Chapter 2

Getting Started with AutoCAD

CHAPTER OBJECTIVES

In this chapter, you will learn:

- *To draw lines using the LINE command and its options.*
- *Various coordinate systems used in AutoCAD.*
- *To clear the drawing area by using the ERASE command.*
- *The two basic object selection methods: Window and Crossing.*
- *To draw circles using the options of the CIRCLE command.*
- *To use the ZOOM and PAN display commands.*
- *To set up units using the UNITS command.*
- *To set up and determine limits for a given drawing.*
- *To plot drawings using the basic plotting options.*
- *To use the Options dialog box to specify settings.*

KEY TERMS

Line
Coordinate Systems
Absolute Coordinate Systems
Relative Coordinate Systems
Direct Distance Entry
Erase
Object Selection
Circle
Zoom
Pan
Units Format
Limits
Options
Plot

DRAWING LINES IN AUTOCAD

Ribbon: Home > Draw > Line **Toolbar:** Draw > Line **Menu Bar:** Draw > Line
Tool Palettes: Draw > Line **Command:** LINE or L

A line is defined as a straight curve. The most fundamental object in a drawing is line. In AutoCAD, a line is drawn between two points by using the **LINE** command. You can invoke the **LINE** command by choosing the **Line** button from the **Draw** panel of the **Home** tab in the **Ribbon**, as shown in Figure 2-1. Besides this, you can choose the **Line** button from the **Draw** tab of the **Tool Palettes**. To invoke the **Tool Palettes**, choose **View > Palettes > Tool Palettes** from the **Ribbon**, as shown in Figure 2-2. Alternatively, you can invoke the **LINE** command by choosing the **Line** button from the **Draw** toolbar, as shown in Figure 2-3. However, the **Draw** toolbar is not displayed by default. To invoke this toolbar, choose **Tools > Toolbars > AutoCAD > Draw** from the menu bar.

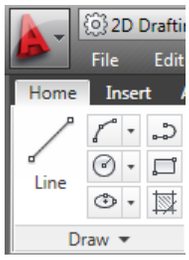

Figure 2-1 *The **Line** button in the **Draw** panel*

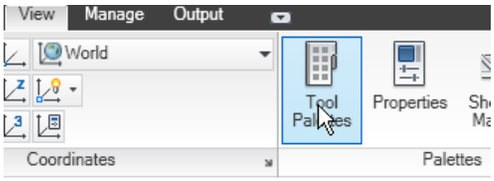

Figure 2-2 *Invoking **Tool Palettes** from the **Ribbon**.*



Figure 2-3 *The **LINE** button in the **Draw** toolbar*

You can also invoke the **LINE** command by entering **LINE** or **L** (L is the command alias of the LINE) at the Command prompt. On invoking the **LINE** command, you will be prompted to specify the starting point of the line. Select a point using the mouse or enter its coordinates at the command prompt. After specifying the first point, you will be prompted to enter the second point. Specify the second point; a line will be drawn. Now, you may continue to specify points and draw lines or terminate the **LINE** command by pressing ENTER, ESC, or SPACEBAR. You can also right-click to display the shortcut menu and choose the **Enter** or **Cancel** options to exit from the **LINE** command. After terminating the **LINE** command, AutoCAD will again display the Command prompt. The prompt sequence for the drawing shown in Figure 2-4 is given next.

Start a new file with the *acad.dwt* template in the **2D Drafting & Annotation** or **AutoCAD Classic** workshop.

Command: LINE Enter

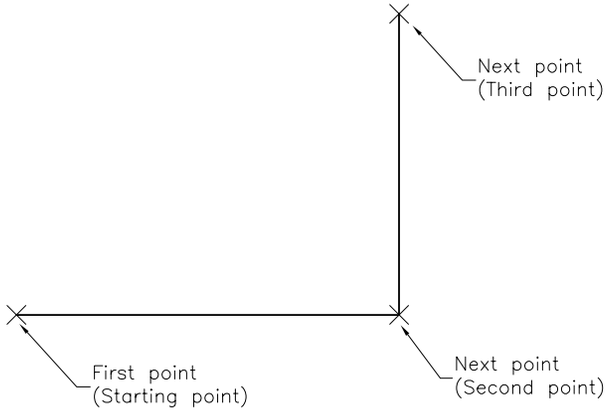

Figure 2-4 *Drawing lines using the **LINE** command*

Specify first point: *Move the cursor (mouse) and left-click to specify the first point.*

Specify next point or [Undo]: *Move the cursor and left-click to specify the second point.*

Specify next point or [Undo]: *Specify the third point.*

Specify next point or [Close/Undo]: Enter *(Press ENTER to exit the **LINE** command.)*



Note

When you select the start point of the line by pressing the left mouse button, a rubber-band line stretches between the selected point and the current position of the cursor. This line is sensitive to the movement of the cursor and helps you to select the direction and the placement of the next point for the line.



Tip

*To clear the drawing area and draw new drawings, choose the **Erase** button from the **Modify** panel in the **Home** tab of the **Ribbon** or type **ERASE** at the Command prompt and press ENTER. The crosshairs will change into a box called pick box and you will be prompted to select objects. You can select an object by positioning the pick box on it and then by pressing the left mouse button. After selecting the objects, press ENTER to terminate the **ERASE** command and the selected objects will be erased. If you enter **All** at the **Select objects** prompt and press ENTER, all objects in the screen will be erased. You can use the **U (undo)** command to undo the last command. Alternatively, choose the **Undo** button from the **Quick Access** toolbar to undo the last command. (See "Erasing Objects" discussed later in this chapter.)*

The **LINE** command has the following two options and are discussed next:

Close Undo

The Close Option

After drawing two continuous lines using the **Line** command, you will notice the **Close** option is displayed at the Command prompt. The **Close** option is used to join the current point to the start point of the first line when two or more continuous lines are drawn. If you are specifying the endpoint using the mouse, then click at the start point of the first line or enter C at the command prompt as given in the Command prompt given below.

Command: ***LINE*** Enter

_line Specify first point: *Pick first point.*

Specify next point or [Undo]: *Pick second point.*

Specify next point or [Undo]: *Pick third point.*

Specify next point or [Close/Undo]: *Pick fourth point.*

Specify next point or [Close/Undo]: *C* Enter *(Joins the fourth point with the first point) See Figure 2-5.*

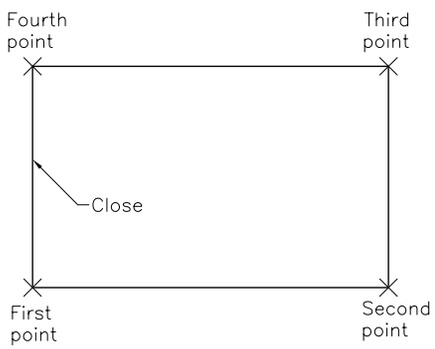

***Figure 2-5** Using the **Close** option with the **LINE** command*

You can also choose the **Close** option from the shortcut menu, which appears when you right-click in the drawing area.



Tip

After exiting the **LINE** *command, you may want to draw another line starting from the endpoint of the previous line. In such cases, press Enter twice; a new line will start from the endpoint of the previous line. You can also type the @ symbol to start the line from the last point. For example, if you have drawn a circle and then immediately start the Line command, the @ will snap to the center point of the circle.*

The Undo Option

While drawing a line, if by mistake you have specified a wrong endpoint, then you can remove that line using the **Undo** option of the **LINE** command. You can use this option multiple times and remove as much as line you want. To use this option, type **Undo** (or just **U**) at the **Specify next point or [Undo]** prompt. You can also right-click to display the shortcut menu and choose the **Undo** option.



Note

By default, whenever you open a new drawing, you need to modify the drawing display area. To modify the display area, type ***ZOOM*** *at the Command prompt and press ENTER. In the command sequence, type* ***ALL*** *and press ENTER; the drawing display is modified. You will learn more about the* ***ZOOM*** *command later in this chapter.*

COORDINATE SYSTEMS

In AutoCAD the location of a point is specified in terms of Cartesian coordinates. In this system, each point in a plane is specified by a pair of numerical coordinates. To specify a point in a plane, take two mutually perpendicular lines as references. The horizontal line is called the *X* axis, and the vertical line is called the *Y* axis. The *X* and *Y* axes divide the *XY* plane into four parts, generally known as quadrants. The point of intersection of these two axes is called the origin and the plane is called XY plane. he origin has the coordinate values of X = 0, Y = 0. The origin is taken as the reference for locating a point on the *XY* plane. Now, to locate a point, say P, and draw a vertical line such that it intersects the X axis. The horizontal distance between the origin and the intersection point is called as X coordinate of P. It will be denoted as P(x). The *X* coordinate specifies how far the point is to the left or right from the origin along the *X* axis. Now, draw a horizontal line such that it intersects the Y axis. The vertical distance between the origin and the intersection point is called as Y coordinate of P. It is denoted by P(y). The *Y* coordinate specifies how far the point is to the top or bottom from the origin along the *Y* axis. The coordinates of the point is denoted as P(x,y). T The *X* coordinate is positive, if measured to the right of the origin, and negative if measured to the left of the origin. The *Y* coordinate is positive, if measured above the origin, and negative if measured below the origin, see Figure 2-6.

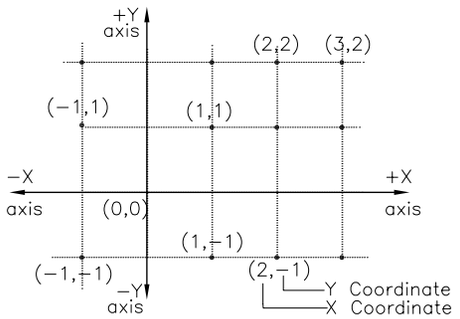

Figure 2-6 *Cartesian coordinate system*

In AutoCAD, the default origin is located at the lower left corner of the drawing area. AutoCAD uses the following coordinate systems to locate a point in an *XY* plane.

1. Absolute coordinates
2. Relative coordinates
    a. Relative rectangular coordinates
    b. Relative polar coordinates
3. Direct distance entry

If you are specifying a point by entering its location at the Command prompt then you need to use any one of the coordinate system.

Absolute Coordinate System

In the absolute coordinate system, the points are located with respect to the origin (0,0). For example, a point with X = 4 and Y = 3 is measured 4 units horizontally (distance along the *X* axis) and 3 units vertically (distance along the *Y* axis) from the origin, as shown in Figure 2-7. In AutoCAD, the absolute coordinates are specified at the Command prompt by entering *X* and *Y* coordinates, separated by a comma. However, remember that if you are specifying the coordinates using the **Dynamic Input** mode, you need to add # as the suffix to the X coordinate value. For example, enter #1,1 in the dynamic input boxes to use absolute coordinate system. The following example illustrates the use of absolute coordinates at the Command prompt to draw the rectangle shown in Figure 2-8.

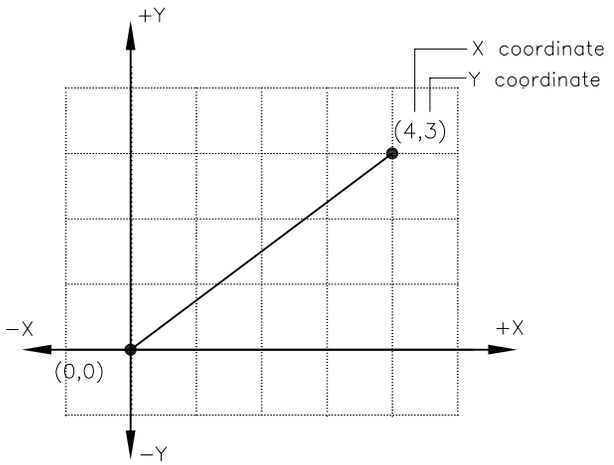

Figure 2-7 *Absolute coordinate system*



Figure 2-8 *Lines created using the absolute coordinates*

Command: **LINE** Enter

_line Specify first point: **1,1** Enter *(X = 1 and Y = 1.)*

Specify next point or [Undo]: **4,1** Enter *(X = 4 and Y = 1.)*

Specify next point or [Undo]: **4,3** Enter

Specify next point or [Close /Undo]: **1,3** Enter

Specify next point or [Close/Undo]: **C** Enter

EXAMPLE 1 ***Absolute Coordinate System***

Draw the profile shown in Figure 2-9 by using the Absolute Coordinate system. The absolute coordinates of the points are given in the following table. Save the drawing under the name *C01_Exam1.dwg*.

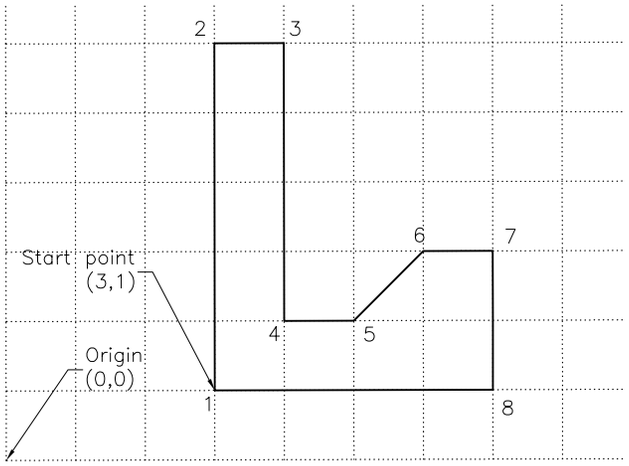

Figure 2-9 *Drawing a figure using the absolute coordinates*

Point	Coordinates	Point	Coordinates
1	3,1	5	5,2
2	3,6	6	6,3
3	4,6	7	7,3
4	4,2		

Start a new file with the ***acad.dwt*** template in the **2D Drafting & Annotation** or **AutoCAD Classic** workspace. Once the coordinates of the points are known, you can draw the sketch by using the **LINE** command. But before you proceed with drawing the object, you need to turn off the **Dynamic Input** mode, if it is on by default, so that you can view the command you are entering in the command window. The prompt sequence is given next.

Choose the **Dynamic Input** button from the Status Bar, if it is already chosen.

Command: ***ZOOM*** Enter

Specify corner of window, enter a scale factor (nX or nXP), or [All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: ***All*** Enter

Command: ***LINE*** Enter

_line Specify first point: *3,1* Enter *(Start point.)*

Specify next point or [Undo]: *3,6* Enter

Specify next point or [Undo]: *4,6* Enter

Specify next point or [Close/Undo]: *4,2* Enter

Specify next point or [Close/Undo]: *5,2* Enter

Specify next point or [Close/Undo]: *6,3* Enter

Specify next point or [Close/Undo]: *7,3* Enter

Specify next point or [Close/Undo]: *7,1* Enter

Specify next point or [Close/Undo]: *C* Enter

Choose the **Save** button from the **Quick Access** toolbar to display the **Save Drawing As** dialog box. Enter the name **Exam1** in the **File name** edit box to replace *Drawing1.dwg* and then choose the **Save** button. The drawing will be saved with the specified name in the default *Documents* folder.

EXERCISE 1 ***Absolute Coordinate System***

Draw the profile shown in Figure 2-10. The distance between the dotted lines is 1 unit. Enter the absolute coordinates of the points given in the following table. Then, use these coordinates to draw the same figure.

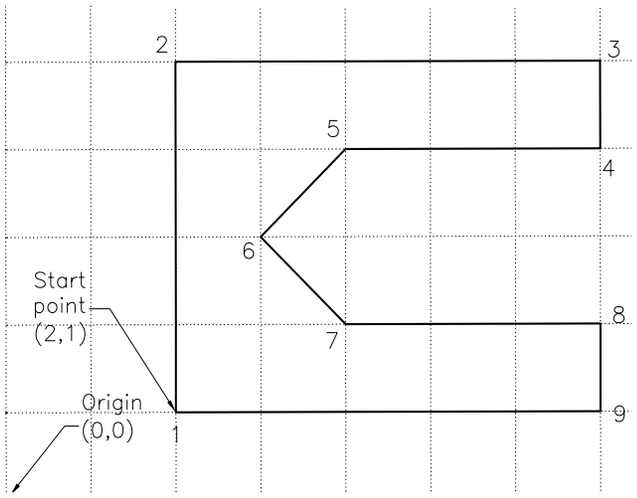

Figure 2-10 *Drawing for Exercise 1*

Point	Coordinates	Point	Coordinates
1	2, 1	6	_____
2	_____	7	_____
3	_____	8	_____
4	_____	9	_____
5	_____		

Relative Coordinate System

There are two types of relative coordinates: relative rectangular and relative polar.

Relative Rectangular Coordinates

In the relative rectangular coordinate system, the location of a point is specified with respect to the previous point and not with respect to the origin. To enter the coordinate values in terms of the Relative Rectangular Coordinate system, check whether the Dynamic Input is on or not. If the Dynamic Input is turned on, then by default the profile will be drawn using the Relative Rectangular Coordinate system. Therefore, in this case, enter the X coordinate, then type comma (,), and then enter the Y coordinate. However, if the Dynamic Input is turned off, the coordinate values has to be prefixed by the symbol @. Then, the profile will be drawn using the Relative Rectangular Coordinate system. For example, to draw a rectangle (see Figure 2-11) of length 4 units and width 3 units and the lower left corner at the point (1,1) using the Relative Rectangular Coordinate system, you need to use the following prompt sequence:

Command: ***LINE*** Enter

_line Specify first point: ***1,1*** Enter *(Start point)*

Specify next point or [Undo]: ***@4,0*** Enter

Specify next point or [Undo]: ***@0,3*** Enter

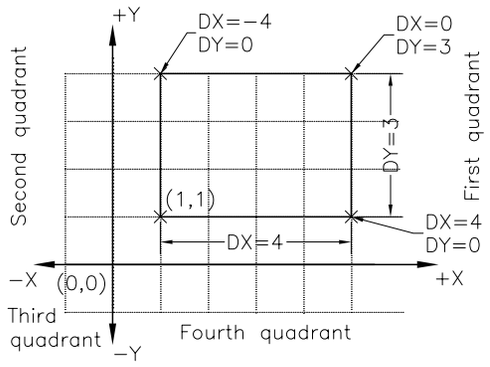

Figure 2-11 *Drawing lines using the relative rectangular coordinates*

Specify next point or [Close/Undo]:***@-4,0*** Enter

Specify next point or [Close/Undo]: ***@0,-3*** Enter

Specify next point or [Close/Undo]: Enter

Remember that if **Dynamic Input** is on, you need to use a comma (,) after entering the first value in the Dynamic Input boxes. Else, AutoCAD will take coordinates in relative polar form.

Sign Convention. As just mentioned, in the relative rectangular coordinate system the distance along the X and Y axes are measured with respect to the previous point. To understand the sign convention, imagine a horizontal line and a vertical line passing through the previous point so that you get four quadrants. If the new point is located in the first quadrant, then both the distances (DX and DY) will be specified as positive values. If the new point is located in the third quadrant, then both the distances (DX and DY) will be specified as negative values. In other words, the point will have a positive coordinate value, if it is located above or right of the an axis. Similarly, the point will have a negative coordinate value, if it is located below or left of an axis.

EXAMPLE 2 *Relative Rectangular Coordinates*

Draw the profile shown in Figure 2-12 using the relative rectangular coordinates. The coordinates of the points are given in the table below.

Point	Coordinates	Point	Coordinates
1	3,1	8	@-1,-1
2	@4,0	9	@-1,1
3	@0,1	10	@-1,0
4	@-1,0	11	@0,-2
5	@1,1	12	@1,-1
6	@0,2	13	@-1,0
7	@-1,0	14	@0,-1

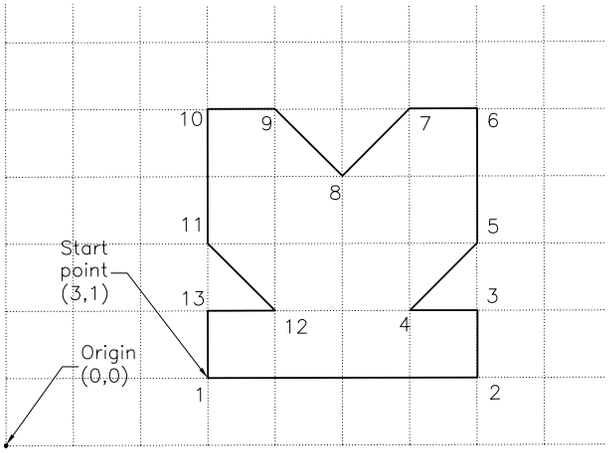

Figure 2-12 *Profile for Example* ***2***

Start a new file with the ***acad.dwt*** template in the **2D Drafting & Annotation** or **AutoCAD Classic** workspace. Before you proceed, you need to make sure the Dynamic Input is turned on.

Command: ***ZOOM*** Enter

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: ***All*** Enter

Command: ***LINE*** Enter

_line Specify first point: *Type* ***3,1*** *in the dynamic input boxes and press* Enter *(Start point)*

Specify next point or [Undo]: *Type* ***4,0*** *in the dynamic input boxes and press* Enter

Specify next point or [Undo]: *Type* ***0,1*** *in the dynamic input boxes and press* Enter

Specify next point or [Close/Undo]: *Type* ***-1,0*** *in the dynamic input boxes and press* Enter

Specify next point or [Close/Undo]: *Type* ***1,1*** *in the dynamic input boxes and press* Enter

Specify next point or [Close/Undo]: *Type* ***0,2*** *in the dynamic input boxes and press* Enter

Specify next point or [Close/Undo]: ***-1,0*** *and press* Enter

Specify next point or [Close/Undo]: ***-1,-1*** *and press* Enter

Specify next point or [Close/Undo]: ***-1,1*** *and press* Enter

Specify next point or [Close/Undo]: ***-1,0*** *and press* Enter

Specify next point or [Close/Undo]: ***0,-2*** *and press* Enter

Specify next point or [Close/Undo]: ***1,-1*** *and press* Enter

Specify next point or [Close/Undo]: ***-1,0*** *and press* Enter

Specify next point or [Close/Undo]: *0,-1 and press* Enter

Specify next point or [Close/Undo]: Enter

EXERCISE 2 *Absolute Coordinates*

For Figure 2-13, enter the relative rectangular coordinates of the points given in the following table. Then, use these coordinates to draw the figure. The distance between the dotted lines is 1 unit.

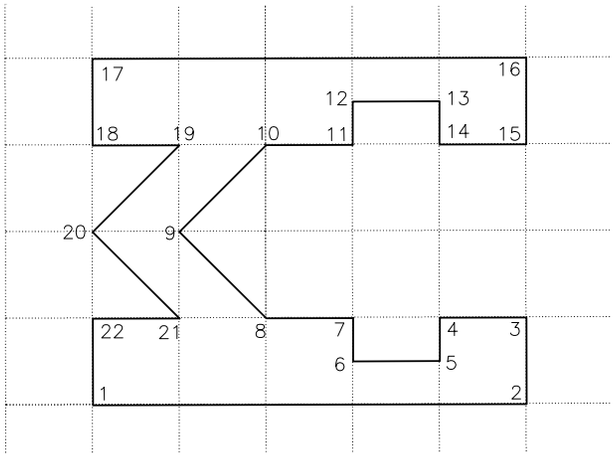

Figure 2-13 *Drawing for Exercise* ***2***

Point	Coordinates	Point	Coordinates
1	2, 1	12	_____
2	_____	13	_____
3	_____	14	_____
4	_____	15	_____
5	_____	16	_____
6	_____	17	_____
7	_____	18	_____
8	_____	19	_____
9	_____	20	_____
10	_____	21	_____
11	_____	22	_____

Relative Polar Coordinates

In the relative polar coordinate system, the location of a point is specified by defining the distance of the point from the current point and the angle between the two points with respect to the positive *X* axis. The prompt sequence to draw a line of length 5 units whose start point is at 1,1 and inclined at an angle of 30-degree to the *X* axis, as shown in Figure 2-14, is given below.

Command: ***LINE*** Enter
Specify first point: ***1,1*** Enter
Specify next point or [Undo]: ***@5<30*** Enter

If the **Dynamic Input** mode is on, by default the relative polar coordinate mode is activated. Therefore, when you choose the **LINE** command and specify the start point; two input boxes will be displayed. The second input box shows the angle value, preceded by the < symbol. Now, enter the distance value, then press the TAB key to shift to the second input box, and then enter the angle value.
Sign Convention. By default, in the relative polar coordinate system, the angle is measured from the horizontal axis as the zero degree. Also, the angle is positive if measured in a counterclockwise direction and negative if measured in a clockwise direction. Here, it is assumed that the default setup of the angle measurement has not been changed.

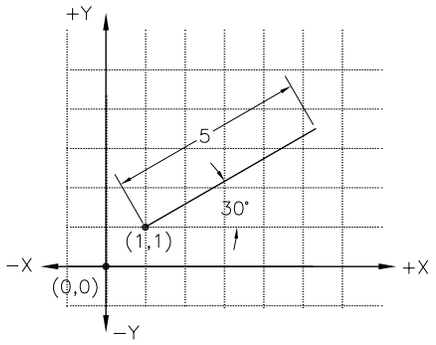

Figure 2-14 *Drawing a line using relative polar coordinates*



Note

You can modify the default settings of the angle measurement direction using the **UNITS** *command, which is discussed later.*

EXAMPLE 3 *Relative Rectangular Coordinates*

Draw the profile shown in Figure 2-15 using the relative polar coordinates. The relative coordinate values of each point is given in the table. The start point is located at 1.5, 1.75. Save this drawing as Exam3.dwg. The dimensions and the numbering are for reference only.

Point	Coordinates	Point	Coordinates
1	1.5,1.75	7	@1.0<180
2	@1.0<90	8	@0.5<270
3	@2.0<0	9	@1.0<0
4	@2.0<30	10	@1.25<270
5	@0.75<0	11	@0.75<180
6	@1.25<-90 (or <270)	12	@2.0<150

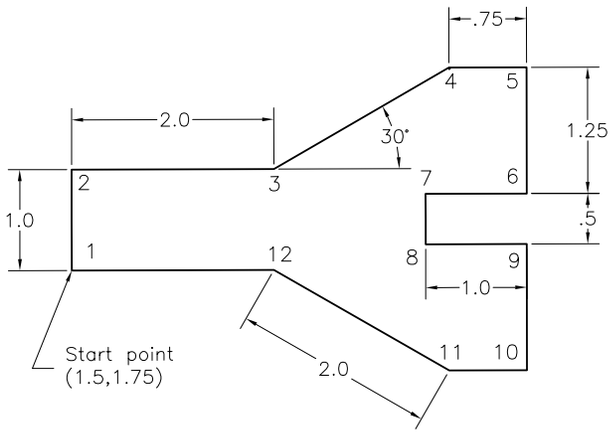

Figure 2-15 *Drawing for Example* ***3***

Start a new file with the ***acad.dwt*** template in the **2D Drafting & Annotation** or **AutoCAD Classic** workspace. Now, you need to modify the drawing display area. To do so, choose the **Zoom Extents** button from the Heads-Up toolbar. Next, turn off the dynamic input by choosing the **Dynamic Input** button from the Status Bar.

Command: *LINE* Enter
_line Specify first point: *1.5,1.75* Enter *(Start point)*
Specify next point or [Undo]: *@1<90* Enter
Specify next point or [Undo]: *@2.0<0* Enter
Specify next point or [Close/Undo]: *@2<30* Enter
Specify next point or [Close/Undo]: **@0.75<0** Enter
Specify next point or [Close/Undo]: *@1.25<-90* Enter
Specify next point or [Close/Undo]: *@1.0<180* Enter
Specify next point or [Close/Undo]: *@0.5<270* Enter
Specify next point or [Close/Undo]: *@1.0<0* Enter
Specify next point or [Close/Undo]: *@1.25<270* Enter
Specify next point or [Close/Undo]: *@0.75<180* Enter
Specify next point or [Close/Undo]: *@2.0<150* Enter
Specify next point or [Close/Undo]: *C* Enter *(Joins the last point with the first point.)*

Save this drawing by choosing the **Save** button from the **Quick Access** toolbar. The **Save Drawing As** dialog box will be displayed. Enter the name *Exam3* in the **File name** edit box and then choose the **Save** button. The drawing will be saved with the specified name in the *My Documents* folder.

EXERCISE 3

Draw the profile shown in Figure 2-16 by specifying the points using the absolute, relative rectangular, and relative polar coordinate systems. Do not dimension the profile. They are for reference only.

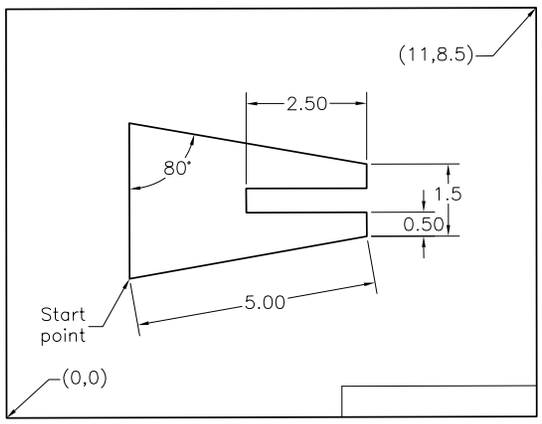

Figure 2-16 *Drawing for Exercise* ***3***

Direct Distance Entry

The easiest way to draw a line in AutoCAD is by using the Direct Distance Entry method. To draw a line by using this method, ensure that the Dynamic Input button is chosen in the Status Bar and choose the **Line** button; you will be prompted to specify the start point. Enter the coordinate values in the text box and press ENTER; you will be prompted to specify the next point. Now enter the absolute length of the line and its angle with respect to the current position of the cursor in the corresponding text boxes, as shown in Figure 2-17. Note that you can use the TAB key to toggle between the text boxes. If the **Ortho** mode is on, while drawing lines using this method, you can position the cursor only along the *X* or *Y* axis. If the **Dynamic Input** button is not chosen, then you need to enter the length of the line at the Command prompt. Therefore,

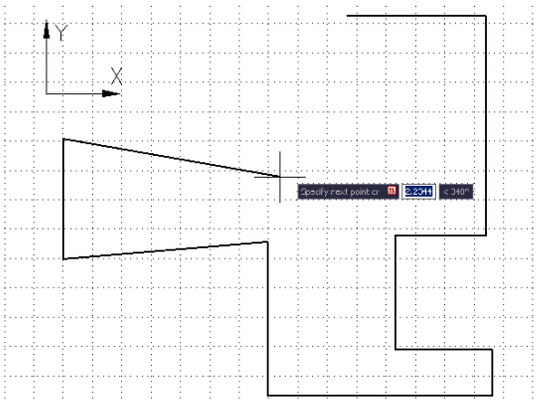

Figure 2-17 *Drawing lines using the* ***Direct Distance Entry*** *method*

position the cursor at the desired angle and type the length at the Command prompt and press ENTER, as given below.

Command: *LINE* Enter

_line Specify first point: *Start point.*

Specify next point or [Undo]: *Position the cursor and then enter distance.*

Specify next point or [Undo]: *Position the cursor and then enter distance.*

EXAMPLE 4 — Direct Distance Entry

In this example, you will draw the profile shown in Figure 2-18, using direct distance entry method. The starting point is 2,2.

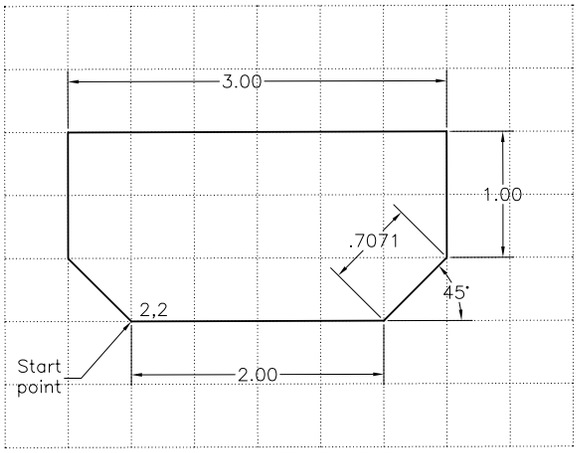

Figure 2-18 *Drawing for Example* ***4***

In this example, you will use the polar tracking option to draw lines. The polar tracking option allows you to track lines that are drawn at the specified angles. The default angle that is specified for polar tracking is 90-degree. Therefore, by default you can track lines at an angle that are in multiples 90, such as 90, 180, 270, and 360. In this example, you need to draw lines at angles that are in multiples of 45-degree such as 45, 90, 135, and so on. So, first you need to set the polar tracking angle as 45-degree.

Note

You will learn more about polar tracking in Chapter 4, Working with Drawing Aids.

1. Start a new file with the *acad.dwt* template in the **2D Drafting & Annotation** or **AutoCAD Classic** workspace.

2. To add a 45-degree angle to polar tracking, right-click on the **Polar Tracking** button on the Status Bar and choose 45 from the shortcut menu. Again choose the Polar Tracking button in the Status Bar to turn the polar tracking on.

3. Choose the **Line** button from the **Draw** panel of the **Home** tab; you are prompted to specify the start point.

4. Enter **2,1** at the command prompt and press ENTER; you are prompted to specify next point.

5. Move the cursor horizontally toward the right and when the tooltip appears as polar angle 0, type **2** and press ENTER; you are prompted to specify next point.

6. Move the cursor at an angle close to 45-degree and when the tooltip appears as polar angle 45, type **0.7071** and press ENTER; you are prompted to specify next point

7. Move the cursor vertically upward and when the tooltip appears as polar angle 90, type **1** and press ENTER; you are prompted to specify next point.

8. Move the cursor horizontally toward the left and when the tooltip appears as polar angle 180, type **3** and press ENETR; you are prompted to specify next point.

9. Move the cursor vertically downward and when the tooltip appears as polar angle 90, type **1** and press ENTER; you are prompted to specify next point.

10. Type **C** and press ENTER.



Tip

To add the angular values other than those in the shortcut menu of the polar tracking, choose **Settings** *from the shortcut menu. Next, select the* ***Additional angles*** *check box in the* ***Polar Angle Settings*** *area and choose the* ***New*** *button. Enter a new value in the field that appears and then press ENTER. Choose* ***OK*** *to close the dialog box.*

EXERCISE 4 ***Direct Distance Entry***

Use the direct distance entry method to draw a parallelogram. The base of the parallelogram equals 4 units, the side equals 2.25 units, and the angle equals 45-degree. Draw the same parallelogram using the absolute, relative, and polar coordinates. Note the differences and the advantages of using Direct Distance Entry.

ERASING OBJECTS

Ribbon: Home > Modify > Erase
Menu Bar: Modify > Erase
Command: ERASE or E
Toolbar: Modify > Erase
Tool Palettes: Modify > Erase

After drawing some objects, you may need to erase some of them. Erasing unwanted objects is done using the **ERASE** command. This command is used exactly the same way as an eraser is used in manual drafting to delete the unwanted lines. To erase, choose the **Erase** button from the **Modify** panel of the **Ribbon**, see Figure 2-19. You can also choose the **Erase** button from the **Modify** toolbar, as shown in Figure 2-20. To invoke the **Modify** toolbar, choose **Tools > Toolbars > AutoCAD > Modify** from the menu bar. When you invoke the **ERASE** command,

a small box, known as pick box, replaces the screen cursor. To erase an object, select the object using the pick box (see Figure 2-21); the selected object is displayed in dashed lines and the Select objects prompt returns. You can either continue selecting the objects or press Enter to terminate the object selection process and erase the selected objects. The prompt sequence is given next.

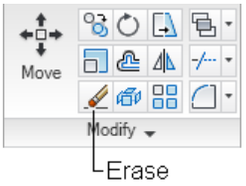

Figure 2-19 *The* ***Erase*** *button in the* ***Ribbon***

Command: *ERASE*

Select objects: *Select first object.*

Select objects: *Select second object.*

Select objects:

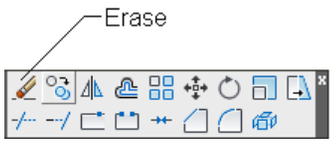

Figure 2-20 *The* ***Erase*** *button in the* ***Modify*** *toolbar*

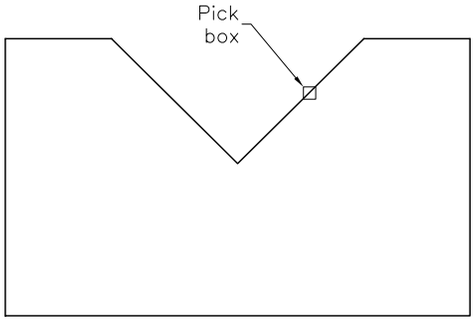

Figure 2-21 *Selecting object by positioning the pick box at the top of the object*

If you enter **All** at the **Select objects** prompt, all objects in the drawing are will be selected, even if the objects are outside the display area. Now, if you press ENTER, all selected objects will be erased.

You can also first select the objects to be erased from the drawing and then right-click in the drawing area to display the shortcut menu. From this shortcut menu, choose the Erase option.

CANCELING AND UNDOING A COMMAND

If you erase an object by mistake, then to restore the erased object enter the **OOPS** or **UNDO** command. The **OOPS** command is used to restore the erased objects immediately **ERASE** command, refer to Figure 2-24. The **U** (Undo) command is used to undo any command.

To cancel or exit from a command, press the ESC (Escape) key on the keyboard.

OBJECT SELECTION METHODS

The usual method to select objects is selecting them individually. But this will be time-consuming, if you have a number of objects to select. This problem can be solved by creating a selection set that enables you to select several objects at a time. The selection set options can be used with those commands that require object selection, such as **ERASE** and **MOVE**. There are many object selection methods, such as **All**, **Last**, **Add**, and so on. In this chapter, you will learn two options: **Window** and **Crossing**. The remaining options are discussed in Chapter 5.

The Window Option

The window option is one of the selection options in which an object or group of objects are selected by drawing a window. The objects that are completely enclosed within the window are selected and those objects that lie partially inside the boundaries of the window are not selected.

To select the objects using the **Window** option after invoking a command, type **W** at the **Select objects** prompt and press ENTER; you will prompted to specify the first corner of the window. Select the first corner and then move the cursor to specify the opposite corner. As you move the cursor, a blue color window of continuous line is displayed that changes in size as you move the cursor. Specify the opposite corner of the window; the objects that are enclosed in this window are displayed as dashed objects. Figure 2-22 shows a window drawn to select the objects by using the **Window** option. The objects that are to be selected are shown in dashed lines.

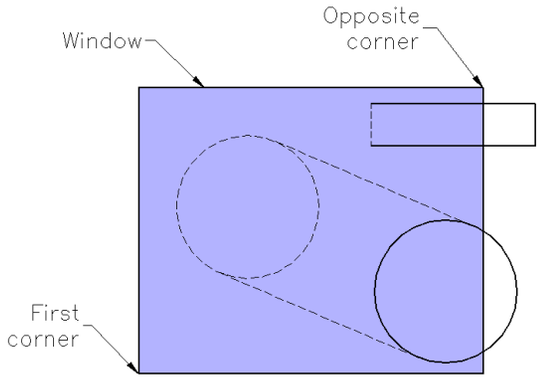

Figure 2-22 *Selecting objects using the **Window** option*

You can also invoke the **Window** option without entering **W** at the Command prompt. To do so, select a blank point on the screen at the **Select objects** prompt. This is considered as the first corner of the window. Moving the cursor to the right will display a blue shaded window. After enclosing the required objects to be selected, specify the other corner of the window. The objects that are completely enclosed within the window will be selected and displayed in dashed line. The following is the prompt sequence for automatic window selection with the **ERASE** command.

Command: ***ERASE*** Enter

Select objects: *Select a blank point as the first corner of the window.*

Specify opposite corner: *Drag the cursor to the right to select the other corner of the window.*

Select objects: Enter



Tip

*In AutoCAD, the entities are highlighted when you move the cursor over them. This feature is known as the selection preview. To set the selection preview, invoke the **Options** dialog box, then choose the **Selection** tab, and then the **Visual Effects Settings** button in the **Selection Preview** area; the **Visual Effect Settings** dialog box will be displayed. Set the type of selection preview by selecting the appropriate radio button in the Line Highlighting area of this dialog box.*

The Crossing Option

The crossing option is one of the selection methods in which an object or group of objects that are partially enclosed by the selection window are selected. The objects to be selected should be touching the window boundaries or completely enclosed within it. To select the objects by using the crossing option after invoking a command, type **C** at the **Select objects** prompt and press ENTER; you are prompted to select the first corner of the window. Select the first corner and then move the cursor to specify the opposite corner. As you move the cursor, a green color window of dashed line is displayed that changes in size as you move the cursor. Specify the opposite corner of the window; the objects that touches the window boundaries and that are enclosed by this window are selected as displayed as dashed objects. Figure 2-23 shows a window drawn to select the objects by using the **Crossing** option. The objects that are to be selected are shown in dashed lines.

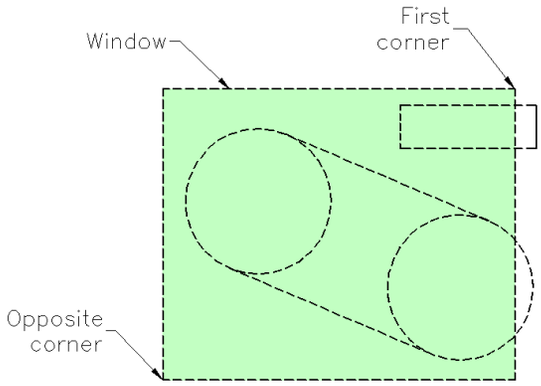

Figure 2-23 *Selecting objects using the* ***Crossing*** *option*

You can also invoke the **Crossing** option without entering **C** at the Command prompt. To do so, specify a point in the drawing area at the **Select objects** prompt and move the cursor to the left. The first point you selected becomes the first corner of the crossing window and you will be prompted to select the other corner. As you move the cursor, a green color window of dashed lines is displayed that changes in size as you move the cursor. Specify the opposite corner of the window; the objects that touches the window boundaries and that are enclosed by this window are selected as displayed as dashed objects. The prompt sequence for automatic crossing option when you choose the **Erase** command is given next.

Select objects: *Select a blank point as the first corner of the crossing window.*

Specify opposite corner: *Drag the cursor to the left to select the other corner of the crossing window.*

Select objects: Enter



Tip

You can also select the objects using the window or crossing options before invoking a command. To do so, specify the start point of the selection window and drag the cursor to enclose the objects in a window. If you have moved the cursor to the left of starting point, crossing option will be activated and that objects that are touched by the window boundaries and that are enclosed by this window are selected. Similarly, if you have moved the cursor to the right of the starting point, the window option is activated and the objects that are enclosed in this window are selected.

DRAWING CIRCLE

Command:	**CIRCLE or C**	**Toolbar:**	**Draw > Circle**
Menu Bar:	**Draw > Circle**	**Tool Palettes:**	**Draw > Circle**
Ribbon:	**Home > Draw > Center, Radius**		

A circle is drawn using the **CIRCLE** command. In AutoCAD, you can draw a circle by using six different buttons. All these buttons are grouped together in the Draw panel of the Ribbon. To view all these buttons, choose the down arrow next to the Center,Radius button in the **Draw** panel of the **Ribbon**; all buttons will be listed in a flyout, as shown in Figure 2-24. Note that the button that was last chosen will be displayed in the **Draw** panel. You can also invoke **CIRCLE** command by choosing the **Circle** button from the **Draw** toolbar or the **Tool Palettes**. The different methods to draw a circle are discussed next.

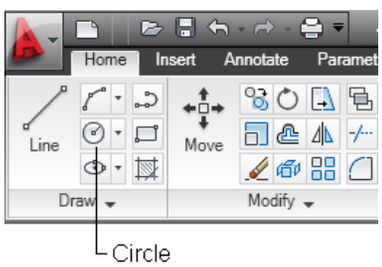

Figure 2-24 *The* ***Circle*** *button in the* ***Ribbon***

Drawing Circle by Specifying the Center and Radius

Ribbon: Home > Draw > Center, Radius

To draw a circle by specifying the center and the radius of the circle, ensure the Dynamic Input button is chosen and choose the Center, Radius button from the Draw panel; you will be prompted to specify the center. Type the coordinates and press ENTER or specify the center using the left mouse button. After specifying the center point of the circle and move the cursor to define its radius. The current radius of the circle will be displayed in the dimension input box, as shown in Figure 2-25. This radius value will change as you move the cursor. Type a radius value in the dimension input box or click to define the radius; a circle is drawn of specified radius value.

Drawing a Circle by Specifying the Center and Diameter

Ribbon: Home > Draw > Center, Radius > Center, Diameter

To draw a circle by specifying the center and the diameter of the circle, ensure the **Dynamic Input** button is chosen and choose **Center, Radius > Center, Diameter** from the **Draw** panel; you will be prompted to specify the center. Type the coordinates and press ENTER or specify the center using the left mouse button. After specifying the center point of the circle, move the cursor to define its diameter. The current diameter of the circle will be displayed in the dimension input box, as shown in Figure 2-26. This diameter value will change as you move the cursor. Type a diameter value in the dimension input box or click to define the diameter; a circle is drawn of specified diameter value.

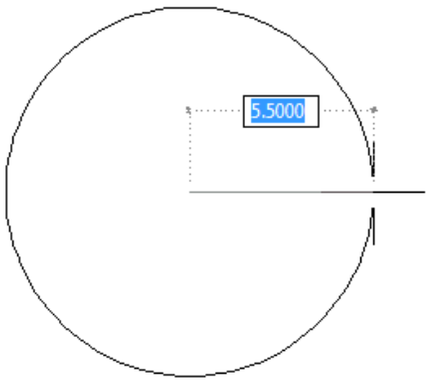

Figure 2-25 *Drawing a circle by specifying the center and radius*

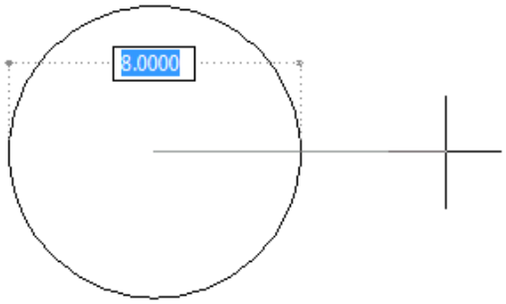

Figure 2-26 *Drawing a circle by specifying the center and diameter*

Drawing a Circle by Specifying Two Diametrical Ends

Ribbon: Home > Draw > Center, Radius > 2-Point
Command: C > 2P

You can also draw a circle, by specifying two diametrical ends of a circle, see Figure 2-27. To do so, ensure the **Dynamic Input** button is chosen and choose **Center, Radius > 2-Point** from the **Draw** panel; you will be prompted to specify the first end of the diameter. Type the coordinates and press ENTER or specify the center using the left mouse button. After specifying the center point of the circle, move the cursor to define its diameter. Now, you can type the coordinates or diameter in the dimension input box.

Drawing a Circle by Specifying Three Points on a Circle

Ribbon: Home > Draw > Center, Radius > 3-Point
Command: C > 3P

To draw a circle by specifying three points that are on the periphery of a circle, choose Center,

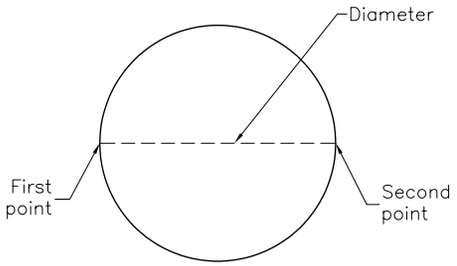

Figure 2-27 *A circle drawn using the* ***Two-Point*** *option*

Radius > 3-Point from the Draw panel and specify the three points in succession. You can type the coordinates of the points or specify using the left mouse button. The prompt sequence to type the three coordinates on choosing the 3-Point button is given below.

Specify center point for circle or [3P/2P/Ttr(tan tan radius)]: _3p Specify first point on circle: *3,3*

Specify second point on circle: *#3,1*

Specify third point on circle: *#4,2 (see Figure 2-28)*

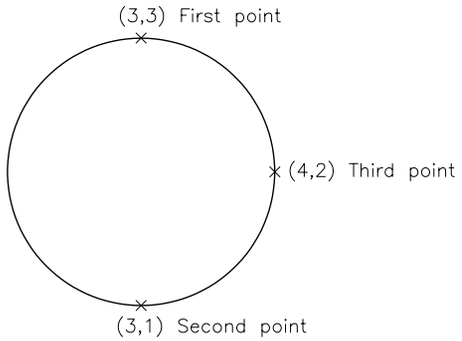

Figure 2-28 *A circle drawn using the* ***Three-Point*** *option*

You can also use relative rectangular coordinates to define the points.

Drawing a Circle Tangent to Two Objects

Ribbon: Home > Draw > Center, Radius > Tan,Tan,Tan Command: C > Ttr

A tangent is an object (line, circle, or arc) that contacts the circumference of a circle at only one point. To draw a circle of specific radius and tangent to two objects, ensure the **Dynamic Input** button is chosen and choose **Center, Radius > Tan,Tan, Radius** from the **Draw** panel; you will be prompted to specify a point on object for first tangent of circle. Move the cursor near a object that is to be tangent to the circle; a tangent symbol will be displayed. Specify the first point; you will be prompted to specify a point on object for second tangent of circle. Move the cursor near the second object that is to be tangent to the circle; a tangent symbol will be displayed. Specify the second point; you will be prompted to specify the radius. Type the radius in the dimension input box and press ENTER; a circle will be drawn of specified radius and tangent to two objects.

In Figures 2-29 through 2-32, the dotted circles represent the circle that is tangent to two objects. The circle actually drawn depends on how you select the objects that are to be tangent to the new circle. The figures show the effect of selecting different points on the objects. If you specify too small or large radius, you may get unexpected results or the "**Circle does not exist**" prompt.

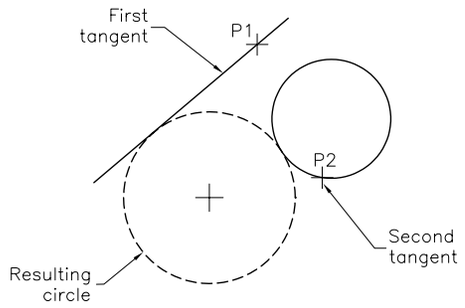

Figure 2-29 *Drawing a circle tangent to two objects*

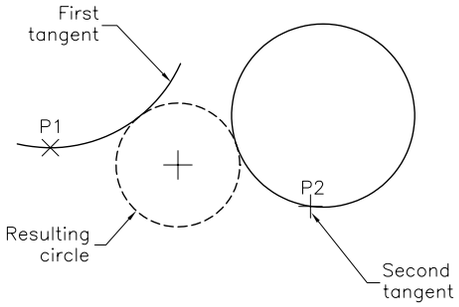

Figure 2-30 *Drawing a circle tangent to two objects*

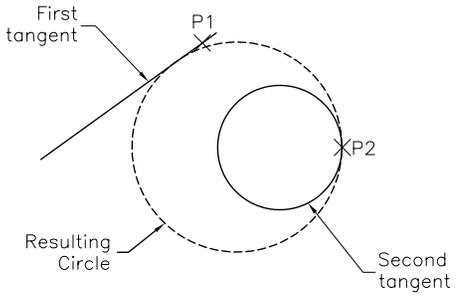

Figure 2-31 *Drawing a circle tangent to two objects*

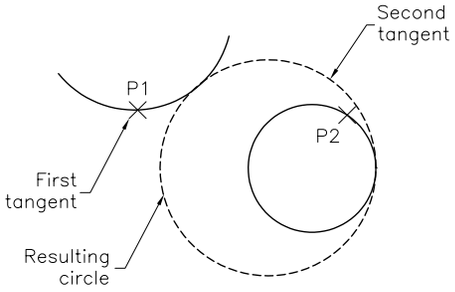

Figure 2-32 *Drawing a circle tangent to two objects*

Drawing a Circle Tangent to Three Objects

Ribbon: **Home > Draw > Center, Radius > Tan,Tan,Tan**

You can also draw a circle that is tangent to three objects. To do so, ensure the **Dynamic Input** button is chosen and choose **Center, Radius > Tan,Tan,Tan** from the **Draw** panel and select three objects in succession to which the resulting circle is to be tangent; the circle will be drawn, as shown in Figure 2-33.

EXERCISE 5 *Line and Circle*

Draw Figure 2-34 using various options of the **LINE** and **CIRCLE** commands. Use absolute, relative rectangular, or relative polar coordinates for drawing the triangle. The vertices of the triangle will be used as the center of the circles. The circles can be drawn using the **Center, Radius**, **Center, Diameter**, or **Tan, Tan, Tan** options. Do not apply dimensions; they are for reference only.

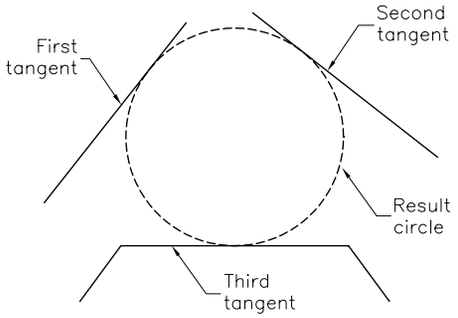

Figure 2-33 *Drawing a circle tangent to three objects*

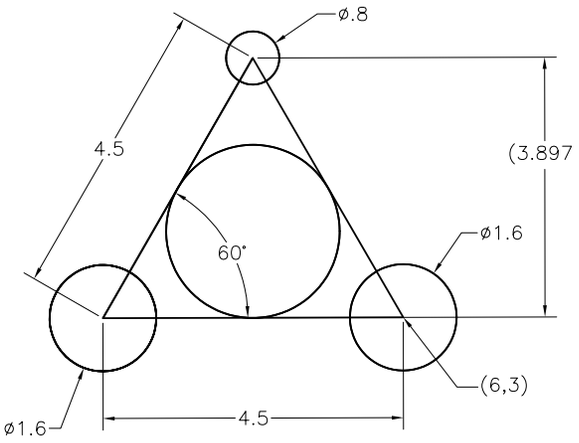

Figure 2-34 *Drawing for Exercise* ***4***

BASIC DISPLAY COMMANDS

Sometimes while drawing a sketch, it will be very difficult to view and alter minute details. You can overcome this problem by viewing only a specific portion of the drawing. This is done using the **ZOOM** command. This command lets you enlarge or reduce the size of the drawing displayed on the screen. Similarly, you may need to slide the drawing view. This is done using the **PAN** command. These are called as display command.

Zooming Drawings

The **Zoom** command is used to enlarge or reduce the view of a drawing on the screen, without affecting the actual size of entities. In AutoCAD 2011, the **ZOOM** options grouped together and are available in the **Navigator Bar**. To invoke the different **ZOOM** options, press the down arrow next to the **Zoom Extents** button in the **Navigator Bar**; all options of the **ZOOM** command will be displayed in the flyout, as shown in Figure 2-35. You can also invoke the **ZOOM** command by choosing **View > Zoom** from the menu bar. To display the menu bar, left-click on the arrow in the **Quick Access** toolbar and then select **Show Menu Bar**. Some **ZOOM** commands are also available in the **Standard** toolbar.

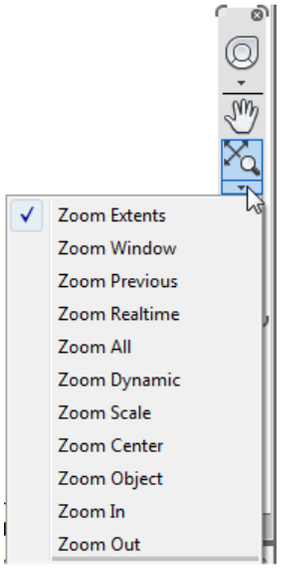

Figure 2-35 *The* ***Zoom*** *options in the* ***Navigator Bar***

Zoom Extents



Choose the **Zoom Extents** button to increase or decrease the drawing display area so that all sketched entities or dimensions are fitted inside the current view.

Zoom Window



This is the most commonly used option of the **ZOOM** command. On choosing this option, you need to draw a window by specifying its two opposite corners. The center of the zoom window becomes the center of the new display area and the objects in this window are magnified.

Zoom Realtime



The **Zoom Realtime** option is used to dynamically zoom into or out of the drawing. When you choose this option, the cursor will be replaced by the zoom cursor. To zoom

out of a drawing, press and hold the left mouse button and drag the cursor in the downward direction. Similarly, to zoom into a drawing, press and hold the left mouse button and drag the cursor in the upward direction. As you drag the cursor, the drawing display will be modified dynamically. After you get the desired view, exit this tool by right-clicking and choosing Exit from the shortcut menu or by pressing the ESC key. If you have a mouse with scroll wheel, then scroll the wheel to zoom into/out of the drawing.

Zoom Previous



While working on a complex drawing, you may need to zoom multiple times into a drawing to edit some minute details. After completing the editing if you want to view the previous views, choose the **Previous** option. You can view up to the last ten views using the **Previous** option.

Zoom In / Zoom Out



Choose the **Zoom In** / **Zoom Out** options to double the view of the image size or decrease the view of the image size by half, respectively.



Note

You will learn about the other options of the ZOOM command in detail in Chapter 6.

Moving the View



You can use **Pan** command to move the view, by sliding the view and placing it at the required position. To pan a drawing view, invoke the command from the **Navigator Bar**; an image of a hand is displayed. Drag the cursor in any direction to move the drawing. To exit **Pan** command, right-click and choose **Exit**. You can also press ESC or the ENTER key to exit the command.

SETTING UNITS TYPE AND PRECISION

Application Menu: Drawing Utilities > Units **Command: UNITS**

In the previous chapter, you learned to set units while starting a drawing by using the **Use a Wizard** option in the **Startup** dialog box. But, if you are drawing a sketch in an existing template or a new template, you need to change the format of the units for distance and angle measurements. This is done using the **UNITS** command. To set the units format, enter **UNITS** at the Command prompt; the **Drawing Units** dialog box will be displayed, as shown in Figure 2-36. You can also invoke this dialog box by choosing **Drawing Utilities > Units** from the **Application Menu** or by choosing **Format > Units** from the menu bar. The procedure to change the units format is discussed next.

Specifying the Format

In the **Drawing Units** dialog box, you can select a desired format of units from the **Type** drop-down list. You can select one of the following five formats.

Architectural (0'-01/16")	Decimal (0.00)	Engineering (0'-0.00")
Fractional (0 1/16)	Scientific (0.00E+01)	

If you select the scientific, decimal, or fractional format, you can enter the distances or coordinates in any of these three formats, but not in engineering or architectural units. If you choose the engineering or architectural format, you can enter the distances or coordinates in any of the five formats.

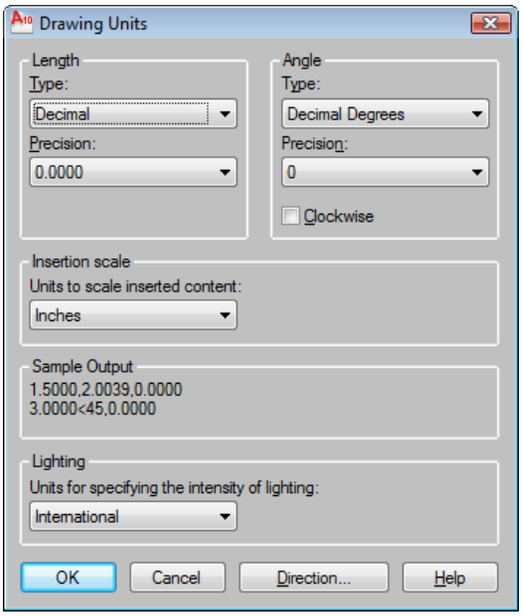

Figure 2-36 *The* ***Drawing Units*** *dialog box*



Note

The inch symbol (") is optional. For example, 1'1-3/4" is the same as 1'1-3/4, and 3/4" is the same as 3/4.

Specifying Angle Format

You can select one of the following five angle measuring formats.

1. Decimal Degrees (0.00)
2. Deg/min/sec (0d00'00")
3. Grads (0.00g)
4. Radians (0.00r)
5. Surveyor's Units (N 0d00'00" E)

If you select any one of the first four measuring formats, you can enter the angle in the **Decimal**, **Degrees/minutes/seconds**, **Grads**, or **Radians** system, but you cannot enter the angle in **Surveyor's Units**. However, if you select **Surveyor's Units**, you can enter the angles in any of the five systems. To enter the value in another system, use the appropriate suffixes and symbols, such as r (Radians), d (Degrees), or g (Grads). If you enter an angle value without any indicating the symbol of a measuring system, it is taken in the current system.

In Surveyor's units, you must specify the angle that the line makes with the north-south direction, as shown in Figure 2-37. For example, if you want to define an angle of 60-degree with north, in the Surveyor's units the angle will be specified as N 60d E. Similarly, you can specify angles such as S 50d E, S 50d W, and N 75d W, refer to Figure 2-43. You cannot specify an angle that exceeds 90-degree (N 120 E). The angles can also be specified in radians or grads, for example, 180-degree is equal to ϖ (3.14159) radians. You can convert degrees into radians or radians into degrees using the equations given below.

radians = degrees X 3.14159/180;
degrees = radians X 180/3.14159

Grads are generally used in land surveys. There are 400 grads or 360-degree in a circle. 90-degree angle is equal to 100 grads.

In AutoCAD, if an angle is measured in the counterclockwise direction, then it is positive. Also, the angles are measured from the positive X axis, see Figure 2-38. If you want the angles to be

measured as positive in the clockwise direction, select the **Clockwise** check box from the **Angle** area. You can specify the precision for the length and angle in the respective **Precision** drop-down lists in this dialog box.

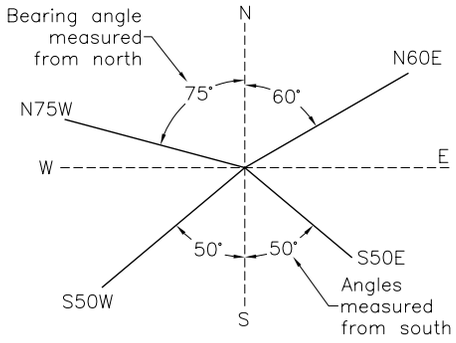

Figure 2-37 *Specifying angle in Surveyor's Units*

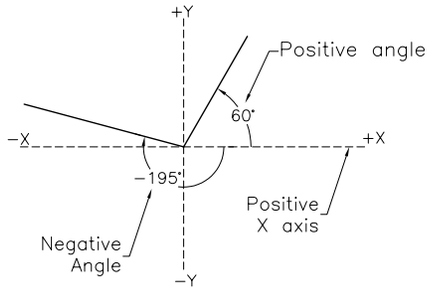

Figure 2-38 *Measuring angles*

Setting the Direction for Angle Measurement

As mentioned above the angles are measured about the positive X axis. This means that base angle (0-degree) is set along the east direction, see Figure 2-39. To change this, choose the **Direction** button in the **Drawing Units** dialog box, the **Direction Control** dialog box will be displayed, as shown in Figure 2-40. Select the appropriate radio button to specify the direction for the base angle (0-degree).

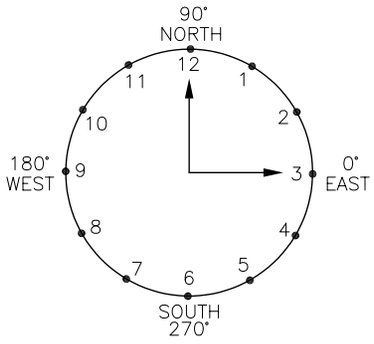

Figure 2-39 *North, South, East, and West directions*

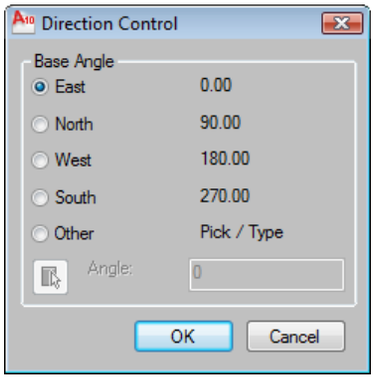

Figure 2-40 *The* ***Direction Control*** *dialog box*

If you select the **Other** option, you can set your own direction for the base angle (0-degree) by entering a value in the **Angle** edit box or by choosing the **Pick an Angle** button and picking two points to specify the angle. After specifying the base angle direction choose the **OK** button to apply the settings.



Tip

If you are entering the values using the dimension input boxes that is displayed when Dynamic Input option is chosen, then you no need to set the base angle.

Specifying the Units for the Drawing or Block to be Inserted

To set the units for a block or a drawing that has to be inserted, select a unit from the **Units to scale inserted content** drop-down list. Now, if you insert a block or a drawing from the **DesignCenter**, the specified unit will be applied to the . Even if the block was created using a different measuring unit, AutoCAD scales it and inserts it using the specified measuring unit.

If you choose **Unitless** from the drop-down list, then the units specified in the **Insertion Scale** area in the **User Prefernces** tab of the **Options** dialog box will be used.



Note

The insertion of blocks into a drawing is discussed in detail in Chapter 16.

Sample Output

The **Sample Output** area in this dialog box shows an example of the current format of the units and angles. When you change the type of length and angle measure in the **Length** and **Angle** areas of the **Drawing Units** dialog box, the corresponding example is displayed in this **Sample Output** area.

Specifying the Unit for Lighting

You can also specify the units to be used for the measurement of intensity of the photometric lights. The photometric lights are used for the rendering of the drawings. Select the desired unit system from the **Units for specifying the intensity of lighting** drop-down list in the **Lighting** area of the **Drawing Units** dialog box. While using the photometric lights in rendering, you cannot use the **Generic** unit for the measurement of the intensity of light. The concept of rendering and lights is discussed in Chapter 30.

EXAMPLE 5 *Setting Units*

In this example, you will set the units of a drawing according to the given specifications and then draw Figure 2-41. You also need to do the following.

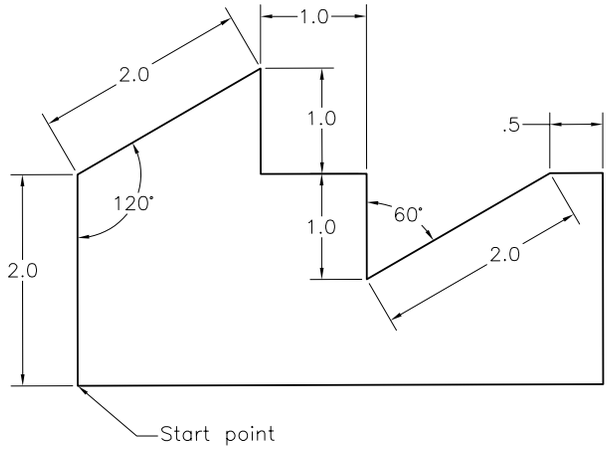

Figure 2-41 *Drawing for Example* ***5***

a. Set the units of length to fractional, with the denominator of the smallest fraction equal to 32.
b. Set the angular measurement to surveyor's units, with the number of fractional places for display of angles equal to zero.
c. Set the base angle (0-degree) to North and the direction of measurement of angles to clockwise.

The procedure of completing this example is given in the following steps.

1. Start a new file with the *acad.dwt* template in the **2D Drafting & Annotation** or the **AutoCAD Classic** workspace and invoke the **Drawing Units** dialog box by choosing the **Drawing Utilities > Units** from the **Application Menu**. You can also invoke this dialog box by entering **UNITS** at the Command prompt.

2. In the **Length** area of the dialog box, select **Fractional** from the **Type** drop-down list. From the **Precision** drop-down list, select **0 1/32**.

3. In the **Angle** area of the dialog box, select **Surveyor's Units** from the **Type** drop-down list. From the **Precision** drop-down list, select **N 0d E**, if it is not already selected. Also, select the **Clockwise** check box to set the clockwise angle measurement as positive.

4. Choose the **Direction** button to display the **Direction Control** dialog box. Next, select the **North** radio button. Choose the **OK** button to exit the **Direction Control** dialog box.

5. Choose the **OK** button to exit the **Drawing Units** dialog box.

6. With the units set, draw Figure 2-41 using the relative polar coordinates. Here, the units are fractional and the angles are measured from north (90-degree axis). Also, the angles are measured as positive in the clockwise direction and negative in the counterclockwise direction.

7. To modify the drawing display area, choose the **Zoom All** button from the **Navigator Bar**.

8. Turn off the dynamic input. The prompt sequence to complete the sketch is as follows:
   Command: ***LINE***
   Specify first point: ***2,2***
   Specify next point or [Undo]: ***@2.0<0***
   Specify next point or [Undo]: ***@2.0<60***
   Specify next point or [Close/Undo]: ***@1<180***
   Specify next point or [Close/Undo]: ***@1<90***
   Specify next point or [Close/Undo]: ***@1<180***
   Specify next point or [Close/Undo]: ***@2.0<60***
   Specify next point or [Close/Undo]: ***@0.5<90***
   Specify next point or [Close/Undo]: ***@2.0<180***
   Specify next point or [Close/Undo]: ***C***

SETTING LIMITS OF THE DRAWING

Command: LIMITS

In AutoCAD, the limits represent the drawing area and it is endless. Therefore, you need to define the drawing area before starting the drawing. In the previous chapter, you learned to set the limits while starting a drawing by using the **Use a Wizard** option in the **Startup** dialog box. If you are working in a drawing using the default template, you need to change the limits. For example, the template like *acad.dwt* has the default limits 12,9. To draw a rectangle of dimension 150x75 in this template, you need to change its limits to 200x100. This will done using the LIMITS command. The following is the prompt sequence of the **LIMITS** command for setting the limits to 24,18 for the *acad.dwt* template, which has the default limits 12,9

Command: ***LIMITS***
Reset Model space limits:
Specify lower left corner or [ON/OFF]<0.0000,0.0000>: ***0,0***
Specify upper right corner <12.0000,9.0000>: ***24,18***



Tip

Whenever you reset the drawing limits, the display area does not change automatically. You need to use the **All** *option of the* **ZOOM** *command to display the complete drawing area.*

The limits of the drawing area are usually determined by the following factors.

1. The actual size of the drawing.
2. The space needed for adding the dimensions, notes, bill of materials, and other necessary details.
3. The space between various views so that the drawing does not look cluttered.
4. The space for the border and title block, if any.

Setting Limits

To get a good idea of how to set up limits, it is always better to draw a rough sketch of the drawing. This will help in calculating the drawing area needed. For example, if an object has a front view size of 5 X 5, a side view size of 3 X 5, and a top view size of 5 X 3, the limits should be set so that they can accommodate the drawing and everything associated with it. In Figure 2-42, the space between the front and side views is 4 units and between the front and top views is 3 units. Also, the space between the border and the drawing is 5 units on the left, 5 units on the right, 3 units at the bottom, and 2 units at the top. (The space between the views and between the borderline and the drawing depends on the drawing.)

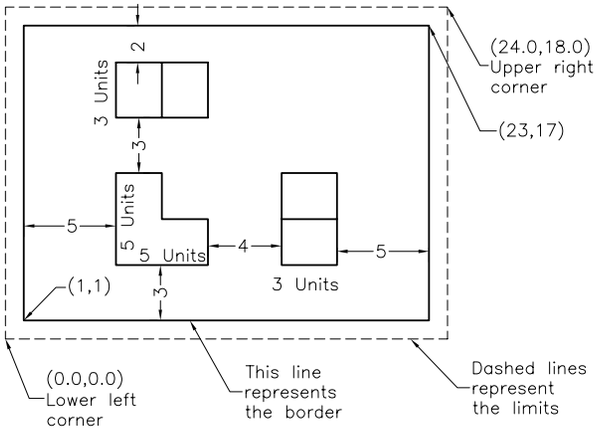

Figure 2-42 *Setting limits in a drawing*

After knowing the size of different views, the space required between views, between the border and the drawing, and between the borderline and the edges of the paper, you can calculate the space you need as follows.

Space along (*X* axis) = 1 + 5 + 5 + 4 + 3 + 5 + 1 = 24
Space along (*Y* axis) = 1 + 3 + 5 + 3 + 3 + 2 + 1 = 18

Thus, the limits you need to set for this drawing is 24 X 18. Once you have determined the space, select the sheet size that can accommodate your drawing. In the case just explained, you will select a D size (34 X 22) sheet. Therefore, the actual drawing limits will be 34,22.



Tip

To display the grid, choose the ***Grid Display*** *button in the Status Bar. By default, the grid will be displayed beyond the limits. To display the grid up to the limits, use the* ***GRID*** *command and set the* ***Display grid beyond Limits [Yes/No] <Yes>****: option to No. Now, the grids will be displayed only up to the limits set.*

Limits for Architectural Drawings

Most architectural drawings are drawn at a scale of 1/4" = 1', 1/8" = 1', or 1/16" = 1'. You must set the limits accordingly. The following example illustrates how to calculate the limits in architectural drawings.

Given

Sheet size = 24 X 18
Scale is 1/4" = 1'

Calculate limits

Scale is 1/4" = 1'
or 1/4" = 12"
or 1" = 48"
X limit = 24 X 48
= 1152" or 1152 Units
= 96'
Y limit = 18 X 48
= 864" or 864 Units
= 72'

Thus, the scale factor is 48 and the limits are 1152",864", or 96',72'.

EXAMPLE 6 *Setting Limits*

In this example, you will calculate the limits and determine an appropriate drawing scale factor for Figure 2-43. The drawing is to be plotted on a 12" X 9" sheet. Assume the missing dimensions.

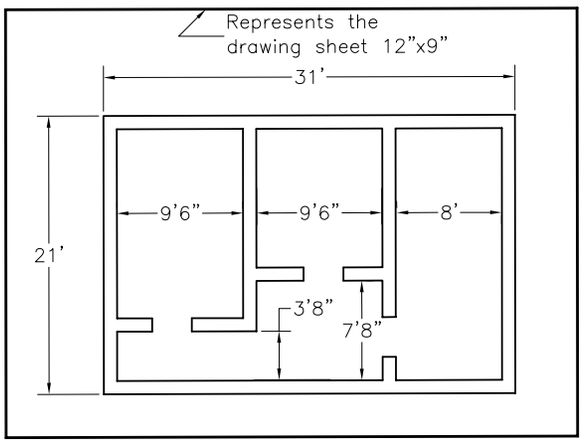

Figure 2-43 *Drawing for Example* ***6***

The calculation for the scale factor is given next.

Given or known

Overall length of the drawing = 31'
Length of the sheet = 12"
Approximate space between the drawing and the edges of the paper = 2"

Calculate scale factor

To calculate the scale factor, you have to try various scales until you find one that satisfies the given conditions. After some experience, you will find this fairly easy to do. For this example, assume a scale factor of 1/4" = 1'.

Scale factor 1/4" = 1' or 1" = 4'

Thus, a line 31' long will be = 31'/4' = 7.75" on paper. Similarly, a line 21' long = 21'/4' = 5.25". Approximate space between the drawing and the edges of paper = 2"

Therefore, total length of the sheet = 7.75 + 2 + 2 = 11.75"

Similarly, total width of the sheet = 5.25 + 2 + 2 = 9.25"

Because you selected the scale 1/4" = 1', the drawing will definitely fit on the given sheet of paper (12" x 9"). Therefore, the scale for this drawing is 1/4" = 1'.

Calculate limits

Scale factor = 1" = 48" or 1" = 4'
The length of the sheet is 12"
Therefore, X limit = 12 X 4' = 48' and Y limit = 9 X 4' = 36'

Limits for Metric Drawings

When the drawing units is in metric, you must use **standard metric size sheets** or calculate the limits in millimeters (mm). For example, if the sheet size you decide to use is 24 X 18, the limits, after conversion to the metric system, will be 609.6,457.2 (multiplying length and width by 25.4). You can round these numbers to the nearest whole numbers 610,457. Note that metric drawings do not require any special setup, except for the limits. Metric drawings are like any other drawings that use decimal units. Similar to architectural drawings, you can draw metric drawings to a scale. For example, if the scale is 1:20, you must calculate the limits accordingly. The following example illustrates how to calculate the limits for metric drawings.

Given

Sheet size = 24" X 18"
Scale = 1:20

Calculate limits

Scale is 1:20
Therefore, scale factor = 20
X limit = 24 X 25.4 X 20 = 12192 units
Y limit = 18 X 25.4 X 20 = 9144 units

EXERCISE 6 *Setting Units and Limits*

Set the units of the drawing according to the following specifications and then make the drawing shown in Figure 2-44 (leave a space of 3 to 5 units around the drawing for dimensioning and title block). The space between the dotted lines is 1 unit.

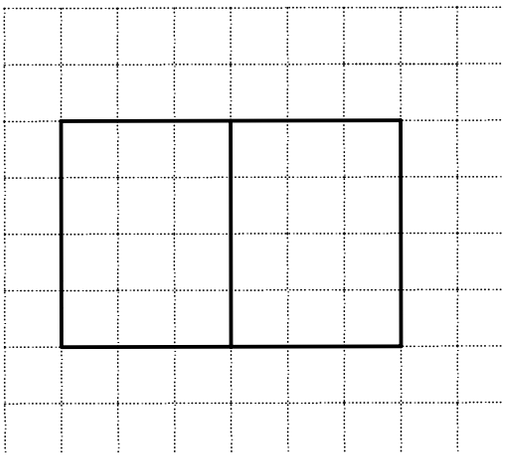

Figure 2-44 *Drawing for Exercise* ***6***

1. Set UNITS to decimal units, with two digits to the right of the decimal point.
2. Set the angular measurement to decimal degrees, with the number of fractional places for display of angles equal to 1.
3. Set the direction to 0-degree (east) and the direction of measurement of angles to counterclockwise (angles measured positive in a counterclockwise direction).
4. Set the limits leaving a space of 3 to 5 units around the drawing for dimensioning and title block.

INTRODUCTION TO PLOTTING DRAWINGS

Ribbon: Output > Plot > Plot **Toolbar:** Quick Access > Plot
Application Menu: Print > Plot **Command:** PLOT or PRINT

After creating a drawing such as an architectural plan or a mechanical component, you may need to send it to the client or have a hard copy for reference. To do so, you need to plot the drawing. To plot a drawing, choose **Plot** from the **Quick Access** toolbar; the **Plot** dialog box will be displayed. If the dialog box is not expanded by default, choose the **More Options** button at the lower right corner of the dialog box to expand it, as shown in Figure 2-45.

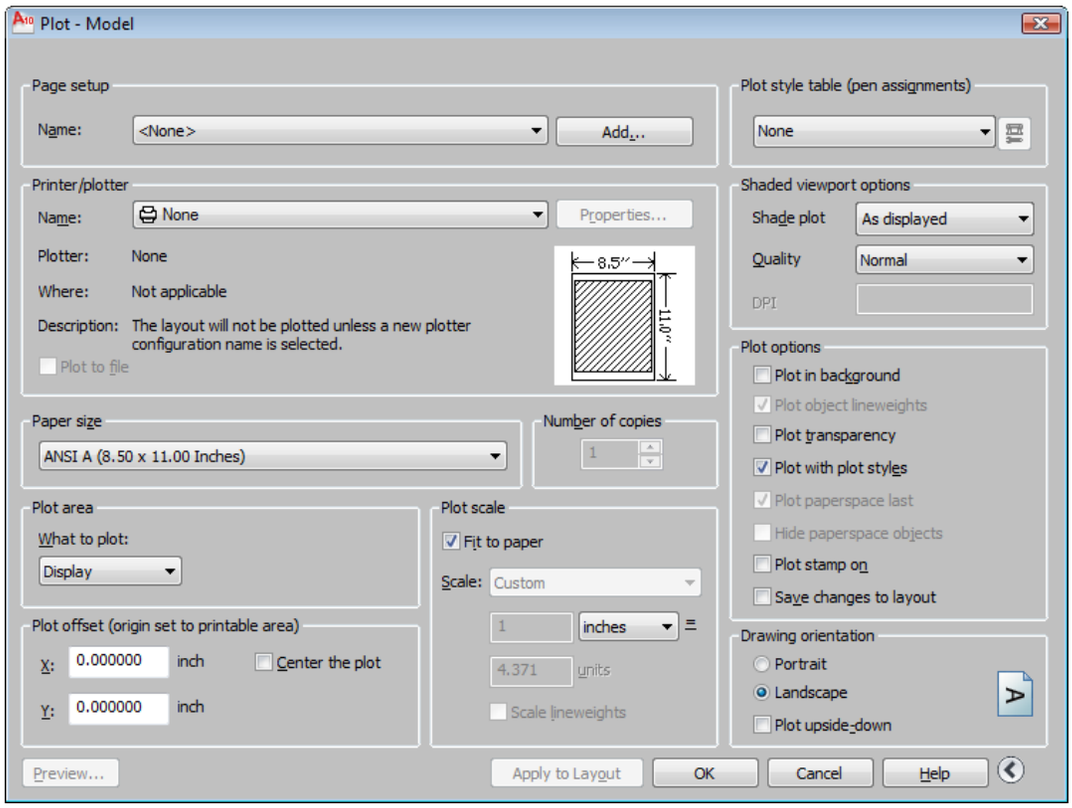

Figure 2-45 *The* ***Plot*** *dialog box*

Basic Plotting

In this section, you will learn to set up the basic plotting parameters. Later, you will learn about the advance options to plot according your plot drawing specifications. Basic plotting involves selecting the correct output device (plotter), specifying the area to plot, selecting paper size, specifying the plot origin, orientation, and the plot scale.

To learn the basic plotting, we will plot the drawing drawn in Example 3 of this chapter. It is assumed that AutoCAD is configured for two output devices: Default System Printer and Linotronic 300v52.3.

1. Invoke the **Plot** dialog box by choosing the **Plot** button in the **Quick Access** toolbar.

2. The name of the default system printer is displayed in the **Name** drop-down list in the **Printer/plotter** area. Select **Linotronic 300v52.3** in the **Name** drop-down list.

3. Select the **Window** option from the **What to plot** drop-down list in the **Plot** area area. The dialog box is temporarily closed and the drawing area will appear. Now, select two opposite corners to define a window such that it specifies the area you want to plot. Note that the complete drawing, along with the dimensions should be enclosed in the window. Once you have defined the two corners, the **Plot** dialog box will reappear.

4. To set the size of the plot, you need to select a paper size from the drop-down list in the **Paper size** area. Once you select a paper size, you need to set the orientation of the paper. To set the orientation, expand the **Plot** dialog box by choosing the **More Options** button at the lower right corner of the dialog box. You can set the orientation as **Landscape** or **Portrait** by selecting the appropriate radio button from the **Drawing orientation** area. The sections in the **Plot** dialog box related to paper size and orientation are automatically revised to reflect the new paper size and orientation. For this example, you will specify **A4** paper size and **Portrait** orientation.

5. You can also modify values for the plot offset from the **Plot offset** area; the default values for X and Y are 0. For this example, you can select the **Center the plot** check box to get the drawing in the center of the paper.

6. In AutoCAD, you can enter values for the plot scale from the **Plot scale** area. Clear the **Fit to paper** check box, if selected and then click the **Scale** drop-down list in the **Plot scale** area to display various scale factors. From this list, you can select a scale factor that you want to use. For example if you select the scale factor 1/4" = 1'-0", the edit boxes below the drop-down list will show 0.25 inches = 12 units. If you want the drawing to be plotted so that it fits on the specified sheet of paper, select the **Fit to paper** check box. When you select this check box, AutoCAD will determine the scale factor and display it in the edit boxes. For this example, we will plot the drawing so that it scales to fit the paper. Therefore, select the **Fit to paper** check box and notice the change in the edit boxes. You can also enter your own values in the edit boxes.

7. To preview a plot, choose the **Preview** button; you can preview the plot on the specified paper size. In the preview window, the realtime zoom icon will be displayed. If needed, you can zoom into or zoom out of the preview image for better visualization.

8. If the plot preview is satisfactory, you can plot your drawing by choosing **Plot** from the shortcut menu. If you want to make some changes in the settings, choose Exit in the shortcut menu or press the ESC or the ENTER key to get back to the dialog box. Change the parameters and choose the **OK** button in the dialog box to plot the drawing.

MODIFYING AUTOCAD SETTINGS USING THE OPTIONS DIALOG BOX

Application Menu: Options **Command:** OPTIONS

You can use the **Options** dialog box to change the default settings customize them to your requirements. For example, you can use this dialog box to turn off the settings to display the shortcut menu, change the display color of the objects, or specify the support directories that contain the files you need.

To invoke the **Options** dialog box, right-click at the Command prompt or in the drawing area when no command is active or no object is selected and choose **Options** from the shortcut menu; the dialog box is displayed, as shown in Figure 2-46. The current profile and current drawing

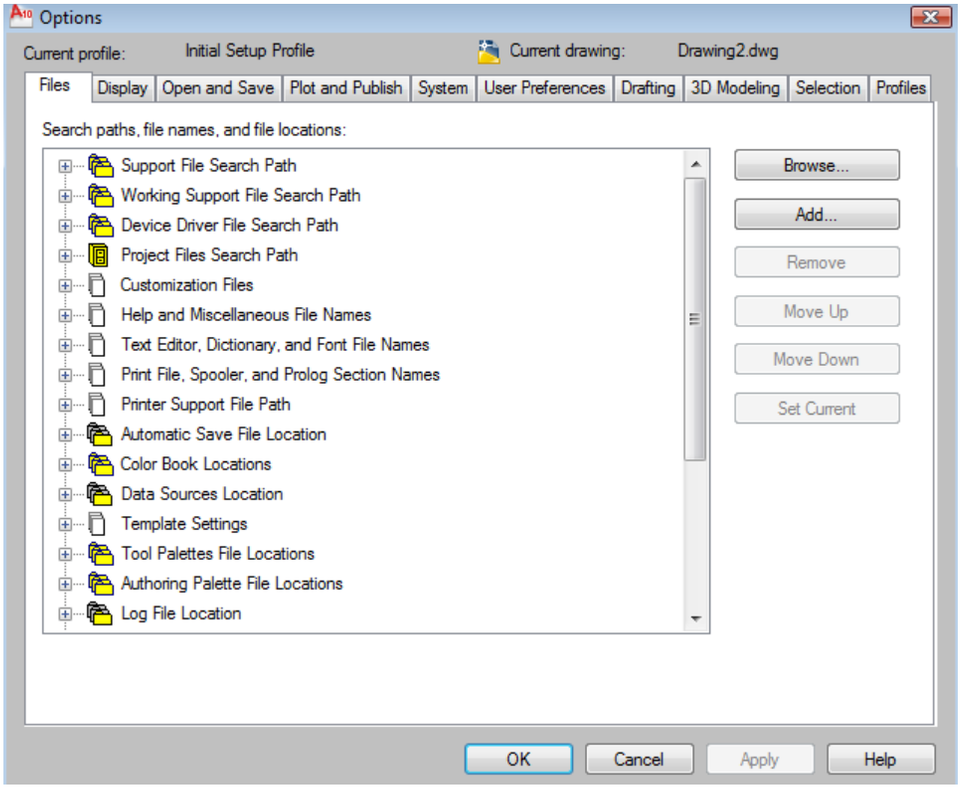

Figure 2-46 *The* ***Options*** *dialog box*

names are displayed below the name of the title bar. You can save a set of custom settings in a profile to be used later for other drawings.

Files

This tab stores the directories in which AutoCAD looks for the driver, support, menu, project, template, and other files. It uses three icons: folder, paper stack, and file cabinet. The folder icon is for a search path, the paper stack icon is for files, and the file cabinet icon is for a specific folder. Suppose you want to know the path of the font mapping file. You can select the Text Editor, Dictionary, and Font File Names folder and then select the Font Mapping Files icon to display the path, see Figure 2-46. Similarly, you can define a custom hatch pattern file and then add its search path.

Display

This tab controls the drawing and window settings like screen menu display and scroll bar. For example, if you want to display the screen menu, select the Display screen menu check box in the **Window Elements** area. You can change the color of the graphics window background, layout window background, command line background, and also the color of the command line text by using the **Drawing Window Colors** dialog box that is displayed on choosing the **Colors** button. In this tab you can specify the parameters to set the display resolution and display performance. You can also set the smoothness and resolutions of certain objects such as the circle, arc, rendered object, and polyline curve. You can also toggle on and off the display performance such as the pan and zoom with raster images, apply the solid fills, and so on. In this tab, you can toggle on and off the various layout elements such as the layout tabs on the screen, margins, paper background, and so on.

Open and Save

This tab controls the parameters related to opening and saving of files in AutoCAD. You can specify the file type for saving while using the **SAVEAS** command. The various formats are

AutoCAD 2010 Drawing (*.dwg), AutoCAD 2007 Drawing (*.dwg), AutoCAD 2004/LT 2004 Drawing (*.dwg), AutoCAD 2000/LT2000 Drawing (*.dwg), AutoCAD R14/LT98/LT97 Drawing (*.dwg), AutoCAD Drawing Template(*.dwt), AutoCAD 2007 DXF (*.dxf), AutoCAD 2004/LT2004 DXF (*.dxf), AutoCAD 2000/LT2000 DXF(*.dxf), AutoCAD R12/LT2 DXF (*.dxf), and so on. You can also set the various file safety precautions such as the Automatic Save feature, or the creation of a backup copy. To add a password and digital signatures to your drawing while saving, choose the **Security Options** button in the **File Safety Precautions** area and set the password. On selecting the **Display digital signature information** check box, you can view the digital signature information when a file with a valid digital signature is opened. You can change the number of recently saved files to be displayed in the **File Open** area. You can also set the various parameters for external references and the ObjectARX applications.

Plot and Publish

The options in this tab controls the parameters related to the plotting and publishing of the drawings in AutoCAD. You can set the default output device and also add a new plotter. You can set the general parameters such as the layout or plot device paper size and the background processing options while plotting or publishing. It is possible to select the spool alert for the system printer and also the OLE plot quality. You can also set the parameters for the plot style such as using the color-dependent plot styles or the named plot styles.

System

This tab contains AutoCAD system settings options such as the 3D graphics display and pointing device settings options where you can choose the pointing device driver. Here you can also set the system parameters such as the single drawing mode instead of MDE, the display of the Startup option while opening a new session of AutoCAD and the OLE Properties dialog box, and beep for wrong user input. You also have options to set the parameters for database connectivity.

User Preferences

The parameters in this tab is used to control the settings such as the right-click customization to change the shortcut menus according to the user preferences. You can set the units parameters for the blocks or the drawings that are inserted as well as the priorities for various coordinate data entry methods. You can also set the order of object sorting methods and also set the lineweight options.

Drafting

The options in this tab are used to control the settings such as the autosnap settings and the aperture size. Here you can also set the toggles on and off for the various autotracking settings. Using this tab, you can also set the tool tip appearance in the **Model** tab and layouts for **Dynamic Input** mode.

3D Modeling

This tab controls settings related to 3D modeling, such as the display of cursor in the 3D modeling environment, visual styles, 3D navigation, and so on.

Selection

This tab controls settings related to the methods of object selection such as the grips, which enable you to change various grip colors and the grip size. You can also set the toggles on or off for the various selection modes.

Profiles

This tab saves and restores the system settings. To save a profile and have different settings, choose the **Add to List** button; the **Add Profile** dialog box will be displayed. Enter the profile name and description. Then choose the **Apply & Close** button. Next, make the new profile current and then change the settings in each tab. The settings are saved in the new profile and can be restored anytime by making the profile current.



Note

The options in various tabs of the Options dialog box have been discussed throughout the book wherever applicable.



Tip

Some options in the Options dialog box have a drawing file icon in front of them. For example, the options in the Display resolution area of the Display tab have drawing file icons. This specifies that these parameters are saved with the current drawing only and therefore it affects only that drawing. The options without the drawing file icon are saved with the current profile and affect all drawings present in that AutoCAD session or future sessions.

EXAMPLE 7 *Modifying the Default Options*

In the following example, you will use the **Options** dialog box to create a profile that contains the specified settings.

1. Choose **Options** from the **Application Menu** to invoke the **Options** dialog box. Alternatively, right-click in the drawing area to display the shortcut menu and choose **Options** to invoke this dialog box.

2. Choose the **Profiles** tab and then choose the **Add to List** button to display the **Add Profile** dialog box. Enter the name of the new profile as **CADCIM** and **Profile created for chapter 2, Tutorial 8** as the description of the new profile, and then choose the **Apply & Close** button to exit.

3. Select the **CADCIM** profile and then choose the **Set Current** button to make this current. You will notice that the **Current Profile** name above the tabs displays *CADCIM*.

4. Choose the **Display** tab and then choose the **Colors** button; the **Drawing Window Colors** dialog box is displayed. Select the **2D model space in the Context** area and **Uniform background** in the **Interface element** area. Select **White** from the **Color** drop-down list, see Figure 2-60. This changes the color of the model tab background to white. Choose the **Apply & Close** button to return to the **Options** dialog box. Notice that the screen color changes to white.

5. Choose the **Drafting** tab and change the **AutoSnap Marker Size** to the maximum using the slider bar. Choose the **Apply** button and then the **OK** button to exit the dialog box.

6. Draw a line and then choose the **Object Snap** button from the Status Bar. Again, invoke the **LINE** command and move the cursor on the previously drawn line; a marker is displayed at the endpoint. Notice the size of the marker now.

7. Invoke the **Options** dialog box again and choose the **Profiles** tab. Double-click on the default profile to reload the default settings. The screen settings will change as specified in the default profile.

Self-Evaluation Test

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

1. You can draw a line by specifying the length of the line and its direction, using **Direct Distance Entry** method. (T/F)
2. While using the **Crossing** method of object selection, the objects that are completely enclosed within the boundaries of the crossing box are selected. (T/F)
3. The **3-Point** option of the **CIRCLE** command lets you draw a circle by specifying the two endpoints of the circle's diameter. (T/F)
4. If you choose the engineering or architectural format for units in the **Drawing Units** dialog box, you can enter the distances or coordinates in any of the five formats. (T/F)
5. You can erase a previously drawn line using the _____ option of the **LINE** command.
6. The _____option of the **CIRCLE** command can be used to draw a circle, if you want the circle to be tangent to two previously drawn objects.
7. The _____ command enlarges or reduces the view of the drawing without affecting the actual size of the entities.
8. After increasing the drawing limits, you need to choose the _____ button from the **Navigator Bar** to display the complete area inside the drawing area.
9. In _____ units, you must specify the bearing angle that the line makes with the north-south direction.
10. You can preview the plot before the actual plotting using the _____ button in the **Plot** dialog box.

REVIEW QUESTIONS

Answer the following questions:

1. In the Relative rectangular coordinate system, the displacements along the X and Y axes (DX and DY) are measured with respect to the previous point and not with respect to the origin. (T/F)
2. In AutoCAD, by default the angles are measured along the positive X axis and it will be positive if measured in the counterclockwise direction. (T/F)
3. You can also invoke the **PLOT** command by choosing **Plot** from the shortcut menu, which is displayed by right-clicking on the Command window. (T/F)
4. The **Files** tab of the **Options** dialog box stores the directories, in which AutoCAD looks for the driver, support, menu, project, template, and other files. (T/F)
5. Which of the following keys on the keyboard is used to terminate the **LINE** command at the **Specify next point or [Close/Undo]:** prompt?

(a) SPACEBAR (b) BACKSPACE
(c) ENTER (d) ESC

6. Which of the following **ZOOM** commands, zooms to the drawing limits or the extents, whichever is greater?

   (a) **Zoom Previous** (b) **Zoom Window**
   (c) **Zoom All** (d) **Zoom Realtime**

7. How many formats of units can you choose from the **Drawing** Units dialog box?

   (a) Three (b) Five
   (c) Six (d) Seven

8. Which of the following input methods cannot be used to invoke the **OPTIONS** command that displays the **Options** dialog box?

   (a) Menu (b) Toolbar
   (c) Shortcut menu (d) Command prompt

9. When you define the direction by specifying the angle, the output of the angle does not depend on which one of the following factors?

   (a) Angular units (b) Angle value
   (c) Angle direction (d) Angle base

10. The _____ option of the **LINE** command can be used to join the current point with the initial point of the first line when two or more lines are drawn in continuation.

11. The _____ option of drawing the circle cannot be invoked by entering the command at the Command prompt.

12. When you select any type of unit and angle in the **Length** or **Angle** area of the **Drawing Units** dialog box, the corresponding example is displayed in the _____ area of the dialog box.

13. If you want the drawing to be plotted so that it fits on the specified sheet of paper, select the _____ option in the **Plot** dialog box.

14. The _____ tab in the **Options** dialog box stores the details of all profiles available in the current drawing.

15. You can use the _____ command to change the settings that affect the drawing environment or the AutoCAD interface.

EXERCISE 7 *Relative Rectangular and Absolute Coordinates*

Use the following relative rectangular and absolute coordinate values in the **LINE** command to draw an object.

Point	Coordinates	Point	Coordinates
1	3.0, 3.0	5	@3.0,5.0
2	@3,0	6	@3,0
3	@-1.5,3.0	7	@-1.5,-3
4	@-1.5,-3.0	8	@-1.5,3

EXERCISE 8 *Relative Rectangular and Polar Coordinates*

Draw the profile shown in Figure 2-47, using the relative rectangular and relative polar coordinates of the points in the following table. The distance between the dotted lines is 1 unit. Save this drawing as *C02_E xer8.dwg*.

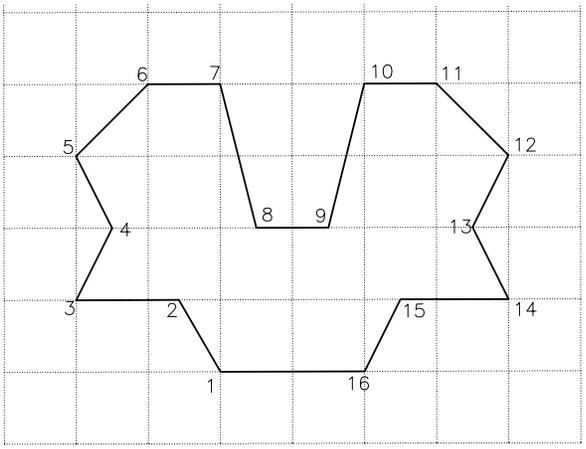

Figure 2-47 *Drawing for Exercise* 8

Point	Coordinates	Point	Coordinates	Point	Coordinates
1	3.0, 1.0	7	_____	13	_____
2	_____	8	_____	14	_____
3	_____	9	_____	15	_____
4	_____	10	_____	16	_____
5	_____	11	_____		
6	_____	12	_____		

EXERCISE 9 *Relative Polar Coordinates*

For the drawing shown in Figure 2-48, enter the relative polar coordinates of the points in the following table. Then use these coordinates to draw the figure. Do not dimension the drawing.

Point	Coordinates
1	
2	1.0, 1.0
3	_____
4	_____
5	_____
6	_____
7	_____
8	_____
9	_____

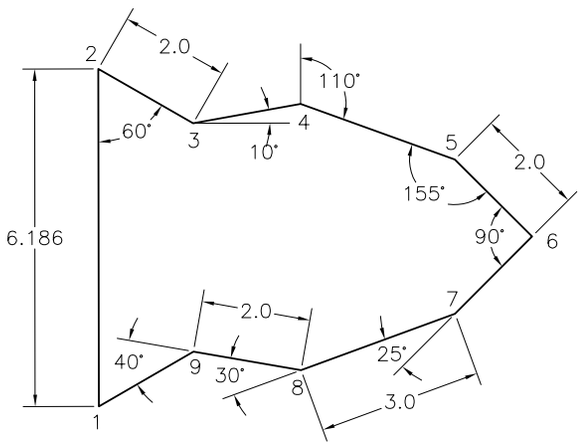

Figure 2-48 *Drawing for Exercise 9*

EXERCISE 10 — Line and Circle

Draw the sketch shown in Figure 2-49, using the **LINE** and **CIRCLE** commands. The distance between the dotted lines is 1.0 units.

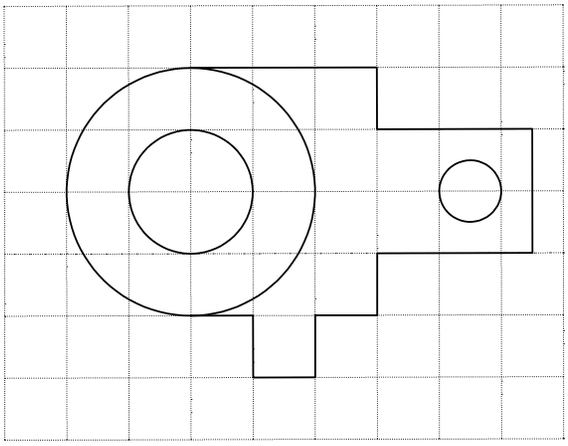

Figure 2-49 *Drawing for Exercise* ***10***

EXERCISE 11 — Line and Circle Tangent to Two objects

Draw the sketch shown in Figure 2-50 using the **LINE** command and the **Ttr** option of the **CIRCLE** command.

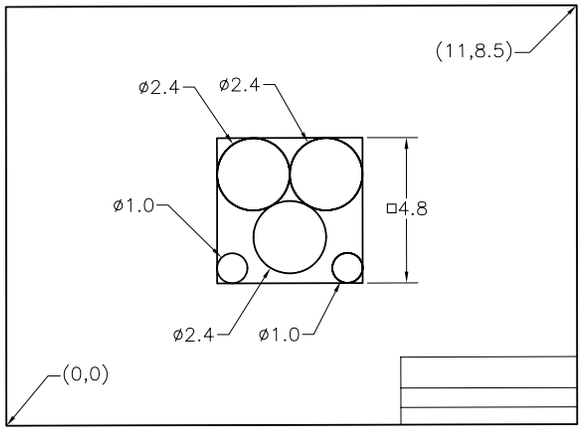

Figure 2-50 *Drawing for Exercise* ***11***

EXERCISE 12 — Setting Units

Set the units for a drawing according to the following specifications.

1. Set the UNITS to architectural, with the denominator of the smallest fraction equal to 16.
2. Set the angular measurement to degrees/minutes/seconds, with the number of fractional places for display of angles equal to 0d00'.
3. Set the direction to 0-degree (east) and the direction of measurement of angles to counterclockwise (angles measured positive in a counterclockwise direction).

EXERCISE 13 — Setting Units

Based on Figure 2-51, determine and set the limits of the drawing. The scale for this drawing is 1/4" = 1'. Leave enough space around the drawing for dimensioning and title block. (HINT: Scale factor = 48; sheet size required is 12 x 9; therefore, the limits are 12 X 48, 9 X 48 = 576, 432. Use the **ZOOM** command and then select the **All** option to display the new limits.)

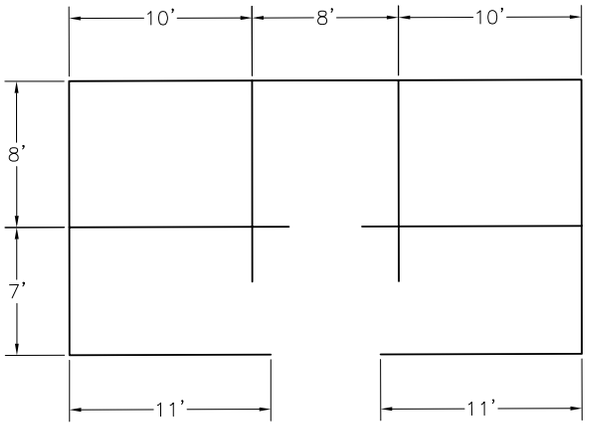

Figure 2-51 *Drawing for Exercise* ***13***

EXERCISE 14 — Using Architectural Units

Draw the sketch shown in Figure 2-52. The distance between the dotted lines is 10 feet. Determine the limits for this drawing and use the Architectural units with 0'-01/32" precision.

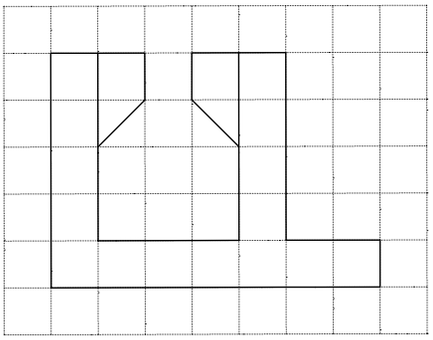

Figure 2-52 *Drawing for Exercise* ***14***

EXERCISE 15 — Using Fractional Units

Draw the object shown in Figure 2-53. The distance between the dotted lines is 5 inches. Determine the limits for this drawing and use the Fractional units with 1 1/16 precision.

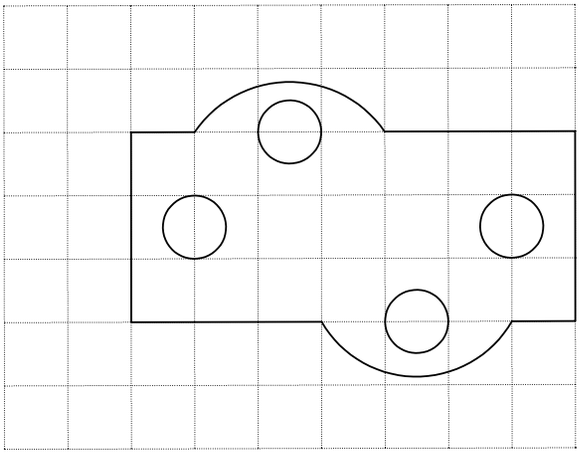

Figure 2-53 *Drawing for Exercise* ***15***

EXERCISE 16 — Using Decimal Units

Draw the object shown in Figure 2-54. The distance between the dotted lines is 1 unit. Determine the limits for this drawing and use the Decimal units with 0.00 precision.

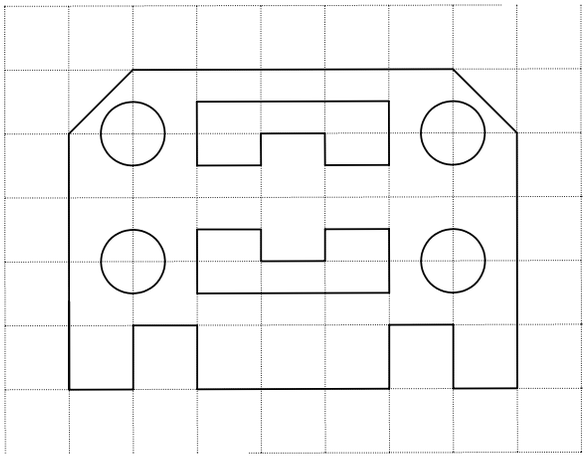

Figure 2-54 *Drawing for Exercise* ***16***

EXERCISE 17 — Using Engineering Units

Draw the object shown in Figure 2-55. The distance between the dotted lines is 10 feet. Determine the limits for this drawing and use the Engineering units with 0'0.00" precision.

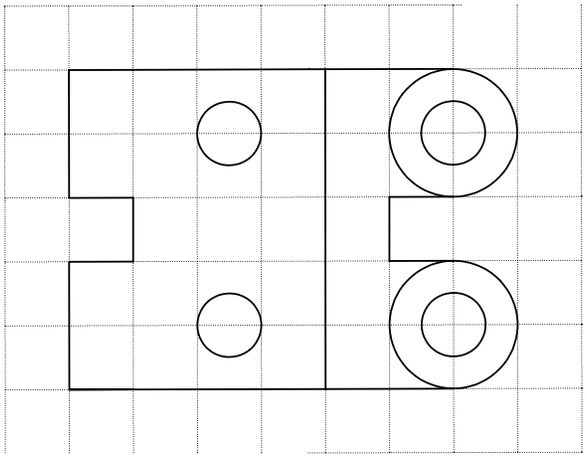

Figure 2-55 *Drawing for Exercise* ***17***

Problem-Solving Exercise 1

Draw the object shown in Figure 2-56, using the **LINE** and **CIRCLE** commands. In this exercise only the diameters of the circles are given. To draw the lines and small circles (Dia 0.6), you need to find the coordinate points for the lines and the center points of the circles. For example, if the center of concentric circles is at 5,3.5, then the *X* coordinate of the lower left corner of the rectangle is 5.0 - 2.4 = 2.6.

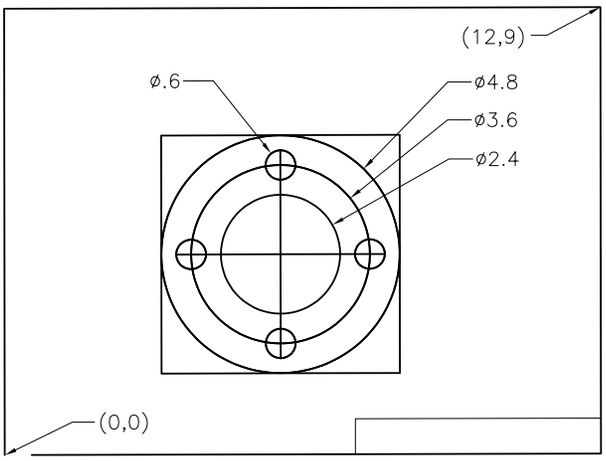

Figure 2-56 *Drawing for Problem-Solving Exercise* ***1***

Problem-Solving Exercise 2

Draw the object shown in Figure 2-57 using various options of the **CIRCLE** and **LINE** commands. In this exercise, you have to find the coordinate points for drawing the lines and circles. Also, you need to determine the best and easiest method to draw the 0.85 diameter circles along the outermost circle.

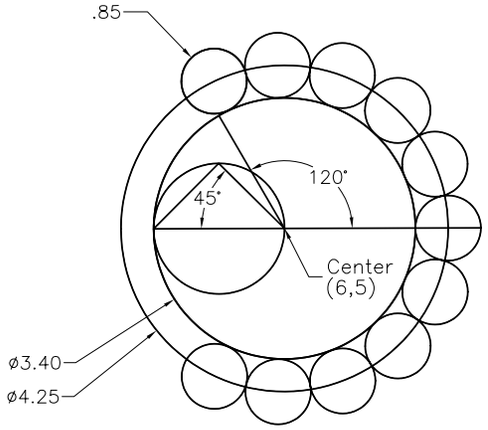

Figure 2-57 *Drawing for Problem-Solving Exercise* ***2***

Problem-Solving Exercise 3

Draw the object shown in Figure 2-58 using the absolute, relative rectangular, or relative polar coordinate system. Draw according to the dimensions shown in the figure, but do not dimension the sketch. The dimensions are only for your reference.

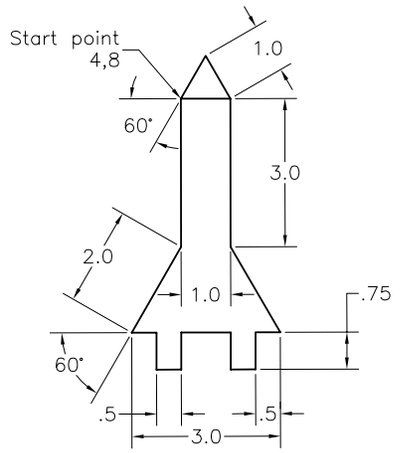

Figure 2-58 *Drawing for Problem-Solving Exercise* ***3***

Answers to Self-Evaluation Test

1. T, 2. F, 3. F, 4. T, 5. Undo, 6. Tan, Tan, Radius, 7. ZOOM, 8. All, 9. Surveyor's, 10. Preview