



# Chapter 3

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## Starting with the Advanced Sketching

### Learning Objectives

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

- Draw arcs using various options.
- Draw rectangles, ellipses, and elliptical arcs.
- Draw polygons such as hexagons and pentagons.
- Draw polylines and donuts.
- Draw points and change point style and point size.
- Draw infinite lines and create simple text.

## DRAWING ARCS

<b>Ribbon:</b>	Home > Draw > Arc
<b>Menu Browser:</b>	Draw > Arc
<b>Toolbar:</b>	Draw > Arc
<b>Command:</b>	ARC or A

An arc is defined as a part of a circle. In AutoCAD LT, it can be drawn using the **ARC** command. You can invoke the **ARC** command from the **Draw** toolbar, as shown in Figure 3-1. AutoCAD LT provides eleven different options to draw an arc. To view these options, click on the down arrow on the **3-Point** button on the **Draw** panel of the **Home** tab in the **Ribbon**; a flyout will be displayed, as shown in Figure 3-2. The default option for drawing an arc is the **3-Point** option. Other options can be invoked by entering the appropriate letter in the command window or by right-clicking and choosing the appropriate option from the flyout. The option that was used last will be displayed in the **Draw** panel. Also, the last parameter specified in any arc generation is automatically dragged into the relevant location. This is dependent on the **DRAGMODE** variable, which should be set to **Auto** (default). The various options to draw the Arcs are discussed next.

### The 3-Point Option



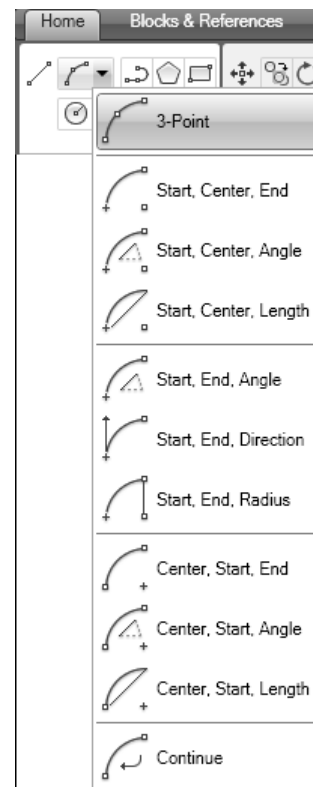
When you choose the **Arc** button from the **Draw** toolbar, or enter **ARC** at the Command prompt, the **3-Point** option automatically gets activated. You can also choose the **3-Point** button from the **Draw** panel of the **Home** tab in the **Ribbon**. This option requires you to specify the start point, second point, and endpoint of the arc, see Figure 3-3. The arc can be drawn in a clockwise or counterclockwise direction by dragging the arc with the cursor. The following is the prompt sequence to draw an arc with a start point at (2,2), second point at (3,3), and an endpoint at (3,4). (You can also specify them on the screen by moving the cursor.)

Command: Turn off the dynamic input and then choose the **3-Point** button from the **Draw** panel in the **Ribbon** (**3-Point** is the default option).

\_arc Specify start point of arc or [Center]: 2,2   
Specify second point of arc or [Center/End]: 3,3   
Specify end point of arc: 3,4



**Figure 3-1** Invoking the **ARC** command from the **Draw** toolbar



**Figure 3-2** The options in the **Ribbon** for drawing an arc

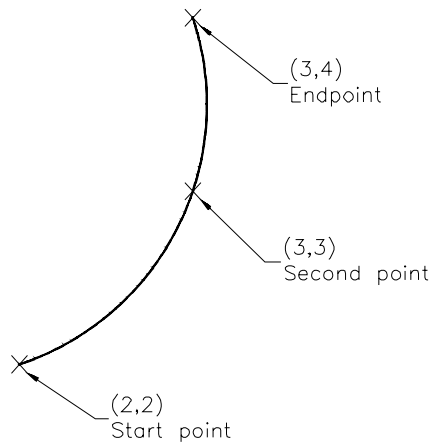


Figure 3-3 Drawing an arc using the **3-Point** option

## Exercise 1

General

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

## The Start, Center, End Option

This option is slightly different from the **3-Point** option. In this option, instead of entering the second point, you need to enter the center of the arc. Choose this option, when you know the start point, endpoint, and center point of the arc. The arc is drawn in a counterclockwise direction from the start point to the endpoint around the specified center. The endpoint specified need not be on the arc and is used only to calculate the angle at which the arc ends. The radius of the arc is determined by the distance between the center point and start point. The prompt sequence for drawing an arc, shown in Figure 3-4, with a start point at (3,2), center point at (2,2), and endpoint at (2,3.5), is given next.

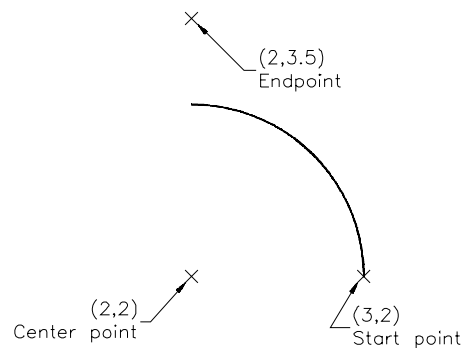


Figure 3-4 Drawing an arc using the **Start, Center, End** option

Command: Turn off the dynamic input and then enter **ARC**

\_arc Specify start point of arc or [Center]: **3,2**

Specify second point of arc or [Center/End]: **C**

Specify center point of arc: **2,2**

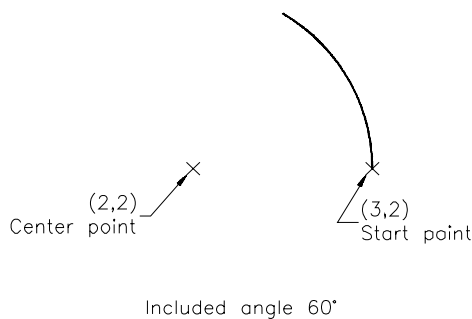
Specify end point of the arc or [Angle/chord Length]: **2,3.5**

**Note**

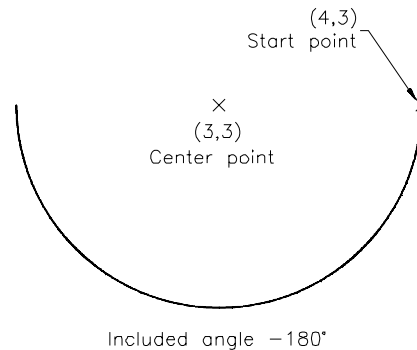
If you choose **Draw > 3-Point > Start, Center, End** from the **Home** tab of the **Ribbon**, the **Center** option is automatically invoked. This way you bypass the prompt **Specify second point of arc or [Center/End]**. Therefore, you need to specify only the center point.

## The Start, Center, Angle Option

This option is the best choice, if you know the **included angle** of the arc. The included angle is the angle formed by the start and endpoint of the arc with the specified center. This option draws an arc in a counterclockwise direction with the specified center and start point spanning the indicated angle, see Figure 3-5. If the specified angle is negative, the arc is drawn in a clockwise direction, see Figure 3-6.



**Figure 3-5** Drawing an arc using the **Start, Center, Angle** option



**Figure 3-6** Drawing an arc using a negative angle in the **Start, Center, Angle** option

The prompt sequence for drawing an arc with center at (2,2), start point at (3,2), and an included angle of 60-degree, as shown in Figure 3-5, is given next.

Command: Turn off the dynamic input and then enter **ARC**.

Specify start point of arc or [Center]: **3,2**

Specify second point of arc or [Center/End]: **C**

Specify center point of arc: **2,2**

Specify end point of the arc or [Angle/chord Length]: **A**

Specify included angle: **60**

You can draw arcs with negative angle values in the **Start, Center, Included Angle** (St,C,Ang) option by entering “-” (negative sign), followed by the required angle at the **Specify included angle** prompt (Figure 3-6). The following prompt sequence is displayed, when you invoke the **ARC** command:

\_arc Specify start point of arc or [Center]: **4,3**

Specify second point of arc or [Center/End]: **C**

Specify center point of arc: **3,3**

Specify end point of the arc or [Angle/ chord Length]: **A**

Specify included angle: **-180**

**Note**

If you choose **Draw > 3-Point > Start, Center, Angle** from the **Home** tab of the **Ribbon**, you need to specify the start point, the center point, and the angle.

**Exercise 2***Mechanical*

- Draw an arc using the **St, C, Ang** option. The start point is (6,3), the center point is (3,3), and the angle is 240-degree.
- Make the drawing shown in Figure 3-7. The distance between the dotted lines is 1.0 unit. Create the radii, by using the arc command options indicated in the drawing.

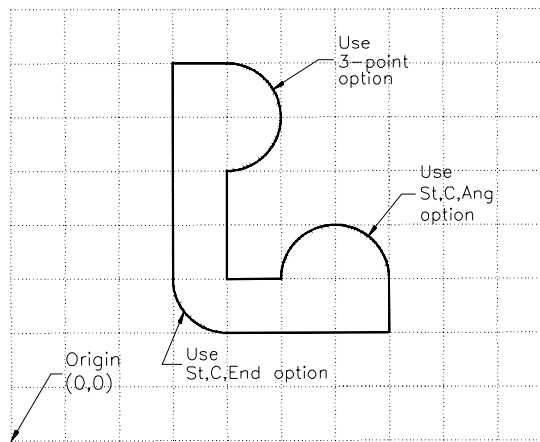


Figure 3-7 Drawing for Exercise 2

**The Start, Center, Length Option**

In this option, you are required to specify the start point, center point, and length of the chord. A chord is defined as the straight line connecting the start point and endpoint of an arc. The chord length needs to be specified so that AutoCAD LT can calculate the ending angle. Identical start, center, and chord length specifications can be used to define four different arcs. AutoCAD LT always draws this type of arc counterclockwise from the start point. Therefore, a positive chord length gives the smallest possible arc with that length. This arc is known as the minor arc. The minor arc is less than 180-degree. A negative value for the chord length results in the largest possible arc, also known as the major arc. The chord length can be determined, by using the standard chord length tables or the mathematical relation ( $L = 2 * \text{Sqrt} [h(2r-h)]$ ). For example, an arc of radius 1 unit, with an included angle of 30-degree, has a chord length of 0.51764 units. The prompt sequence for drawing an arc, shown in Figure 3-8 that has a start point (3,1), center (2,2), and the chord length of 2 units is given next.

Command: Turn off the dynamic input and choose **Draw > 3-Point > Start, Center, Length** from the **Home** tab of the **Ribbon**.

\_arc Specify start point of arc or [Center]: **3,1**

Specify second point of arc or [Center/End]: **\_c** Specify center point of arc: **2,2**

Specify end point of the arc or [Angle/ chord Length]: **\_l** Specify length of chord: **2**

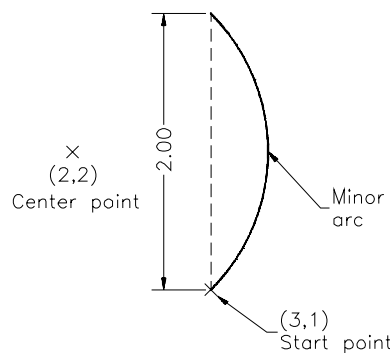
You can draw the major arc by defining the length of the chord as negative, see Figure 3-9. In this case, the arc with a start point at (3,1), center point at (2,2), and a negative chord length of -2, is drawn with the following prompt sequence.

Command: Choose **Draw > 3-Point > Start, Center, Length** from the **Home** tab of the **Ribbon**

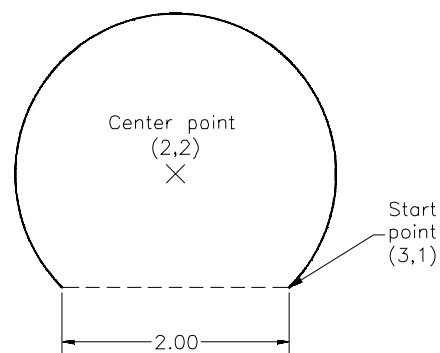
\_arc Specify start point of arc or [Center]: **3,1**

Specify second point of arc or [Center/End]: **\_c** Specify center point of arc: **2,2**

Specify end point of the arc or [Angle/ chord Length]: **\_l** Specify length of chord: **-2**



**Figure 3-8** Drawing an arc using the **Start, Center, Length** option



**Figure 3-9** Drawing an arc using a negative chord length in the **Start, Center, Length** option



#### Note

The points that have been specified for drawing the arcs using different options are quite close to each other and therefore the arcs created might overlap each other. You can use the **ERASE** command to erase some of the previously drawn arcs and gain space to draw new arcs.

### Exercise 3

General

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

### The Start, End, Angle Option

With this option, you can draw an arc by specifying the start point of the arc, endpoint, and the included angle. A positive included angle value draws an arc in a counterclockwise direction from the start point to the endpoint, spanning the included angle; a negative included angle value draws the arc in a clockwise direction, as shown in Figure 3-10. The prompt sequence for drawing an arc with a start point at (3,2), endpoint at (2,4), and included angle of 120-degree is given next.

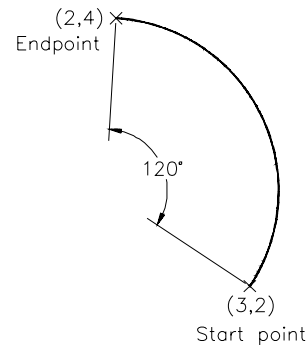
Command: Turn off the dynamic input and choose **Draw > 3-Point > Start, End, Angle** from the **Home** tab of the **Ribbon**.

\_arc Specify start point of arc or [Center]: **3,2**

Specify second point of arc or [Center/End]: **\_e**

Specify end point of arc: **2,4**

Specify center point of arc or [Angle/Direction/Radius]: **\_a** Specify included angle: **120**



### The Start, End, Direction Option

**Figure 3-10** Drawing an arc using the **Start, End, Angle** option

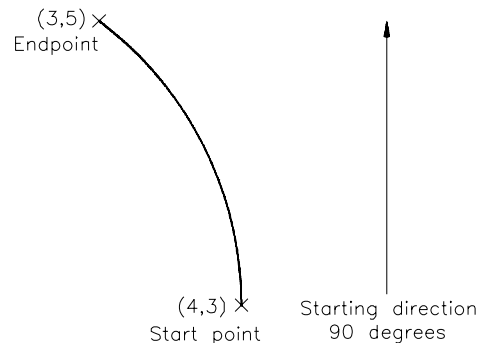
In this option, you can draw an arc by specifying the start point, endpoint, and starting direction of the arc, in degrees. In other words, the arc starts in the direction you specify (the start of the arc is established tangent to the direction you specify). This option can be used to draw a major or minor arc, in a clockwise or counterclockwise direction, whose size and position are determined by the distance between the start point and endpoint and the direction specified. To illustrate the positive direction option shown in Figure 3-11, the prompt sequence for an arc with a start point at (4,3), endpoint at (3,5), and direction of 90-degree is given next.

Command: Turn off the dynamic input and choose **Draw > 3-Point > Start, End, Direction** from the **Home** tab of the **Ribbon**.

\_arc Specify start point of arc or [Center]: **4,3**

Specify second point of arc or [Center/End]: **\_e** Specify end point of arc: **3,5**

Specify center point of arc or [Angle/Direction/Radius]: **\_d** Specify tangent direction for the start point of arc: **90**



**Figure 3-11** Drawing an arc using the **Start, End, Direction** option

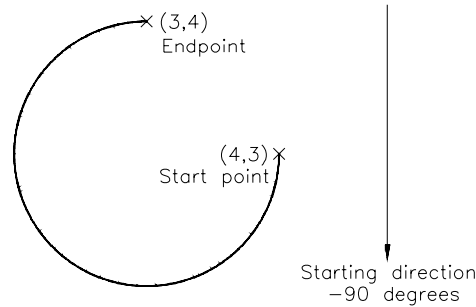
To illustrate the option of using a negative direction degree specification, shown in Figure 3-12, the prompt sequence for an arc with a start point at (4,3), endpoint at (3,4), and direction of -90-degree, is given next.

Command: Choose **Draw > 3-Point > Start, End, Direction** from the **Home** tab of the **Ribbon**.

\_arc Specify start point of arc or [Center]: **4,3**

Specify second point of arc or [Center/End]: **\_e** Specify end point of arc: **3,4**

Specify center point of arc or [Angle/Direction/Radius]: **\_d** Specify tangent direction for the start point of arc: **-90**



**Figure 3-12** Drawing an arc using a negative direction in the **Start, End, Direction** option



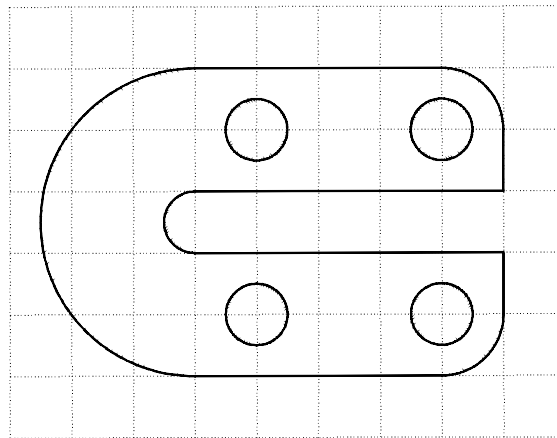
**Note**

With the **Start, End, Direction** option, if you do not specify a start point but press **ENTER** at the **Specify start point of arc or [Center]** prompt, the start point and direction of the arc will be taken from the **endpoint and ending direction** of the previous line or arc drawn on the current screen. You are then required to specify only the endpoint of the arc.

## Exercise 4

*Mechanical*

- Specify the directions and coordinates of two arcs such that they form a circular figure.
- Make the drawing shown in Figure 3-13. Create the curves using the **ARC** command. The distance between the dotted lines is 1.0 unit and the diameter of the circles is 1 unit.

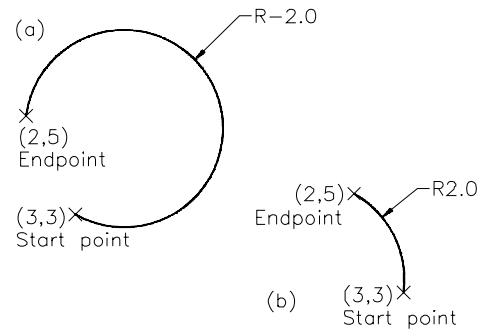


**Figure 3-13** Drawing for Exercise 4



## The Start, End, Radius Option

This option is used when you know the start point, endpoint, and radius of the arc. The same values for the three variables (start point, endpoint, and radius) can result in four different arcs. AutoCAD LT resolves this by always drawing this type of arc in a counterclockwise direction from the start point. Therefore, a negative radius value results in a major arc (the largest arc between two endpoints), Figure 3-14(a), while a positive radius value results in a minor arc (smallest arc between the start point and endpoint), Figure 3-14(b). The prompt sequence to draw a major arc with a start point at (3,3), endpoint at (2,5), and radius of -2, Figure 3-14(a), is given next.



**Figure 3-14** Drawing an arc using the **Start, End, Radius** option

Command: Turn off the dynamic input and choose **3-Point > Start, End, Radius** from the **Draw** panel of the **Ribbon**.

\_arc Specify start point of arc or [Center]: **3,3**

Specify second point of arc or [Center/End]: **\_e** Specify end point of arc: **2,5**

Specify center point of arc or [Angle/Direction/Radius]: **\_r** Specify radius of arc: **-2**

The prompt sequence to draw a minor arc with its start point at (3,3), endpoint at (2,5), and radius as 2, as shown in Figure 3-14(b) is given next.

Command: Choose **3-Point > Start, End, Radius** from the **Draw** panel of the **Ribbon**.

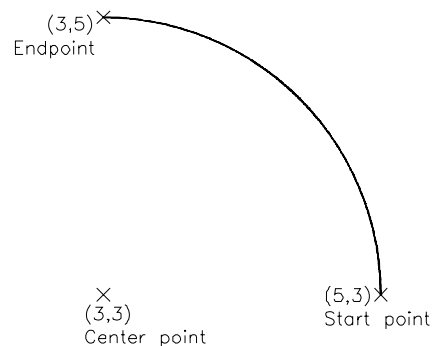
\_arc Specify start point of arc or [Center]: **3,3**

Specify second point of arc or [Center/End]: **\_e** Specify end point of arc: **2,5**

Specify center point of arc or [Angle/Direction/Radius]: **\_r** Specify radius of arc: **2**

## The Center, Start, End Option

The **Center, Start, End** option is a modification of the **Start, Center, End** option. Use this option, 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 3-15, that has a center point at (3,3), start point at (5,3), and endpoint at (3,5) is given next.



**Figure 3-15** Drawing an arc using the **Center, Start, End** option

Command: Choose **3-Point > Center, Start, End** from the **Draw** panel of the **Ribbon**.

\_arc Specify start point of arc or [Center]: \_c Specify center point of arc: 3,3

Specify start point of arc: 5,3

Specify end point of arc or [Angle/chord Length]: 3,5

## The Center, Start, Angle Option

This option is a variation of the **Start, Center, Angle** option. Use this option, whenever it is easier to draw an arc by giving the center first. The prompt sequence for drawing the arc shown in Figure 3-16, that has a center point at (4,5), start point at (5,4), and included angle of 120-degree is given next.

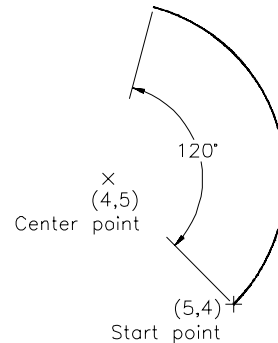
Command: Turn off the dynamic input and choose **3-Point > Center, Start, Angle** from the **Draw** panel of the **Ribbon**.

\_arc Specify start point of arc or [Center]: \_c

Specify center point of arc: 4,5

Specify start point of arc: 5,4

Specify end point of arc or [Angle/chord Length]: \_a Specify included angle: 120



**Figure 3-16** Drawing an arc using the **Center, Start, Angle** option

## The Center, Start, Length Option

The **Center, Start, Length** option is a modification of the **Start, Center, Length** option. This option 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 3-17, that has a center point at (2,2), start point at (4,3), and length of chord of 3 is given next.

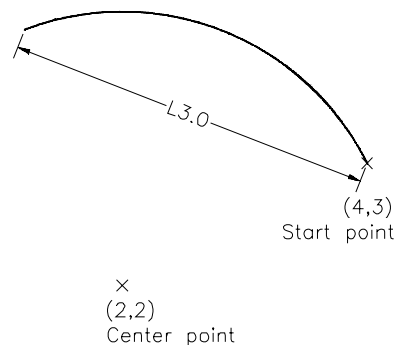
Command: Turn off the dynamic input and choose **3-Point > Center, Start, Length** from the **Draw** panel of the **Ribbon**.

\_arc Specify start point of arc or [Center]: \_c

Specify center point of arc: 2,2

Specify start point of arc: 4,3

Specify end point of arc or [Angle/chord Length]: \_l Specify length of chord: 3



**Figure 3-17** Drawing an arc using the **Center, Start, Length** option

## The Continue Option

With this option, you can continue drawing an arc from a previously drawn arc or line. When you choose **3-Point > Continue** from the **Draw** panel of the **Ribbon**, the start point and direction of the arc will be taken from the endpoint and ending direction of the previous line or arc. When this option is used to draw arcs, each successive arc will be tangent to the

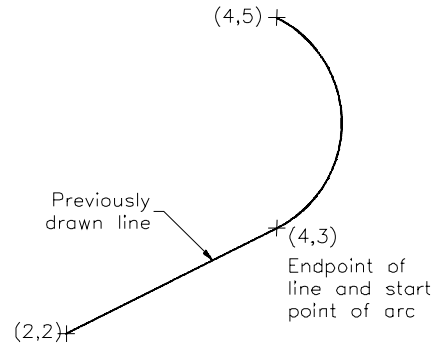
previous one. Most often, this option is used to draw arcs tangent to a previously drawn line. The prompt sequence to draw the arc shown in Figure 3-18, which is tangent to an earlier drawn line, using the **Continue** option is given next.

Command: **LINE**   
 Specify first point: **2,2**   
 Specify next point or [Undo]: **4,3**   
 Specify next point or [Undo]:

Command: Choose **3-Point > Continue** option from the **Draw** panel of the **Ribbon**.

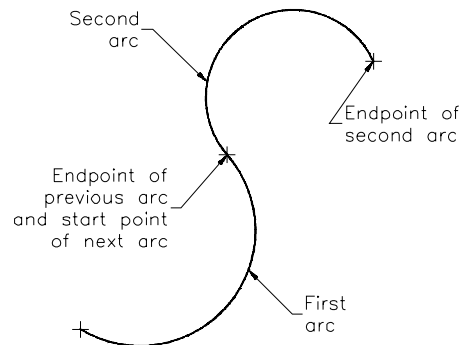
Command: **\_arc** Specify start point of arc or [Center]:

Specify endpoint of arc: **4,5**



The prompt sequence to draw an arc continued from a previously drawn arc, as shown in Figure 3-19, is given next.

Command: **ARC**   
 Specify start point of arc or [Center]: **2,2**   
 Specify second point of arc or [Center/End]: **E**   
 Specify end point of arc : **3,4**   
 Specify center point of arc or [Angle/Direction/Radius]: **R**   
 Specify radius of arc: **2**



Command: Choose **3-Point > Continue** option from the **Draw** panel of the **Ribbon**.

Command: **\_arc** Specify start point of arc or [Center]:

Specify end point of arc: **5,4**

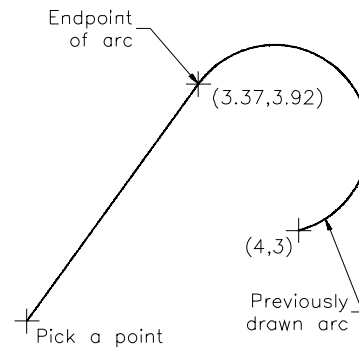


### Tip

You can also invoke the **Continue** option automatically. To do so, first draw a line or an arc and choose any option of the **ARC** command from the **Draw** panel of the **Ribbon**. Now, press **ENTER** at the **Specify start point of arc or [Center]** prompt; the **Continue** option will be invoked automatically. The endpoint of the line or the arc drawn previously will be selected as the start point of the arc. In addition, you will be prompted to specify the endpoint to complete the arc.

## The Continue (LineCont:) Option

This option is used when you want to continue drawing a line from the endpoint of a previously drawn arc. When you use this option, the start point and direction of the line will be taken from the endpoint and ending direction of the previous arc. In other words, the line will be tangent to the arc drawn on the current screen. This option is invoked when you press ENTER at the **Specify first point** prompt of the **LINE** command. The prompt sequence to draw a line, tangent to an earlier drawn arc, as shown in Figure 3-20, is given next.



**Figure 3-20** Drawing a line from the endpoint of an arc

Command: **ARC**

Specify start point of arc or [Center]: **4,3**

Specify second point of arc or [Center/End]: **E**

Specify end point of arc: **3.37,3.92**

Specify center point of arc or [Angle/Direction/Radius]: *Specify the center point.*

Command: **LINE**

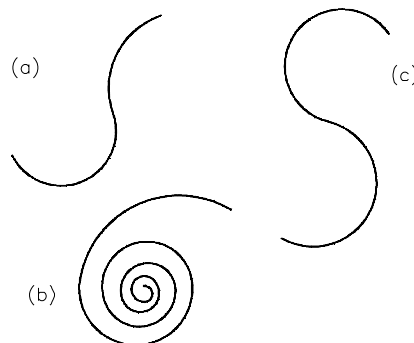
Specify first point:

Length of line: *Enter a value or pick a point.*

### Exercise 5

General

- Use the **Center**, **Start**, **Angle** and **Continue** options to draw the figures shown in Figure 3-21.
- Make the drawing shown in Figure 3-22. The distance between the dotted lines is 1.0 units. Create the radii as indicated in the drawing, by using the **ARC** command options.



**Figure 3-21** Drawing for Exercise 5(a)

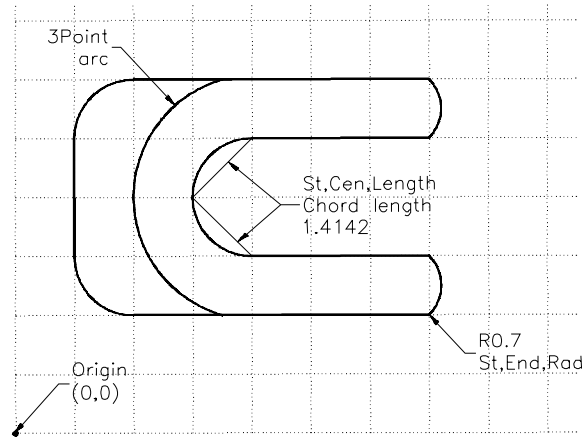


Figure 3-22 Drawing for Exercise 5(b)

## DRAWING RECTANGLES

<b>Ribbon:</b>	Home > Draw > Rectangle
<b>Menu Browser:</b>	Draw > Rectangle
<b>Toolbar:</b>	Draw > Rectangle
<b>Tool Palette:</b>	Draw > Rectangle
<b>Command:</b>	RECTANG

A rectangle can be drawn by choosing the **Rectangle** button from the **Draw** panel of the **Ribbon**, as shown in Figure 3-23, or by choosing the **Rectangle** button from the **Draw** toolbar. You can also enter **RECTANG** at the Command prompt. In AutoCAD LT, 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 to draw rectangles are discussed next.

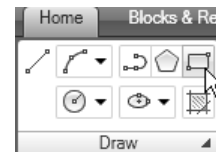
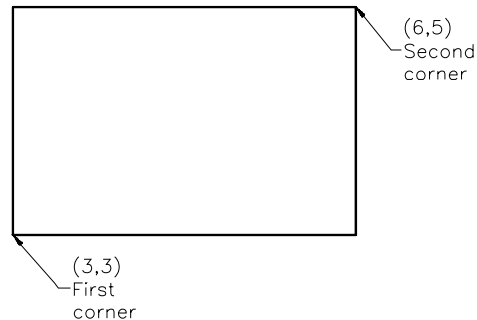


Figure 3-23 Invoking the **RECTANGLE** command from the **Ribbon**

### Drawing Rectangles Using Two Opposite Corners

On invoking the rectangle command, you are prompted to specify the first corner of the rectangle. Here, you can enter the coordinates of the first corner or specify the desired point with the pointing device. The first corner can be any one of the four corners. Next, you are prompted to specify the other corner. This corner is taken as the corner diagonally opposite to the first corner. You can specify the coordinates for the other corner or simply move the cursor to specify it, refer to Figure 3-24. The prompt sequence for drawing a rectangle with (3,3) as its lower left corner coordinate and (6,5) as its upper right corner (Figure 3-24), is given next. It is presumed that the dynamic input is turned off.



**Figure 3-24** Drawing a rectangle by specifying two opposite corners

Command: **RECTANG**

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **3,3**  (Lower left corner location.)

Specify other corner point or [Area/Dimensions/Rotation]: **6,5**  (Upper right corner location)

### Drawing Rectangles by Specifying the Area and One Side

This option allows you to draw the rectangle by specifying its area and the value of one of the sides. This can be done by choosing the **Area** option from the shortcut menu at the **Specify other corner point or [Area/Dimensions/Rotation]** prompt. The following is the prompt sequence to use this option:

Command: **RECTANG**

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **3,3**

Specify other corner point or [Area/Dimensions/Rotation]: **A**

Enter area of rectangle in current units <100.000>: **15**

Calculate rectangle dimensions based on [Length/Width] <Length>: **L**

Enter rectangle length <10.0000>: **5**

Here, the area and length of the rectangle were entered. The system automatically calculates its width 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 specify the dimensions of the rectangle to draw it. This can be done by choosing the **Dimensions** option from the shortcut menu at the **Specify other corner point or**

**[Area/Dimensions/Rotation]** prompt, which then allows you to enter the length and width of the rectangle. The prompt sequence for drawing a rectangle with a length of **5** units and width of **3** units is given next.

Command: **RECTANG**   
 Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **3,3**   
 Specify other corner point or [Area/Dimensions/Rotation]: **D**   
 Specify length for rectangles <0.0000>: **5**   
 Specify width for rectangles <0.0000>: **3**   
 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 locations. Depending on the location of the cursor, the specified first corner point 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 then click to place the rectangle.

## Drawing Rectangle at an Angle

You can also draw a rectangle inclined at an angle. This can be done by choosing the **Rotation** option from the **Specify other corner point or [Area/Dimensions/Rotation]** prompt, which then allows you to enter the rotation angle. After entering the rotation angle, you can continue sizing the rectangle using any of the methods. Once you have specified the rotation angle, next time whenever you draw the rectangle, it will be drawn at an angle. Set the rotation angle to Zero, if you do not want to draw the rectangle at an angle. The prompt sequence for drawing the rectangle at a rotation of 45 degree is:

Command: **RECTANG**   
 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**   
 Specify rotation angle or [Pick points] <current>: **45**   
 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 the different quadrants and select a point in the quadrant, in which you need to draw the rectangle. Figure 3-25 shows a rectangle drawn at an angle of 45-degree.

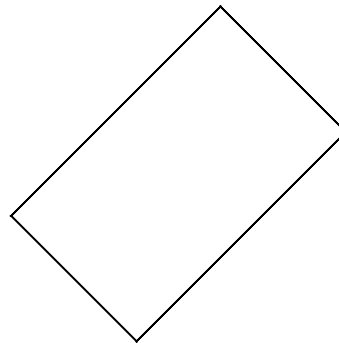



Figure 3-25 Rectangle drawn at an angle

The remaining options of the **RECTANG** command are discussed next.

## Chamfer

The **Chamfer** option creates a chamfer, which is an angled corner, by specifying the chamfer distances, see Figure 3-26. The chamfer is created at all the four corners. You can give two different chamfer values to create an unequal chamfer.

Command: **RECTANG** 

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **C** 

Specify first chamfer distance for rectangles <0.0000>: *Enter a value.*

Specify second chamfer distance for rectangles <0.0000>: *Enter a value.*

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]: *Select a point as upper right corner location.*




### Note

*The first corner point that you specify need not be the location of the lower left corner. While selecting the other corner, you can select a location such that the first corner point becomes the lower right corner or the upper left corner, or the upper right corner.*

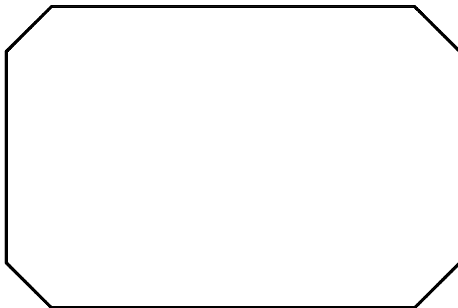
## Fillet

The **Fillet** option allows you to create a filleted rectangle, see Figure 3-27. You can specify the required fillet radius. The following is the prompt sequence for specifying the fillet.

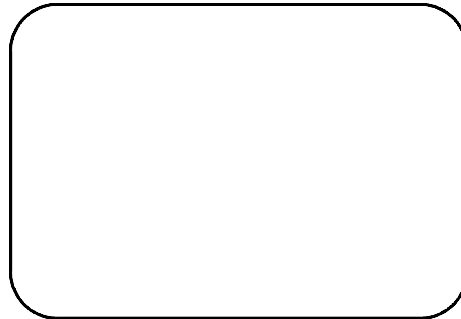
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **F** 

Specify fillet radius for rectangles <0.0000>: *Enter a value.*

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 LT will draw a rectangle without fillets.



**Figure 3-26** Drawing a rectangle with chamfers



**Figure 3-27** Drawing a rectangle with fillets




**Note**

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


**Width**

The **Width** option allows you to create a rectangle whose line segments have some specified width, as shown in Figure 3-28.

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

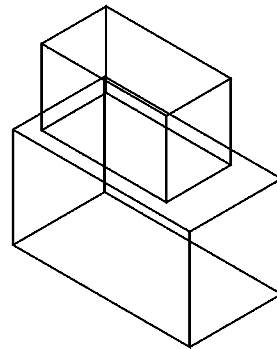
**Thickness**

The **Thickness** option allows you to draw a rectangle that is extruded in the Z direction by the specified value of thickness. For example, if you draw a rectangle with thickness of 2 units, you will get a rectangular box whose height is 2 units, see Figure 3-29. To view the box, choose **View > 3D Views > SE Isometric** from the **Menu Browser**. You need to restore the view back to the plan view by choosing **View > 3D Views > Top** from the **Menu Browser**.

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




**Figure 3-28** Drawing a rectangle with a specified width



**Figure 3-29** Drawing rectangles with thickness and elevation specified

**Elevation**

The **Elevation** option allows you 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 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, see Figure 3-29.

Chamfer/Elevation/Fillet/Thickness/Width/<First corner>: **E**   
Specify elevation for rectangles <0.0000>: *Enter a value.*

To view the objects in 3D space, change the viewpoint by choosing **View > 3D Views > SE Isometric** from the **Menu Browser**. To restore the top view, choose **View > 3D Views > Top** from the **Menu Browser**.

**Note**

*The value you enter for fillet, width, elevation, and thickness options becomes the current value for the subsequent **RECTANG** command. Therefore, you must reset the values, if they are different from the current values. The thickness can be controlled by its thickness settings.*

*The rectangle generated on the screen is treated as a single object. Therefore, an individual side can be edited only after the rectangle has been exploded using the **EXPLODE** command.*

**Tip**

*You can combine different options in one **RECTANG** command and then draw the rectangle with the specified characteristics. When you invoke the **RECTANG** command again, the previously set options and their values are displayed before the first prompt. This allows you to change the settings according to the new specifications.*

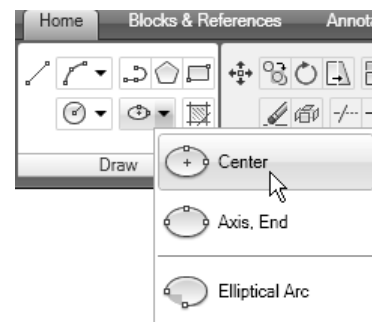
**Exercise 6***General*

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

**DRAWING ELLIPSES**

**Ribbon:** Home > Draw > Ellipse  
**Menu Browser:** Draw > Ellipse  
**Toolbar:** Draw > Ellipse  
**Tool Palettes:** Draw > Ellipse  
**Command:** ELLIPSE

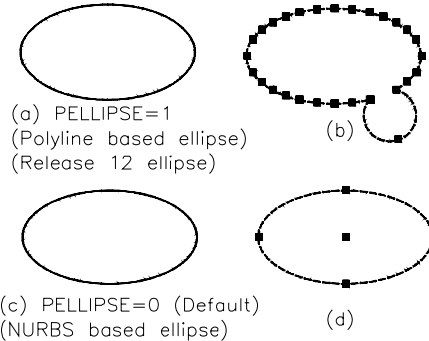
If a circle is observed from an angle, the shape seen is called an ellipse, which can be created in AutoCAD LT using the **ELLIPSE** command. An ellipse can be created using various options and all these options are grouped together in the **Draw** panel of the **Ribbon**, refer to Figure 3-30. AutoCAD LT creates a true ellipse, also known as a NURBS-based (Non-Uniform Rational Bezier Spline) ellipse. The true ellipse has a center and quadrant points. If you select it, the grips (small blue squares) will be displayed at the center and quadrant points of the ellipse. If you move one of the grips located on the perimeter of the ellipse, the major or minor axis will change, which changes the size of the ellipse, as shown in Figure 3-31(d). The creation of a true ellipse is dependent on the **PELLIPSE** system variable, which has a default value of 0.



**Figure 3-30** Invoking the **ELLIPSE** command from the **Ribbon**

**Note**

Until Release 12, ellipses were based on polylines. They were made of multiple polyarcs and as a result, it was difficult to edit an ellipse. For example, if you select a polyline-based ellipse, the grips will be displayed at the endpoints of each polyarc. On moving a vertex point, you get the shape shown in Figure 3-31(b). Also, you cannot snap to the center or quadrant points of a polyline-based ellipse. In AutoCAD LT 2009, you can still draw the polyline-based ellipse by setting the value of the **PELLIPSE** system variable to **1**, which is **0** (true ellipse) by default.



**Figure 3-31** Drawing polyline and NURBS-based ellipses

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

**Note**

The **Isocircle** option is not available by default in the **ELLIPSE** command. To display this option, select the **Isometric snap** radio button in the **Snap and Grid** tab of the **Drafting Settings** dialog box.

The **Arc** option is not available, if you set the value of the **PELLIPSE** system variable to **1** for drawing the polyline-based ellipse.

## Drawing an Ellipse Using the Axis and End Option

In this option, you draw an ellipse by specifying one of its axes and the endpoint of the other axis. To use this option, acknowledge the **Specify axis endpoint of ellipse or [Arc/Center]** prompt by specifying a point, either by using a pointing device or by entering its coordinates. This is the first endpoint of one axis of the ellipse. AutoCAD LT will then respond with the prompt **Specify other endpoint of axis**. Here, specify the other endpoint of the axis. The angle at which the ellipse is drawn depends on the angle made by these two axis endpoints. Your response to the next prompt determines whether the axis is the **major axis** or the **minor axis**.

The next prompt is **Specify distance to other axis or [Rotation]**. If you specify a distance, it is presumed as half the length of the second axis. You can also specify a point. The distance from this point to the midpoint of the first axis is again taken as half the length of this axis. The ellipse will pass through the selected point only if it is perpendicular to the midpoint of the first axis. To visually analyze the distance between the selected point and the midpoint of the first axis, AutoCAD LT appends an elastic line to the crosshairs, with one end fixed at the midpoint of the first axis. You can also drag the point, dynamically specifying half of the

other axis distance. This helps you to visualize the ellipse, refer to Figure 3-32. The prompt sequence for drawing an ellipse with one axis endpoint located at (3,3), the other at (6,3), and the distance of the other axis being 1 is given next.

Command: **ELLIPSE**

**Specify axis endpoint of ellipse or [Arc/Center]:** 3,3

**Specify other endpoint of axis:** 6,3

**Specify distance to other axis or [Rotation]:** 1

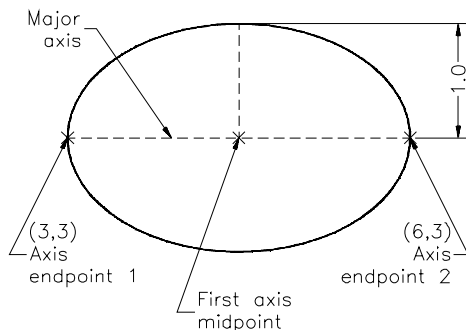
Another example for drawing an ellipse using this option is shown in Figure 3-33 and illustrated by the following prompt sequence.

Command: **ELLIPSE**

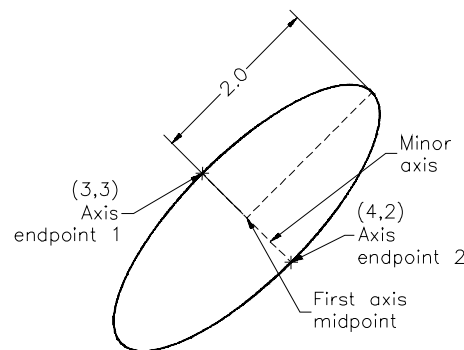
**Specify axis endpoint of ellipse or [Arc/Center]:** 3,3

**Specify other endpoint of axis:** 4,2

**Specify distance to other axis or [Rotation]:** 2



**Figure 3-32** Drawing an ellipse using the *Axis and Endpoint* option



**Figure 3-33** Drawing an ellipse using the *Axis and Endpoint* option

If you enter **Rotation** or **R** at the **Specify distance to other axis or [Rotation]** prompt, the first axis specified is automatically taken as the major axis of the ellipse. The next prompt is **Specify rotation around major axis**. The major axis is taken as the diameter line of the circle, and the rotation takes place around this diameter line into the third dimension. The ellipse is formed, when AutoCAD LT projects this rotated circle into the drawing plane. You can enter the rotation angle value in the range of 0 to 89.4-degree only, because an angle value greater than 89.4-degree changes the circle into a line. Instead of entering a definite angle value at the **Specify rotation around major axis** prompt, you can specify a point relative to the midpoint of the major axis. This point can be dragged to specify the ellipse dynamically. The following is the prompt sequence for a rotation of 0-degree around the major axis, as shown in Figure 3-34(a):

Command: **ELLIPSE**

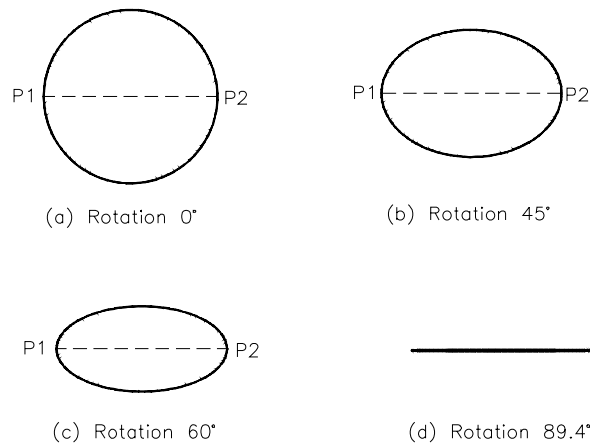
**Specify axis endpoint of ellipse or [Arc/Center]:** *Select point (P1).*

**Specify other endpoint of axis:** *Select another point (P2).*

**Specify distance to other axis or [Rotation]:** **R**

**Specify rotation around major axis:** **0**

Figure 3-34 also shows rotations of 45-degree, 60-degree, and 89.4-degree.



**Figure 3-34** Rotation about the major axis

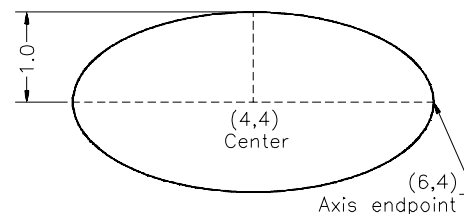
## Exercise 7

General

Draw an ellipse whose major axis is 4 units and the rotation around this axis is 60-degree. Draw another ellipse, whose rotation around the major axis is 15-degree.

## Drawing an Ellipse Using the Center Option

In this option, you can construct an ellipse by specifying the center point, endpoint of one axis, and length of the other axis. The only difference between this method and the ellipse by axis and endpoint method is that instead of specifying the second endpoint of the first axis, the center of the ellipse is specified. The center of an ellipse is defined as the point of intersection of the major and minor axes. In this option, the first axis need not be the major axis. For example, draw an ellipse with center at (4,4), axis endpoint at (6,4), and length of the other axis as 1 unit, as shown in Figure 3-35.



**Figure 3-35** Drawing an ellipse using the **Center** option

The prompt sequence is given next.

Command: **ELLIPSE**   
 Specify axis endpoint of **ellipse** or **[Arc/Center]**: **C**   
 Specify center of ellipse: **4,4**   
 Specify endpoint of axis: **6,4**   
 Specify distance to other axis or **[Rotation]**: **1**

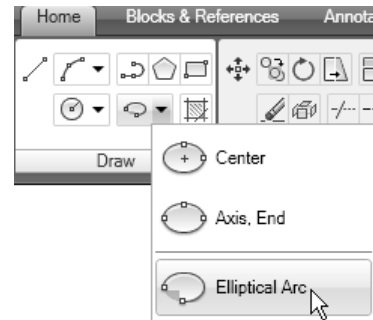
Instead of entering the distance, you can enter **Rotation** or **R** at the **Specify distance to other axis [Rotation]** prompt. This takes the first axis specified as the major axis. The next prompt, **Specify rotation around major axis**, prompts you to enter the rotation angle value. The rotation takes place around the major axis, which is taken as the diameter line of the circle.

## Drawing Elliptical Arcs

**Ribbon:** Draw > Center > Elliptical Arc  
**Menu Browser:** Draw > Ellipse > Arc  
**Toolbar:** Draw > Ellipse Arc  
**Tool Palette:** Draw > Ellipse Arc  
**Command:** ELLIPSE > Arc

AutoCAD LT allows you to draw elliptical arcs using the **Ellipse Arc** button in the **Draw** toolbar or by choosing **Center > Elliptical Arc** from the **Draw** panel of the **Ribbon**, as shown in Figure 3-36. When you choose this button, the **ELLIPSE** command is invoked with the **Arc** option selected and you will be prompt to enter information about the geometry of the ellipse and the arc limits. You can define the arc limits by using the following options:

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



**Figure 3-36** Invoking the **Elliptical Arc** command from the **Ribbon**

The angles are measured from the first point and in counterclockwise direction, if the default setup of AutoCAD LT is not changed. The next example illustrates the use of these options.

### Example 1

*General*

Draw the following elliptical arcs, as shown in Figures 3-37 and 3-38.

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

**Specifying Start and End Angle of the Elliptical Arc** [Figure 3-37(a)]

Command: Choose **Center > Elliptical Arc** from the **Draw** panel of the **Ribbon**.

Specify axis endpoint of ellipse or [Arc/Center]: **\_a**

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**

Specify end angle or [Parameter/Included angle]: **135**  (*Angle where arc ends.*)

**Specifying Start and Included Angle of the Elliptical Arc** [Figure 3-37(b)]

Command: Choose **Center > Elliptical Arc** from the **Draw** panel of the **Ribbon**.

Specify axis endpoint of ellipse or [Arc/Center]: **\_a**

Specify axis endpoint of elliptical arc or [Center]: *Select the first endpoint.*

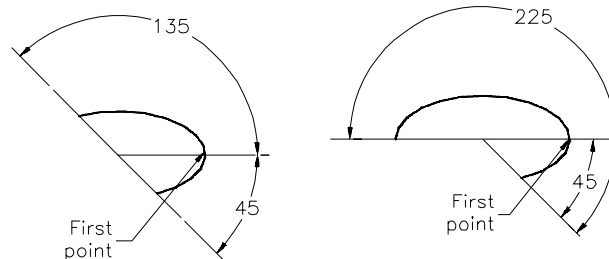
Specify other endpoint of axis: *Select the second point.*

Specify distance to other axis or [Rotation]: *Select a point or enter a distance.*

Specify start angle or [Parameter]: **-45**

Specify end angle or [Parameter/Included angle]: **I**

Specify included angle for arc<current>: **225**  (*Included angle.*)



(a) Start angle=-45  
End angle =135

(b) Start angle=-45  
Included angle=225

**Figure 3-37** Drawing elliptical arcs

**Specifying Start and End Parameters** (Figure 3-38)

Command: Choose **Center > Elliptical Arc** from the **Draw** panel of the **Ribbon**.

Specify axis endpoint of ellipse or [Arc/Center]: **\_a**

Specify axis endpoint of elliptical arc or [Center]: *Select the first endpoint.*

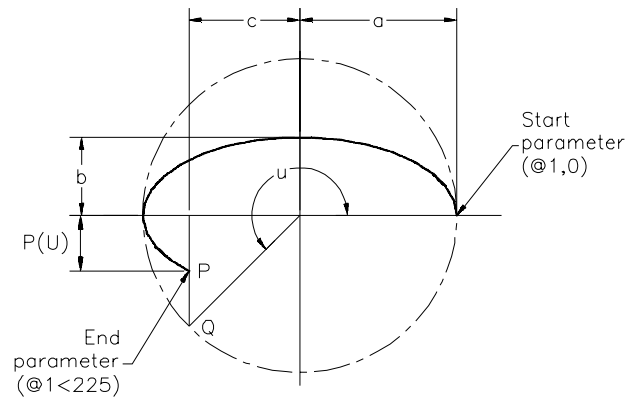
Specify other endpoint of axis: *Select the second endpoint.*

Specify distance to other axis or [Rotation]: *Select a point or enter a distance.*

Specify start angle or [Parameter]: **P**

Specify start parameter or [Angle]: @1,0

Specify end parameter or [Angle/ Included angle]: @1<225

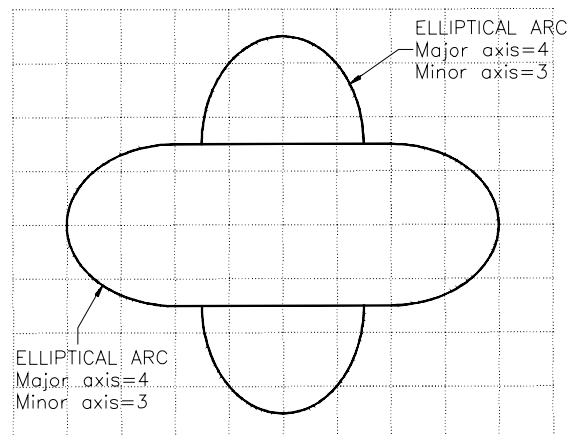


**Figure 3-38** Drawing an elliptical arc by specifying the start and end parameters

## Exercise 8

General

- Construct an ellipse with center at (2,3), axis endpoint at (4,6), and the other axis endpoint at a distance of 0.75 units from the midpoint of the first axis.
- Make the drawing, as shown in Figure 3-39. The distance between the dotted lines is 1.0 unit. Create the elliptical arcs using the **ELLIPSE** command options.



**Figure 3-39** Drawing for Exercise 8



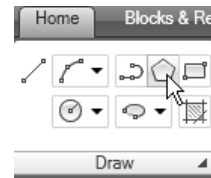
**Note**

From now on, all command sequences in this book presume that the **Dynamic Input** button is turned off.

## DRAWING REGULAR POLYGONS

**Ribbon:** Home > Draw > Polygon  
**Menu Browser:** Draw > Polygon  
**Toolbar:** Draw > Polygon  
**Tool Palette:** Draw > Polygon  
**Command:** POLYGON

A regular polygon is a closed geometric figure with equal sides. The number of sides varies from 3 to 1024. For example, a triangle is a three-sided polygon and a pentagon is a five-sided polygon. In AutoCAD LT, the **POLYGON** command is used to draw regular 2D polygons. Choose the **Polygon** button from the **Draw** panel of the **Ribbon** to invoke this command, refer to Figure 3-40. The characteristics of a polygon drawn in AutoCAD LT are those of a closed polyline having a 0 width. You can change the width of the polyline forming the polygon. The prompt sequence is given next.



**Figure 3-40** Invoking the **POLYGON** command from the **Ribbon**

Command: **POLYGON** Enter

Enter number of sides <4>: *Specify the required number of sides of the polygon.*

Once you invoke the **POLYGON** command, it prompts you to enter the number of sides. The number of sides determine the type of polygon (for example, six sides define a hexagon). The default value for the number of sides is 4. You can change the number of sides to your requirement and then the new value becomes the default value. You can also set a different default value for the number of sides, by using the **POLYSIDES** system variable.

### The Center of Polygon Option

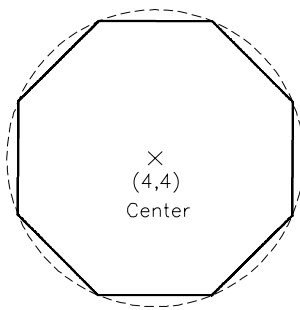
After you specify the number of sides, the next prompt is **Specify center of polygon or [Edge]**, where the default option prompts you to select a point that is taken as the center point of the polygon. The next prompt is **Enter an option [Inscribed in circle/Circumscribed about circle] <I>**. A polygon is said to be **inscribed** when it is drawn inside an imaginary circle and its vertices (corners) touch the circle, see Figure 3-41. Similarly, a polygon is **circumscribed** when it is drawn outside the imaginary circle and the sides of the polygon are tangent to the circle (midpoint of each side of the polygon will lie on the circle), see Figure 3-42. If you want to have an inscribed polygon, choose **Inscribed in circle** option from the shortcut menu. The next prompt issued is **Specify radius of circle**. Here, you are required to specify the radius of the circle on which all the vertices of the polygon will lie. Once you specify it, a polygon will be generated. If you want to select the circumscribed option, choose the **Circumscribed about circle** option from the shortcut menu at the **Enter an option [Inscribed in circle/Circumscribed**

**about circle] <I>** prompt. You can also select these options from the dynamic preview. After this, enter the radius of the circle. The inscribed or circumscribed circle is not drawn on the screen. The radius of the circle can be dynamically dragged instead of a numerical value being entered. The prompt sequence for drawing an inscribed octagon shown in Figure 3-41, with the center at (4,4) and the radius of 1.5 units, is given next.

Command: **POLYGON**   
 Enter number of sides<4>: **8**   
 Specify center of polygon or [Edge]: **4,4**   
 Enter an option[Inscribed in circle/Circumscribed about circle ]<I>: **I**   
 Specify radius of circle: **1.5**

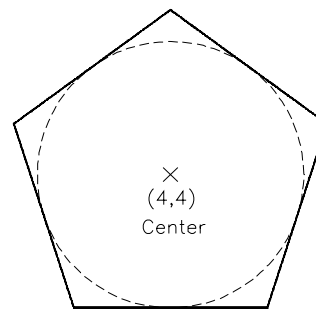
The prompt sequence for drawing a circumscribed pentagon shown in Figure 3-42 with center at (4,4) and a radius of 1.5 units, is given next.

Command: **POLYGON**   
 Enter number of sides<4>: **5**   
 Specify center of polygon or [Edge]: **4,4**   
 Enter an option[Inscribed in circle/Circumscribed about circle ]<I>: **C**   
 Specify radius of circle: **1.5**



Inscribed octagon

**Figure 3-41** Drawing an inscribed polygon using the **Center of Polygon** option



Circumscribed pentagon

**Figure 3-42** Drawing a circumscribed polygon using the **Center of Polygon** option



#### Note

*If you select a point to specify the radius of an inscribed polygon, one of the vertices is positioned on the selected point. In case of circumscribed polygons, the midpoint of an edge is placed on the point you have specified. In this manner, you can specify the size and rotation of the polygon.*

*In case of the numerical specification of the radius, the bottom edge of the polygon is rotated by the prevalent snap rotation angle.*

## Exercise 9

General

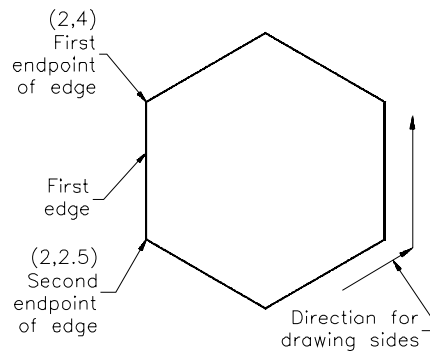
Draw a circumscribed polygon of eight sides. The polygon should be drawn by the **Center of Polygon** method.

## The Edge Option

The other method for drawing a polygon is to select the **Edge** option. This can be done by entering E at the **Specify center of polygon or [Edge]** prompt. The next two prompts issued are **Specify first endpoint of edge** and **Specify second endpoint of edge**. Here, you need to specify the two endpoints of an edge of the polygon. The polygon is drawn in a counterclockwise direction, with the two entered points defining its first edge. To draw a hexagon (six-sided polygon), as shown in Figure 3-43, using the **Edge** option, with the first endpoint of the edge at (2,4) and the second endpoint of the edge at (2,2.5), the prompt sequence is given next.

```
Command: POLYGON 
  _polygon Enter number of
sides<4>: 6 
Specify center of polygon or
[Edge]: E 
Specify first endpoint of edge: 2,4

Specify second endpoint of edge:
2,2.5 
```

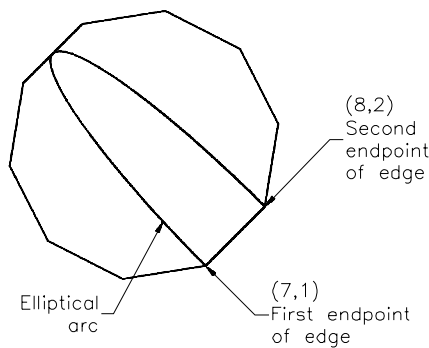


**Figure 3-43** Drawing a polygon (hexagon) using the **Edge** option

## Exercise 10

General

Draw a polygon with ten sides using the **Edge** option and an elliptical arc, as shown in Figure 3-44. Let the first endpoint of the edge be at (7,1) and the second endpoint be at (8,2).

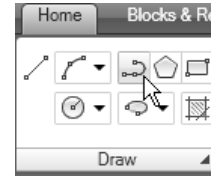


**Figure 3-44** Polygon and elliptical arc for Exercise 10

## DRAWING POLYLINES

**Ribbon:** Home > Draw > Polyline  
**Menu Browser:** Draw > Polyline  
**Toolbar:** Draw > Polyline  
**Tool Palettes:** Draw > Polyline  
**Command:** PLINE (or PL)

A polyline is a line that has different features. Invoke the **PLINE** command by choosing the **Polyline** button from the **Draw** panel in the **Ribbon**, see Figure 3-45. The term POLYLINE can be broken into two parts: POLY and LINE. POLY means “many”. This signifies that a polyline can have many lines. Some of the features of the Polylines are listed next.



**Figure 3-45** Invoking the **POLYLINE** command from the **Ribbon**

1. Polylines can be thick lines with a desired width. They are very flexible and can be used to draw any shape, such as a filled circle or a doughnut.
2. Polylines can be used to draw objects in any linetype (for example, hidden linetype).
3. Advanced editing commands can be used to edit them (for example, the **PEDIT** command).
4. A single polyline object can be formed by joining polylines and polyarcs of different thicknesses.
5. It is easy to determine the area or perimeter of a polyline feature. Also, it is easy to offset when drawing walls.

The **PLINE** command functions fundamentally like the **LINE** command, except that it has some additional options are provided in it. All the segments of the polyline form a single object. After invoking the **PLINE** command, the following prompt is displayed:

Specify start point: *Specify the starting point or enter its coordinates.*  
 Current line width is nn.nnnn

**Current line width is nn.nnnn** is displayed automatically, which indicates that the polyline drawn will have nn.nnnn width. If you want a different width, invoke the **Width** option at the next prompt and set it. Next, the following prompt is displayed.


Specify next point or [Arc/Halfwidth/Length/Undo/Width]: *Specify the next point or enter an option.*

Depending on your requirements, the options that can be invoked at this prompt are as follows.

### Next Point of Line


This option is maintained as the default option and is used to specify the next point of the current polyline segment. If additional polyline segments are added to the first polyline,

AutoCAD LT automatically makes the endpoint of the previous polyline segment the start point of the next polyline segment. The prompt sequence is given next.









Command: **PLINE**   
Specify start point: *Specify the starting 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 command.*

## Width





You can change the current polyline width by entering **W** (width option) at the last prompt. You can also right-click and choose the **Width** option from the shortcut menu. Next, you are prompted for the starting and ending width of the polyline.

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **W**   
Specify starting width <current>: *Specify the starting width.*  
Specify ending width <starting width>: *Specify the ending width.*

The starting width value is taken as the default value for the ending width. Therefore, to have a uniform polyline, you need to press ENTER at the **Specify ending width <>** prompt. To draw a uniform polyline, shown in Figure 3-46, with a width of 0.25 units, start point at (4,5), endpoint at (5,5), and the next endpoint at (3,3), use the following prompt sequence:

Command: **PLINE**   
Specify start point: **4,5**   
Current line-width is 0.0000  
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **W**   
Specify starting width <current>: **0.25**   
Specify ending width <0.25>:   
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **5,5**   
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **3,3**   
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 

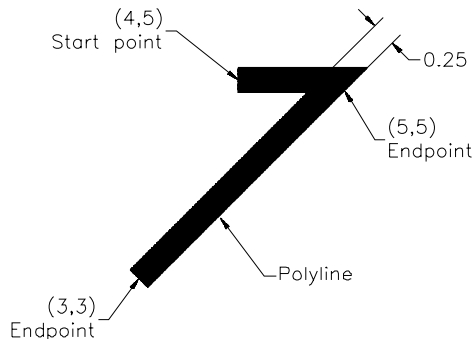
You can get a tapered polyline, by entering two different values at the starting width and the ending width prompts. To draw a tapered polyline, shown in Figure 3-47, with a starting width of 0.5 units and an ending width of 0.15 units, a start point at (2,4), and an endpoint at (5,4), use the prompt sequence given next.

Command: **PLINE**   
Specify start point: **2,4**   
Current line-width is 0.0000  
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **W**   
Specify starting width <0.0000>: **0.50** 

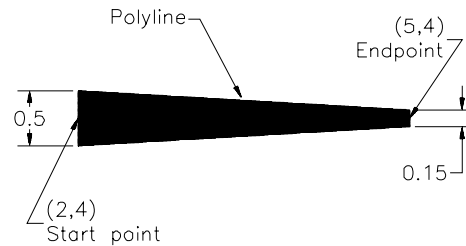
Specify ending width <0.50>: **0.15**

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **5,4**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:



**Figure 3-46** Drawing a uniform polyline using the **PLINE** command



**Figure 3-47** Drawing a tapered polyline using the **PLINE** command

## Halfwidth

With this option, you can specify the starting and ending halfwidth of a polyline. This halfwidth distance is equal to half of the actual width of the polyline. This option can be invoked by entering **H** or choosing **Halfwidth** from the shortcut menu at the following prompt:

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **H**

Specify starting half-width <0.0000>: **0.12**  (Specify desired starting halfwidth)

Specify ending half-width <0.1200>: **0.05**  (Specify desired ending halfwidth)

## Length

This option prompts you to enter the length of a new polyline segment. The new polyline segment will be the length you have entered. It will be drawn at the same angle as the last polyline segment or tangent to the previous polyarc segment. This option can be 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**

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** at the following prompt.

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **U**

You can use this option repeatedly until you reach the start point of the first polyline segment.


Further use of **Undo** option evokes the message **All segments already undone.**

## Close

This option is available when at least one segment of the polyline is drawn. It closes the polyline by drawing a polyline segment from the most recent endpoint to the initial start point. At the same time, it exits from the **PLINE** command. The width of the closing segment can be changed by using the **Width/Halfwidth** option, before invoking the **Close** option.

## Arc

This option is used to switch from drawing polylines to drawing polyarcs, and provides you the options associated with drawing polyarcs. The prompt sequence is given next.

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: A   
Specify endpoint of arc or [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: *Enter an option.*

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. The preceding prompt contains options associated with the **PLINE Arc** such as **Angle**, **CEnter**, **CLose**, etc. The detailed explanation of each of these options is as follows.

## Angle

This option prompts you to enter the included angle for the arc. If you enter a positive angle, the arc is drawn in a counterclockwise direction from the start point to the endpoint. If the angle specified is negative, the arc is drawn in a clockwise direction. The prompts are given next.

Specify included angle: *Specify the included angle.*  
Specify endpoint of arc or [Center/Radius]: *Specify the endpoint of arc or choose an option.*

Center refers to the center of the arc segment, Radius refers to the radius of the arc, and Endpoint draws the arc.

## CEnter

This option prompts you to specify the center of the arc to be drawn. As mentioned before, usually the arc segment is drawn such that it is tangent to the previous polyline segment; in such cases AutoCAD LT determines the center of the arc automatically. Therefore, the **CEnter** option provides the freedom to choose the center of the arc segment. This option can be invoked by entering **CE** at the **Specify end point of arc or [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]** prompt. Once you specify the center point, AutoCAD LT issues the following prompt.

Specify endpoint of arc or [Angle/Length]: *Specify the endpoint of arc or choose an option.*

Angle refers to the included angle, Length refers to the length of the chord, and Endpoint refers to the endpoint of the arc.

**Close**

This option is used to join the last point of a polyline with its start point in the form of an arc. This option is available only when you specify two or more than two points for creating the polyline segments.

**Direction**

Usually, the arc drawn, with the **PLINE** command, is tangent to the previous polyline segment. In other words, the starting direction of the arc is the ending direction of the previous segment. The Direction option allows you to specify the tangent direction of your choice for the arc segment to be drawn. You can 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: *Specify the endpoint of arc.*

**Halfwidth**

This option is the same as for the **PLINE** and prompts you to specify the starting and ending halfwidth of the arc segment.

**Line**

This option takes you back to the **Line** mode. You can draw polylines only in this mode.

**Radius**

This option prompts you to specify the radius of the arc segment. The prompt sequence is given next.

Specify radius of arc: *Specify the radius of the arc segment.*

Specify endpoint of arc or [Angle]: *Specify the endpoint of arc or choose an option.*

If you specify a point, the arc segment is drawn. If you enter an angle, 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<current>** prompts, respectively.

**Second pt**

This option selects the second point of an arc in the three-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.*

**Undo**

This option reverses the changes made in the previously drawn segment.

**Width**

This option prompts you to enter the width of the arc segment. To draw a tapered arc segment, you can enter different values at the starting width and ending width prompts. The prompt



sequence is identical to that for the polyline. Also, a specified point on a polyline refers to the midpoint on its width.

## Endpoint of Arc

This option is maintained as the default and prompts you to specify the endpoint of the current arc segment. The following is the prompt sequence for drawing an arc, shown in Figure 3-48, with start point at (3,3), endpoint at (3,5), starting width of 0.50 units, and ending width of 0.15 units.

Command: **PLINE**   
 Specify start point: **3,3**   
 Current line-width is 0.0000  
 Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **A**   
 Specify endpoint of arc or [Angle/CEnter/CLOSE/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **W**   
 Specify starting width <current>: **0.50**   
 Specify ending width <0.50>: **0.15**   
 Specify endpoint of arc or [Angle/CEnter/CLOSE/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **3,5**   
 Specify endpoint of arc or [Angle/CEnter/CLOSE/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:

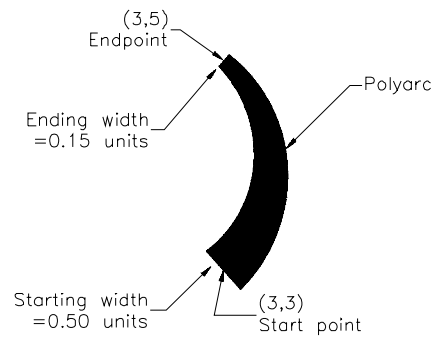


Figure 3-48 Drawing a polyarc



### Tip

After invoking the **PLINE** command and specifying the start point, you can right-click to display the shortcut menu. You can choose any of the options under the **PLINE** command, directly from the shortcut menu, instead of entering the appropriate letters at the Command prompt. Similarly, after invoking the **ARC** option of the **PLINE** command, you can right-click to display the shortcut menu and choose any polyarc option.



### Note

If **FILL** is on or if **FILLMODE** is 1, the polylines drawn are filled. If you change **FILL** to off or **FILLMODE** to 0, only the outlines are drawn for the new plines and previously drawn plines are also changed from filled to no-fill. However, note that the change is effective only on regeneration. Similarly, it works in reverse also.

Also, the **PLINEGEN** system variable controls the linetype pattern between the vertex points of a 2D polyline. A value of 0 centers the linetype for each polyline segment and 1 makes it continuous.

## Exercise 11

General

Draw the objects shown in Figures 3-49 and 3-50. Approximate the width of different polylines.

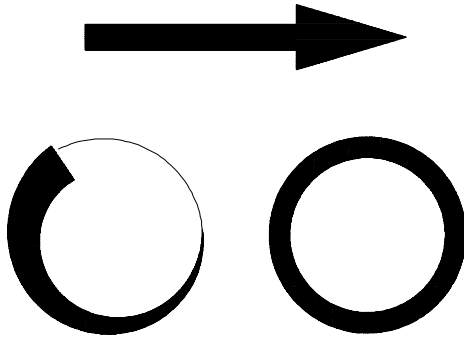


Figure 3-49 Drawing for Exercise 11

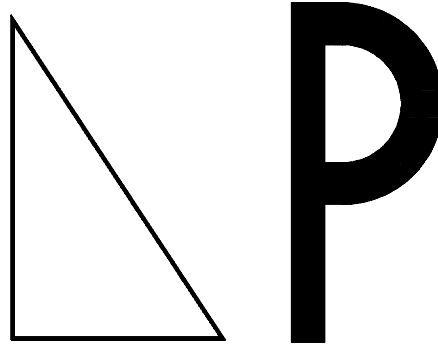


Figure 3-50 Drawing for Exercise 11

## DRAWING DONUTS

**Ribbon:** Home > Draw > Donut  
**Menu Browser:** Draw > Donut  
**Command:** DONUT

In AutoCAD LT, the **DONUT** or **DOUGHNUT** command is issued to draw an object that looks like a filled circle ring called a donut. Actually, AutoCAD LT's donuts are made of two semicircular polylines with a certain width. Therefore, the **DONUT** command allows you to draw a thick circle. The donuts can have any inside and outside diameters. If **FILLMODE** is off, the donuts look like circles (if the inside diameter is zero) or concentric circles (if the inside diameter is not zero). After specifying the two diameters, the donut gets attached to the crosshairs. You can select a point for the center of the donut anywhere on the screen with the help of a pointing device, and then place the donut. You can place it by clicking your pointing device. The prompt sequence for drawing donuts is given next.

Command: **DONUT**

Specify inside diameter of donut <current>: *Specify the inner diameter of the donut.*

Specify outside diameter of donut <current>: *Specify the outer diameter of the donut.*

Specify center of donut or <exit>: *Specify the center of the donut.*

Specify center of donut or <exit>: *Specify the center of the donut to draw more donuts of previous specifications or give a null response to exit.*

The defaults for the inside and outside diameters are the respective diameters of the most recent donut drawn. The values for the inside and outside diameters are saved in the **DONUTID** and **DONUTOD** system variables. A solid-filled circle is drawn by specifying the inside diameter as zero (**FILLMODE** is on). Once the diameter specification is completed, the donuts are formed at the crosshairs and can be placed anywhere on the screen. For the location, you can enter the coordinates of the point or specify the point by dragging the center point. Once you have specified the center of the donut, AutoCAD LT repeats the **Specify center of donut or <exit>** prompt. As you go on specifying the locations for the center point, donuts with the specified diameters are drawn at specified locations. To end

the **DONUT** command, give a null response to this prompt by pressing ENTER. Since donuts are circular polylines, the donut can be edited with the **PEDIT** command or any other editing command that can be used to edit polylines.

## Example 2

General

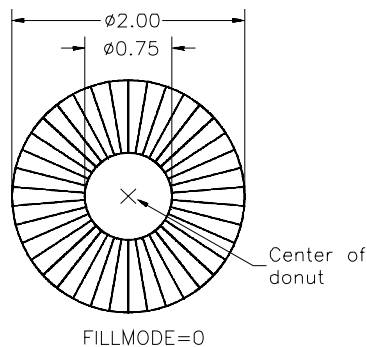
You will draw an unfilled donut shown in Figure 3-50 with an inside diameter of 0.75 units, an outside diameter of 2.0 units, and centered at (2,2). You will also draw a filled donut and a solid-filled donut with the given specifications.

The following is the prompt sequence to draw an unfilled donut shown in Figure 3-51.

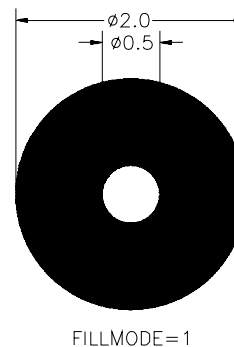
Command: **FILLMODE**   
 New value for FILLMODE <1>: **0**   
 Command: **DONUT**   
 Specify inside diameter of donut <0.5000>: **0.75**   
 Specify outside diameter of donut <1.000>: **2**   
 Specify center of donut or <exit>: **2,2**   
 Specify center of donut or <exit>:

The prompt sequence for drawing a filled donut, shown in Figure 3-52, is given next, with an inside diameter of 0.5 units, outside diameter of 2.0 units, centered at a specified point.

Command: **FILLMODE**   
 Enter new value for FILLMODE <0>: **1**   
 Command: **DONUT**   
 Specify inside diameter of donut <0.5000>: **0.50**   
 Specify outside diameter of donut <1.000>: **2**   
 Specify center of donut or <exit>: *Specify a point.*  
 Specify center of donut or <exit>:

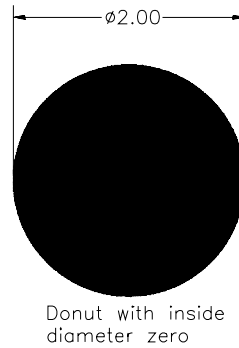


**Figure 3-51** Drawing an unfilled donut using the **DONUT** command



**Figure 3-52** Drawing a filled donut using the **DONUT** command

To draw a solid-filled donut, shown in Figure 3-53, with an outside diameter of 2.0 units, use the following prompt sequence.



**Figure 3-53** Solid-filled donut

Command: **DONUT**   
 Specify inside diameter of donut <0.50>: **0**   
 Specify outside diameter of donut <1.0>: **2**   
 Specify center of donut or <exit>: *Specify a point.*  
 Specify center of donut or <exit>:

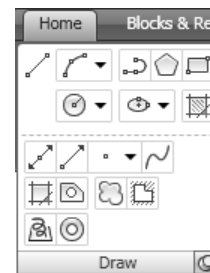
## DRAWING POINTS

**Ribbon:** Home > Draw > Point  
**Menu Browser:** Draw > Point  
**Toolbar:** Draw > Point  
**Tool Palette:** Draw > Point  
**Command:** POINT

The point is the basic drawing object. Points are invaluable in building a drawing file. To draw a point anywhere on the screen, AutoCAD LT provides the **POINT** command.

Command: **POINT**   
 Current point modes: PDMODE=n PDSIZE=n.nnnn  
 Specify a point: *Specify the location where you want to place the point.*

You can invoke the **POINT** command from the **Draw** toolbar or the **Menu Browser**, or from the **Ribbon (Multiple Point** option), as shown in Figure 3-54. You can draw as many points as you desire in a single command. Then press ESC to exit from the **POINT** command. If you invoke this command by entering **POINT** at the Command prompt or use the **Single Point** option from the menu, you can draw only one point in a single point command.



**Figure 3-54** Invoking the **POINT** command from the **Ribbon**



**Note**  
It is possible to have a temporary construction marker for the point known as blip. A mark appears on the screen where you place the point. Then this blip mark can be cleared, once the screen is redrawn using the **REDRAW** command, and the point is left on the screen. The visibility of a blip can be controlled using the **BLIPMODE** system variable.

Changing the Point Type

**Menu Browser:** Format > Point Style  
**Command:** DDPTYPE

The point type can be set from the **Point Style** dialog box shown in Figure 3-55. There are twenty combinations of point types. Choose **Format > Point Style** from the **Menu Browser**; the **Point Style** dialog box will be invoked. You can choose a point style in this dialog box, which is indicated by highlighting that particular point style. Next, choose the **OK** button. Now all the points will be drawn in the selected style, until you change it to a new style. The type of point drawn is stored in the **PDMODE** (Point Display MODE) system variable. You can change the point style by entering a numeric value in the **PDMODE** variable.

Figure 3-56 shows the **PDMODE** values for different point type areas.

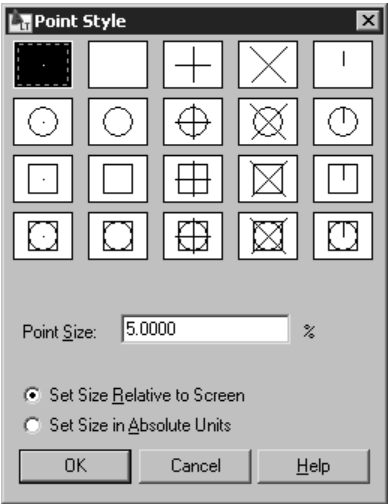


Figure 3-55 The Point Style dialog box

Pdmode Value	Point Style	Pdmode Value	Point Style
0	.	64+0=64	□
1		64+1=65	□
2	+	64+2=66	⊕
3	×	64+3=67	⊗
4		64+4=68	⌈
32+0=32	○	96+0=96	◻
32+1=33	○	96+1=97	◻
32+2=34	⊕	96+2=98	⊕
32+3=35	⊗	96+3=99	⊗
32+4=36	⊙	96+4=100	⊙

Exercise 12

General

Check what types of points are drawn for each value of the **PDMODE** variable.

## Changing the Point Size

**Menu Browser:** Format > Point Style  
**Command:** DDPTYPE

The size of a point can be set from the **Point Style** dialog box by entering the desired point size in the **Point Size** edit box, refer to Figure 3-55. You can generate the point at a specified percentage of the graphics area height or define an absolute size for the point. An absolute size for the point can be specified by selecting the **Set Size in Absolute Units** radio button in the **Point Style** dialog box and then entering a value in the **Point Size** edit box. The point size can also be set by changing the value of **PDSIZE** (Figure 3-57). The variable **PDSIZE** governs the size of the point (except for the **PDMODE** values of 0 and 1). You can set the size in absolute units by specifying a positive value for the **PDSIZE** variable.

If the **Set Size Relative to Screen** radio button is selected in the **Point Style** dialog box, the size is taken as a percentage of the viewport size. This can also be set by entering a negative value for the **PDSIZE** variable. For example, a setting of 5 makes the point 5 units high; a setting of -5 makes the point 5 percent of the current drawing area.



**Figure 3-57** Changing the point size using the **PDSIZE** variable

### Exercise 13

General

- Try various combinations of the **PDMODE** and **PDSIZE** variables.
- Check the difference between the points generated from negative values of **PDSIZE** and the points generated from positive values of **PDSIZE**.

## DRAWING INFINITE LINES

The **XLINE** and **RAY** commands can be used to draw construction or projection lines. These are the lines that aid in construction or projection and are drawn very lightly, when drafting manually. An **xline** (construction line) is a 3D line that extends to infinity at both ends. As the line is infinite in length, it does not have any endpoints. A **ray** is a 3D line that extends to infinity at only one end. The other end of the ray has 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 commands that change the drawing extents, such as the **ZOOM** command with the **All** option. Most of the object snap modes work with both xlines and rays, with some limitations: You cannot 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**

*Xlines and rays plot like any other objects in a drawing may create confusion. Therefore, it is a good idea to create the construction lines in a different layer altogether, so that you can recognize them easily. You will learn about layers in later chapters.*

## Drawing XLINE

**Ribbon:** Home > Draw > Construction Line  
**Menu Browser:** Draw > Construction Line  
**Toolbar:** Draw > Construction Line  
**Tool Palettes:** Draw > Construction Line  
**Command:** XLINE

You can choose the **Construction Line** button from the **Draw** panel of the **Ribbon** to invoke the **XLINE** command, refer to Figure 3-58. The prompt sequence on invoking the **XLINE** command is as follows:

Command: **XLINE**

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

The various options of the **XLINE** command are discussed next.

### Point

If you use the default option, AutoCAD LT will prompt you to select two points through which the xline shall pass at the **Specify a point** and the **Specify through point** prompts. After you select the first point, AutoCAD LT will dynamically rotate the xline through the specified point, as you move the cursor. When you select the second point, an xline will be created that passes through the first and second points (Figure 3-59).

Command: **XLINE**

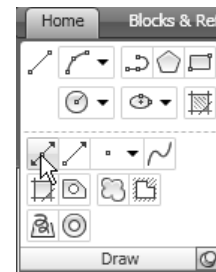
Specify a point or [Hor/Ver/Ang/Bisect/Offset]: *Specify a point.*

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 end the command.

### Horizontal

This option will 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, see Figure 3-60. As you invoke this option, the horizontal xline gets attached to the cursor. You are prompted to select only one point through which the horizontal xline passes. You can continue selecting points to draw horizontal xlines and right-click or press ENTER to end the command.



**Figure 3-58** Choosing the **Construction Line** button from the **Ribbon**

## Vertical

This option will 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, see Figure 3-60. As you invoke this option, the vertical xline gets attached to the cursor. You are prompted to select only one point through which the vertical xline passes. You can continue selecting points to draw vertical xlines and right-click or press ENTER to end the command.

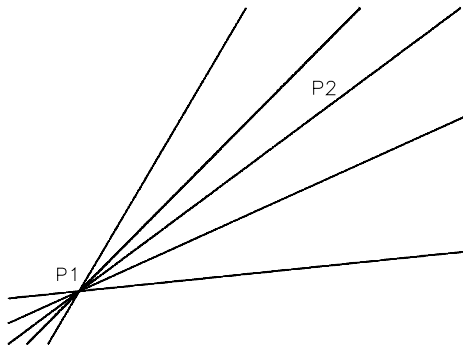


Figure 3-59 Drawing xlines

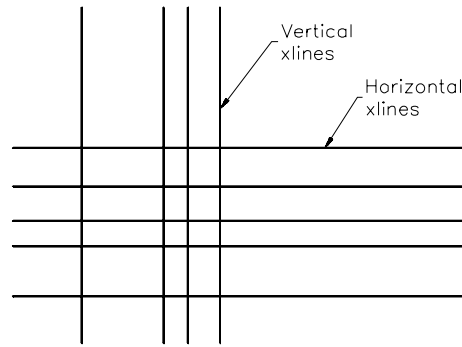


Figure 3-60 Horizontal and vertical xlines

## Angular

This option will create xlines of infinite length that pass through the selected point at a specified angle (in Figure 3-61, the angle specified is 38-degree). The angle can be specified by entering a value at the keyboard. You can also use 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.

Command: **XLINE**

\_xline Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **A**

Enter angle of xline (0) or [Reference]: **R**  (Here you use the **Reference** method for specifying 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.*

## Bisect

This option will create an xline that passes through the angle vertex and bisects the angle you specify by selecting two points. The xline created using this option will lie in the plane defined by the selected points. You can use the object snaps to select the points on the existing objects. The following is the prompt sequence for this option, refer to Figure 3-62.

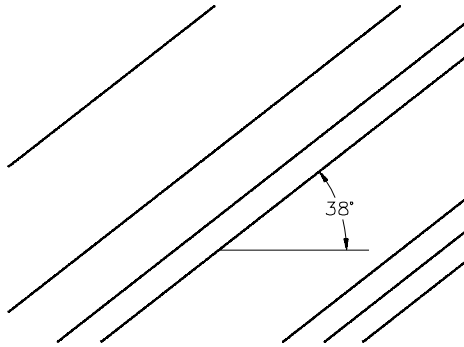
Command: **XLINE**

\_xline Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **B**

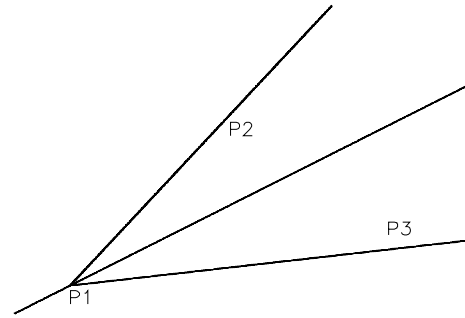
Specify angle vertex point: *Enter a point (P1).*



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 command.*



**Figure 3-61** The angular xlines



**Figure 3-62** Using the **Bisect** option to draw xlines

## Offset

The **Offset** option creates xlines that are parallel to the 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 select the **Through** option, the offset line will pass through the selected point. This option works like the **OFFSET** editing command. The prompts at the command line are as follows.

Command: **XLINE**

\_xline Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **O**

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 at a specified distance.*


Specify through point: *Select a point through which the xline should pass.*

If you specify the offset distance, and after you have selected a line object, you are prompted to specify the direction in which the xline is to be offset. You can continue drawing xlines or right-click or press ENTER to end the command.

## Drawing RAY

<b>Ribbon:</b>	Home > Draw > Ray
<b>Menu Browser:</b>	Draw > Ray
<b>Command:</b>	RAY

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

Command: **RAY** 

Specify start point: *Select the starting point for the ray.*

Specify through point: *Specify the second point.*

Press ENTER or right-click to exit the command.



#### Note


*When you trim an xline, it gets converted into a ray, and when a ray is trimmed at the end that is infinite, it gets converted into a line object.*

## WRITING A SINGLE LINE TEXT

**Ribbon:** Annotation > Single Line Text  
**Menu Browser:** Draw > Text > Single Line Text  
**Toolbar:** Text > Single Line Text  
**Command:** TEXT



The **TEXT** command lets you write a single line text in the drawing. Although you can write more than one lines of text using this command, but each line will be a separate text entity. After invoking this command, you need to specify the start point for the text. Then you need to specify the text height and also the rotation angle. The characters appear on the screen, as you enter them. When you press ENTER, after typing a line, the cursor automatically places itself at the start of the next line and repeats the prompt for entering another line. You can end the command by pressing the ENTER key. You can use the BACKSPACE key to edit the text on the screen while you are writing it. The prompt sequence is given next.

Command: **TEXT** 

Current text style: "Standard" Text height: 0.2000 Annotative: No

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 the first line of text in the text box displayed in the drawing window.*

*Enter the second line of text in the text box displayed in the drawing window.*

*Press ENTER.*



#### Note

*The other commands to enter text are discussed in detail in Chapter 7.*

*To move the objects, use the **MOVE** command and then select the objects, specify the base point, and the second point of displacement. The **MOVE** command is discussed in detail in Chapter 5.*

### Self-Evaluation Test

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

1. A negative value for a chord length in the **Start, Center, Length** option of the **ARC** command results in the largest possible arc, also known as the major arc. (T/F)
2. In the **ARC** command, if you do not specify a start point but just press ENTER or choose the **Continue** option, the start point and direction of the arc is taken from the endpoint and ending direction of the previous line or arc drawn on the current screen. (T/F)
3. If the **PELLIPSE** is set to 1, AutoCAD LT creates a true ellipse, also known as NURBS based (Non-Uniform Rational Bezier Spline) ellipse. (T/F)
4. 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 the ellipse. (T/F)
5. The \_\_\_\_\_ option of the **RECTANG** command allows you to draw a rectangle at a specified distance from the *XY* plane along the *Z* axis.
6. If the **FILLMODE** is set to \_\_\_\_\_, only the outlines are drawn for the new polyline.
7. You can get a \_\_\_\_\_ polyline by entering two different values at the starting width and the ending width prompts.
8. In case of a(n) \_\_\_\_\_ polyline, the vertices are not stored as separate entities, but as a single object with an array of information.
9. If you invoke the \_\_\_\_\_ command from the **Draw** panel of the **Ribbon**, you can draw as many points as you desire in a single command.
10. The size of a point is taken as a percentage of the viewport size, if you enter a \_\_\_\_\_ value for the **PDSIZE** variable.

### Review Questions

Answer the following questions:

1. Using the **Start, End, Angle** option of the **ARC** command, a negative included angle value draws the arc in a clockwise direction. (T/F)
2. When the **Continue** option of the **ARC** command is used to draw arcs, each successive arc is perpendicular to the previous one. (T/F)

3. If you specify the chamfer distances first and then specify the fillet radius in the same **RECTANG** command, the rectangle will be drawn with chamfers only. (T/F)
4. Using the **RECTANG** command, the rectangle drawn is treated as a combination of different objects, therefore the individual sides can be edited independently. (T/F)
5. Using the **Start, Center, Length** option of the **ARC** command, a positive chord length generates the smallest possible arc (minor arc) with this length, and the arc is always less than
- (a) 90-degree
  - (b) 180-degree
  - (c) 270-degree
  - (d) 360-degree
6. Which one of the following options of the **RECTANG** command allows you to draw a rectangle that is extruded in the Z direction by the specified value?
- (a) **Elevation**
  - (b) **Thickness**
  - (c) **Extrude**
  - (d) **Width**
7. Which of the following commands draws a line anywhere in 3D space, which starts from a point that you specify and the other end extending to infinity?
- (a) **PLINE**
  - (b) **RAY**
  - (c) **XLINE**
  - (d) **MLINE**
8. If the old 2D polylines should not get converted to lightweight polylines, when opening the drawings in AutoCAD LT 2009, and also the new polylines drawn should be 2D polylines, the **PLINETYPE** variable should be set to which of the following values?
- (a) 0
  - (b) 1
  - (c) 2
  - (d) 3
9. Which of the following values should be assigned to the **PDMODE** variable such that a cross mark (X) is generated through the specified point?
- (a) 1
  - (b) 3
  - (c) 5
  - (d) 7
10. A polygon is said to be \_\_\_\_\_, when it is drawn inside an imaginary circle and its vertices (corners) touch the circle.
11. If additional polyline segments are added to the first polyline, AutoCAD LT automatically makes the \_\_\_\_\_ of the first polyline segment the start point of the next polyline segment.

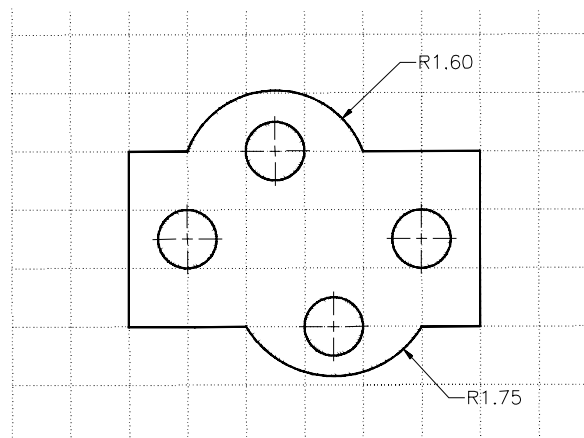
12. To create a solid filled circle by the **DONUT** command, the value of the inside diameter of the circle should be \_\_\_\_\_.
13. With the **DONUT** command, you can draw a solid-filled circle by specifying the inside diameter as \_\_\_\_\_ and keeping the **FILLMODE** on.
14. The visibility of blips can be controlled using the \_\_\_\_\_ system variable.
15. An absolute size for the point can be specified by entering a \_\_\_\_\_ value for the **PDSIZE** variable.
16. The \_\_\_\_\_ option of the **XLINE** command creates xlines of infinite length that are parallel to the *Y* axis of the current UCS.
17. You can use the \_\_\_\_\_ key to edit the text on the screen, while you are writing it using the **TEXT** command.

## Exercises

### Exercise 14

*Mechanical*

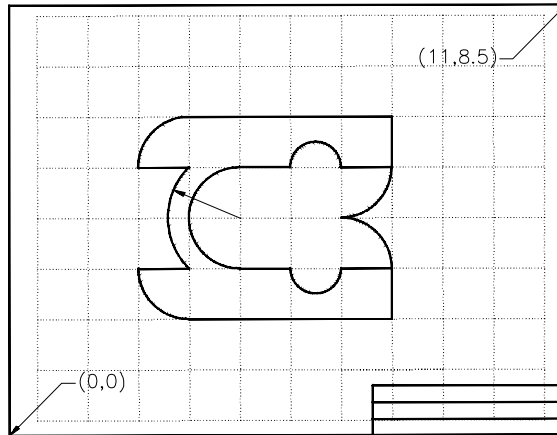
Draw the sketch shown in Figure 3-63. The distance between the dotted lines is 1.0 unit. Create the radii, using appropriate **ARC** command options.



**Figure 3-63** Drawing for Exercise 14

**Exercise 15***Graphics*

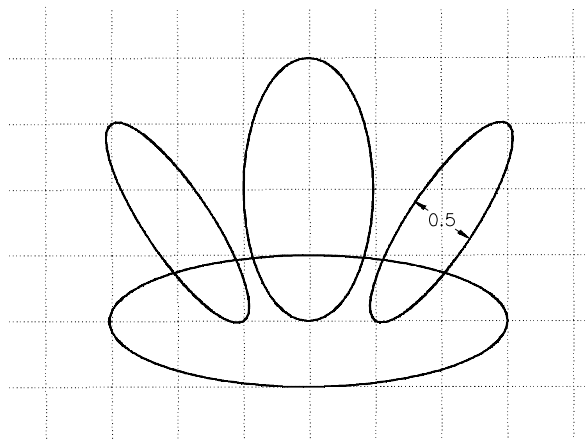
Draw the sketch shown in Figure 3-64. The distance between the dotted lines is 1.0 unit. Create the arcs, using appropriate **ARC** command options.



*Figure 3-64 Drawing for Exercise 15*

**Exercise 16***Mechanical*

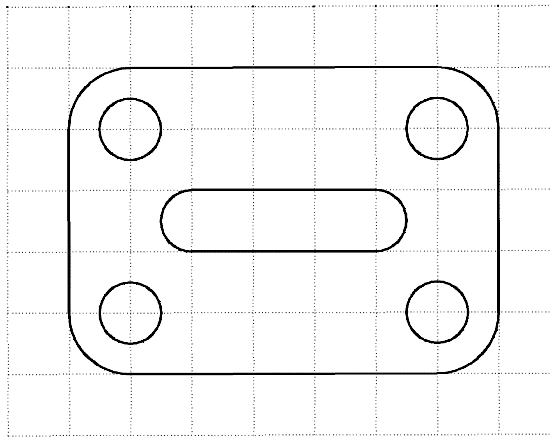
Draw the sketch shown in Figure 3-65. The distance between the dotted lines is 0.5 unit. Create the ellipses, using the **ELLIPSE** command.



*Figure 3-65 Drawing for Exercise 16*

**Exercise 17***Mechanical*

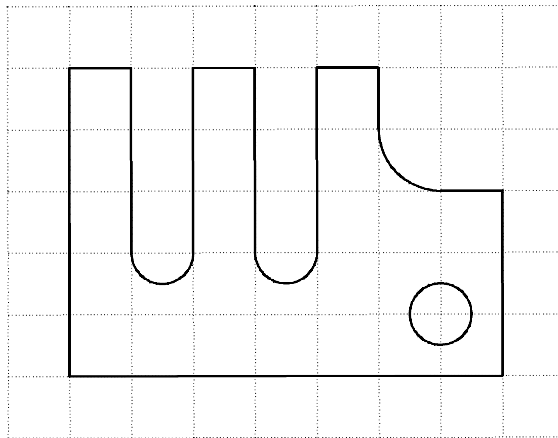
Draw Figure 3-66 using the **LINE**, **CIRCLE**, and **ARC** commands. The distance between the dotted lines is 1.0 unit and the diameter of the circles is 1.0 units.



*Figure 3-66 Drawing for Exercise 17*

**Exercise 18***Mechanical*

Draw Figure 3-67 using the **LINE**, **CIRCLE**, and **ARC** commands or their options. The distance between the grid lines is 1.0 unit and the diameter of the circle is 1.0 unit.



*Figure 3-67 Drawing for Exercise 18*

### Problem-Solving Exercise 1

*Mechanical*

Draw the sketch shown in Figure 3-68. Create the radii by using the arc command options indicated in the drawing. (Use the @ symbol to snap to the previous point. Example: Specify start point of arc or [Center]: @)

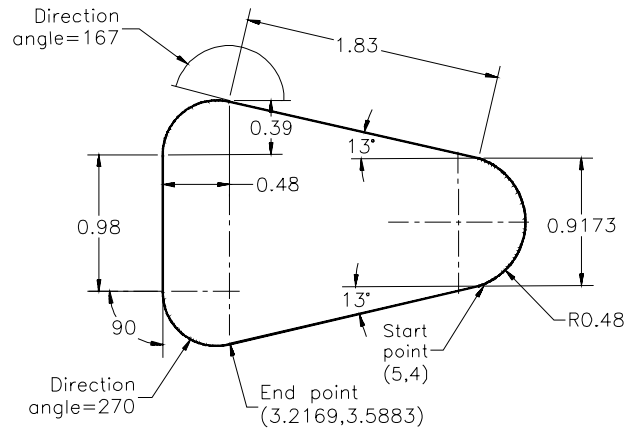


Figure 3-68 Drawing for Problem-Solving Exercise 1

### Problem-Solving Exercise 2

*Mechanical*

Draw the sketch shown in Figure 3-69. Create the radii by using the arc command options. The distance between the dotted lines is 0.5 units.

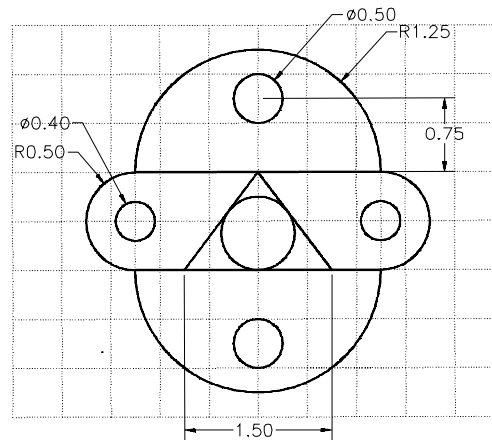


Figure 3-69 Drawing for Problem-Solving Exercise 2



### Problem-Solving Exercise 3

*Mechanical*

Draw the sketch shown in Figure 3-70. Create the radii by using the **ARC** command options. The distance between the dotted lines is 1.0 unit.

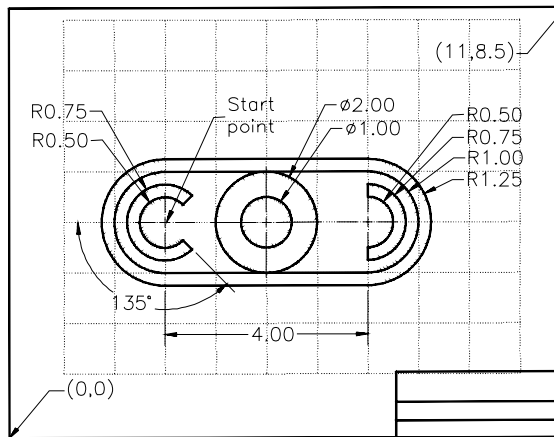


Figure 3-70 Drawing for Problem-Solving Exercise 3

### Problem-Solving Exercise 4

*Mechanical*

Draw the sketch shown in Figure 3-71 using the **POLYGON**, **CIRCLE**, and **LINE** commands.

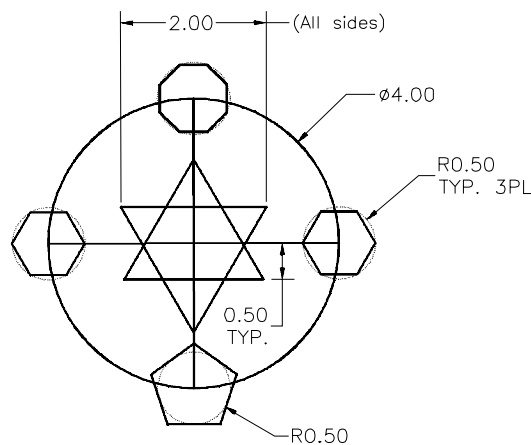


Figure 3-71 Drawing for Problem-Solving Exercise 4

**Problem-Solving Exercise 5***Mechanical*

Draw the sketch shown in Figure 3-72 using the draw commands. Note,  $\sin 30 = 0.5$ ,  $\sin 60 = 0.866$ . The distance between the dotted lines is 1 unit.

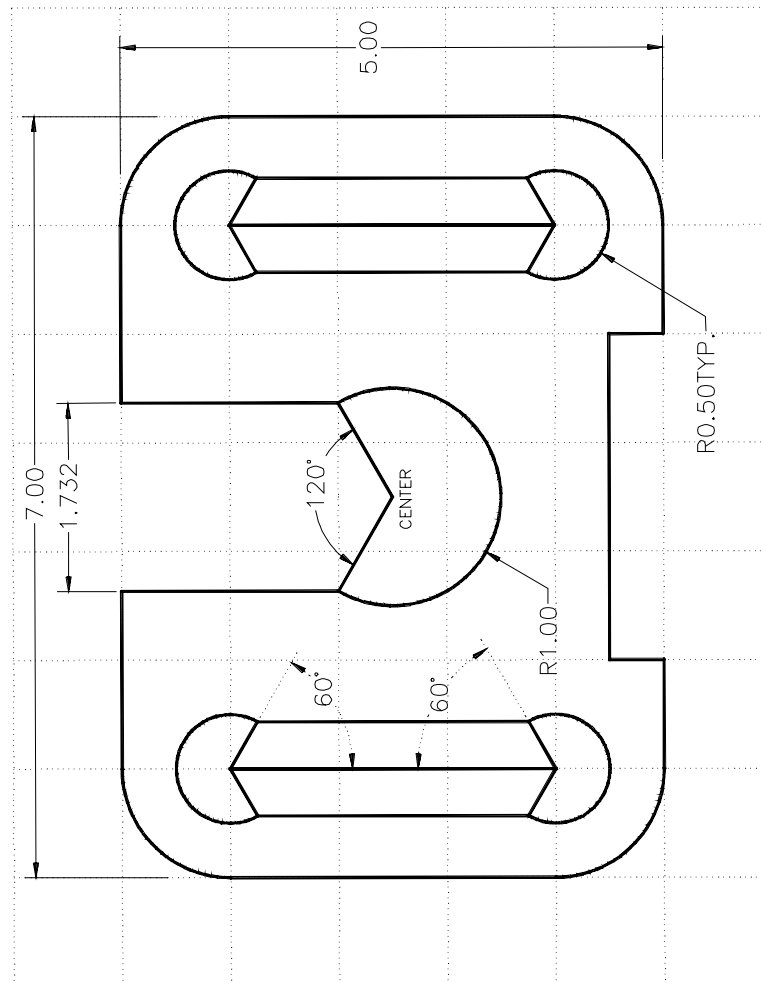


Figure 3-72 Drawing for Problem-Solving Exercise 5

**Answers to Self-Evaluation Test**

1. T, 2. T, 3. F, 4. F, 5. Elevation, 6. 0, 7. tapered, 8. optimized, 9. Multiple Point, 10. negative