5.

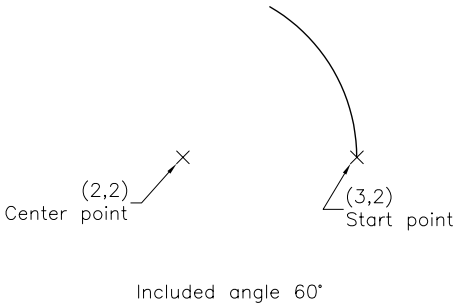


Figure 3-4 *Drawing an arc using the* ***Start, Center, Angle*** *tool*

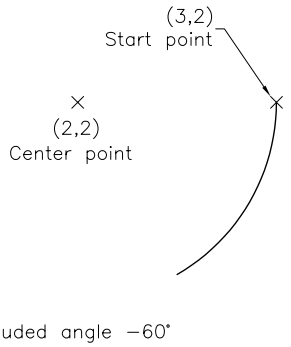


Figure 3-5 *Drawing an arc by specifying a negative angle using the* ***Start, Center, Angle*** *tool*

In AutoCAD, while creating an arc, you can flip the direction of the arc for specifying a point on the arc circumference using the Ctrl key. To do so, while creating the arc if you press and hold the Ctrl key and move the cursor, you will notice that the direction of arc has been flipped. Now, you can specify the parameters of the arc in different direction. Note that if you release the Ctrl key, the direction of arc creation will switch back to the previous direction.

Exercise 2 — *Start, Center, Angle*

a. Draw an arc whose start point is at 6,3, center point is at 3,3, and the included angle is 240 degrees.

b. Draw the profile shown in Figure 3-6. The distance between the dotted lines is 1.0 unit. Create the arcs by using different arc command options as indicated in the figure.

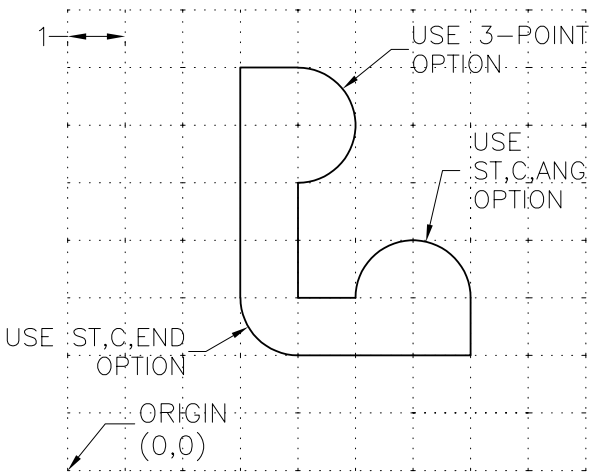


Figure 3-6 *Drawing for Exercise 2*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Drawing an Arc by Specifying the Start Point, Center Point, and Chord Length

A chord is defined as a straight line connecting the start point and endpoint of an arc. To draw an arc by specifying its chord length, choose the **Start, Center, Length** tool from the **Arc** drop-down in the **Draw** panel and specify the start point, center point, and length of the chord in succession. On specifying the chord length, AutoCAD will calculate the included angle and an arc will be drawn in the counterclockwise direction from the start point. A positive chord length gives the smallest possible arc with that length, as shown in Figure 3-7. This arc is known as minor arc. The included angle in a minor arc is less than 180 degrees. A negative value for the chord length results in the largest possible arc also known as major arc, as shown in Figure 3-8.

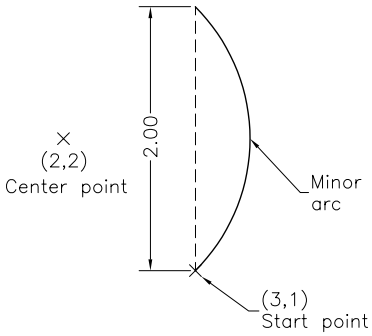


Figure 3-7 *Drawing a minor arc by specifying a positive chord length using the* ***Start, Center, Length*** *tool*

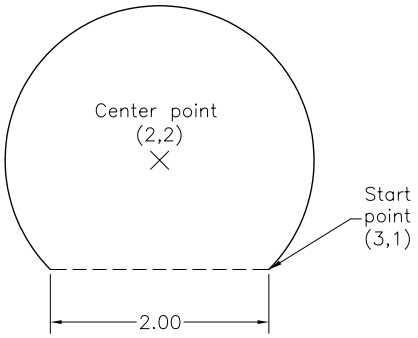


Figure 3-8 *Drawing a major arc by specifying a negative chord length using the* ***Start, Center, Length*** *tool*

Drawing an Arc by Specifying its Start Point, Endpoint, and Included Angle

To draw an arc by specifying its start point, endpoint, and the included angle, choose the **Start, End, Angle** tool from the **Arc** drop-down in the **Draw** panel and specify the start point, endpoint, and the included angle in succession; the arc will be drawn. A positive included angle value draws an arc in a counterclockwise direction from the start point to the endpoint, as shown in Figure 3-9. Similarly, a negative included angle value draws the arc in clockwise direction.

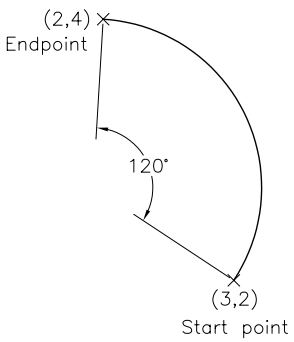


Figure 3-9 *Drawing an arc using the* ***Start, End, Angle*** *tool*

Drawing an Arc by Specifying its Start Point, Endpoint, and Direction

This option is used to draw a major or minor arc, whose size and position are determined by the distance between the start point and endpoint and the direction specified. You can specify the direction by selecting a point on a line that is tangent to the start point or by entering an angle between the start point of the arc and the end point of the tangent line.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

To draw an arc by specifying its direction, choose the **Start, End, Direction** tool from the **Arc** drop-down in the **Draw** panel and specify the start and end points in succession; you will be prompted to specify the direction. Specify a point on the line that is tangent to the start point or enter an angle between the start point of the arc and the end point of the tangent line; an arc will be drawn.

In other words, on using this option, the arc will start in the direction you specify (the start of the arc is established tangent to the direction you specify). The prompt sequence to draw an arc, whose start point is at 3,6, endpoint is at 4.5,5, and direction of -40 degrees is given next.

*Choose the **Start, End, Direction** tool from the **Arc** drop-down (Ensure that dynamic input is off)*
Specify start point of arc or [Center]: **3,6** Enter
Specify end point of arc: **4.5,3** Enter
Specify tangent direction for the start point of arc (hold Ctrl to switch direction): **-40** Enter, refer to Figure 3-10.

The prompt sequence to draw an arc, whose start point is at 4,3, endpoint is at 3,5, and direction of 90 degrees, is given next.

*Choose the **Start, End, Direction** tool from the **Arc** drop-down (Ensure that dynamic input is off)*
Specify start point of arc or [Center]: **4,3** Enter
Specify end point of arc: **3,5** Enter
Specify tangent direction for the start point of arc (hold Ctrl to switch direction): **90** Enter, refer to Figure 3-11.

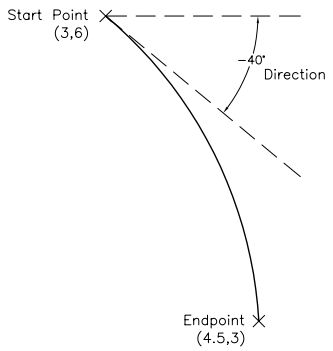


Figure 3-10 *Drawing an arc in negative direction using the **Start, End, Direction** tool*

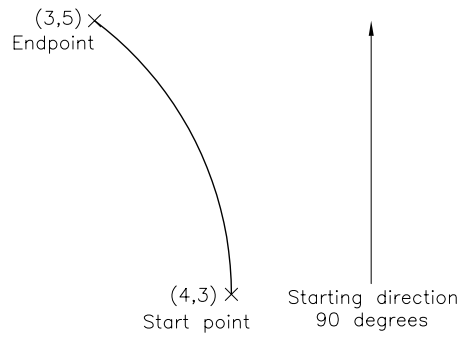


Figure 3-11 *Drawing an arc in positive direction using the **Start, End, Direction** tool*

Exercise 3 — Start, End, Direction

a. Specify the directions and coordinates of two arcs in such a way that they form a circular figure.

b. Draw the profile shown in Figure 3-12. Create the curves by using the **Start, End, Direction** tool. The distance between the dotted lines is 1.0 unit and the diameter of the circles is 1 unit.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

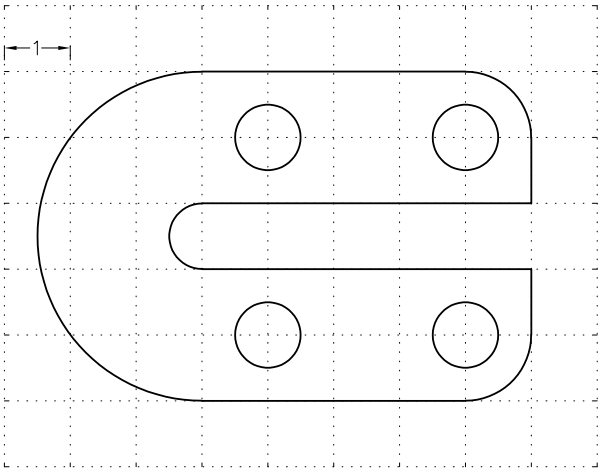


Figure 3-12 *Drawing for Exercise 3*

Drawing an Arc by Specifying its Start Point, Endpoint, and Radius

If you know the location of the start point, endpoint, and radius of an arc, choose the **Start, End, Radius** tool from the **Arc** drop-down in the **Draw** panel and specify the start and end points; you will be prompted to specify the radius. Enter the radius value; the arc will be drawn. In this case, the arc will be drawn in counterclockwise direction from the start point. This means that a negative radius value results in a major arc (the largest arc between the two endpoints), refer to Figure 3-13(a). A positive radius value results in a minor arc (smallest arc between the start point and the endpoint), refer to Figure 3-13(b).

Drawing an Arc by Specifying its Center Point, Start Point, and Endpoint

The **Center, Start, End** tool is the modification of the **Start, Center, End** tool. Use this tool, whenever it is easier to start drawing an arc by establishing the center first. Here, the arc is always drawn in a counterclockwise direction from the start point to the endpoint, around the specified center. The prompt sequence for drawing the arc shown in Figure 3-14, which has a center point at (3,3), start point at (5,3), and endpoint at (3,5), is given next.

*Choose the **Center, Start, End** tool from the **Draw** panel (Ensure that dynamic input is off)*
Specify center point of arc: **3,3** Enter
Specify start point of arc: **5,3** Enter
Specify end point of arc (hold Ctrl to switch direction) or [Angle/chord Length]: **3,5** Enter

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

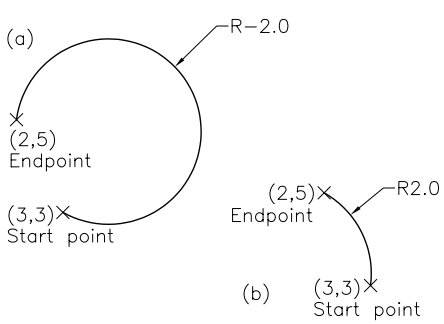


Figure 3-13 *Drawing arcs using the* ***Start, End, Radius*** *tool*

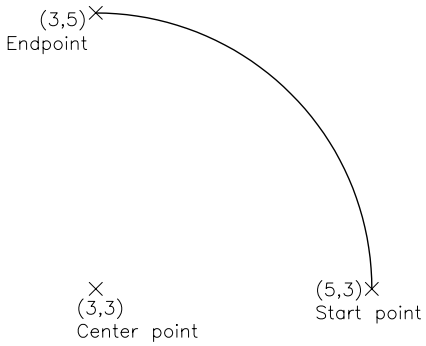


Figure 3-14 *Drawing an arc using the* ***Center, Start, End*** *tool*

Drawing an Arc by Specifying its Center Point, Start Point, and Angle

You can use the **Center, Start, Angle** tool if you need to draw an arc by specifying the center first. The prompt sequence for drawing the arc shown in Figure 3-15, which has a center point at (4,5), start point at (5,4), and included angle of 120 degrees, is given next.

Choose the ***Center, Start, Angle*** *tool from the* ***Arc*** *drop-down (Ensure that dynamic input is off)*
Specify center point of arc: **4,5** Enter
Specify start point of arc: **5,4** Enter
Specify included angle (hold Ctrl to switch direction): **120** Enter, refer to Figure 3-15.

Drawing an Arc by Specifying the Center Point, Start Point, and Chord Length

The **Center, Start, Length** tool is used whenever it is easier to draw an arc by establishing the center first. The prompt sequence for drawing the arc shown in Figure 3-16, which has a center point at (2,2), start point at (4,3), and length of chord as 3, is given next.

Choose the ***Center, Start, Length*** *tool from the* ***Arc*** *drop-down (Ensure that dynamic input is off)*
Specify center point of arc: **2,2** Enter
Specify start point of arc: **4,3** Enter
Specify length of chord (hold Ctrl to switch direction): **3** Enter, refer to Figure 3-16.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

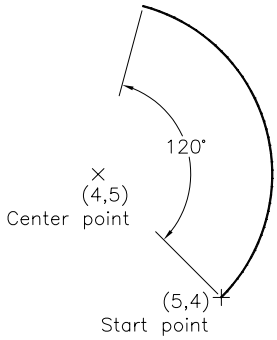


Figure 3-15 *Drawing an arc using the* ***Center, Start, Angle*** *tool*

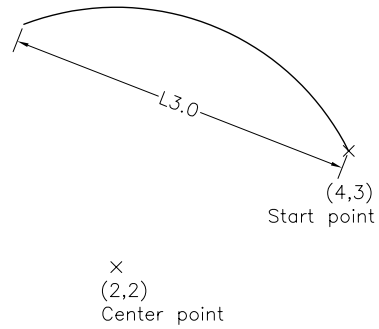


Figure 3-16 *Drawing an arc using the* ***Center, Start, Length*** *tool*

Exercise 4 — Center, Start, Length

Draw a minor arc with the center point at (3,4), start point at (4,2), and chord length of 4 units.

Drawing an Arc by Using the Continue Tool

To create an arc that is tangent to a previously drawn arc or line, choose the **Continue** tool from the **Arc** drop-down in the **Draw** panel; the start point and the direction of the arc will be taken from the endpoint and the ending direction of the previous line or arc. Next, specify the endpoint to draw an arc.

Tip

The ***Continue*** *tool can also be invoked automatically by first drawing a line or an arc and then choosing a tool from the* ***Arc*** *drop-down in the* ***Draw*** *panel. To do so, first draw a line or arc, then choose any tool from the* ***Arc*** *drop-down list (Other than* ***Center, Start, End****;* ***Center, Start, Angle****; and* ***Center, Start, Length*** *tools); you will be prompted to specify the start point of arc. Press Enter at this prompt; the* ***Continue*** *tool will be invoked automatically and an arc will get attached to the endpoint of the previously created line or arc.*

Exercise 5

a. Use the **Center**, **Start**, **Angle**, and **Continue** tools to draw the profiles shown in Figure 3-17.

b. Draw the profile shown in Figure 3-18. The distance between the dotted lines is 1.0 unit. Create the radii as indicated in the drawing by using the **Arc** tools.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

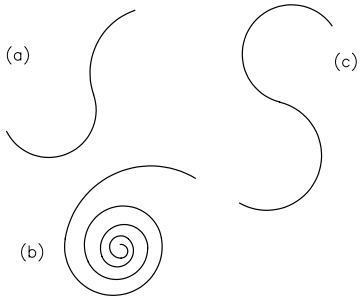


Figure 3-17 Drawing for Exercise 5(a)

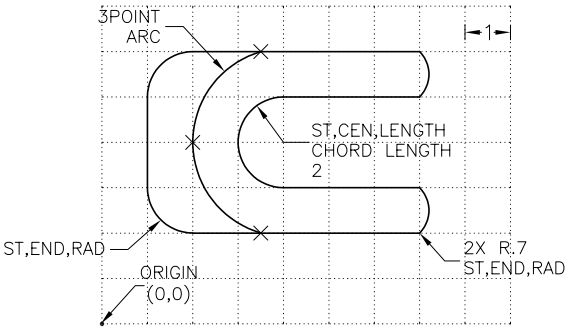


Figure 3-18 Drawing for Exercise 5(b)

DRAWING RECTANGLES

Ribbon: Home > Draw > Rectangle **Toolbar:** Draw > Rectangle
Tool Palettes: Draw > Rectangle **Command:** RECTANG/REC

A rectangle is drawn by choosing the **Rectangle** tool, refer to Figure 3-19, from the **Draw** panel of the **Home** tab. In AutoCAD, you can draw rectangles by specifying two opposite corners of the rectangle by specifying the area and the size of one of the sides or by specifying the dimensions of the rectangle. All these methods of drawing rectangles are discussed next.

Drawing Rectangles by Specifying Two Opposite Corners

On invoking the **Rectangle** tool, you will be prompted to specify the first corner of the rectangle. Enter the coordinates of the first corner or specify the first corner by using the mouse; you will be prompted to specify the other corner. The first corner can be any one of the four corners. Specify the diagonally opposite corner by entering the coordinates or by using the left mouse button, as shown in Figure 3-20.

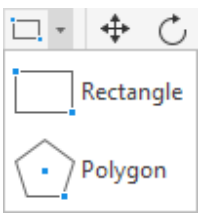


*Figure 3-19 Invoking the **Rectangle** tool from the **Draw** panel*

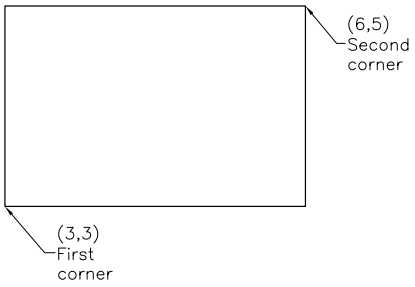


Figure 3-20 Drawing a rectangle by specifying two opposite corners

Drawing Rectangles by Specifying the Area and One Side

To draw a rectangle by specifying its area and the length of one of the sides, first specify the start point. Next, invoke the shortcut menu by right-clicking and then choose the **Area** option. Next, specify the parameters; the rectangle is drawn. Following is the prompt sequence to draw a rectangle whose start point is at 3,3, has an area of 15 units and length of 5 units:

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

*Choose the **Rectangle** tool from the **Draw** panel*
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **3,3** Enter
Specify other corner point or [Area/Dimensions/Rotation]: **A** Enter
Enter area of rectangle in current units <100.000>: **15** Enter
Calculate rectangle dimensions based on [Length/Width] <Length>: **L** Enter
Enter rectangle length <10.0000>: **5** Enter

In the above case, the area and length of the rectangle were entered. The system automatically calculates the width of the rectangle by using the following formula:

Area of rectangle = Length x Width
Width = Area of rectangle/Length
Width =15/5
Width =3 units

Drawing Rectangles by Specifying their Dimensions

You can also draw a rectangle by specifying its dimensions. This can be done by choosing the **Dimensions** option from the shortcut menu at the **Specify other corner point or [Area/Dimensions/Rotation]** prompt and entering the length and width of the rectangle. The prompt sequence for drawing a rectangle whose start point is at 3,3 with a length of **5** units and width of **3** units is given next.

*Choose the **Rectangle** tool from the **Draw** panel*
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **3,3** Enter
Specify other corner point or [Area/Dimensions/Rotation]: **D** Enter
Specify length for rectangles <0.0000>: **5** Enter
Specify width for rectangles <0.0000>: **3** Enter
Specify other corner point or [Area/Dimensions/Rotation]: *Click on the screen to specify the orientation of rectangle.*

Here, you are allowed to choose any one of the four locations for placing the rectangle. You can move the cursor to see the four quadrants. Depending on the location of the cursor, the corner point that is specified first holds the position of either the lower left corner, the lower right corner, the upper right corner, or the upper left corner. After deciding the position, you can click to place the rectangle.

Drawing Rectangle at an Angle

You can also draw a rectangle at an angle. This can be done by choosing the **Rotation** option from the shortcut menu, at the **Specify other corner point or [Area/Dimensions/Rotation]** prompt and entering the rotation angle. After entering the rotation angle, you can continue sizing the rectangle using any one of the above discussed methods. The prompt sequence for drawing a rectangle at an angle of 45-degree is:

*Choose the **Rectangle** tool from the **Draw** panel*
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: *Select a point as lower left corner location.*
Specify other corner point or [Area/Dimensions/Rotation]: **R** Enter

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Specify rotation angle or [Pick points] <current>: **45** Enter
Specify other corner point or [Area/Dimensions/Rotation]: *Select a diagonally opposite point.*

While specifying the other corner point, you can place the rectangle in any of the four quadrants. Move the cursor in different quadrants and then select a point in the quadrant in which you need to draw the rectangle. Figure 3-21 shows a rectangle drawn at an angle of 45-degree.

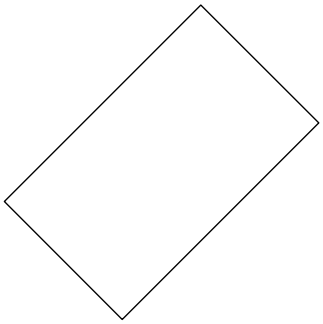


Figure 3-21 *Rectangle drawn at an angle*

Note

Once the rotation angle has been specified, the subsequent rectangles will be drawn at the specified angle. If you do not want to draw the subsequent rectangles at an angle, you need to reset the rotation angle to zero.

You can also set some of the parameters of a rectangle before specifying the start point. These parameters are available as options in the command Prompt and are discussed next.

Chamfer

The **Chamfer** option is used to create a chamfer, which is an angled corner, by specifying the chamfer distances, refer to Figure 3-22. The chamfer is created at all four corners. You can give two different chamfer values to create an unequal chamfer.

*Choose the **Rectangle** tool from the **Draw** panel*
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **C** Enter
Specify first chamfer distance for rectangles <0.0000>: *Enter a value, d*1.
Specify second chamfer distance for rectangles <d1>: *Enter a value, d*2.
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: *Select a point as lower left corner.*
Specify other corner point or [Area/Dimensions/Rotation]: *Select a point as upper right corner.*

Note

The sequence of d1 is calculated clockwise while the sequence of d2 is calculated counterclockwise from the corners of the rectangle.

Fillet

The **Fillet** option is used to create a filleted rectangle, refer to Figure 3-23. You can specify the required fillet radius. The following is the prompt sequence for specifying the fillet:

*Choose the **Rectangle** tool from the **Draw** panel*
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **F** Enter
Specify fillet radius for rectangles <0.0000>: *Enter a value.*
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: *Select a point as lower left corner.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Specify other corner point or [Area/Dimensions/Rotation]: *Select a point as upper right corner.*

Note that the rectangle will be filleted only if the length and width of the rectangle are equal to or greater than twice the value of the specified fillet. Otherwise, AutoCAD will draw a rectangle without fillets.



Note

You can draw a rectangle with chamfers or fillets. If you specify the chamfer distances first and then specify the fillet radius in the same rectangle, the rectangle will be drawn with fillets only.

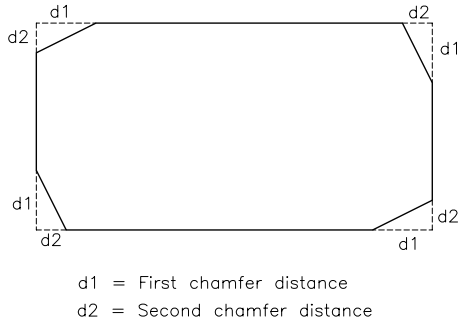


Figure 3-22 *Drawing a rectangle with chamfers*

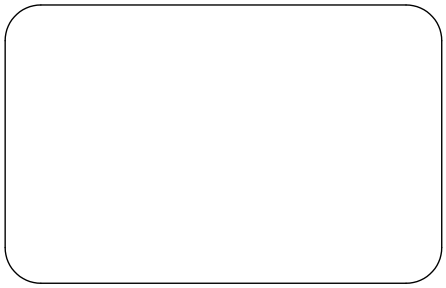


Figure 3-23 *Drawing a rectangle with fillets*

Width

The **Width** option is used to create a rectangle whose line segments have some specified width, as shown in Figure 3-24.

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **W** Enter
Specify line width for rectangles <0.0000>: *Enter a value.*

Thickness

The **Thickness** option is used to draw a rectangle that is extruded in the Z direction by a specified value of thickness. For example, if you draw a rectangle with a thickness of 2 units, you will get a cuboid whose height is 2 units, refer to Figure 3-25. To view the cuboid, choose the **Home** option of the **ViewCube** from the drawing area or choose the **SE Isometric** option from the **Restore View** drop-down of the **Named Views** panel in the **View** tab. To restore the view to the plan view, choose the **Top** option from the **Restore View** drop-down of the **Named Views** panel or choose **Top** from the **ViewCube** in the drawing area.

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **T** Enter
Specify thickness for rectangles <0.0000>: *Enter a value.*

Elevation

The **Elevation** option is used to draw a rectangle at a specified distance from the *XY* plane along the *Z* axis. For example, if the elevation is 2 units, the rectangle will be drawn two units above

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

the *XY* plane. If the thickness of the rectangle is 1 unit, you will get a rectangular box of 1 unit height, located 2 units above the *XY* plane, refer to Figure 3-25.

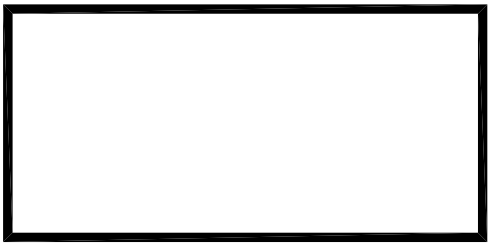


Figure 3-24 *Drawing a rectangle of specified width*

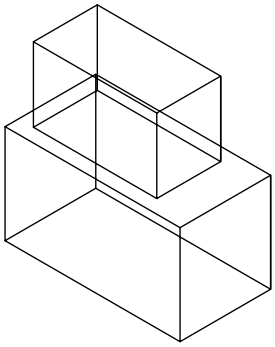


Figure 3-25 *Drawing rectangles with thickness and elevation specified*

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **E** Enter
Specify elevation for rectangles <0.0000>: *Enter a value.*

Note

1. The values you enter for chamfer, fillet, width, elevation, and thickness become the current values for the rectangles drawn subsequently. Therefore, you need to reset the values based on the requirement.

2. A rectangle generated by using the ***Rectangle*** *tool is treated as a single object. To edit the individual lines of the rectangle, you need to explode the rectangle by using the* ***Explode*** *tool from the* ***Modify*** *panel of the* ***Home*** *tab and then edit them.*

Exercise 6 *Rectangle*

Draw a rectangle of length 4 units, width 3 units, and start point at (1,1). Draw another rectangle of length 2 units and width 1 unit, with its first corner at 1.5,1.5, and which is at an angle of 65-degree.

DRAWING ELLIPSES

Ribbon: Home > Draw > Ellipse drop-down **Toolbar:** Draw > Ellipse
Tool Palettes: Draw > Ellipse **Command:** ELLIPSE/EL

If you cut a cone by a cutting plane at an angle and view the cone perpendicular to the cutting plane, the shape created is called an ellipse. An ellipse can be created by using different tools available in the **Ellipse** drop-down of the **Draw** panel, refer to Figure 3-26. In AutoCAD, you can create a true ellipse also known as a NURBS-based (Non-Uniform Rational Bezier Spline) ellipse. A true ellipse has center and quadrant points. If you select it, grips (small blue squares) will be displayed at the center and

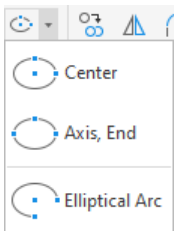


Figure 3-26 *Tools in the* ***Ellipse*** *drop-down of the* ***Draw*** *panel*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

quadrant points of the ellipse. If you move one of the grips located on the perimeter of the ellipse, the size of the ellipse will be changed.

Once you invoke the **Ellipse** tool, the **Specify axis endpoint of ellipse or [Arc/Center]** or **Specify axis endpoint of ellipse or [Arc/Center/Isocircle]** (if isometric snap is ON) prompt will be displayed. The response to this prompt depends on the option you choose. The options are explained next.



Note

*By default, the **Isocircle** option is not available for the **Ellipse** tool. To display this option, you have to select the **Isometric snap** radio button in the **Snap and Grid** tab of the **Drafting Settings** dialog box or choose the **Isometric Drafting** button in the Status Bar.*

Drawing Ellipse Using the Center Option



To draw an ellipse by specifying its center point, endpoint of one of its axes, and endpoint of other axis, choose the **Center** tool from the **Ellipse** drop-down in the **Draw** panel; you will be prompted to specify the center of the ellipse. The center of an ellipse is defined as the point of intersection of the major and minor axes. Specify the center point or enter coordinates; you will be prompted to specify the endpoint. Specify the endpoint of the major or minor axis; you will be prompted to specify the distance of the other axis. Specify the distance; the ellipse will be drawn.

After specifying the endpoint of one axis, you can enter **R** at the **Specify distance to other axis or [Rotation]** prompt to specify the rotation angle around the major axis. In this case, the first axis specified is considered as the major axis. On specifying the rotation angle, the ellipse will be drawn at an angle with respect to the major axis. Note that the rotation angle should range between 0 and 89.4-degrees. Figure 3-27 shows the ellipse created at different rotation angles.

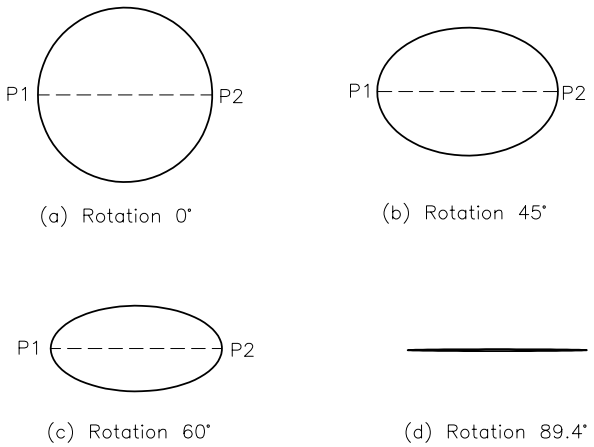


Figure 3-27 *Rotation about the major axis*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Drawing an Ellipse by Specifying its Axis and Endpoint

To draw an ellipse by specifying one of its axes and the endpoint of the other axis, choose the **Axis, End** tool from the **Ellipse** drop-down of the **Draw** panel; you will be prompted to specify the axis end point. Specify the first endpoint of one axis of the ellipse; you will be prompted to specify the other endpoint of the axis. Specify the other endpoint of the axis. Now, you can specify the distance to other axis from the center or specify the rotation angle around the specified axis.

Figure 3-28 shows an ellipse with one endpoint of the axis located at (3,3), the other at (6,3), and the distance to the other axis as 1 unit. Figure 3-29 shows an ellipse with one endpoint of the axis located at (3,3), the other at (4,2), and the distance to the other axis as 2 units.

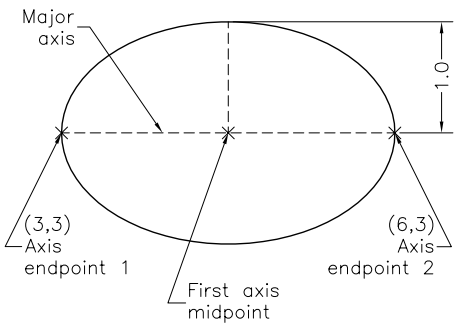


Figure 3-28 *Drawing an ellipse using the Minor axis and end points on the Major Axis*

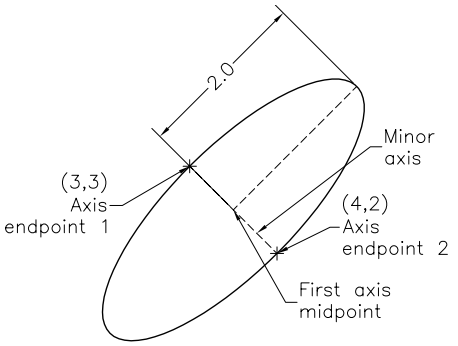


Figure 3-29 *Drawing an ellipse using the Major axis and end points on the Minor Axis*

Exercise 7 *Ellipse*

Draw an ellipse whose major axis is 4 units, minor axis is 3 units, and the rotation around minor axis is 60 degrees. Draw another ellipse with same dimensions and rotation around the major axis as 15 degrees.

Drawing Elliptical Arcs

Ribbon: Home > Draw > Ellipse drop-down > Elliptical Arc **Toolbar:** Draw > Ellipse Arc
Tool Palettes: Draw > Ellipse Arc **Command:** ELLIPSE > Arc

In AutoCAD, you can draw an elliptical arc by choosing the **Elliptical Arc** tool from the **Ellipse** drop-down of the **Draw** panel. On choosing this tool, you can specify the endpoints of one of the axes, the distance to other axis from the center and any one of the following information:

1. Start and End angles of the arc.
2. Start and Included angles of the arc.
3. Start and End parameters.

Remember that in case of elliptical arcs, angles are measured from the first point and in a counterclockwise direction.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

In this section, you will draw an elliptical arc by using the information given below.

a. Start angle = -45, end angle = 135
b. Start angle = -45, included angle = 225
c. Start parameter = @1,0, end parameter = @1<225

Specifying the Start and End Angles of the Elliptical Arc [Figure 3-30(a)]

*Choose the **Elliptical Arc** tool from the **Ellipse** drop-down*
Specify axis endpoint of elliptical arc or [Center]: *Select the first endpoint*
Switch on the Ortho mode by pressing F8, if it is not already chosen
Specify other endpoint of axis: *Select the second point to the left of the first point*
Specify distance to other axis or [Rotation]: *Select a point or enter a distance*
Specify start angle or [Parameter]: **-45** Enter
Specify end angle or [Parameter/Included angle]: **135** Enter *(Angle where arc ends)*

Specifying the Start and Included Angles of the Elliptical Arc [Figure 3-30(b)]

*Choose the **Elliptical Arc** tool from the **Ellipse** drop-dow*n
Specify axis endpoint of elliptical arc or [Center]: *Select the first endpoint*
Specify other endpoint of axis: *Select the second point to the left of the first point*
Specify distance to other axis or [Rotation]: *Select a point or enter a distance*
Specify start angle or [Parameter]: **-45** Enter
Specify end angle or [Parameter/Included angle]: **I** Enter
Specify included angle for arc<current>: **225** Enter *(Included angle)*

Specifying the Start and End Parameters [Figure 3-30(c)]

*Choose the **Elliptical Arc** tool from the **Ellipse** drop-down*
Specify axis endpoint of elliptical arc or [Center]: *Select the first endpoint*
Specify other endpoint of axis: *Select the second endpoint to the left of the first point*
Specify distance to other axis or [Rotation]: *Select a point or enter a distance*
Specify start angle or [Parameter]: **P** Enter
Specify start parameter or [Angle]: **@1,0** Enter
Specify end parameter or [Angle/ Included angle]: **@1<225** Enter

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

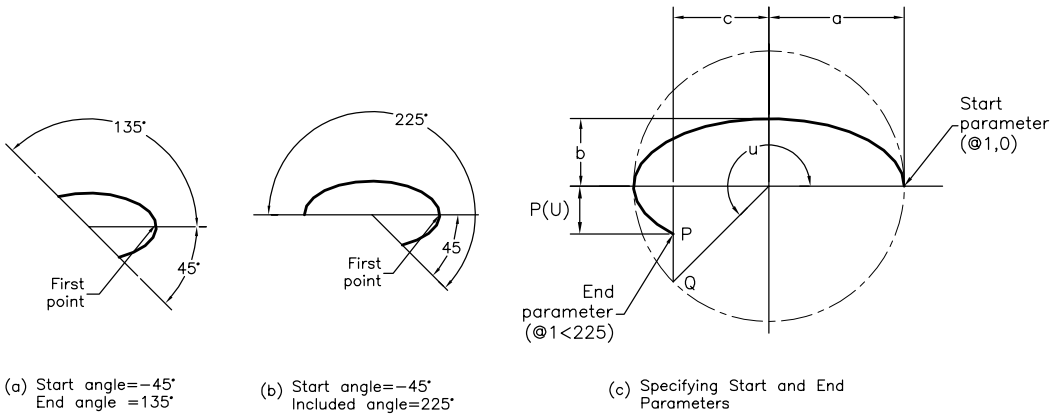


Figure 3-30 *Drawing elliptical arcs*

Exercise 8 — *Elliptical Arc*

a. Construct an ellipse with center at (2,3), axis endpoint at (4,6), and the other axis endpoint at a distance of 0.75 unit from the midpoint of the first axis.

b. Draw the profile shown in Figure 3-31. The distance between the dotted lines is 1.0 unit.

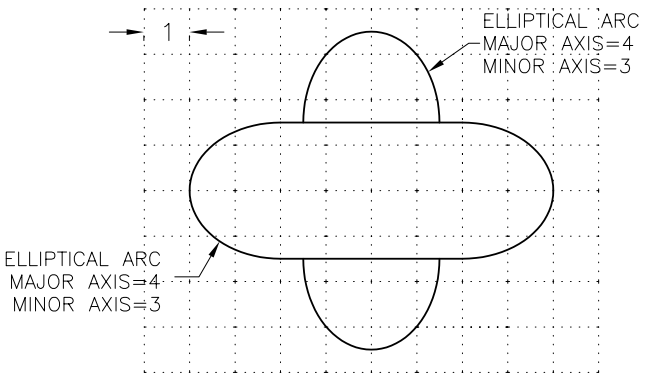


Figure 3-31 *Drawing for Exercise 8*

DRAWING REGULAR POLYGONS

Ribbon: Home > Draw > Rectangle drop-down > Polygon **Toolbar:** Draw > Polygon
Tool Palettes: Draw > Polygon **Command:** POLYGON/POL

A regular polygon is a closed geometric entity with equal sides. The number of sides of a polygon vary from 3 to 1024. For example, a triangle is a three-sided polygon and a pentagon is a five-sided polygon. To draw a regular 2D polygon, choose the **Polygon** tool from the **Rectangle** drop-down of the **Draw** panel; you will be prompted to specify the number of sides. Type the number of sides and press Enter. Now, you can draw the polygon by specifying

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

the length of an edge or by specifying the center of the polygon. Both these methods are discussed next.

Drawing a Polygon by Specifying the Center of Polygon

After you specify the number of sides and press Enter, you will be prompted to specify the center of polygon. Specify the center point; you will be prompted to specify whether the polygon to be drawn is inscribed in a circle or circumscribed about an imaginary circle. A polygon is said to be inscribed when it is drawn inside an imaginary circle such that the vertices of the polygon touch the circle, refer to Figure 3-32. A polygon is said to be circumscribed when it is drawn outside the imaginary circle such that the sides of the polygon are tangent to the circle, refer to Figure 3-33. Type **I** or **C** to draw an inscribed or a circumscribed polygon respectively, and press Enter; you will be prompted to specify the radius. Specify the radius and press Enter; a polygon will be created.

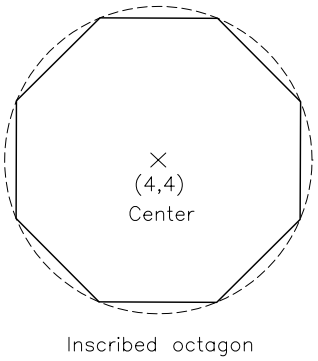


Figure 3-32 *An inscribed polygon*

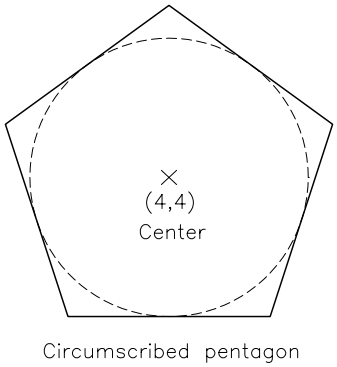


Figure 3-33 *A circumscribed polygon*

Drawing a Polygon by Specifying an Edge

To draw a polygon by specifying the length of an edge, you need to type **E** at the **Specify center of polygon or [Edge]** command prompt and press Enter. Next, specify the first and second endpoints of the edge in succession; the polygon will be drawn in a counterclockwise direction, as shown in Figure 3-34.

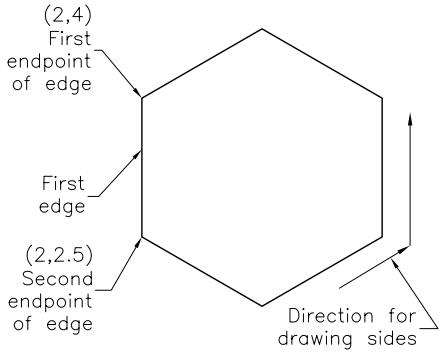


Figure 3-34 *Drawing a polygon (hexagon) using the* ***Edge*** *option*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Exercise 9

Draw a circumscribed polygon of eight sides by using the **Center of Polygon** method.

Exercise 10

Draw a polygon of ten sides by using the **Edge** option and an elliptical arc, as shown in Figure 3-35. The first endpoint of the edge is at (7,1) and the second endpoint is at (8,2).

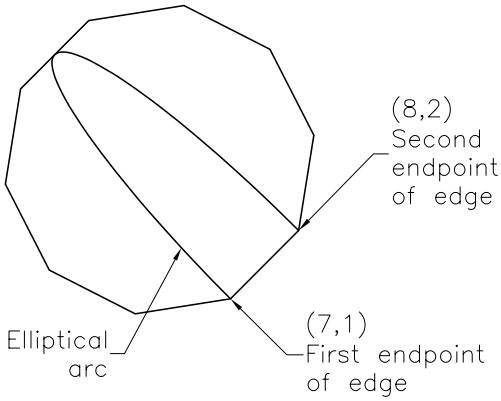


Figure 3-35 *Polygon and elliptical arc for Exercise 10*

DRAWING POLYLINES

Ribbon: Home > Draw > Polyline **Toolbar:** Draw > Polyline
Tool Palettes: Draw > Polyline **Command:** PLINE/PL

Generally, polylines means many lines. Some of the features of a polyline are listed below.

1. Polylines are thick lines with desired width. They are very flexible and can be used to draw any shape, such as a filled circle or a donut.
2. Polylines can be used to draw objects in any linetype (for example, hidden linetype).
3. Polylines are edited by using the advanced editing tools such as the **Edit Polyline** tool from the **Modify** panel of the **Home** tab.
4. A single polyline object can be formed by joining polylines and polyarcs of different thickness.
5. It is easy to determine the area or perimeter of a polyline feature.

To draw a polyline, choose the **Polyline** tool from the **Draw** panel, refer to Figure 3-36. The **Polyline** tool functions fundamentally like the **Line** tool except that some additional options are provided and all segments of the polyline form a single object. After invoking the **Polyline** tool and specifying a start point, the following prompt is displayed.

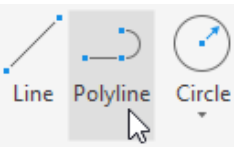


Figure 3-36 *The **Polyline** tool in the **Draw** panel*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Specify start point: *Specify the starting point or enter its coordinates.*
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]:

Note that the message **Current line-width is 0.0000** is displayed automatically indicating that the polyline drawn will have 0.0000 width.

When you are prompted to specify the next point, you can continue specifying the next point and draw a polyline, or depending on your requirements, the other options can be invoked. All these options are discussed next.

Next Point of Line

This is the default set option that is displayed after specifying the start point and is used to specify the next point of the current polyline segment. If additional polyline segments are added to the first polyline, AutoCAD automatically makes the endpoint of the previous polyline segment as the start point of the next polyline segment. The prompt sequence is given next.

*Choose the **Polyline** tool from the **Draw** panel*
Specify start point: *Specify the start point of the polyline.*
Current line-width is 0.0000.
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: *Specify the endpoint of the first polyline segment.*
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: *Specify the endpoint of the second polyline segment, or press Enter to exit the tool.*

Width

To change the current width of a polyline, enter **W** (width option) at the last prompt. You can also right-click and choose the **Width** option from the shortcut menu. On doing so, you will be prompted to specify the width at the start and end of the polyline. Specify the widths of the polyline and press Enter; the polyline will be drawn with the specified width.

The starting width value is taken as the ending width value by default. Therefore, to have a uniform polyline, you need to press Enter at the **Specify ending width <>** prompt. However, if you specify a different value for the ending width, the resulting polyline will be tapered.

For example, to draw a polyline of uniform width 0.25 unit, start point at (4,5), endpoint at (5,5), and the next endpoint at (3,3), use the following prompt sequence:

*Choose the **Polyline** tool from the **Draw** panel*
Specify start point: **4,5** Enter
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **W** Enter
Specify starting width <current>: **0.25** Enter
Specify ending width <0.25>: Enter
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **5,5** Enter
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **3,3** Enter
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: Enter. *Refer to Figure 3-37.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Similarly, to draw a tapered polyline of starting width 0.5 units and an ending width 0.15 units, a start point at (2,4), and an endpoint at (5,4), use the prompt sequence given next.

*Choose the **Polyline** tool from the **Draw** panel*
Specify start point: **2,4** Enter
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **W** Enter
Specify starting width <0.0000>: **0.50** Enter
Specify ending width <0.50>: **0.15** Enter
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **5,4** Enter
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: Enter. *Refer to Figure 3-38.*

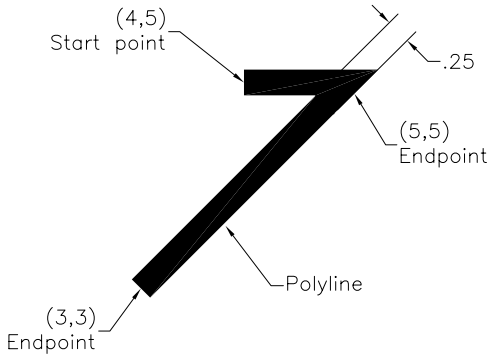


Figure 3-37 *Drawing a uniform polyline using the **Polyline** tool*

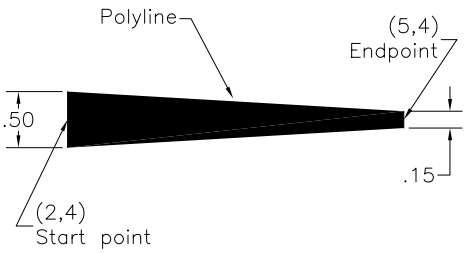


Figure 3-38 *Drawing a tapered polyline using the **Polyline** tool*

Halfwidth

The halfwidth is equal to the half of the actual width of a polyline. This option is invoked by entering **H** or choosing **Halfwidth** from the shortcut menu. The prompt sequence to specify the starting and ending halfwidths of a polyline is given next.

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **H** Enter
Specify starting half-width <0.0000>: **0.12** Enter *(Specify the desired starting halfwidth)*
Specify ending half-width <0.1200>: **0.05** Enter *(Specify the desired ending halfwidth)*

Length

The **Length** option is used to draw a new polyline segment of specified length and at the same angle as the last polyline segment or tangent to the previous polyarc segment. This option is invoked by entering **L** at the following prompt or by choosing **Length** from the shortcut menu.

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **L** Enter
Specify length of line: *Specify the desired length of the Pline*

Undo

This option erases the most recently drawn polyline segment. It can be invoked by entering **U**

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

at the **Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]** prompt. You can use this option repeatedly until you reach the start point of the first polyline segment. If you use this option again, the message **All segments already undone** will be displayed.

Close

This option will be available only when at least one segment of a polyline is drawn. It closes the polyline by drawing a polyline segment from the most recent endpoint to the initial start point.

Arc

This option is used to switch from polylines to polyarcs. You can also set the parameters associated with polyarcs. By default, the arc segment is drawn tangent to the previous segment of the polyline. The direction of the previous line, arc, or polyline segment is the default direction for the polyarc. On invoking the arc option, you need to choose the sub-options to draw the polyarc. Some of the sub-options to draw a polyarc are similar to that of drawing an arc. The sub-options that are different are discussed next.

Angle

To invoke this option, first specify the first point of the polyline to be drawn. Next enter **A**; **Arc** will be selected in the command prompt. Enter **A** again or choose **Angle** from the shortcut menu. The prompts are given next.

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **A**
Specify endpoint of arc (hold Ctrl to switch direction) or
[Angle/CEnter/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **A**
Specify included angle: *Specify the desired angle*
Specify endpoint of arc (hold Ctrl to switch direction) or [CEnter/Radius]: *Specify the desired endpoint* Enter

Center

To invoke this option, first specify the first point of the polyline to be drawn. Next enter **A**; **Arc** will be selected in the command prompt. Enter **C** or choose **Center** from the shortcut menu. The prompts are given next.

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **A**
Specify endpoint of arc (hold Ctrl to switch direction) or
[Angle/CEnter/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **CE**
Specify center point of arc: *Specify the desired center point*
Specify endpoint of arc (hold Ctrl to switch direction) or [Angle/Length]: *Specify the desired endpoint* Enter

CLose

This option will be available only when you specify two or more than two points for creating the polyline segments. To join the start and last points of a polyline in the form of an arc, type **CL** at the **Specify endpoint of arc (hold Ctrl to switch direction) or [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]** command prompt and press Enter; an arc will be created that will close the loop.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Direction

Usually, the arc drawn by using the **Arc** option of the **Polyline** tool is tangent to the previously drawn polyline segment. In other words, the starting direction of the arc depends upon the ending direction of the previous segment. The **Direction** option is used to specify the direction of the tangent for the arc segment to be drawn. You need to specify the direction by specifying a point. The prompts are given next.

Specify tangent direction for the start point of arc: *Specify the direction.*
Specify endpoint of arc (hold Ctrl to switch direction): *Specify the endpoint of arc.*

Halfwidth

The use of this option is similar to the option used for the polyline segment. It is used to specify the starting and ending halfwidths of an arc segment.

Line

This option is used to invoke the **Line** mode again.

Radius

This option is used to specify the radius of the arc segment. The prompt sequence for specifying the radius is given next.

Specify endpoint of arc or [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/
Second pt/Undo/Width]: **R**
Specify radius of arc: *Specify the radius of the arc segment.* Enter
Specify endpoint of arc (hold Ctrl to switch direction) or [Angle]: *Specify the endpoint of arc or choose an option.*

If you specify a point, the arc segment will be drawn. If you use the angle option, you will have to specify the angle and the direction of the chord at the **Specify included angle** and **Specify direction of chord for arc (hold Ctrl to switch direction) <current>** prompts, respectively.

Second pt

This option is used to select the second point of an arc while using the **3-Point arc** option. The prompt sequence is given next.

Specify second point of arc: *Specify the second point on the arc*
Specify end point of arc: *Specify the third point on the arc*

Width

This option is used to enter the width of the arc segment. The prompt sequence for specifying the width is the same as that of the polyline. To draw a tapered arc segment, as shown in Figure 3-39,

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

you need to enter different values at the starting width and ending width prompts.

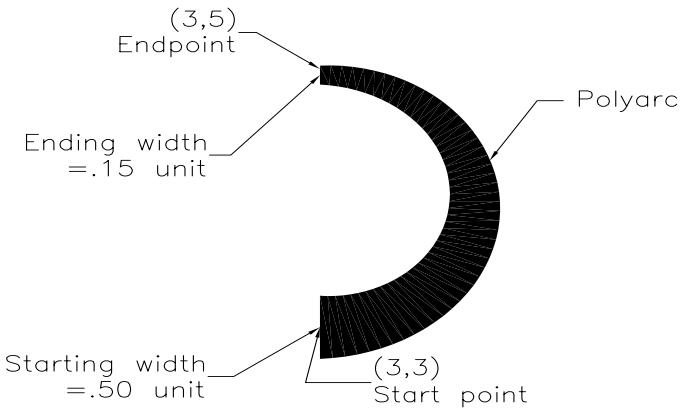


Figure 3-39 *Drawing a polyarc*

The prompt sequence to draw an arc, whose start point is at (3,3), endpoint is at (3,5), starting width is 0.50 unit, and ending width is 0.15 unit, is given next.

*Choose the **Polyline** tool from the **Draw** panel*
Specify start point: **3,3** Enter
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **A** Enter
Specify endpoint of arc (hold Ctrl to switch direction) or [Angle/CEnter/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **W** Enter
Specify starting width <current>: **0.50** Enter
Specify ending width <0.50>: **0.15** Enter
Specify endpoint of arc (hold Ctrl to switch direction) or [Angle/CEnter/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **R** Enter
Specify radius of arc: 1 Enter
Specify endpoint of arc (hold Ctrl to switch direction) or [Angle/CEnter/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **3,5** Enter
Specify endpoint of arc (hold Ctrl to switch direction) or [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: Enter



Tip

*In AutoCAD, you can close an open polyline using the **Fillet** tool (by defining radius as zero). To do so, choose the **Fillet** tool from the **Modify** panel of the **Home** tab and select the two open ends of the polyline. Figure 3-40 shows an open polyline and Figure 3-41 shows closed polyline after selecting both ends.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

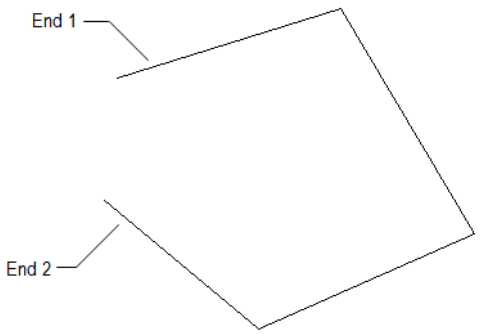


Figure 3-40 *Open polyline with two ends*

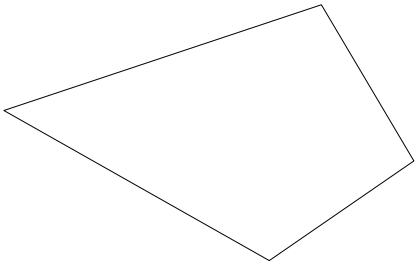


Figure 3-41 *Final closed polyline*

Exercise 11

Draw the objects shown in Figures 3-42 and 3-43 by using the polylines of different widths.

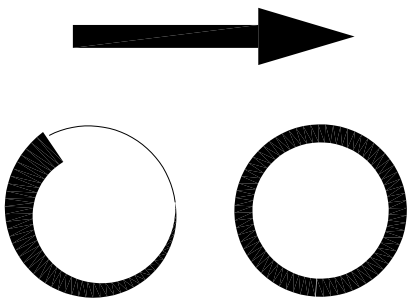


Figure 3-42 *Drawing for Exercise 11*

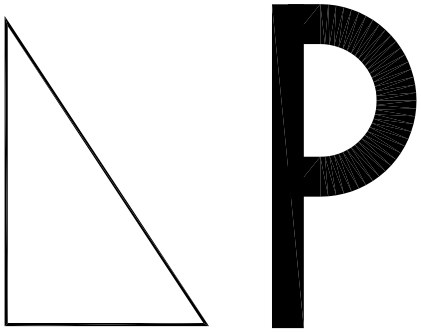


Figure 3-43 *Drawing for Exercise 11*

DRAWING DONUTS

Ribbon: Home > Draw > Donut **Command:** DONUT/DOUGHNUT/DO

In AutoCAD, the **Donut** tool is used to draw an object that looks like a filled circular ring called donut. The donuts in AutoCAD are made up of two semicircular polyarcs with a certain width. Therefore, the **Donut** tool allows you to draw a thick circle. The donuts can have any inside and outside diameters. If **FILLMODE** is off, a donut will look like a circle (if the inside diameter is zero) or a concentric circle (if the inside diameter is not zero). On invoking the **Donut** tool, you will be prompted to specify the diameters. After specifying the two diameters, the donut gets attached to the crosshairs. Specify a point for the center of the donut in the drawing area to place the donut. In this way, you can place as many donuts as required without exiting the tool. Press Enter to exit the tool.

The default values for the inside and outside diameters of donuts are saved in the **DONUTID** and **DONUTOD** system variables. A solid-filled circle is drawn by specifying the inside diameter as zero, if **FILLMODE** is on.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Example 1 *Donut*

You will draw an unfilled donut shown in Figure 3-44 with an inside diameter of 0.75 unit, an outside diameter of 2.0 units, and centered at (2,2). You will also draw a filled donut and a solid-filled donut with the given specifications.

The following is the prompt sequence to draw an unfilled donut shown in Figure 3-44.

Command: **FILLMODE** Enter
New value for FILLMODE <1>: **0** Enter
Command: *Choose the **Donut** tool from the **Draw** panel*
Specify inside diameter of donut<0.5000>: **0.75** Enter
Specify outside diameter of donut <1.000>: **2** Enter
Specify center of donut or <exit>: **2,2** Enter
Specify center of donut or <exit>: Enter

The prompt sequence for drawing a filled donut with an inside diameter of 0.5 unit, outside diameter of 2.0 units, which is centered at a specified point is given below:

Command: **FILLMODE** Enter
Enter new value for FILLMODE <0>: **1** Enter
Command: *Choose the **Donut** tool from the **Draw** panel*
Specify inside diameter of donut<0.5000>: **0.5** Enter
Specify outside diameter of donut <1.000>: **2** Enter
Specify center of donut or <exit>: *Specify a point*
Specify center of donut or <exit>: Enter. *Refer to Figure 3-45*

To draw a solid-filled donut with an outside diameter of 2.0 units, use the following prompt sequence:

*Choose the **Donut** tool from the **Draw** panel*
Specify inside diameter of donut <0.50>: **0** Enter
Specify outside diameter of donut <1.0>: **2** Enter
Specify center of donut or <exit>: *Specify a point*
Specify center of donut or <exit>: Enter. *Refer to Figure 3-46*

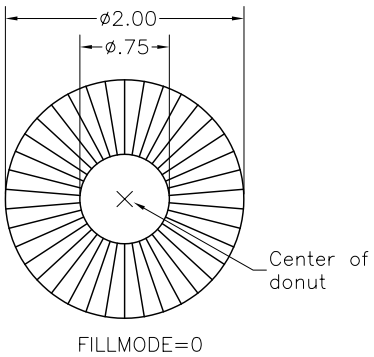


Figure 3-44 *Unfilled donut*

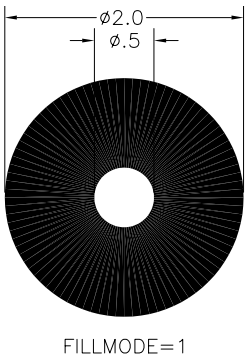


Figure 3-45 *Filled donut*

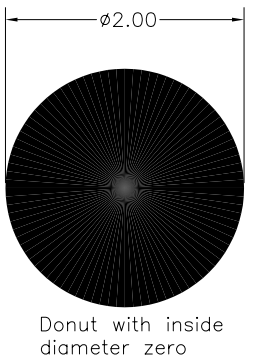


Figure 3-46 *Solid-filled donut*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

PLACING POINTS

A point is one of the basic drawing objects and is specified as a dot (a period). A point is defined as a geometric object that has no dimension and properties, except location. However, in AutoCAD, you can control the size and appearance (style) of a point. You will first learn to change the point style and size and then about various methods to place a point.

Changing the Point Style and Size

Ribbon: Home > Utilities > Point Style **Command:** DDPTYPE or PTYPE

To change the style and size of a point, choose the **Point Style** tool from the **Utilities** panel of the **Home** tab; the **Point Style** dialog box will be displayed, as shown in Figure 3-47. Select any one of the twenty point styles in the **Point Style** dialog box. You can specify the point size as a specified percentage of the drawing area or as an absolute size by selecting the **Set Size Relative to Screen** or **Set Size in Absolute Units** radio button respectively. After selecting the appropriate radio button, enter the point size in the **Point Size** edit box. Choose the **OK** button after specifying the parameters. Now, the points will be drawn according to the selected style and size, until you change the style and size. You can also change the point style and the point size by using the **PDMODE** and **PDSIZE** system variables, respectively. Different values of **PDMODE** and the resulting style are shown in Figure 3-48.

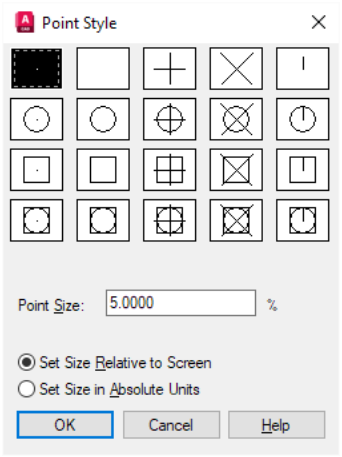


Figure 3-47 *The* ***Point Style*** *dialog box*

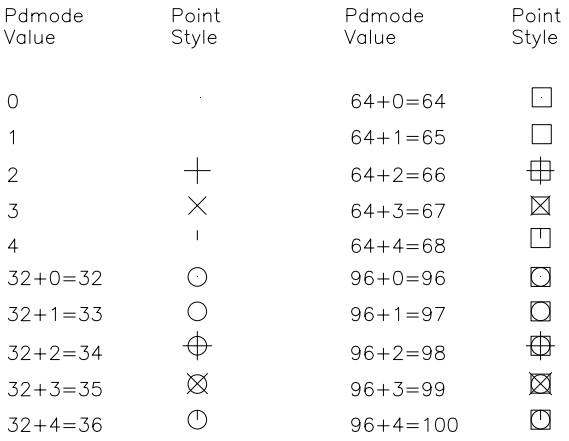


Figure 3-48 *Different point styles for* ***PDMODE*** *values*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Placing Multiple Points

Ribbon: Home > Draw > Multiple Points
Menu Bar: Draw > Point > Multiple Point
Toolbar: Draw > Point
Tool Palettes: Draw > Point

To place points, choose the **Multiple Points** tool from the **Draw** panel, refer to Figure 3-49; the current **PDMODE** and **PDSIZE** values will be displayed and you will be prompted to specify a point. Left-click to place a point.

Next, continue placing as many points as needed and then press Esc to exit the command. You can also enter the coordinate value and place a point at a particular location.

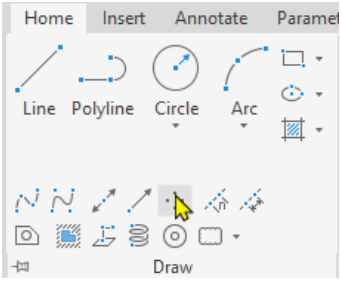


Figure 3-49 *Choosing the* ***Multiple Points*** *tool from the* ***Draw*** *panel*

Placing Points at Equal Distance

Ribbon: Home > Draw > Divide **Command:** DIVIDE/DIV

To place points at an equal distance on an object, choose the **Divide** tool from the **Draw** panel; you will be prompted to select an object to divide. Select an object; you will be prompted to specify the number of segments. Enter the number of segments; the points will be created on the object.

Placing Points at Specified Intervals

Ribbon: Home > Draw > Measure **Command:** MEASURE/ME

You can place points at specified intervals on an object by selecting the object and specifying the length of the segment between two points. To do so, choose the **Measure** tool from the **Draw** panel; you will be prompted to select the object to measure. Select the object; you will be prompted to specify the length of the segment. Specify the length of the segment; the points will be placed at specified intervals.

Exercise 12 *PDMODE and PDSIZE*

a. Try various combinations of the **PDMODE** and **PDSIZE** variables.

b. Check the difference between the points generated by using the negative values of **PDSIZE** and the points generated by using positive values of **PDSIZE**.

DRAWING AN INFINITE LINE

In AutoCAD, you can draw a construction line or a ray that aids in construction or projection. A construction line (xline) is a 3D line that extends to infinity at both ends. As line is infinite in length and does not have any endpoint, Whereas, a ray is a 3D line that extends to infinity at only one end. The other end of the ray is a finite endpoint. The xlines and rays have zero extents. This means that the extents of the drawing will not change if you use the **Zoom All** tool. Most of the object snap modes work with both xlines and rays with some limitations. You cannot

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

use the **Endpoint** object snap with the xline because by definition an xline does not have any endpoints. However, for rays, you can use the **Endpoint** snap on one end only. Also, xlines and rays take the properties of the layer in which they are drawn.



Tip

Sometimes, Xlines and rays when plotted like any other object may create confusion. Therefore, it is recommended to create the construction lines in a different layer altogether so that you can recognize them easily. You will learn about layers in the later chapters.

Drawing Construction Lines

Ribbon: Home > Draw > Construction Line
Tool Palettes: Draw > Construction Line
Toolbar: Draw > Construction Line
Command: XL/XLINE



To draw a construction line that extends to infinity from both sides, invoke the **Construction Line** tool from the **Draw** panel. The prompt sequence that is displayed on choosing this tool is as follows:

Specify a point or [Hor/Ver/Ang/Bisect/Offset]: *Specify an option or select a point through which the xline will pass.*

The various options displayed on invoking the **Construction Line** tool are discussed next.

Point

If you use the default option, you need to specify two points through which the xline will pass. After specifying the first point, move the cursor; a line will be attached to the cursor. On specifying the second point, an xline will be created that passes through the first and second points, refer to Figure 3-50.

Specify a point or [Hor/Ver/Ang/Bisect/Offset]: *Specify a point.*
Specify through point: *Specify the second point.*

You can continue to select more points to create more xlines. All these xlines will pass through the first point you had selected at the **Specify a point** prompt. This point is also called the root point. Right-click or press Enter to exit the tool.

Horizontal

This option is used to create horizontal xlines of infinite length that pass through the selected points. The xlines will be parallel to the *X* axis of the current UCS, refer to Figure 3-51. On invoking this option, a horizontal xline will be attached to the cursor and you will be prompted to select a point through which the horizontal xline will pass. Specify a point. You can continue specifying more points to draw more horizontal xlines. To exit the tool, right-click or press Enter.

Vertical

This option is used to create vertical xlines of infinite length that pass through the selected points. The xlines will be parallel to the *Y* axis of the current UCS, refer to Figure 3-51.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

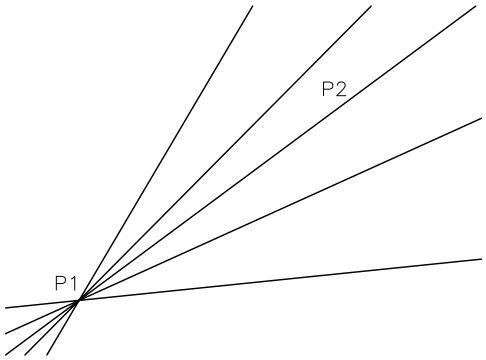


Figure 3-50 *Drawing the xlines*

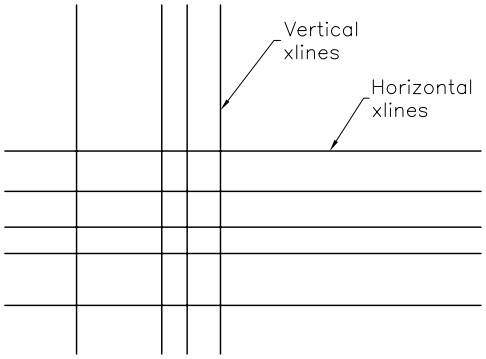


Figure 3-51 *Horizontal and vertical xlines*

Angular

This option is used to create xlines of infinite length that pass through the selected point at a specified angle, refer to Figure 3-52. The angle can be specified by entering a value. You can also invoke the **Reference** option by selecting an object and then specifying an angle relative to it. The **Reference** option is useful when the actual angle is not known, but the angle relative to an existing object can be specified.

*Choose the **Construction Line** tool from the **Draw** panel*
XLINE Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **A** Enter
Enter angle of xline (0) or [Reference]: **R** Enter *(Use the **Reference** method to specify the angle)*
Select a line object: *Select a line.*
Enter angle of xline <0>: *Enter angle (the angle will be measured counterclockwise with respect to the selected line)*
Specify through point: *Specify the second point* Enter.

Bisect

This option is used to create an xline that bisects an angle. In this case, you need to specify the vertex, start point, and endpoint of the angle. The xline will pass through the vertex and bisect the angle specified by selecting two points. The xline created using this option will lie on the plane defined by the selected points. The following is the prompt sequence for this option, refer to Figure 3-53.

*Choose the **Construction Line** tool from the **Draw** panel*
XLINE Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **B** Enter
Specify angle vertex point: *Enter a point (P1).*
Specify angle start point: *Enter a point (P2).*
Specify angle end point: *Enter a point (P3).*
Specify angle end point: *Select more points or press Enter or right-click to end the tool.*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

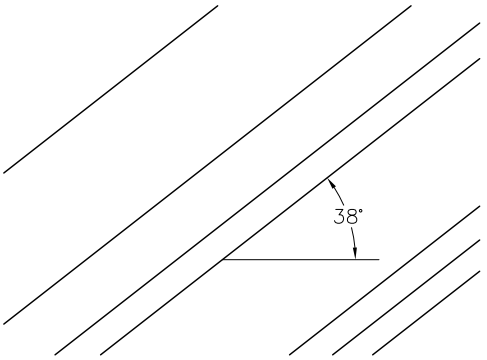


Figure 3-52 *The angular xlines*

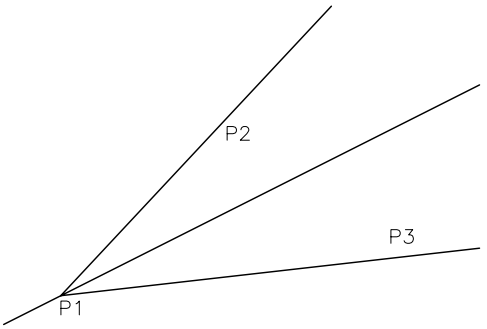


Figure 3-53 *Using the* ***Bisect*** *option to draw xlines*

Offset

The **Offset** option is used to create xlines that are parallel to a selected line/xline at a specified offset distance. You can specify the offset distance by entering a numerical value or by selecting two points on the screen. If you choose the **Through** option, the offset line will pass through the selected point. The following is the prompt sequence for this option:

Choose the ***Construction Line*** *tool from the* ***Draw*** *panel*
XLINE Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **O** Enter
Specify offset distance or [Through] <Through>: *Press Enter to accept the* ***Through*** *option or specify a distance from the selected line object at which the xline shall be drawn.*
Select a line object: *Select the object to which the xline is drawn parallel and at a specified distance.*
Specify through point: *Select a point through which the xline should pass.*

After specifying the offset distance and selecting a line object, you need to specify the direction in which the xline has to be offset. You can continue drawing construction lines or right-click or press Enter to exit the tool.

Drawing Ray

Ribbon: Home > Draw > Ray **Command:** RAY

A ray is a 3D line similar to the xline with the difference being that it extends to infinity only in one direction. It starts from a specified point and extends to infinity through the specified point. The prompt sequence is given next.

Specify start point: *Select the start point for the ray.*
Specify through point: *Specify the second point.*

Press Enter or right-click to exit the tool.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

WRITING A SINGLE LINE TEXT

Ribbon: Home > Annotation > Multiline Text drop-down > Single Line
Command: DT/TEXT

A The **Single Line** tool is used to write a single line text. Although you can write more than one line of text using this command, but each line will be a separate text entity. To write text, choose the **Single Line** tool from **Home > Annotation > Multiline Text** drop-down. This tool is also available in the **Multiline Text** drop-down available in the **Text** panel of the **Annotate** tab. After invoking this tool, you need to specify the start point for the text. Next, you need to specify the text height and the rotation angle. As you enter the characters, they start appearing on the screen. After typing a line if you press Enter, the cursor will be automatically placed at the start of the next line and the prompt for entering another line will be repeated. Type another line or press Enter again to exit the tool. You can use the Backspace key to edit the text on the screen while writing it. The prompt sequence is given next.

*Choose the **Single Line** tool from the **Annotation** panel*
Current text style: "Standard" Text height: 0.2000 Annotative: No Justify: Left
Specify start point of text or [Justify/Style]: *Specify the starting point of the text.*
Specify height<current>: *Enter the text height.*
Specify rotation angle of text <0>: Enter
Enter the first line of the text in the text box displayed in the drawing window.
Enter the second line of the text in the text box displayed in the drawing window.
Press Enter twice to exit.



Note
The other tools to enter text are discussed in detail in Chapter 7.

WORKING WITH TEMPLATES

In AutoCAD, you can use various templates to make your drawings. AutoCAD comes with standard drawing templates such as *acad.dwt, acadiso.dwt, acad3D.dwt, acadiso3D.dwt, acad-Named Plot Styles.dwt, acadISO-Named Plot Styles.dwt,* and so on. The ISO template drawings are based on the drawing standards developed by ISO (International Organization for Standardization). To use these template drawings, select the required template from the **New** drop-down in the **Start** tab, refer to Figure 3-54. You will learn more about templates in Chapter 14.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

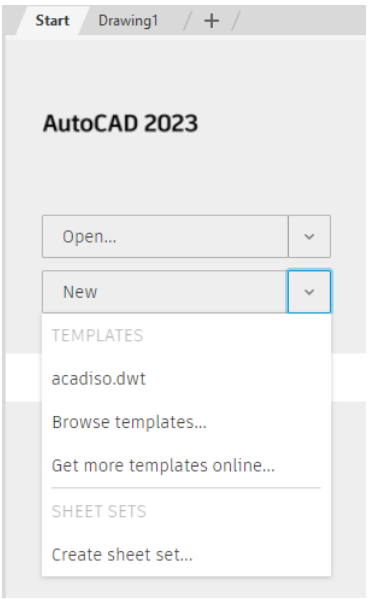


Figure 3-54 *Various templates in the* ***Templates*** *drop-down*

Self-Evaluation Test

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

1. The _____ option of the **Rectangle** tool is used to draw a rectangle at a specified distance from the XY plane along the Z axis.

2. If the **FILLMODE** is set to _____, only the outlines for the new polyline will be drawn.

3. You can get a _____ polyline by entering two different values at the starting width and ending width prompts.

4. Choose the _____ tool from the **Draw** panel to place points at specified intervals on an object.

5. Choose the _____ tool from the **Draw** panel to draw as many points as you want in a single command.

6. The size of a point will be taken as a percentage of the viewport size if you enter a _____ value for the **PDSIZE** variable.

7. While drawing an arc by choosing the **Start, Center, Angle** tool if you enter the included angle with a negative value, an arc will be drawn in a clockwise direction. (T/F)

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

8. After choosing the **Arc** tool if you press Enter instead of specifying the start point, the start point and direction of the arc will be taken from the endpoint and ending direction of the previous line or the arc drawn. (T/F)

9. By using the **Single Line** tool, you can write more than one line of text. (T/F)

10. The start and end parameters of an elliptical arc are determined by specifying a point on the circle whose diameter is equal to the minor diameter of an ellipse. (T/F)

Review Questions

Answer the following questions:

1. On drawing an arc by choosing the **Start, Center, Length** tool, a positive chord length generates the smallest possible arc (minor arc), and the arc is always less than:

   (a) 90 degree (b) 180 degree
   (c) 270 degree (d) 360 degree

2. Which of the following options of the **Rectangle** tool is used to draw a rectangle that is extruded in the Z direction by a specified value?

   (a) **Elevation** (b) **Thickness**
   (c) **Extrude** (d) **Width**

3. Which of the following tools should be used to draw a line in 3D space that starts from a specified start point and the other end extends to infinity?

   (a) **Polyline** (b) **Ray**
   (c) **Construction Line** (d) **Multiline Text**

4. A polygon is said to be _____ when it is drawn inside an imaginary circle and its vertices touch the circle.

5. If additional polyline segments are added to the first polyline, the _____ of the first polyline segment becomes the start point of the next polyline segment.

6. By using the **Donut** tool, you can draw a solid-filled circle by specifying the inside diameter as _____ and keeping the **FILLMODE** on.

7. The _____ option of the **Construction Line** tool is used to create construction lines of infinite length that are parallel to the *Y* axis of the current UCS.

8. You can use the _____ key to edit text on the screen while writing it using the **Single Line** tool.

9. While drawing an arc by choosing the **Start, End, Angle** tool, a negative included angle value draws the arc in clockwise direction. (T/F)

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

10. When the **Continue** option of the **Arc** tool is used to draw arcs, each successive arc is perpendicular to the previous one. (T/F)

11. If you specify the chamfer distances first and then specify the fillet radius of the same **Rectangle** tool, the rectangle will be drawn with chamfers only. (T/F)

12. A rectangle drawn using the **Rectangle** tool is treated as a combination of different objects, therefore, the individual sides can be edited independently. (T/F)

Exercise 13 *Arc*

Draw the sketch shown in Figure 3-55. The distance between the dotted lines is 1.0 unit. Create arcs by choosing appropriate tools from the **Arc** drop-down.

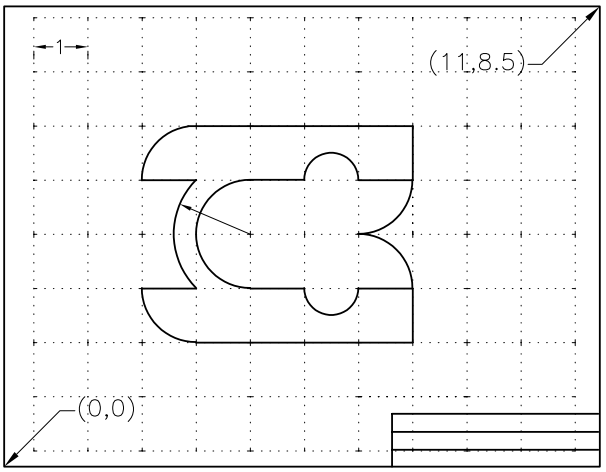


Figure 3-55 *Drawing for Exercise 13*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Exercise 14 *Ellipse*

Draw the sketch shown in Figure 3-56. The distance between the dotted lines is 0.5 unit. Create ellipses using the tools in the **Ellipse** drop-down.

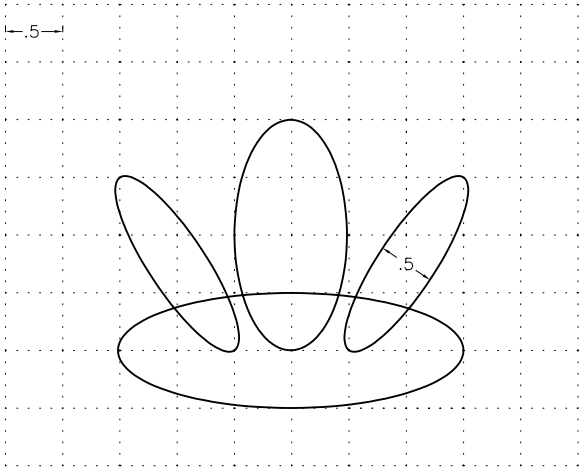


Figure 3-56 *Drawing for Exercise 14*

Exercise 15

Draw the sketch shown in Figure 3-57 using the **Line**, **Circle**, and **Arc** tools. The distance between the dotted lines is 1.0 unit and the diameter of the circles is 1.0 unit.

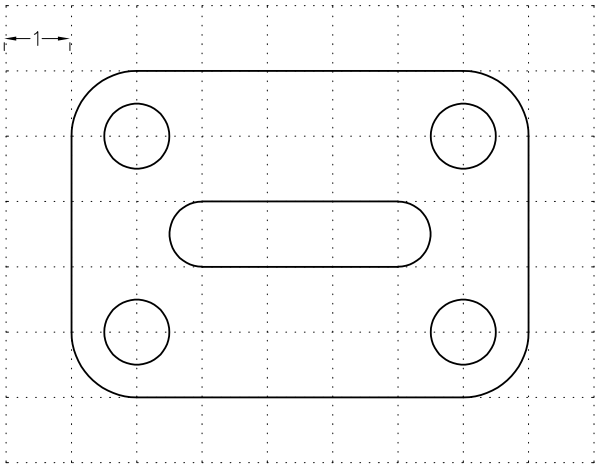


Figure 3-57 *Drawing for Exercise 15*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Exercise 16

Draw the sketch shown in Figure 3-58 using the **Line**, **Circle**, and **Arc** tools or their options. The distance between the grid lines is 1.0 unit and the diameter of the circle is 1.0 unit.

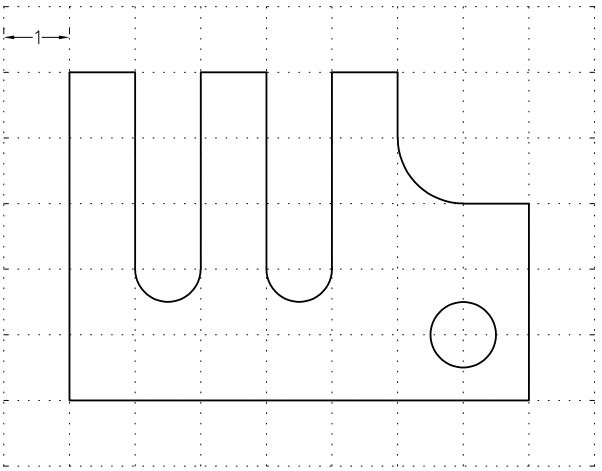


Figure 3-58 *Drawing for Exercise 16*

Exercise 17

Draw the object shown in Figure 3-59. The distance between the dotted lines is 5 inches. Determine the limits for this drawing and use the Fractional units with 0 1/16 precision.

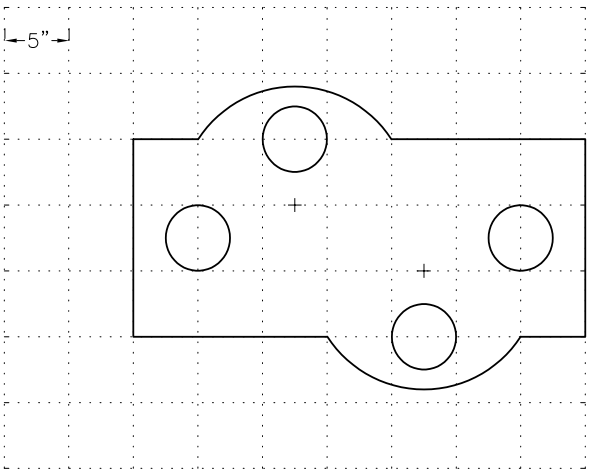


Figure 3-59 *Drawing for Exercise 17*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Problem-Solving Exercise 1 *Arc*

Draw the sketch shown in Figure 3-60. Next, create arcs by using the **Arc** tools indicated in the drawing. (Use the @ symbol to snap to the previous point. Example: Specify start point of arc or [Center]: @)

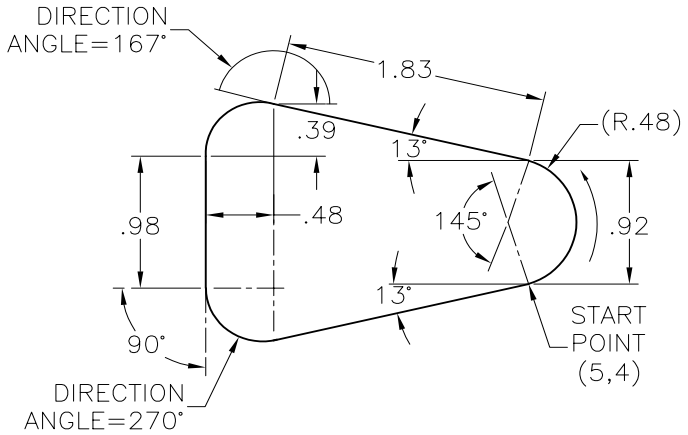


Figure 3-60 *Drawing for Problem-Solving Exercise 1*

Problem-Solving Exercise 2 *Arc*

Draw the sketch shown in Figure 3-61. Create arcs by using the **Arc** tools. The distance between the dotted lines is 0.5 unit.

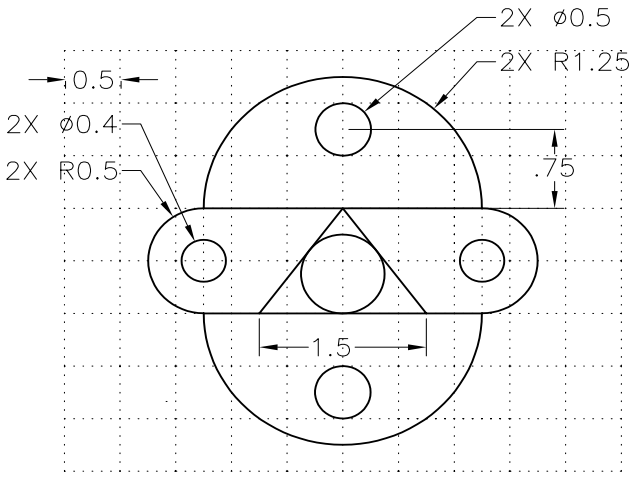


Figure 3-61 *Drawing for Problem-Solving Exercise 2*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Problem-Solving Exercise 3 *Arc*

Draw the sketch shown in Figure 3-62 by using different tools in the **Draw** panel. Note, Sin30=0.5 and Sin60=0.866. The distance between the dotted lines is 1 unit.

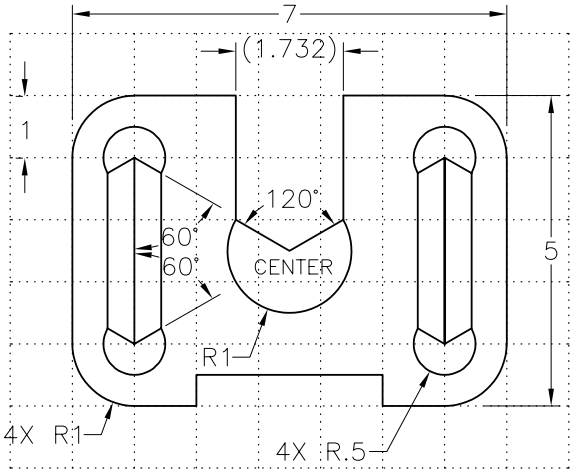


Figure 3-62 *Drawing for Problem-Solving Exercise 3*

Problem-Solving Exercise 4

Draw the sketch shown in Figure 3-63 using the **Polygon**, **Circle**, and **Line** tools.

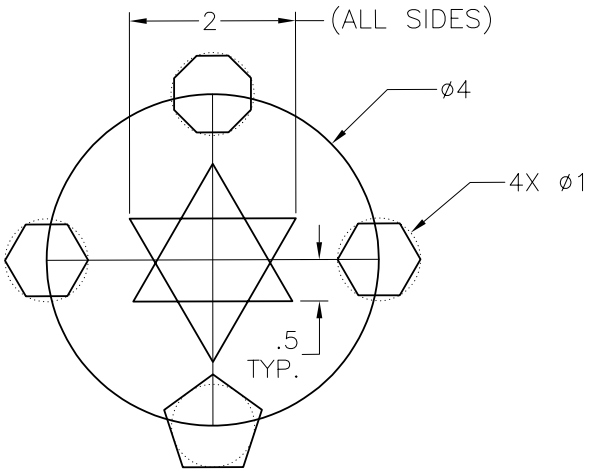


Figure 3-63 *Drawing for Problem-Solving Exercise 4*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Problem-Solving Exercise 5

Draw the sketch shown in Figure 3-64 by using different tools in the **Draw** panel. Also, draw the hidden lines and center lines as continuous lines. Note that the dimensions are given only for your reference.

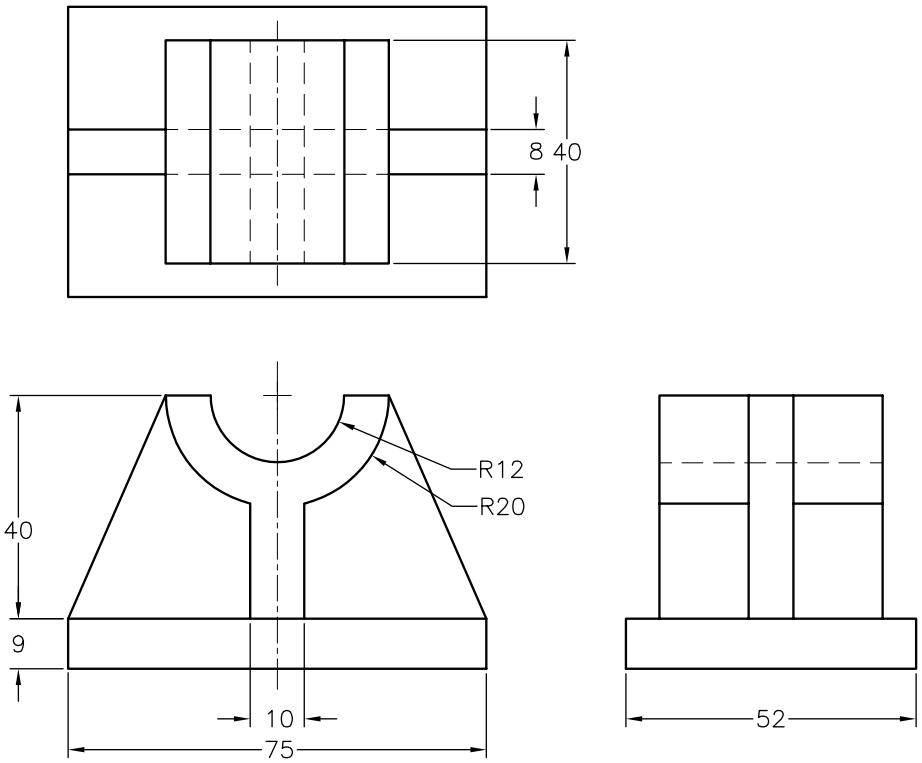


Figure 3-64 *Drawing for the Problem-Solving Exercise 5*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Problem-Solving Exercise 6

Draw the sketch shown in Figure 3-65 in the paper space area of the **Manufacturing Metric** template. Note that the dimensions are given only for your reference.

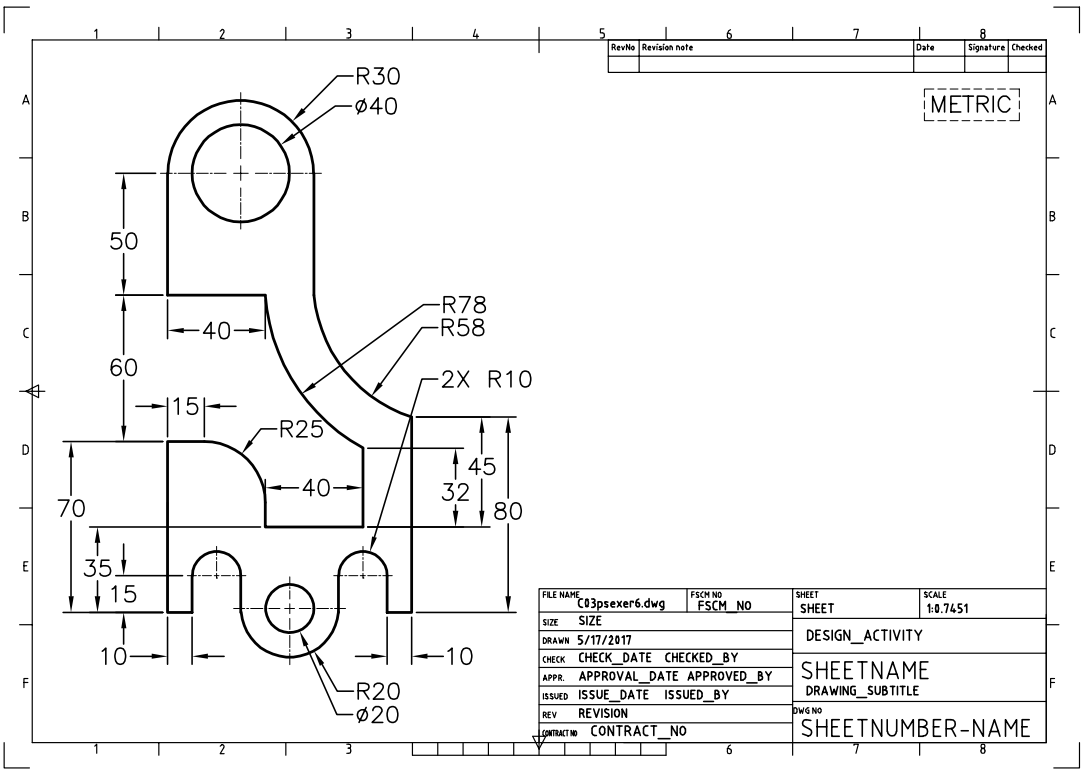


Figure 3-65 *Drawing for the Problem-Solving Exercise 6*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Problem-Solving Exercise 7

Draw the sketch shown in Figure 3-66.

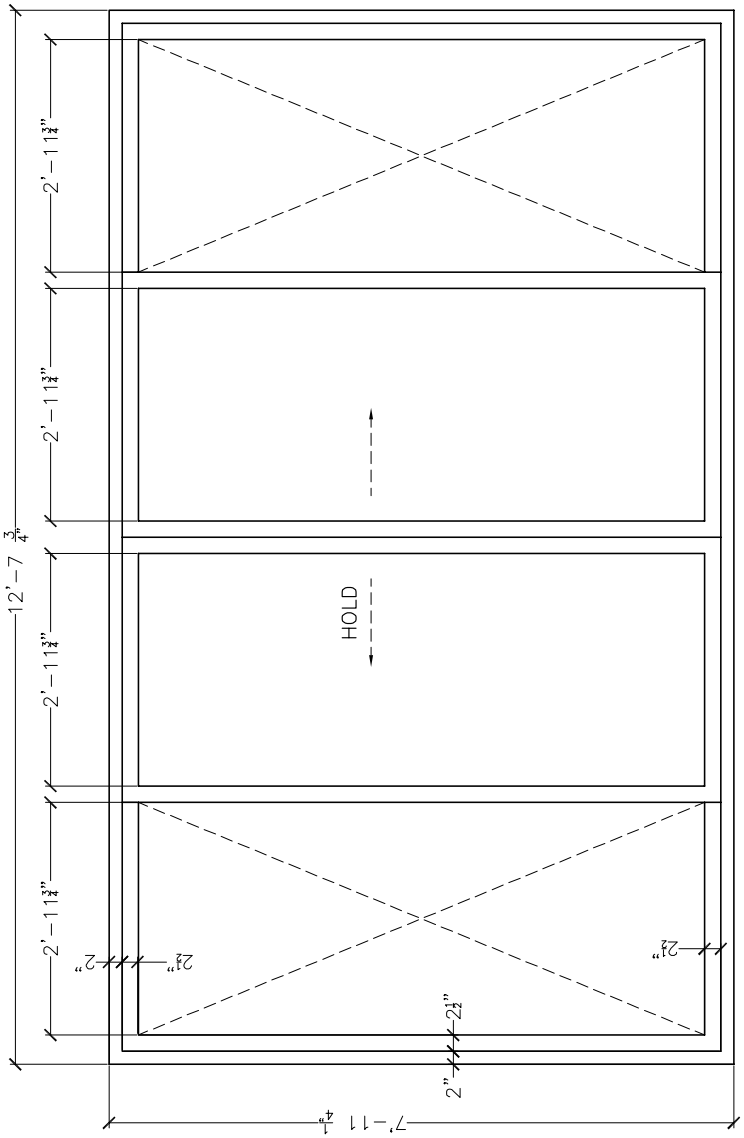


Figure 3-66 *Drawing for the Problem-Solving Exercise 7*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Answers to Self-Evaluation Test

1. Elevation, **2.** 0, **3.** Tapered, **4. Measure**, **5. Multiple Point**, **6.** Negative, **7.** T, **8.** T, **9.** T, **10.** F

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com