Chapter 2

Getting Started with AutoCAD LT

Learning Objectives

After completing this chapter, you will be able to:

- *Draw lines using the **LINE** command and its options.*
- *Understand various coordinate systems used in AutoCAD LT.*
- *Use the **ERASE** commands to clear the drawing area.*
- *Understand the two basic object selection methods: Window and Crossing options.*
- *Draw circles using the options of the **CIRCLE** command.*
- *Use the **ZOOM** and **PAN** display commands.*
- *Set up units using the **UNITS** command.*
- *Set up and determine limits for a given drawing.*
- *Plot drawings using the basic plotting options.*
- *Use the **Options** dialog box to specify settings.*

DRAWING LINES IN AutoCAD LT

Toolbar:	Draw > Line
Tool Palettes:	Command Tools > Line
Menu:	Draw > Line
Command:	LINE or L

Figure 2-1 *Invoking the* ***LINE*** *command from the* ***Draw*** *toolbar*

The most fundamental object in a drawing is the line. A line can be drawn between any two points by using the **LINE** command. You can invoke the **LINE** command by choosing the **Line** button in the **Draw** toolbar (Figure 2-1), by choosing it from the **TOOL PALETTES** (Figure 2-2), by choosing it from the **Draw** menu, or by entering **LINE** at the Command prompt. Once you have invoked the **LINE** command, the next prompt, **Specify first point,** requires you to specify the starting point of the line. You can either select a point using the pointing device or you can enter its coordinates. After the first point is selected, AutoCAD LT will prompt you to enter the second point at the **Specify next point or [Undo]** prompt. At this point, you may continue to select points or terminate the **LINE** command by pressing ENTER, ESC, or the 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 LT will again display the Command prompt. The prompt sequence for the drawing shown in Figure 2-3 is as follows:

Figure 2-2 *Invoking the* ***LINE*** *command from the* ***TOOL PALETTES***

Command: **LINE** Enter
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.)*

The **LINE** command has the following three options.

Continue **Close** **Undo**

Tip

When you select the start point of the line by pressing the left mouse button, a rubber-band line appears that 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.

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Figure 2-3 *Drawing lines using the* ***LINE*** *command*

Note

To clear the drawing area to gain space to work out the exercises and examples, choose the ***Erase*** *button from the* ***Modify*** *toolbar or type* ***ERASE*** *at the Command prompt and press ENTER. The screen crosshairs will change into a box called a pick box and AutoCAD LT will prompt you to select objects. You can select the object by positioning the pick box anywhere on it and then pressing the pick button of the pointing device. Once you have finished 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, AutoCAD LT will erase all objects from the screen. (See "Erasing Objects" discussed later in this chapter.) You can use the* ***U*** (undo) *command to undo the last command by choosing the* ***Undo*** *button from the* ***Standard*** *toolbar.*

Command: **ERASE or E** Enter *(**E** is the command alias of the **ERASE** command.)*
Select objects: *Select objects.* *(Select objects using the pick box.)*
Select objects: Enter

Command: **ERASE** Enter
Select objects: **ALL** Enter
Select objects: Enter
Command: **U** Enter *(The **U** command will undo the last command.)*

The Continue Option

After exiting the **LINE** command, you may want to draw another line starting from the point where the previous line ended. In such cases, you can use the **Continue** option. This option enables you to grab the endpoint of the previous line and continue drawing the line from that point (Figure 2-4). The prompt sequence for the **Continue** option is given next.

Command: **LINE or L** Enter *(**L** is the command alias of the **LINE** command.)*

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Specify first point: *Pick first point of the line.*
Specify next point or [Undo]: *Pick second point.*
Specify next point or [Undo]: Enter

Command: **LINE** Enter *(Or choose **Repeat LINE** from the shortcut menu.)*
Specify first point: Enter *(Press ENTER or right-click to continue the line from the last line.)*
Specify next point or [Undo]: *Pick second point of second line (third point in Figure 2-4).*
Specify next point or [Undo]: Enter

Figure 2-4 *Using the **Continue** option with the **LINE** command*

Tip

*You can also type the @ symbol to start the line from the **last point**. For example, if you draw a circle and then immediately start the **LINE** command, the @ will snap to the center point of the circle.*

*The **Continue** option snaps to the endpoint of the last line or arc even if other points have been defined to draw entities such as circles, ellipses, and so on, after the line was drawn.*

Command: **LINE** Enter
Specify first point: *Pick first point of the line.*
Specify next point or [Undo]: *Pick second point.*
Specify next point or [Undo]: Enter

Command: **LINE or L** Enter *(**L** is the command alias of the **LINE** command)*
Specify first point: @ Enter *(Continues drawing the line from the last point.)*
Specify next point or [Undo]: *Pick second point of the second line.*
Specify next point or [Undo]: Enter

The Close Option

The **Close** option 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**. For example, this option can be used when an open figure needs one more line to close it and make a polygon (a polygon is a closed figure with at least three sides, for example, a triangle or rectangle). The following is the prompt sequence for the **Close** option (Figure 2-5).

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

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Command: **LINE** Enter
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.)*

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

The Undo Option

If you draw a line, and then realize that you made an error, you can remove the line using the **Undo** option of the **LINE** command. If you need to remove more than one line, you can use this option multiple times and go as far back as you want. In this option, you can type **Undo** (or just **U**) at the **Specify next point or [Undo]** prompt. You can also right-click to display the shortcut menu, which gives you the **Undo** option. The following example illustrates the use of the **Undo** option (Figure 2-6).

Figure 2-6 *Removing lines using the* ***Undo*** *option of the* ***LINE*** *command*

Command: **LINE or L** Enter *(**L** is the command alias of the **LINE** command)*
Specify first point: *Pick first point (Point 1 in Figure 2-6).*
Specify next point or [Undo]: *Pick second point (Point 2).*
Specify next point or [Undo]: *Pick third point.*
Specify next point or [Close/Undo]: *Pick fourth point.*
Specify next point or [Close/Undo]: **U** Enter *(Removes last line from Point 3 to Point 4.)*
Specify next point or [Close/Undo]: **U** Enter *(Removes next line from Point 2 to Point 3.)*
Specify next point or [Close/Undo]: Enter

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Tip

*AutoCAD LT allows you to enter the command aliases in place of the complete command name. For example, you can enter **L** instead of **LINE** at the Command prompt to invoke the **LINE** command.*

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 that appears, type ALL and press ENTER. The drawing display is modified. You will learn more about the ZOOM command later in this chapter.

COORDINATE SYSTEMS

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 point of intersection of these two axes is called the **origin**. The *X* and *Y* axes divide the *XY* plane into four parts, generally known as quadrants. The *X* coordinate measures the horizontal distance from the origin (how far left or right) on the *X* axis. The *Y* coordinate measures the vertical distance from the origin (how far up or down) on the *Y* axis. The origin has the coordinate values of X = 0, Y = 0. The origin is taken as the reference for locating any point on the *XY* plane. 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. This method of specifying points is called the **Cartesian coordinate system**, see Figure 2-7. In AutoCAD LT, the default origin is located at the lower left corner of the graphics area of the screen. AutoCAD LT uses the following coordinate systems to locate a point in an *XY* plane.

Figure 2-7 *Cartesian coordinate system*

1. **Absolute coordinates**

2. **Relative coordinates**
   a. **Relative rectangular coordinates**
   b. **Relative polar coordinates**

3. **Direct distance entry**

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 (displacement along the *X* axis) and 3 units vertically (displacement along the *Y* axis) from the origin, as shown in Figure 2-8.

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Figure 2-8 *Absolute coordinate system*

In AutoCAD LT, the absolute coordinates are specified by entering *X* and *Y* coordinates, separated by a comma. The following example illustrates the use of absolute coordinates to draw the rectangle shown in Figure 2-9.

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

Figure 2-9 *Drawing lines using the absolute coordinates*

Example 1 — *General*

For Figure 2-10, enter the absolute coordinates of the points in the following table. Then draw the figure using the absolute coordinates. Save the drawing under the name *Exam1.dwg*.

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	8	7,1

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Figure 2-10 *Drawing a figure using the absolute coordinates*

Once the coordinates of the points are known, you can draw the figure by using the **LINE** command. But before you proceed with drawing the object, you need to modify the drawing display area. The prompt sequence is given next.

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

Save this drawing. Choose the **Save** button from the **Standard** toolbar. The **Save Drawing As** dialog box is displayed. 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 given name in the default *My Documents* folder.

Exercise 1 *General*

For Figure 2-11, enter the absolute coordinates of the points in the following table, and then use these coordinates to draw the same figure. The distance between the dotted lines is 1 unit.

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

Figure 2-11 *Drawing for Exercise 1*

Relative Coordinate System

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

Relative Rectangular Coordinates

In the relative rectangular coordinate system, the displacements along the *X* and *Y* axes (DX and DY) are measured with reference to the previous point rather than to the origin. In AutoCAD LT, the relative coordinate system is designated by the symbol @ and it should precede any relative entry. The following prompt sequence illustrates the use of the relative rectangular coordinate system to draw a rectangle with the lower left corner at the point (1,1). The length of the rectangle is 4 units and the width is 3 units (Figure 2-12).

Figure 2-12 *Drawing lines using the relative rectangular coordinates*

Command: **LINE** Enter
Specify first point: **1,1** Enter *(Start point)*
Specify next point or [Undo]: **@4,0** Enter *(Second point DX = 4, DY = 0.)*

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Specify next point or [Undo]: **@0,3** Enter *(Third point DX = 0, DY = 3.)*
Specify next point or [Close/Undo]: **@-4,0** Enter *(Fourth point DX = -4, DY = 0.)*
Specify next point or [Close/Undo]: **@0,-3** Enter *(Start point DX = 0, DY = -3.)*
Specify next point or [Close/Undo]: Enter

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

Sign Convention. As just mentioned, in the relative rectangular coordinate system the displacements along the *X* and *Y* axes are measured with respect to the previous point. 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, the displacements DX and DY are both positive. If the new point is located in the third quadrant, the displacements DX and DY are both negative. In other words, up or right are positive and down or left are negative.

Example 2 *General*

Draw Figure 2-13 using the relative rectangular coordinates of the points given in the table that follows.

Figure 2-13 *Using relative rectangular coordinates with the* ***LINE*** *command*

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

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Once you know the coordinates of the points, you can draw the figure using the **LINE** command and entering the coordinates of the points. But before you proceed, you need to modify the drawing display area, if not already done.

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
Specify first point: **3,1** Enter *(Start point)*
Specify next point or [Undo]: **@4,0** Enter
Specify next point or [Undo]: **@0,1** Enter
Specify next point or [Close/Undo]: **@-1,0** Enter
Specify next point or [Close/Undo]: **@1,1** Enter
Specify next point or [Close/Undo]: **@0,2** Enter
Specify next point or [Close/Undo]: **@-1,0** Enter
Specify next point or [Close/Undo]: **@-1,-1** Enter
Specify next point or [Close/Undo]: **@-1,1** Enter
Specify next point or [Close/Undo]: **@-1,0** Enter
Specify next point or [Close/Undo]: **@0,-2** Enter
Specify next point or [Close/Undo]: **@1,-1** Enter
Specify next point or [Close/Undo]: **@-1,0** Enter
Specify next point or [Close/Undo]: **@0,-1** Enter
Specify next point or [Close/Undo]: Enter

Exercise 2

General

For Figure 2-14, enter the relative rectangular coordinates of the points in the following table, and then use these coordinates to draw the figure. The distance between the dotted lines is 1 unit.

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	_____

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Figure 2-14 *Drawing for Exercise 2*

Relative Polar Coordinates

In the relative polar coordinate system, a point is located by defining both the distance of the point from the current point and the angle that the line between the two points makes with the positive *X* axis. The prompt sequence to draw a line from a point at 1,1 to a point at a distance of 5 units from the point (1,1), and at an angle of 30-degree to the *X* axis (Figure 2-15) is given next.

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

Figure 2-15 *Drawing a line using relative polar coordinates*

Note

*If the **Dynamic Input** mode is on **and you press the** @ key, the relative polar coordinate mode is activated and the second input box shows the angle value, preceded by the < symbol. Therefore, you do not need to input the @ symbol. You can simply enter the distance value and then press the TAB key to shift to the second input box for specifying the angle value. To enter values in the relative polar coordinates, you need to input a comma after the first value.*

Sign Convention. By default, in the relative polar coordinate system, the angle is measured from the horizontal axis (3 'O' clock) as the zero degree baseline. 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.

Note

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

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Example 3

General

For Figure 2-16, enter the relative polar coordinates of each point in the table, and then draw the sketch. Use absolute coordinates for the start point (1.5, 1.75). The dimensions are shown in the drawing. Also, save this drawing as *Exam3.dwg*.

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 2-16 *Drawing for Example 3*

Once you know the coordinates of the points, you can draw the sketch using the **LINE** command. However, before drawing the sketch, modify the drawing display area using the **ZOOM > All** command.

Command: **LINE** Enter
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

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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 **Standard** toolbar. The **Save Drawing As** dialog box is displayed. Enter the name *Exam3* in the **File name** edit box and then choose the **Save** button. The drawing will be saved with the given name in the default *My Documents* folder.

Exercise 3

General

Draw the object shown in Figure 2-17 using the absolute, relative rectangular, and relative polar coordinate systems to locate the points. Do not draw the dimensions; they are for reference only.

Figure 2-17 *Drawing for Exercise 3*

Direct Distance Entry

You can draw a line by specifying the length of the line and its direction using **Direct Distance Entry** (Figure 2-18). The direction is determined by the position of the cursor, and the length of the line is entered from the keyboard. If the **Ortho** mode is on, you can draw lines along the *X* or *Y* axis by specifying the length of the line and positioning the cursor along the ortho direction. You can also use it with the other draw commands like the **RECTANGLE** command. You can also use Direct Distance Entry with polar tracking and **SNAPANG**. For example, if **SNAPANG** is 45-degree and **Ortho** mode is off, you can draw a line at 45 or 135-degree direction by positioning the cursor and

Figure 2-18 *Using the Direct Distance Entry method to draw lines*

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entering the distance from the keyboard. Similarly, if the polar tracking is on, you can position the cursor at the predefined angles and then enter the length of the line from the keyboard.

Command: **LINE** Enter
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

General

In this example, you will draw the object, as shown in Figure 2-19, using Direct Distance Entry. The starting point is 2,2.

Figure 2-19 *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. As a result, you can use the polar tracking to draw lines at an angle that is divisible by 90, such as 90, 180, 270, and 360. This is the reason you first need to add another angle of 45-degree to the polar tracking that will allow you to track the lines drawn at an angle divisible by 45, such as 45, 90, 135, and so on.

To add a 45-degree angle to polar tracking, right-click on the **POLAR** button on the status bar and choose **Settings** from the shortcut menu. Select the **Additional angles** check box in the **Polar Angle Settings** area and then choose the **New** button. Enter **45** in the field that appears and then press ENTER. Choose **OK** to close the dialog box. Now, to turn the polar tracking on, choose the **POLAR** button in the status bar. You can also turn polar tracking on or off while you are in a command. As you move the cursor to draw lines now, AutoCAD LT displays a dotted line when the position of the cursor matches one of the predefined angles for polar tracking. Also, a tooltip is displayed that shows the length of the line and the angle at which it is being drawn.

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The following is the Command prompt sequence for drawing the sketch in Figure 2-19. It is presumed that you have modified the drawing display area.

Command: **LINE** Enter
Specify first point: **2,2**
Specify next point or [Close/Undo]: *Move the cursor horizontally toward the right and when the dotted line and tooltip appear, enter* ***2***.
Specify next point or [Close/Undo]: *Move the cursor at an angle close to 45-degree and when the dotted line and tooltip appear, enter* ***0.7071***.
Specify next point or [Close/Undo]: *Move the cursor vertically upward and when the dotted line and tooltip appear, enter* ***1***.
Specify next point or [Close/Undo]: *Move the cursor horizontally toward the left and when the dotted line and tooltip appear, enter* ***3***.
Specify next point or [Close/Undo]: *Move the cursor vertically downward and when the dotted line and tooltip appear, enter* ***1***.
Specify next point or [Close/Undo]: **C**

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

Exercise 4

General

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 advantage of using direct distance entry.

ERASING OBJECTS

Toolbar:	Modify > Erase
Menu:	Modify > Erase
Command:	ERASE

Figure 2-20 *Invoking the* ***ERASE*** *command from the* ***Modify*** *toolbar*

After drawing some objects, you may want to erase some of them from the screen. To erase, you can use the **ERASE** command (Figure 2-20). This command is used exactly the same way as an eraser is used in manual drafting to remove the unwanted information. When you invoke the **ERASE** command, a small box, known as the pick box, replaces the screen cursor. To erase an object, move the pick box so that it touches the object. You can select the object by pressing the pick button of your pointing device (Figure 2-21). AutoCAD LT confirms the selection by changing the selected objects into dashed lines, and the **Select objects** prompt returns. You can either continue selecting objects or press ENTER to terminate the object selection process and erase the selected objects. If you enter the command from the keyboard, you can type **E** or **ERASE**. The prompt sequence is given next.

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Command: **ERASE** Enter
Select objects: *Select first object.*
Select objects: *Select second object.*
Select objects: Enter

Figure 2-21 Selecting objects by positioning the pick box at the top of the object and then pressing the pick button on the pointing device

If you enter All at the **Select objects** prompt, AutoCAD LT will erase all objects in the drawing, even if the objects are outside the screen display area.

Command: **ERASE** Enter
Select objects: **All**
Select objects: Enter

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 menu, you can choose the **Erase** option.

CANCELING AND UNDOING A COMMAND

If you are in a command and you want to cancel or get out of it, press the ESC (Escape) key on the keyboard.

Command: **ERASE** Enter
Select objects: *Press ESC (Escape) to cancel the command.*

*Figure 2-22 Use of the **OOPS** command*

Similarly, sometimes you unintentionally erase some object from the screen. When you discover such an error, you can correct it by restoring the erased object by means of the **OOPS** command. The **OOPS** command restores objects that have been accidentally erased by the previous **ERASE** command, Figure 2-22. You can also use the **U** (Undo) command to undo the last command.

Command: **OOPS** Enter *(Restores erased objects.)*
Command: **U** Enter *(Undoes the last command.)*

OBJECT SELECTION METHODS

One of the ways to select objects is to select them individually. which can be time-consuming, if you have a number of objects to edit. 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. At this point, you will learn two options: **Window** and **Crossing.** The remaining options are discussed in Chapter 5.

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The Window Option*

This option is used to select an object or group of objects by enclosing them in a box or window. The objects to be selected should be completely enclosed within the window; those objects that lie partially inside the boundaries of the window are not selected. You can select the **Window** option by typing W at the **Select objects** prompt. You are prompted to select the two opposite corners of the window. After selecting the first corner, you can select the other corner by moving the cursor to the desired position and specifying the particular point. As you move the cursor, a blue shaded window is displayed that changes in size as you move the cursor. The objects selected by the **Window** option are displayed as dashed objects (Figure 2-23).

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

The prompt sequence for using the **Window** option with the **ERASE** command is given next.

Command: **ERASE** Enter
Select objects: **W** Enter
Specify first corner: *Select the first corner.*
Specify opposite corner: *Select the second corner.*
Select objects: Enter

You can also invoke the **Window** option by selecting a blank point on the screen at the **Select objects** prompt. This is automatically taken as the first corner of the window. Moving the cursor to the right will display a blue shaded window. After getting all the objects to be selected inside this window, you can specify the other corner with your pointing device. The objects that are completely enclosed within the window will be selected and highlighted. 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

The Crossing Option*

This option is used to select an object or group of objects by creating a box or window around them. The objects to be selected should be touching the window boundaries or completely enclosed within it. You can invoke the **Crossing** option by entering **C** at the **Select objects** prompt. After you choose the Crossing option, AutoCAD LT prompts you to select the first corner at the **Specify first corner** prompt. Once you have selected the first corner, a shaded

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dashed box or window of green color is drawn. By moving the cursor, you can change the size of the crossing box, hence putting the objects to be selected within (or touching) the box. Here, you can select the other corner. The objects selected by the Crossing option are highlighted by displaying them as dashed objects, Figure 2-24. The following prompt sequence illustrates the use of the Crossing option when you choose the **Erase** button.

Select objects: **C** Enter
Specify first corner: *Select the first corner of the crossing window.*
Specify opposite corner: *Select the other corner of the crossing window.*
Select objects: Enter

Figure 2-24 *Selecting objects using the Crossing option*

You can also select the **Crossing** option automatically by selecting a blank point on the screen at the **Select objects:** prompt and dragging the cursor to the left. The blank point you selected becomes the first corner of the crossing window and AutoCAD LT will then prompt you to select the other corner. As you move the cursor, a green shaded box or window drawn with dashed lines is displayed. The objects that are touching or completely enclosed within the window will be selected. The objects selected by the Crossing option are highlighted by being displayed as dashed objects. The prompt sequence for automatic crossing selection when you choose the **Erase** button 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

Note

With this release of AutoCAD LT, the entities are highlighted when you move the cursor over them. This feature is known as the ***Selection Preview. The*** *settings for the selection preview are available in the* ***Selection Preview*** *area of the* ***Selection*** *tab of the* ***Options*** *dialog box. The PREVIEWEFFECT variable stores the type of appearance during highlighting. The preview will be a Dashed Line if the variable is set to 1, Thicken Line if it is set to 2, and Dashed Thicken Line, if it is set to 3. The selection preview for different values of the PREVIEWEFFECT variable is shown in Figure 2-25*

Figure 2-25 *The selection preview with* ***Both****,* ***Thicken,*** *and* ***Dash*** *options*

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DRAWING CIRCLES

Toolbar:	Draw > Circle
Tool Palettes:	Command Tools > Line > Circle
Menu:	Draw > Circle
Command:	CIRCLE or C

Figure 2-26 *Invoking the* ***CIRCLE*** *command from the* ***Draw*** *toolbar*

To draw a circle, you can use the **CIRCLE** command. You can invoke the **CIRCLE** command from the **Draw** toolbar (Figure 2-26), from the **TOOL PALETTES**, or from the **Draw** menu (Figure 2-27). The following is the prompt sequence for the **CIRCLE** command.

Command: **CIRCLE** Enter
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:

The options of the **CIRCLE** command are explained in the following sections.

Figure 2-27 *Invoking the* ***CIRCLE*** *command from the* ***TOOL PALETTES***

The Center and Radius Option

In this option, you can draw a circle by defining the center and the radius of the circle, Figure 2-28. After entering the **CIRCLE** command, AutoCAD LT will prompt you to enter the center of the circle, which can be selected by specifying a point on the screen or by entering the coordinates of the center point. Next, you will be prompted to enter the radius of the circle. Here you can accept the default value, enter a new value, or select a point on the circumference of the circle to

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specify the radius. The following is the prompt sequence for drawing a circle with a center at 3,2 and a radius of 1 unit.

Command: **CIRCLE** Enter
Specify center point for circle or [3P/2P/Ttr (tan tan radius): **3,2** Enter
Specify radius of circle or [Diameter]<current>: **1** Enter

Note
*You can also set the radius by assigning a value to the **CIRCLERAD** system variable. The value you assign becomes the default value for the radius.*

The Center and Diameter Option

In this option you can draw a circle by defining the center and diameter of the circle. After invoking the **CIRCLE** command, AutoCAD LT prompts you to enter the center of the circle, which can be selected by specifying a point on the screen or by entering the coordinates of the center point. Next, you will be prompted to enter the radius of the circle. At this prompt, enter **D**. After this, you will be prompted to enter the diameter of the circle. For entering the diameter, you can accept the default value, enter a new value, or drag the circle to the desired diameter and select a point. If you use a menu option to select the **CIRCLE** command with the **Diameter** option, the menu automatically enters the **Diameter** option and prompts for the diameter after you specify the center. The following is the prompt sequence for drawing a circle with the center at (2,3) and a diameter of 2 units, as shown in Figure 2-29.

Command: **CIRCLE** Enter
Specify center point for circle or [3P/2P/Ttr(tan tan radius): **2,3** Enter
Specify radius of circle or [Diameter]<current>: **D** Enter
Specify diameter of circle <current>: **2** Enter

Figure 2-28 *A circle drawn by specifying its center and radius*

Figure 2-29 *A circle drawn by specifying its center and diameter*

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The Two-Point Option

You can also draw a circle using the **Two-Point** option. In this option, AutoCAD LT lets you draw the circle by specifying the two endpoints of the circle's diameter. For example, if you want to draw a circle that passes through the points (1,1) and (2,1), you can use the **CIRCLE** command with 2P option, as shown in the following example (Figure 2-30).

Command: **CIRCLE** Enter
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **2P** Enter
Specify first end point of circle's diameter: **1,1** Enter
Specify second end point of circle's diameter: **2,1** Enter *(You can also use the relative coordinates.)*

The Three-Point Option

For drawing a circle, you can also use the **Three-Point** option by defining three points on its circumference. The three points may be entered in any order. To draw a circle that passes through the points (3,3), (3,1), and (4,2), Figure 2-31, the prompt sequence is given next.

Command: **CIRCLE** Enter
Specify center point for circle or [3P/2P/Ttr(tan tan radius)]: **3P** Enter
Specify first point on circle: **3,3** Enter
Specify second point on circle: **3,1** Enter
Specify third point on circle: **4,2** Enter

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

Command: **CIRCLE** Enter
Specify center point for circle or [3P/2P/Ttr(tan tan radius)]: **3P** Enter
Specify first point on circle: **3,3** Enter
Specify second point on circle: **@0,-2** Enter
Specify third point on circle: **@1,1** Enter

Figure 2-30 *A circle drawn using the* ***2 Point*** *option*

Figure 2-31 *A circle drawn using the* ***3 Point*** *option*

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The Tangent Tangent Radius Option

A tangent is an object (line, circle, or arc) that contacts the circumference of a circle at only one point. In this option, AutoCAD LT uses the Tangent object snap to locate two tangent points on the selected objects that are to be tangents to the circle. Then you have to specify the radius of the circle. The prompt sequence for drawing a circle using the **Ttr** option is given next.

Command: **CIRCLE** Enter
Specify center point for circle or [3P/2P/Ttr(tan tan radius)]: **T** Enter
Specify point on object for first tangent of circle: *Select first line, circle, or arc.*
Specify point on object for second tangent of circle: *Select second line, circle, or arc.*
Specify radius of circle <current>: **0.75** Enter

In Figures 2-32 through 2-35, the dotted circles represent the circles that are drawn by using the **Ttr** option. 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 2-32 *Drawing a circle using the* ***Tangent, Tangent, Radius*** *option*

Figure 2-33 *Drawing a circle using the* ***Tangent, Tangent, Radius*** *option*

Figure 2-34 *Drawing a circle using the* ***Tangent, Tangent, Radius*** *option*

Figure 2-35 *Drawing a circle using the* ***Tangent, Tangent, Radius*** *option*

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The Tangent, Tangent, Tangent Option

You can invoke this option from the menu bar. This option is a modification of the **3P** option. In this option, AutoCAD LT uses the Tangent Object Snap to locate three points on three selected objects to which the circle is drawn tangent. The following is the prompt sequence for drawing a circle using the **Tan, Tan, Tan** option (Figure 2-36).

Command: *Choose **Draw > Circle > Tan, Tan, Tan** from the menu bar.*
_CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: _3p Specify first point on circle: _tan to *Select the first object.*
Specify second point on circle: _tan to *Select the second object.*
Specify third point on circle: _tan to *Select the third object.*

Figure 2-36 *Circle drawn using the **Tan,Tan, Tan** option*

Exercise 5 *Mechanical*

Draw Figure 2-37 using the 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 and Radius, Center and Diameter, or Tan, Tan, Tan options. (Height of triangle = 4.5 X sin 60 = 3.897.) Do not draw the dimensions; they are for reference only.

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Figure 2-37 *Drawing for Exercise 5*

BASIC DISPLAY COMMANDS

Drawing in AutoCAD LT is much simpler than manual drafting in many ways. Sometimes while drawing, it is very difficult to see and alter minute details. In AutoCAD LT, 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. Some of the drawing display commands such as **ZOOM** and **PAN** will be introduced here. A detailed explanation of these commands and other display options appears in Chapter 7.

Zooming the Drawings

Toolbar:	Zoom toolbar, Standard > Zoom Window flyout
Menu:	View > Zoom
Command:	ZOOM

The **ZOOM** command (Figure 2-38) enlarges or reduces the view of the drawing on the screen, but it does not affect the actual size of the entities. After the **ZOOM** command has been invoked, (Figure 2-39) various options can be used to obtain the desired display. If you use a menu, it issues the appropriate option at the initial **ZOOM** prompt. The following is the prompt sequence of the **ZOOM** command:

Figure 2-38 *Selecting The **ZOOM** options from the **Standard** toolbar*

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

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Realtime Zooming

You can use the **Realtime Zoom** to zoom in and zoom out interactively. To zoom in, invoke the command, and then hold the pick button down and move the cursor up. If you want to zoom in further, bring the cursor down, specify a point, and move the cursor up. Similarly, to zoom out, hold the pick button down and move the cursor down. Realtime zoom is the default setting for the **ZOOM** command. At the Command prompt, pressing ENTER after invoking the **ZOOM** command automatically invokes the realtime zoom. To exit the Realtime Zoom, right-click to display the shortcut menu and choose **Exit**. You can also press ESC or the ENTER key to exit the command.

Figure 2-39 *Invoking the* ***Zoom*** *options from the* ***View*** *menu*

Window Option

This is the most commonly used option of the **ZOOM** command. It lets you specify the area you want to zoom in on by letting you specify two opposite corners of a rectangular window. The center of the specified window becomes the center of the new display screen. The area inside the window is magnified in size to fill the drawing area as completely as possible. The points can be specified by selecting them with the help of the pointing device or by entering their coordinates.

Previous Option

While working on a complex drawing, you may need to zoom in on a portion of the drawing to edit some minute details. When you have completed the editing you may want to return to the previous view. This can be done using the **Previous** option of the **ZOOM** command. AutoCAD LT remembers the last ten views that can be restored using the **Previous** option.

All Option

This option zooms to the drawing limits or the extents, whichever is greater. Whenever you increase the limits (**Quick Setup** in **Use a Wizard** dialog box) the current display is not affected and hence does not show. You need to use the **Zoom All** option to display the limits of the drawing. Sometimes it is possible that the objects are drawn beyond the limits. In such a case, the **Zoom All** option zooms to fill the drawn objects in the drawing area, irrespective of its limits.

Note

You will learn about the other options of the ***ZOOM*** *command in detail in Chapter 7.*

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Panning in Realtime

You can use **Pan Realtime** to pan the drawing interactively, by sliding the drawing and placing it at the required position. To pan a drawing, invoke the command and then hold the pick button down and move the cursor in any direction. When you select realtime pan, AutoCAD LT displays an image of a hand, indicating that you are in the PAN mode. You can drag the hand anywhere on the screen to move the drawing. To exit realtime pan, right-click to display the shortcut menu and choose **Exit**. You can also press ESC or the ENTER key to exit the command.

Tip

*You can right-click to display a shortcut menu while the realtime zoom or realtime pan options are active. The **Realtime Zoom**, **Realtime Pan**, **Exit**, and other **ZOOM** command options are available in this shortcut menu.*

SETTING UNITS

Menu:	Format > Units
Command:	UNITS

In Chapter 1 you have already learned to set units while starting a drawing from the **Startup** dialog box using the **Wizards** option. If you want to change the units while you are already working on a drawing, the **UNITS** command can be used.

Setting Units Using the Drawing Units Dialog Box

The **UNITS** command is used to select a format for the units of distance and angle measurement. You can invoke this command using the **Format** menu. The **UNITS** command displays the **Drawing Units** dialog box as shown in Figure 2-40. You can then specify the precision for the units and angles from the corresponding **Precision** drop-down list, see Figure 2-41. You can also set the units from the command line by entering **-UNITS** at the Command prompt.

Specifying Units

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

1. Architectural (0'-01/16")
2. Decimal (0.00)
3. Engineering (0'-0.00")
4. Fractional (0 1/16)
5. 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. In the following example, the units are set as decimal, scientific, fractional, and decimal and fractional to enter the coordinates of different points.

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Figure 2-40 *The* ***Drawing Units*** *dialog box*

Figure 2-41 *Specifying the precision from the* ***Drawing Units*** *dialog box*

Command: **LINE** Enter
Specify from point: **1.75,0.75** Enter (*Decimal.*)
Specify next point or [Undo]: **1.75E+01,3.5E+00** Enter (*Scientific.*)
Specify next point or [Undo]: **10-3/8,8-3/4** Enter (*Fractional.*)
Specify next point or [Close/Undo]: **0.5,17/4** Enter (*Decimal and fractional.*)

If you choose the engineering or architectural format, you can enter the distances or coordinates in any of the five formats. In the following example, the units are set as architectural; hence, different formats are used to enter the coordinates of points.

Command: **LINE** Enter
Specify first point: **1-3/4,3/4** Enter (*Fractional.*)
Specify next point or [Undo]: **1'1-3/4",3-1/4** Enter (*Architectural.*)
Specify next point or [Undo]: **0'10.375,0'8.75** Enter (*Engineering.*)
Specify next point or [Close/Undo]: **0.5,4-1/4"** Enter (*Decimal and engineering.*)

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.

You cannot use the feet (') or inch (") symbols if you have selected scientific, decimal, or fractional unit formats.

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Specifying Angle

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

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 of the first four measuring systems, 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. If you enter an angle value without any indication of a measuring system, it is taken in the current system. To enter the value in another system, use the appropriate suffixes and symbols, such as r (Radians), d (Degrees), g (Grads), or the others shown in the following examples. In the following example, the system of angle measure is Surveyor's units and different systems of angle measure are used to define the angle of the line.

Command: **LINE** Enter
Specify first point: **3,3** Enter
Specify next point or [Undo]: **@3<45.5** Enter *(Decimal degrees.)*
Specify next point or [Undo]: **@3<90d30'45"** Enter *(Degrees/min/sec.)*
Specify next point or [Close/Undo]: **@3<75g** Enter *(Grads.)*
Specify next point or [Close/Undo]: **@3<N45d30'E** Enter *(Surveyor's units.)*

In Surveyor's units, you must specify the bearing angle that the line makes with the north-south direction (Figure 2-42). 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 N60dE. Similarly, you can specify angles such as S50dE, S50dW, and N75dW, as shown in Figure 2-41. You cannot specify an angle that exceeds 90-degree (N120E). The angles can also be specified in radians or grads, for example, 180-degree is equal to **PI** (3.14159) radians. You can convert degrees into radians or radians into degrees using the following equations.

Figure 2-42 *Specifying angle in Surveyor's units*

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. A 90-degree angle is equal to 100 grads.

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Tip

*An example corresponding to the type of unit and angle selected from the **Length** or **Angle** area of the dialog box can be seen in the **Sample Output** area of the dialog box.*

In AutoCAD LT, by default the angles are positive if measured in the counterclockwise direction, and negative if measured in the clockwise direction. Also, the angles are measured from the positive *X* axis, see Figures 2-43 and 2-44. If you want the angles to be measured as positive in the clockwise direction, select the **Clockwise** check box from the **Angle** area. Now, the angles will be positive if measured in the clockwise direction, and the negative if measured in the counterclockwise direction.

Figure 2-43 *North, South, East, and West directions*

Figure 2-44 *Measuring angles counterclockwise from the positive X axis (default)*

When you choose the **Direction** button in the **Drawing Units** dialog box, the **Direction Control** dialog box appears, which gives you an option of selecting the setting for direction of the base angle, see Figure 2-45.

If you select the **Other** option, you can set your own direction for the base angle by entering a value in the **Angle** edit box or by choosing the **Pick an Angle** button to pick two points on the screen to specify the angle. After selecting an angle, you can choose the **OK** button to apply the settings. This will redisplay the **Drawing Units** dialog box.

You can also set the units of measure while inserting a block or a drawing from the **DesignCenter**. In the **Drawing Units** dialog box, choose any measuring unit from the **Units to scale drag-and-drop content** drop-down list, see Figure 2-46. Now, while inserting a block or a drawing from the **DesignCenter**, AutoCAD LT inserts the block with the specified unit. Even if the block was created using a different measuring unit, AutoCAD LT scales it and inserts it using the specified measuring unit. If you want to insert the block with the original units, then choose **Unitless** from the drop-down list.

Note

*The insertion of blocks from the **DesignCenter** into a drawing is discussed in detail in Chapter 14.*

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Figure 2-45 *Setting direction from the* ***Direction Control*** *dialog box*

Figure 2-46 *Selecting the measuring units for inserting drawings and blocks using the drag-and-drop method*

The **Sample Output** area in the **Drawing Units** 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 the **Sample Output** area.

Example 5 *General*

In this example, you will set the units for a drawing according to the following specifications and then draw Figure 2-47. You also need to do the following.

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 direction to 90-degree (north) and the direction of measurement of angles to clockwise (angles measured positive in clockwise direction), Figure 2-47.

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Figure 2-47 *Drawing for Example 5*

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

1. Invoke the **Drawing Units** dialog box by choosing **Forma > Units** from the menu bar. You can also invoke the 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**, see Figure 2-48.

Figure 2-48 *Setting units for Example 5 in the* ***Drawing Units*** *dialog box*

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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. 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 2-47 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, see Figure 2-49.

Figure 2-49 *Angles measured from north (Y axis)*

7. Modify the drawing display area using the **ZOOM > All** command. Invoke the **LINE** command and specify the points as follows.

Command: **LINE** Enter
Specify first point: **2,2** Enter
Specify next point or [Undo]: **@2.0<0** Enter
Specify next point or [Undo]: **@2.0<60** Enter
Specify next point or [Close/Undo]: **@1<180** Enter
Specify next point or [Close/Undo]: **@1<90** Enter
Specify next point or [Close/Undo]: **@1<180** Enter
Specify next point or [Close/Undo]: **@2.0<60** Enter
Specify next point or [Close/Undo]: **@0.5<90** Enter
Specify next point or [Close/Undo]: **@2.0<180** Enter
Specify next point or [Close/Undo]: **C** Enter

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SETTING LIMITS OF THE DRAWING

Menu:	Format > Drawing Limits
Command:	LIMITS

In AutoCAD LT, the drawings must be drawn full scale and, therefore, the limits are needed to size up a 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 putting down 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.

In Chapter 1, you have already learned to set the limits while starting a drawing from the **Startup** dialog box using the **Wizards** option. If you want to change the limits while you are already working in a drawing, the **LIMITS** command can be used. When you start AutoCAD LT using the **Imperial** file, the default limits are 12,9. The following is the prompt sequence of the **LIMITS** command for setting the limits of 24,18.

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

At the preceding two prompts you are required to specify the lower left corner and the upper right corner of the sheet. Normally you choose (0,0) as the lower left corner, but you can enter any other point. If the sheet size is 24 X 18, enter (24,18) as the coordinates of the upper right corner.

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

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 to help calculate the 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 2-50, 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.)

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Figure 2-50 *Setting limits in a drawing*

After you know the sizes of various views and have determined 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 space or work area you need for the 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 are 34,22.

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'

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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 *General*

In this example, you will calculate the limits and determine an appropriate drawing scale factor for Figure 2-51. The drawing is to be plotted on a 12" X 9" sheet.

Figure 2-51 *Drawing for Example 6*

The scale factor can be calculated as follows:

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"

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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'
Also, Y limit = 9 X 4' = 36'

Limits for Metric Drawings

When the drawing units are 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 (multiply 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. As with 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

Thus, the limits are 12192 and 9144.

Exercise 6 *General*

Set the units of the drawing according to the following specifications and then make the drawing shown in Figure 2-52 (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.

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).

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4. Set the limits leaving a space of 3 to 5 units around the drawing for dimensioning and title block.

Figure 2-52 *Drawing for Exercise 6*

INTRODUCTION TO PLOTTING DRAWINGS

Toolbar:	Standard toolbar > Plot
Menu:	File > Plot
Command:	PLOT or PRINT

Once you have created a drawing in the current session of AutoCAD LT, you may need to have its hard copy for your reference or for sending to the client. This hard copy is very useful in the industry and can be created by plotting and printing it on a sheet of paper. Suppose you have drawn an architectural plan on the computer, you can print it and send its hard copy to the site for implementation. Similarly, if you have created a mechanical component, you can print it and send its hard copy to the shop floor for manufacturing. Drawings can be plotted using the **PLOT** command. When you invoke this command, the **Plot** dialog box is displayed, see Figure 2-53. By default, the dialog box is not expanded. To expand the dialog box, choose the **More Options** button at the lower right corner of the dialog box.

The values in this dialog box are the ones that were set during the configuring of AutoCAD LT. If the displayed values conform to your requirements, you can start plotting without making any changes. If necessary, you can make changes in the default values according to your plotting requirements.

Basic Plotting

In this section, you will learn to set up the basic plotting parameters. Later, you will learn about the advance options that allow you to plot according to 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.

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Figure 2-53 *The* ***Plot*** *dialog box*

Example 7 — *General*

You will plot the drawing shown in Figure 2-54 using the **Window** option to select the area to plot. The drawing was drawn in **Example 3** of this chapter and here it is assumed to be open on the screen. Assume that AutoCAD LT is configured for two output devices: Default System Printer and **HP Laserjet 2100 Series PS**.

Figure 2-54 *Drawing created in Example 3*

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1. Invoke the **Plot** dialog box from the **Standard** toolbar, the **File** menu (choose **Plot**), or by entering **PLOT** at the Command prompt. You can also invoke it by choosing **Plot** from the shortcut menu, which is displayed by right-clicking on the **Model/Layout** tabs.

2. The name of the default system printer is displayed in the **Name** drop-down list in the **Printer/plotter** area. In this example, it is **HP Laser jet 2100 Series PS**, see Figure 2-55. You can use any other printer by selecting the name of the device from the **Name** drop-down list.

Figure 2-55 *The **Plot** dialog box with the default printer*

Note

The default system printer varies from system to system. It is the printer chosen as the default printer while configuring your system for the output device.

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 the two opposite corners to define a window that specifies the plot area (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 for the plot, you can select a size from the drop-down list in the **Paper size** area, which lists all the plotting sizes that the present plotter can support. You can select any

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one of the sizes listed in the dialog box or specify a size (width and height) of your own through the **Plotter Manager**. (This option is discussed later in Chapter 12, Plotting Drawings.) Once you select a size, you can also select the orientation of the paper. However, to set the orientation, you need to expand the **Plot** dialog box by choosing the **More Options** button at the lower right corner of the dialog box. The expanded form of this dialog box is shown in Figure 2-56. To set the orientation, select the **Landscape** or **Portrait** radio buttons 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. In this example, you will specify **A4** Paper size and **Portrait** orientation.

Figure 2-56 *Expanded form of the* ***Plot*** *dialog box*

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 LT, you can enter values for the plot scale from the **Plot scale** area. Clear the **Fit to paper** check box, if selected and then open the **Scale** drop-down list in the **Plot scale** area to display the various scale factors. From this list, you can select a scale factor 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 1 inch = 48 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 LT will determine the scale factor and display it in the edit boxes. In

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this example, you 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. You can preview the plot on the specified paper size before actually plotting it. This way you can save time and stationery. To preview a plot, choose the **Preview** button. Once regeneration is complete, the preview image is displayed on the screen, see Figure 2-57. Here, in place of the cursor, a realtime zoom icon is displayed. You can hold the pick button of your pointing device and then move it up to zoom into the preview image and move the cursor down to zoom out of the preview image.

8. If the plot preview is satisfactory, you can directly plot your drawing by choosing **Plot** from the shortcut menu, as shown in Figure 2-57. 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. You can also choose the **OK** button in the dialog box to plot the drawing.

Figure 2-57 *Plot preview with the shortcut menu*

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MODIFYING AutoCAD LT SETTINGS USING THE OPTIONS DIALOG BOX

Menu:	Tools > Options
Command:	OPTIONS

You can use the **Options** dialog box to change the default settings that affect the drawing environment or the AutoCAD LT interface and customizethem to your requirements using the **Options** dialog box. For example, you can use this dialog box to turn off the settings to display the shortcut menu by right-clicking or specify the support directories that contain the files you need. The most convenient way of invoking this dialog box is by right-clicking in the command window or in the drawing area when no command is active or no object is selected and choosing **Options** from the shortcut menu. The **Options** dialog box is shown in Figure 2-58.

Figure 2-58 *The* ***Options*** *dialog box (****Files*** *tab)*

The dialog box contains eight tabs that display the sections to change the various environmental aspects. The current profile and current drawing names are displayed on the top, above the tabs. You can save a set of custom settings in a profile to be used later for other drawings. If you do not specify a profile, the current settings are stored with the name **Unnamed Profile**. The various tabs in the **Options** dialog box are discussed next.

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Files

This tab stores the directories in which AutoCAD LT 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 help file. You can select the **Miscellaneous File Names** folder and then select the **Help File** icon to display the path, see Figure 2-57. Similarly, you can define a custom hatch pattern file and then add its search path. This way AutoCAD LT can locate the custom hatch pattern.

Display

This tab controls the drawing and window settings like tool tips display and scroll bar. For example, if you do not want to display the tool tips, clear the **Show ToolTips** check box in the **Window Elements** area. You can also change the color of the graphics window background, layout window background, command line background, and also the color of the command line text using the **Color Options** dialog box that is displayed by choosing the **Colors** button. This tab also allows you to modify 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. Here you can toggle on and off the various layout elements such as the layout tabs on the screen, margins, paper background, and so on. 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.

Open and Save

This tab controls the parameters related to opening and saving of files in AutoCAD LT. You can specify the file type for saving while using the SAVEAS command. The various formats are **AutoCAD LT 2004 Drawing (*.dwg)**, **AutoCAD LT 2000/LT2000 Drawing (*.dwg)**, **AutoCAD LT Drawing Template(*.dwt)**, **AutoCAD LT 2004 DXF (*.dxf)**, **AutoCAD LT 2000/LT2000 DXF(*.dxf)**, **AutoCAD LT 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. You can add a password and digital signatures to your drawing while saving using the **Security Options** button in the **File Safety Precautions** area. You can control the display of the digital signature information when a file with a valid digital signature is opened with the help of the **Display digital signature information** check box. You can change the number of recently saved files to be displayed in the **File** menu for opening. You can also set the various parameters for external references and the ObjectARX applications.

Plot and Publish

The **Plotting** tab controls the parameters related to the plotting and publishing of the drawings in AutoCAD LT. 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.

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System

This tab contains AutoCAD LT 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 various system parameters such as the single drawing mode instead of MDE, the display of the **Startup** option while opening a new session of AutoCAD LT 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

This tab controls settings that depend on the way the user prefers working on AutoCAD LT, such as the right-click customization where you can change the shortcut menus. You can set the units parameters for the **DesignCenter** as well as the priorities for various data entry methods. Here it is possible to set the order of object sorting methods and also set the lineweight options.

Drafting

This tab controls 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.

Selection

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

Note

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

Tip

*Some options in the various tabs of the **Options** dialog box have a drawing file icon in front of them. For example, the options in the **Display resolution** of the **Display** tab have the drawing file icons. This specifies that these parameters are saved with the current drawing only and therefore affects it. The rest of the options (without the drawing file icon) are saved with the current profile and also affect all the drawings present in that AutoCAD LT session.*

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Self-Evaluation Test

Answer the following questions and then compare your answers 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**. (T/F)

2. Using the **Crossing** method of object selection, only those objects that are completely enclosed within the boundaries of the crossing box are selected. (T/F)

3. The **Three-Point** option of the **CIRCLE** command lets you draw the 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 on the screen, but it does not affect the actual size of the entities.

8. After increasing the drawing limits, you need to use the _____ option of the **ZOOM** command 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 reference to the previous point rather than to the origin. (T/F)

2. In AutoCAD LT, by default the angles are positive if measured in the counterclockwise direction and the angles are measured from the positive *X* axis. (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 LT looks for the driver, support, menu, project, template, and other files. (T/F)

5. You cannot terminate the **LINE** command by pressing which of the following key on the keyboard at the **Specify next point or [Close/Undo]:** prompt?

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

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

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

7. How many formats of units can you choose from in 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, which 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.

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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 the profiles available in the current drawing.

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

Exercises

Exercise 7 — *General*

Use the following relative rectangular and absolute coordinate values in the **LINE** command to draw the 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 — *General*

For the drawing shown in Figure 2-59, enter the relative rectangular and relative polar coordinates of the points in the following table, and then use these coordinates to draw the figure. The distance between the dotted lines is 1 unit. Save this drawing as *Exer8.dwg*.

Figure 2-59 *Drawing for Exercise 8*

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

Exercise 9

Mechanical

For the drawing shown in Figure 2-60, enter the relative polar coordinates of the points in the following table. Then use these coordinates to draw the figure. Do not draw the dimensions.

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

Figure 2-60 *Drawing for Exercise 9*

Exercise 10

Mechanical

Draw the sketch shown in Figure 2-61, using the **LINE** and **CIRCLE** commands. The distance between the dotted lines is 1.0 units.

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Figure 2-61 *Drawing for Exercise 10*

Exercise 11

Mechanical

Draw the sketch shown in Figure 2-62 using the **LINE** command and the **Ttr** option of the **CIRCLE** command.

Figure 2-62 *Drawing for Exercise 11*

Exercise 12

Mechanical

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'.

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3. Set the direction to 0-degree (east) and the direction of measurement of angles to counterclockwise (angles measured positive in a counterclockwise direction).

Based on Figure 2-63, 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 2-63 *Drawing for Exercise 12*

Exercise 13 *Mechanical*

Draw the sketch shown in Figure 2-64. 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 2-64 *Drawing for Exercise 13*

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Exercise 14

General

Draw the object shown in Figure 2-65. 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 2-65 *Drawing for Exercise 14*

Exercise 15

General

Draw the object shown in Figure 2-66. 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 2-66 *Drawing for Exercise 15*

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Exercise 16

General

Draw the object shown in Figure 2-67. 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 2-67 *Drawing for Exercise 16*

Problem Solving Exercise 1

Mechanical

Draw the object shown in Figure 2-68, 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 2-68 *Drawing for Problem Solving Exercise 1*

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Problem Solving Exercise 2 *Mechanical*

Draw the object shown in Figure 2-69 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 2-69 *Drawing for Problem Solving Exercise 2*

Problem Solving Exercise 3 *Mechanical*

Draw the object shown in Figure 2-70 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 2-70 *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**

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