# UNIT

## Basic CAD Drawing Techniques

### **OVERVIEW**

In this unit you will first learn how to set the type and precision of units commonly used in CAD drawings. Next you will explore the use of a template drawing. A template drawing can be used to speed up and standardize the drawing process. You will also learn how to enter coordinate information into your drawing accurately.

#### INTRODUCTION

#### OBJECTIVES

- · Set the display format and precision.
- · Create a template drawing.
- Set the template drawing options.
- · Begin a new drawing by using a template drawing.
- · Create geometry using absolute polar tracking.
- Create geometry using relative polar tracking.
- Create geometry using polar tracking and polar snap.

The manual drafter normally begins with a sheet of drafting paper. Depending on the type of drawing to be completed, he or she will select the appropriate units. The manual drafter will use specific scales for each design project. The architect, for example, will normally use an architectural scale with dimensions given in feet and inches. The mechanical drafter will normally use a decimal scale. Like manual drafting, AutoCAD allows you to use different units in your drawing.

In many cases, the manual drafter uses a sheet of paper that contains a preprinted border and title block. The title block is frequently labeled with the company name and address and has a place for the drawing title, part number, scale, material, and drafter's name, as well as an area for revisions. The manual drafter tapes down the sheet of paper and adds the necessary views, dimensions, and so on. Using drafting paper with a preprinted border and title block ensures standardization within the company. This approach saves time because the manual drafter doesn't have to draw the border and title block with each new drawing.

AutoCAD has a feature called a *template drawing*, which provides a similar function to that of the preprinted border and title block used in manual drafting. A simple template drawing may contain values for limits, grid, and snap. A complex template may contain a border and title block, established text styles, and other drawing variables.

In Unit 2, Creating Your First Drawing, you learned how to set up AutoCAD to use the snap increment and the grid to accurately locate points in the drawing. Another way to specify exact points and create accurate drawings is to use coordinates. AutoCAD uses the Cartesian coordinate system for specifying exact points in a drawing. The second part of this unit discusses the two-dimensional coordinate system for locating points and lines. The absolute and relative polar tracking methods are discussed along with polar snap.

#### OUTLINE

- Setting Display Format and Precision (UNITS)
- **Tutorial 4.1:** Drawing Lines Using the Five Different Units Working with Template Drawings
- Tutorial 4.2: Creating a Template
- Drawing

Coordinate System Basics

- **Tutorial 4.3:** Using Absolute Polar Tracking to Create a Side Elevation Profile
- **Tutorial 4.4:** Using Relative Polar Tracking to Create a Side Elevation Profile
- **Tutorial 4.5:** Using Polar Tracking and Polar Snap to Create a Side Elevation Profile

**Tutorial 4.6:** Creating Property Lines with Polar Tracking, Polar Snap, and a Template Drawing Summary Review Exercises

### SETTING DISPLAY FORMAT AND PRECISION (UNITS)

The AutoCAD Drawing Units dialog box enables you to set the type and precision of units you will use in the drawing. Options within the Drawing Units dialog box control two types of unit display: length and angle. Other options enable you to control the precision with which the unit values appear for length and angle measurement.

The unit settings affect the following:

- · The display of the coordinates on the status bar
- The format of the values when you examine the drawing for an area or distance
- The format for entering coordinates, distances, and angles at the Command: line

You can set the units by accessing the Drawing Units dialog box. You can access the Drawing Units dialog box in one of the following ways:

- · Enter units at the Command: prompt, or
- Choose Units from Format on the menu bar.

#### UNITS REPORT FORMATS

AutoCAD has five different measurement report formats: architectural, decimal, engineering, fractional, and scientific. The examples shown in Table 4.1 show how the value of 15.5 drawing units appears in each format.

Each type is designed for a specific purpose. *You* must determine the units in which to create the drawing. You can change the units report format at any time in a drawing session by selecting the down arrow in the Type window. When the units report format is changed, this change is reflected in the sample output section of the Drawing Units dialog box, as shown in Figure 4.1. Note the five types:



#### FOR THE PROFESSIONAL

A generic unit can be declared as anything you want: an inch, a mile, a millimeter, or a kilometer.

TABLE 4.1	UNITS OF MEASUREMENT		
	<b>Report Format</b>	Example	
	Architectural	1'-3 1/2"	
	Decimal	15.50	
	Engineering	1'-3.50"	
	Fractional	15 1/2	
	Scientific	1.55E+01	

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Length		Angle	
Lype:		1 ype:	
Decimal	<u> </u>	Decimal Degrees	-
Precision:		Precision:	
0.0000	*	0	*
Drawing units When inserting Inches	for DesignCenter g blocks into this	Clockwise	

#### Figure 4.1

Use the Drawing Units dialog box to set the type of units and the precision.

- Architectural units. Dimensions are given in feet, inches, and fractional parts of an inch, such as  $6'-11\frac{1''}{4}$ . Twelve inches is automatically converted to 1 foot. Architectural units are used for residential and commercial planning and construction drawings. The smallest fraction that AutoCAD can display is  $\frac{1}{256}''$ . It is important to note that inches is the default unit. If 6 is entered, it is assumed to be 6 inches. To enter feet, you must include the (') symbol.
- **Decimal units.** Dimensions are given in units, such as 1.75 or 3.625. Decimal units are generally used in mechanical drafting, because ANSI Y14.5M Dimensioning and Tolerancing standards specify that decimal inch or metric units in millimeters be used in engineering drawings. AutoCAD can display a maximum of eight decimal places. Decimal is the default type of unit in AutoCAD.
- Engineering units. Dimensions are given in feet, inches, and decimal parts of an inch, such as 6'-5.25". AutoCAD automatically converts and displays 12 inches as 1 foot. It is important to note that inches is the default unit. If 6 is entered it is assumed to be 6 inches. To enter feet, you must include the (') symbol. Each engineering unit in AutoCAD represents one inch. Engineering units are used primarily in civil drafting. Detailed construction drawings and topographic maps for planning and constructing highways and harbors are examples of typical civil drafting projects.
- Fractional units. Dimensions are given in whole units and parts of a unit as a fraction, such as  $26\frac{5}{8}$ . Fractional units may be any value, such as inches, feet, or miles. You can use fractional representation with any system of measure. As with architectural units, the smallest fraction that AutoCAD can display is  $\frac{1}{256}$ ".
- Scientific units. Dimensions are given as a real number raised to a power of 10, such as 125E+02. You can use scientific notation to represent any system of measure. You use scientific notation primarily when working with very large numbers.

#### DISPLAY PRECISION

The display format and entry format of the units selected is determined by the number of decimal places or the smallest fraction. The easiest method of selecting the units and setting the display precision is with the Drawing Units dialog box (Figure. 4.1). You can access this dialog box in one of two ways:

- · Choose Units from Format on the menu bar, or
- Enter units at the Command: prompt.

The default precision value is shown in the dialog box. After you have determined the type of units you will use, you must decide on the accuracy of the drawing display. For example, if you are using decimal units, two digits may be shown as 2.88, three digits as 2.875, and four digits as 2.8751. As shown in this example for decimal units, Auto-CAD rounds the value of the display.

#### FOR THE PROFESSIONAL



Regardless of the display precision setting, AutoCAD is accurate to 14 places for all types of units. The only values affected by changing the precision are the display of the coordinates on the status bar and the format of the values when you examine the drawing for an area or distance.

When you are working on mechanical drawings, three to four digits are normally used for inch drawings. For metric drawings, one- or two-place decimals are generally used, such as 16.5 or 16.50. This difference is due to the greater distance represented by inches as opposed to millimeters.

With architectural and fractional units, the precision is determined by the size of the fraction's denominator. Selecting 1/8 results in the values being displayed in 1/8-inch increments (or their least common denominator). For example, the display may read  $6'-5\frac{1''}{8}$ . If 1/4 were chosen as the precision, the same dimension would be rounded off to the nearest 1/4-inch increment, or  $6'-5\frac{1''}{4}$ .

#### ANGULAR MEASUREMENT

AutoCAD also has five different angular measurement options. The examples in Table 4.2 show how the value of 45.5 degrees appears in each format.

Descriptions of the angular measurement options for AutoCAD follows:

- Decimal degrees. Displays angular measurement as real numbers with up to eight decimal places. This is the default type of angular measurement in AutoCAD.
- **Deg/Min/Sec.** Displays angular measurement in degrees, minutes, and seconds. This representation uses ASCII characters, such as 30d12'38". Measurements of less than 1 second are displayed as decimal places.
- **Grads.** Displays angular measurement as grads. A lowercase g appears after the value, such as 37g. Ninety degrees equals 100 grads.
- **Radians.** Displays angular measurement as radians. A lowercase *r* appears after the value, such as 6.2832r. A radian is 180/pi degrees.
- Surveyor's units. Displays angular measurement in degrees, minutes, and seconds together with quadrant bearings. An example is N45d12'25"E. Surveyor's units are based on a circle divided into four quadrants, so no angular value can be greater than 90 degrees.



#### FOR THE PROFESSIONAL

If the status bar doesn't have enough room to display the coordinates normally, AutoCAD sometimes displays them in scientific units. This does not mean that the units have changed, only that AutoCAD must use scientific units to display them.

Table 4.2	UNITS OF ANGULAR MEASUREMENT		
	Report Format	Example	
	Decimal degrees	42.5000	
	Deg/Min/Sec	42d30'0"	
	Grads	47.2222g	
	Radians	0.7418r	
	Surveyor's units	N 47d30'0"E	

Figure 4.2

Use the Drawing Units dialog box to set the precision for angles.

Drawing Units			? ×
Length		Angle	
<u>Туре:</u>		Туре:	
Decimal	*	Decimal Degrees	-
Precision:		Precision:	
0.0000	*	0	*
Drawing units for De When inserting block	esignCente ks into this	r bloci drawi 0.0000 0.00000 0.00000	
Sample Output 1.5000,2.0039,0.000 3.0000<45,0.0000	00	0.000000	<u> </u>
OK Ca	incel	Direction	Help

**Angle Display Precision** The degree of accuracy for angles is determined by the drawing requirements. Two-place decimal degrees or degrees and minutes are normally used for mechanical drawings. Mapmaking with civil drawings often requires degrees, minutes, and seconds.

You can easily change the angle display precision in the Units Control dialog box. As noted, you access the Drawing Units dialog box by entering units at the Command: prompt or choosing Format/Units. To change the precision, select the arrow at the right of the Precision text box. The Precision list box appears, as shown in Figure 4.2. Use your pointing device to select the precision you want.

**Angle Direction** By default, AutoCAD assumes that 0 degrees is toward the right (east, or 3 o'clock). Also by default, angles increase in the counterclockwise direction, as shown in Figure 4.3.

You can change the location of 0 degrees and the direction of angular measurement with the Direction Control dialog box. Surveyors often change these defaults because typically they measure angles clockwise from north rather than counterclockwise from east. To change the angle direction and location of 0 degrees, choose <u>Direc-</u> tion in the Units Control dialog box. The Direction Control dialog box appears, as shown in Figure 4.3.

To change the angle 0 direction, select the appropriate location with the corresponding radio button. If you choose Other, you can enter an angle value in the box, which becomes the new angle 0 direction, or you can choose Pick, which allows you to "show" AutoCAD the direction for angle 0 by picking two points in the drawing window.

Direction Cor	itrol ? 🗙
Base Angle	
	0.00
C North	90.00
⊂ <u>W</u> est	180.00
C South	270.00
C Other	Pick / Type
Angle:	0
	OK Cancel

Figure 4.3 Change the angle 0 location and direction of angular measurement in the Direction Control dialog box.

#### TUTORIAL 4.1: DRAWING LINES USING THE FIVE DIFFERENT UNITS

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In this tutorial you will create five parallel lines using the different units discussed in the unit. Once all five lines are created you'll be able to compare them.

- From the Standard toolbar, select New. In the Create New Drawing dialog box, select the Start from Scratch button, English Default setting; then select OK.
- Choose Format/Units. Change the units to Scientific and choose OK.
  - From the Draw toolbar, select Line.
    - At the \_line specify first point: prompt, type 2e00, 2e00 and press Enter.
    - At the Specify next point or [Undo]: prompt, type 1.45e01,2e00 and press Enter.
    - At the Specify next point or [Undo]: prompt, press Enter.
- Choose Format/Units. Change the units to Decimal and choose OK.
  - From the Draw toolbar, select Line.
    - At the \_line specify first point: prompt, type 2, 3 and press Enter.
    - At the Specify next point or [Undo]: prompt, type 14.5,3 and press Enter.
    - At the Specify next point or [Undo]: prompt, press Enter.
- 6. Choose Format/Units.

Change the units to Engineering and choose OK.

- From the Draw toolbar, select Line.
  - At the \_line specify first point: prompt, type 0'2", 04 and press Enter.

For this example we'll use the relative coordinate method to describe the line length.

- At the Specify next point or [Undo]: prompt, type @1'.5", 0 and press Enter.
  - At the Specify next point or [Undo]: prompt, press Enter.
- Choose Format/Units. Change the units to Architectural and choose OK.
- 9. From the Draw toolbar, select Line.
  - At the \_line specify first point: prompt, type 0'2, 0'5 and press Enter.

For this example we'll use the relative polar coordinate method to describe the line length.

- At the Specify next point or [Undo]: prompt, type @1'0-1/2"<0 and press enter.</li>
- At the Specify next point or [Undo]: prompt, press Enter.
- **10.** Choose Format/Units. Change the units to Fractional and choose OK.
  - From the Draw toolbar, select Line.
    - At the \_line specify first point: prompt, type 2,6 and press Enter.

For this example we'll use the polar coordinate method to describe the line length.

- At the Specify next point or [Undo]: prompt, type @12-1/2<0 and press Enter.
- At the Specify next point or [Undo]: prompt, press Enter.



#### FOR THE PROFESSIONAL

Point locations can be entered in architectural units as well as decimal units. When you indicate architectural units, use hyphens only to distinguish fractions from whole inches. Additionally, do not use spaces while giving a dimension. For example, you can specify 6 feet, 5 1/4 inches as 6'5-1/4" or 6'5.25", but not as 6'-51/4".

#### WORKING WITH TEMPLATE DRAWINGS

Whenever you begin a new drawing, that drawing is based on a template. AutoCAD comes with a standard template drawing named **acad.dwt**. By default, this template drawing is used when you select the Start from Scratch option in the Create New Drawing dialog box. If you continuously start from scratch and use the default ACAD template drawing, you will soon discover that you frequently make the same drawing aids adjustments over and over. Units, limits, grid, and snap are drawing tools that need to be set every time you begin a new drawing that is a different size than the template.

A template drawing may be defined as any drawing file that you want to use as the basis for a new drawing you are creating. The template drawing can contain settings for such variables as the limits, snap increment, and grid spacing. After the template drawing is loaded, you create geometry as before. Any changes you make in the new drawing do

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not affect the template. The advantage of using a template drawing is that it frees you from having to change the same settings every time you start a new drawing.



#### FOR THE PROFESSIONAL

There is nothing different about a drawing that is used as a template. Any AutoCAD drawing can be used as a template for any other drawing.

When you initially create a template drawing in AutoCAD, you set the limits, units, snap, and grid variables to your current specifications. In time, you may want to set up a template for each drawing size and type. For example, you might establish different template drawings that set the AutoCAD drafting tools and contain a border and title block for A-, B-, and C-size drawings. AutoCAD contains several predefined templates that meet the ANSI and ISO standards.

You may set up different templates for mechanical, electrical, and architectural drawings. Whenever you create several drawings that contain similar settings, consider making and using a template drawing. To use an existing template, select Use a Template in the Create New Drawing dialog box. Select a template file from the Select a Template list and choose OK.

#### CREATING A NEW DRAWING FROM A TEMPLATE

When AutoCAD starts, the standard AutoCAD graphics window is displayed. As discussed in Unit 2, you can begin creating geometry immediately, without giving your drawing a name. If you want to use a template drawing, it must be assigned before you begin working on the new drawing. You can begin a new drawing with one of these methods:

- · Select New from the Standard toolbar,
- Enter new at the Command: prompt, or
- Choose New from the File menu.

Either method accesses the Create New Drawing dialog box (see Figure 4.4). If you select the Start from Scratch option, the **acad.dwt** template file is used. This is Auto-CAD's default template drawing.

When you click the Use a Template button, a list of possible template drawings appears in the Select a Template window (see Figure 4.5). AutoCAD displays the contents of the C:\Program Files\ACAD2000\Template subdirectory.

ri≍i ™ Scratch
) settings.
OK Cancel

Figure 4.4 The Create New Drawing dialog box with the different options ave

box with the different options available for beginning a new drawing.

	plate	
Select a Template: Acad		
Tamplata Description	Browse	lot Styles.
Normal English (feet and inches) drawing template	e. Uses Named Pl	

If you select the Browse button, you will see the Select a Template File dialog box. This option allows you to change directories and choose a different template drawing (Figure 4.6).

After you select a new template drawing, you are ready to begin working on the drawing. All settings that were part of the template drawing are now part of your new drawing. The template drawing file will *not* be changed in any way when you begin creating geometry in the new drawing.



#### SKILL BUILDER

After you select a template drawing and choose OK, you do not have to enter a drawing name in the Create New Drawing dialog box. If you press Enter after selecting a template drawing, you can create and edit your drawing without giving it a name. You are prompted for a name the first time you save the drawing. Although this is fine for experimental drawings, you should get into the habit of naming your drawings from the beginning.



Figure 4.5 In the Select a Template window, you can select a different template drawing.

Figure 4.6 In the Select a Template File dialog box, you can change directories and select a different template drawing.

#### **TUTORIAL 4.2: CREATING A TEMPLATE DRAWING**

In this tutorial, you create a template drawing. This template will be used in a later exercise to help you draw a site plan. In this template drawing, you will set units, limits, grid, and snap; load a phantom linetype; and set the linetype scale. You begin by creating a new drawing. Follow these steps:

> From the Standard toolbar, select New. In the Create New Drawing dialog box, select Start from Scratch and English Default setting, then choose OK.

- You should now see the standard AutoCAD graphics window. From the Standard toolbar, select Save. In the Save Drawing As dialog box, change to the correct drive and subdirectory where you want to store your template drawings. Change the Save as Type window to Drawing Template File (\*.dwt) and enter aproto as the drawing name, as shown in Figure 4.7. When you have entered the correct information, select OK.
- **3.** When the Template Description dialog box appears, enter **English** template drawing and any other pertinent information.

You should be back to the standard AutoCAD graphics window. First, you need to set the units. For this exercise, you will be using architectural units with a precision of 0'-0 1/16"

Figure 4.7

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Using the Save Drawing As dialog box to create the template file.



Figure 4.8 Setting Units to Architectural, Precision to 1/16", and Angles to Surveyor's Units.

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Length Type:		Angle Type:	
Architectural	*	Surveyor's Units	-
Precision:		Precision:	
0'-0 1/16''	•	N Od E	-
		Clockwise	
When inserting block	ks into this dr	awing, scale them to:	
When inserting block Inches Sample Output 1.1/2".2".0" 3" <n 45d="" e.0"<="" td=""><td>ks into this dr</td><td>awing, scale them to:</td><td></td></n>	ks into this dr	awing, scale them to:	

and surveyor angles with a precision of N Od E. Follow these steps:

- **1.** Choose Format/Units. The Drawing Units dialog box appears.
- **2.** Using the pointing device, adjust the dialog box to reflect the settings shown in Figure 4.8.

Next, set the limits for the template drawing. The lowerleft limits will remain 0'-0", and the upper-right limits will be 144'-0",96'-0". Follow these steps:

- 1. Choose Format/Drawing Limits.
- **2.** Specify lower-left corner or [ON/OFF] <0'-0", 0'-0">: prompt, press Enter.
- 3. At the Specify Upper-right corner <1'-0", 0'-9">: prompt, type 144'-0",96'-0" and press Enter.

To show the entire limits within the View window, you issue the Zoom All option:

1. From the Standard toolbar, choose the Zoom All icon. By default, the Zoom Window icon appears as the top level icon. Select and hold the Zoom Window icon to access the flyout toolbar, and select the Zoom All icon.

After the limits are established, you set the grid to 5' and the snap to 1':

- 1. Choose Tools/Drafting Settings to access the Drafting Settings dialog box.
- 2. Enter 5' for the grid X and Y spacing.
- 3. Enter 1' for the snap X and Y spacing.
- 4. Check the Snap on and Grid On check boxes. Your dialog box should appear as shown in Figure 4.9.

Next, load a phantom line. Up to this point, the only linetypes you have drawn have been continuous lines. To create the site plan, you will need to use a phantom linetype. Although this procedure is introduced here, an in-depth discussion of loading linetypes and modifying linetype scales can be found in Unit 5, Understanding Layers and Linetypes. Follow these steps:

- Choose Linetype from Format on the menu bar 1. to access the Linetype Manager dialog box. THE REAL PROPERTY.
- 2. To load the PHANTOM linetype, select Load. The Load or Reload Linetype dialog box is displayed.
- 3. Use the scroll bar to display Phantom in the Available Linetype list box and then select Phantom.

- 4. Press Enter or select OK to accept the Phantom linetype in the Load or Reload Linetypes dialog box and return to the dialog box.
- 5. In the Linetype Manager dialog box, pick the Phantom linetype displayed in the dialog box to set it as the current linetype. Click directly on the word Phantom to select it, and then choose the Current button located at the top of the dialog box.

The next task is to set the linetype scale to 96 so that the line will be scaled correctly when displayed or plotted. A full description of linetype scaling is covered in Unit 5.

- 6. In the Linetype Manager dialog box, select the Show Details button located at the top of the dialog box.
- 7. Enter 96 for the new Global scale factor. Your dialog box should now look like Figure 4.10.
- 8. Select OK to accept the Linetype Manager dialog box. The Phantom linetype is now loaded and set, and the linetype scale is set.

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The Aproto.dwt template file is now ready to be saved for use later in this unit. To save the drawing, click the Save icon on the Standard toolbar.

Figure 4.9	Dialling Settings	Balance States	and the second second	AV.STAL	?
Setting the Grid to 5' and the Snap	Snap and Grid   Polar Tr	acking   Object Shap	1		
Setting the Grid to 5' and the Snap to 1'.	Snap and Gild Potar Tr Snap On (FS) Snap X spacing Snap Y spacing Angle: X base: X base:	acking   Object Snap  1'  1'  N 90d E  0''	Grid On (F7)     Grid X spacing:     Grid X spacing:     Grid Y spacing:     Snap type & style	5"  5" Ilar snap snap	
	Depions		C Pglar map	incel	Нер
Figure 4.10	Line Prine Managers				1
Use the Linetype Manager dialog box to set the linetype scale for the	Linntype filters Show all linetypes	۲ Inver	t filter	Load Ωumer#	Delete Hide gela
phantom linetype.	Current Linetype: PHANTOM Linetype 6(Layer ByBlock Continuous FEANTOM	Appearance Der	iniplien		
	Details Name PHANTOM Dissolption Phantom IF Line paper space units for	rscaling	Global coals fa	ector scale:	36  1.0000
			<u> </u>	Cancel	Help

#### COORDINATE SYSTEM BASICS

One of the historic landmarks of science and mathematics was the introduction of coordinates into geometry. The key idea behind the coordinate system is how a given set of points is represented. Points are represented as an ordered pair of real numbers, written in such a way that you can distinguish one number as the "first" and the other as "second." The usual notation is (x,y), where x and y are real numbers.

Ordered pairs of real numbers may be associated with points of a coordinate plane. For example, suppose that an ordered pair of numbers (x,y) is given. The first number of the pair is located on horizontal line L1, with the positive direction to the right (the x-axis). The second number of the pair is located on vertical line L2 (the y-axis), with the positive direction upward and the origin the same as that of the x-axis. The point (P) where the two lines intersect is labeled (x,y). The numbers x and y are called the Cartesian coordinates of the point P (Figure 4.11). Cartesian is named in honor of Descartes, a French philosopher who introduced the concept in the seventeenth century.

AutoCAD uses the same Cartesian coordinate system to designate points within a drawing. When you create a 2-D drawing, you can enter points for geometry creation with either the pointing device or the keyboard. The coordinate display in the status bar at the bottom of the screen shows the current location of the crosshairs in the present unit of measure. Note that the readout uses the Cartesian coordinate system. It should also be noted that AutoCAD has full 3-D potential by adding the distance from the origin along the z-axis to the Cartesian coordinates, as in (x,y,z). This concept is used when creating three-dimensional drawings.

#### USING COORDINATES IN AUTOCAD

At this point, you should be familiar with the standard AutoCAD graphics window. Note that an icon is shown in the lower-left corner, consisting of two perpendicular arrows. These arrows are directional indicators for the user coordinate system, or UCS. While creating 2-D drawings, the UCS icon shows the direction of the positive values for the x and y axes. The UCS is based on the Cartesian coordinate system and divides space into four quadrants. Figure 4.12 shows the UCS icon and the division of space based on Cartesian coordinates. You locate points in these quadrants by specifying their location in horizontal (x) and vertical (y) directions along the plane. It is also possible to change the origin point of the UCS.







Figure 4.12 The division of space, using the Cartesian coordinate system, with the UCS icon displayed.



To remove the UCS icon, select the  $\underline{V}iew/Display/\underline{U}CS$  icon. If  $\underline{O}n$  has a check next to it, the UCS icon is displayed. If it is not checked, it is not displayed. You can toggle the display of the UCS icon on and off by clicking  $\underline{O}n$ .

The AutoCAD screen has a direct relationship to the Cartesian coordinate system (see Figure 4.13). As you begin creating lines, AutoCAD asks you to specify First point:. In Unit 2, you entered this information using the pointing device, in association with the snap and grid. You did that by moving the crosshairs to a point on-screen and pressing the Pick button on the pointing device.

Using the pointing device with the snap and grid is excellent for creating certain types of geometry. Because of constraints imposed by the Snap and Grid, however, very specific locations cannot be made. In an attempt to locate more specific points, you could change the Snap and Grid settings to smaller values. When the grid spacing is too small, AutoCAD responds with the message Grid too dense to display. When the



Figure 4.13 The relationship of the drawing screen to the Cartesian coordinate system. Limits are set with the lower-left corner at 0,0 and the upper-right corner at two positive values.

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snap increment is set very small, it becomes difficult, if not impossible, to pick exact locations. Because many disciplines (such as architecture and engineering) often deal with very specific locations, a more exact means of controlling the "to point" is needed. You can control this point by using Polar Tracking and Polar Snap.

#### POLAR TRACKING

Polar tracking is similar to Ortho. With Ortho you can draw lines that are perfectly horizontal or vertical. With Polar Tracking, however, you can set angles other than 90 degrees. When Polar Tracking is activated you can track the cursor along a temporary alignment path defined by polar angles. Since Polar and Ortho are similar they *cannot* be on at the same time.

By default Polar Tracking is activated with alignment paths at 90 degree intervals. As you move the cursor, alignment paths and tool tips are displayed when you move the cursor near the polar angles. You can use this alignment path and tool tip to draw your object. You can activate Polar Tracking by doing one of the following:

- On the status bar click the Polar button,
- · Press the F10 key, or
- Select <u>Tools/Drafting</u> Settings from the menu bar to access the Drafting Settings dialog box. Then select the Polar Tracking tab and click the Polar Tracking radio button.

#### FOR THE PROFESSIONAL

Several of the polar tracking settings can be changed, such as the display of the polar tracking vector and the tool tip. These can be changed by selecting the Options button in the Drafting Settings dialog box. In the Options dialog box select the Drafting tab. The polar tracking settings are in the AutoTrack Settings area.

#### CHANGING THE POLAR ANGLE SETTINGS

While polar tracking is activated with alignment paths at 90 degree intervals, you can also choose from a variety of other angles. This can be accomplished through the Drafting Settings dialog box as shown in Figure 4.14. You can access the Drafting Settings dialog box by doing one of the following:

- Select <u>Tools/Drafting</u> Settings on the menu bar to access the Drafting Settings dialog box, then select the Polar Tracking tab, or
- Right-click the POLAR button on the status bar and select Settings.



Figure 4.14 Select the Polar Tracking tab in the Drafting Settings dialog box to change the polar tracking settings. In the Polar Angle Settings area you can change the angle where the polar tracking will be activated. The following features are available in the Polar Angle Settings area:

- **Increment Angle.** By accessing the drop-down list you can change the angle increments to 90, 45, 30, 22.5, 18, 15, 10, or 5. You can also type in an angle to use. Whatever angle you select you will "snap" to that angle, and multiples of it, for up to 360 degrees. For example, if you choose 90 you can snap to 0, 90, 180, and 270 degrees.
- Additional Angles. By activating this feature you can use angles shown in addition to the incremental angles.
- New. By selecting the New button you can enter in additional angles to snap to. You will be able to snap to this angle only, *not* its multiples. You can add up to 10 additional angles.
- Delete. This will allow you to delete selected additional angles from the list.

#### UNDERSTANDING THE POLAR ANGLE MEASUREMENT

When you are using polar tracking and the cursor approaches one of the polar angles, the polar tracking tool tip will show the word *Polar* followed by the distance and angle from the last point. This information is displayed as a polar coordinate. Polar coordinates can be defined as a distance and angle from a specified point. By default, AutoCAD measures angles in a counterclockwise manner. Zero degrees is toward the positive x axis, to the right of the point being referenced. This means that 90 degrees is straight up, toward the positive y axis; 180 degrees is to the left, toward the negative x axis; and 270 degrees is straight down, toward the negative y axis. Figure 4.15 shows the default AutoCAD angle measurement in a two-dimensional coordinate system. This direction can be changed using the **units** command, if so desired. There are two settings for polar angle measurement: Absolute and Relative to last segment. Each setting has specific characteristics.

#### Absolute Polar Tracking

An absolute polar coordinate is specified by the distance, the < sign, then the angle. For example, a 6-unit-long line at an angle of 45 degrees is specified as 6<45. AutoCAD locates the end point of the line 6 units away at an angle of 45 degrees. All absolute polar coordinates are measured from the origin. Say, for example you set Increment Angle to 90 with no additional angles. You begin the **line** command and draw a 2-unit line at 45 degrees for the first segment. With Absolute polar tracking selected, you can draw the line in increments of 90 degrees relative to the user coordinate system. Figure 4.16 shows a 2-unit line at 45 degree angle, with the other possible line increments (shown as dashed lines) when Absolute



Figure 4.15 The default AutoCAD angle measurement system in a 2-D coordinate system.





polar tracking is selected. Note how the dashed lines are horizontal and vertical, since they were determined by the location of the origin, *not* from the original 45 degree line.

#### RELATIVE POLAR TRACKING

Relative polar coordinates are measured from the last point entered as a distance followed by an angle. They use the same syntax as Absolute polar tracking, however, the angle in relative polar tracking is now in relation to the last point entered. Using the same example used to describe Absolute polar tracking, Figure 4.17 again shows a 2-unit line at a 45 degree angle with the other possible line increments (shown as dashed lines). With Relative polar tracking, note how the tool tip now displays Relative Polar, and the dashed lines are now at 90 degree angles to the *original* 45 degree line.



Figure 4.17 Possible line segments with Relative polar tracking selected.

#### TUTORIAL 4.3: USING ABSOLUTE POLAR TRACKING TO CREATE A SIDE ELEVATION PROFILE

4.

In this tutorial, you create a side elevation profile (see Figure 4.18) using the **line** command and absolute polar tracking. Follow these steps:

1.

From the Standard toolbar, select New. Select the Start from Scratch button, English default setting; then select OK.

- 2. Select <u>Tools/Drafting</u> Settings from the menu bar to access the Drafting Setting dialog box.
- **3.** Select the Polar Tracking tab. Change the Increment angle to 22.5, and make sure your Object Snap Tracking Settings is set to Track using All polar angle settings and the Polar Angle measurement is set to <u>Absolute</u>. Your

Drafting Settings dialog box should look like Figure 4.19. Select OK to close the Drafting Settings dialog box.

- From the Draw toolbar, select Line. You will draw all five lines continuously, starting at point A and moving counterclockwise to points B, C, D, and E, and finally back to point A. You will use absolute polar tracking to locate all points in this exercise.
- 5. At the Specify first point: prompt, type 3,3 and press Enter.

The line starts at point A. Continue using absolute polar tracking to complete the drawing.



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When you are entering the coordinates and notice a mistake, simply type **u** at Specify next point or [Undo]: prompt and press Enter. This will undo the last coordinate and allow you to re-enter the correct coordinates.

Figure 4.18 Creating the side elevation profile, using absolute polar tracking.

ile,		D	
E			
G			$ _{\mathbb{B}}$

#### Figure 4.19

Configuring the Drafting Settings dialog box for an increment angle of 22.5 degrees and absolute polar tracking.

Dbject Snap Tracking Settings C Track orthogonally only
Track using all polar angle setting: Polar Angle measurement Abcolute Belative to last segment

- **6.** To locate point B, move the crosshair to the right. When the crosshair is directly to the right and several units away from the starting point it will snap to 0 degrees, and the tool tip will be displayed as shown in Figure 4.20. Don't worry about the distance, since you will enter the coordinate from the keyboard.
- With the cursor snapped to 0 degrees and the tooltip displayed, enter 6 using the keyboard at the Specify next point or [Undo]: prompt, and press Enter.
- 8. To locate point C, move the cursor to 90 degrees until the cursor snaps and the tooltip is displayed. Type 3 at the Specify next point or [Undo]: prompt, and press Enter.
- 9. To locate point D, move the cursor to 158 degrees and type 3.25 at the Specify next point or [Close/Undo]: prompt, and press Enter.
- 10. To locate point E, move the cursor to 203 degrees and type 3.25 at the Specify next point or [Undo]: prompt, and press Enter.
- **11.** To close the line on starting point A, move the cursor to 270 degrees, type **3** at the Specify next point or [Undo]: prompt, and press Enter.
- **12.** At the Specify next point or [Undo]: prompt press the right mouse button and select Enter from the pop-up menu.



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Point locations can be entered in architectural units as well as decimal units. When you indicate architectural units, use hyphens only to distinguish fractions from whole inches. Additionally, do not use spaces while giving a dimension. For example, you can specify 6 feet, 5 1/4 inches as 6'5-1/4" or 6'5.25", but not as 6'-5 1/4".

Polar: 5.4409 < 0°

Figure 4.20 The tooltip is displayed when the cursor snaps to a 0 degree angle.

TUTORIAL 4.4: USING RELATIVE POLAR TRACKING TO CREATE A SIDE ELEVATION PROFILE

In this tutorial, you create the same side elevation profile (refer to Figure 4.18) using the **line** command, only this time you will use relative polar tracking. Follow these steps:



From the Standard toolbar, select New. Select the Start from Scratch button, English default setting; then select OK.

- 2. Select <u>Tools/Drafting</u> Settings from the menu bar to access the Drafting Setting dialog box.
- **3.** Select the Polar Tracking tab. Change the Polar Angle measurement to <u>Relative to last segment</u>. Your Drafting Settings dialog box should look like Figure 4.21. Select OK to close the Drafting Settings dialog box.



From the Draw toolbar, select Line. You will use relative polar tracking to locate all points in this exercise.

- 5. At the Specify first point: prompt, type 3,3 and press Enter.
- 6. To locate point B, move the crosshair to the right until the cursor snaps at 0 degrees and the tooltip appears.

Enter 6 using the keyboard at the Specify next point or [Undo]: prompt, and press Enter.

- 7. To locate point C, move the cursor to 90 degrees until the cursor snaps and the tooltip is displayed. Note how the tooltip now displays Relative Polar. Type 3 at the Specify next point or [Undo]: prompt, and press Enter.
- 8. To locate point D, move the cursor to 68 degrees and type 3.25 at the Specify next point or [Close/Undo]: prompt, and press Enter. Since we are now using relative polar tracking, the angle is now relative to the previous point instead of absolute.
- 9. To locate point E, move the cursor to 45 degrees and type 3.25 at the Specify next point or [Close/Undo]: prompt, and press Enter.
- 10. To close the line on starting point A, move the cursor to 68 degrees, type 3 at the Specify next point or [Undo]: prompt, and press Enter.
- **11.** At the Specify next point or [Undo]: prompt press the right mouse button and select Enter from the pop-up menu.

	Proteing Settings     Image: Settings       Snap and Grid     Polar Tracking Object Snap       Polar Angle Settings     Object Snap Tracking Settings       Increment angle:     Track orthogonally only       22.5     Image: Track using all polar angle settings       Additional angles     New       Polar Angle measurement       Image: Delate       Polar Angle measurement       Image: Delate       Image: Delate
Figure 4.21 Configuring the Drafting Settings dialog box for relative polar tracking	Optione OK Cancel Help

#### POLAR SNAP

Polar snap is used with polar tracking to make the cursor snap to specific distances along the tracking angles. For example, if you set polar tracking to a 45 degree angle and polar snap to 10, you can draw lines 10, 20, 30, and so on units long at 45 degree angles. To use polar snap both POLAR and SNAP must be on. You can easily confirm this by looking at the status bar. You must also activate Polar Snap in the Snap and Grid tab of the Drafting Settings dialog box. You can access the Drafting Settings dialog box by doing one of the following:

- Select Tools/Drafting Settings on the menu bar to access the Drafting Settings dialog box, then select the Snap and Grid tab, or
- Right-click the SNAP button on the status bar and select Settings.

To activate Polar Snap in the Drafting Settings dialog box select the Polar Snap radio button in the Snap type and style area, and enter a distance in the Polar Distance text box as shown in Figure 4.22.

Even if you have Polar Snap activated, you can still enter a distance at the keyboard. A distance entered at the keyboard will override the Polar Snap setting.



Figure 4.22

Configuring the Drafting Settings dialog box for polar snap.

Snap On (F9)		Grid On (F7)	
Sinap		Grid	<b></b>
	0.2500	Grid X spacing:	0.5000
	0.2500	Grjd Y spacing:	0.5000
	U	Care has t at to	
	0.000	Shap type & style	
	0.0000	Gid snap	
		C Hectory/	
olar spacing		C loogstad at	
Polar distance:	0.2500	Polar snap	

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#### TUTORIAL 4.5: USING POLAR TRACKING AND POLAR SNAP TO CREATE A SIDE ELEVATION PROFILE

In this tutorial, you create the side elevation profile shown in Figure 4.23. You will use polar tracking, polar snap, and the **line** command. Follow these steps:



From the Standard toolbar, select New. Select the Start from Scratch button, English default setting; then select OK.

- On the status bar right-click on the Polar button and select <u>Settings</u>.
- Ensure the polar angle measurement is set to <u>Relative to</u> last segment and the increment angle is set to 22.5. Your Drafting Settings dialog box should look like Figure 4.21.
- 4. Select the Snap and Grid tab in the Drafting Settings dialog box. Select the Polar Snap radio button in the Snap type and style area, and enter 0.2500 in the Polar distance. Your Drafting Settings dialog box should look like Figure 4.22. Select OK to close the Drafting Settings dialog box.



From the Draw toolbar, select Line. You will use relative polar tracking and polar snap to locate all points in this exercise.

- 6. At the Specify first point: prompt, type 3,3 and press Enter.
- **7.** To locate point B, move the crosshair to the right until the cursor snaps and the tooltip reads **Polar: 6.000<0** as shown in Figure 4.24. Select this point by pressing the left mouse button.
- To locate point C, move the cursor to 90 degrees until the cursor snaps and the tooltip reads Relative Polar: 3.0000<90 and select this point.</li>
- To locate point D, move the cursor to 68 degrees until the cursor snaps and the tooltip reads Relative Polar: 3.2500<68 and select this point.</li>
- To locate point E, move the cursor to 45 degrees until the cursor snaps and the tooltip reads **Relative Polar:** 3.2500<45 and select this point.</li>
- **11.** To close the line on starting point A, right-click and select Close in the pop-up window.



#### TUTORIAL 4.6: CREATING PROPERTY LINES WITH POLAR TRACKING, POLAR SNAP AND A TEMPLATE DRAWING

In this tutorial, you create the property lines shown in Figure 4.25. To begin this drawing, you will need to have completed the template drawing exercise at the beginning of the previous unit. That file is used to load default settings for this drawing. After loading this template and beginning a new drawing, you will be ready to create the property lines, using a phantom linetype.

- 1. From the Standard toolbar, select New.
- 2. When the Create New Drawing dialog box appears, select the Use a Template button and adjust your settings to reflect those found in Figure 4.26 to load the Aproto.dwt template drawing.
- **3.** On the status bar, right-click on the Polar button and select <u>Settings</u>. Set the Increment angle to N 60d E. Ensure the Polar Angle measurement is set to <u>Absolute</u> and the Object Snap Tracking Settings is set to Track using all polar angle settings. Your Drafting Settings dialog box should look like Figure 4.27.

4. Select the Snap and Grid tab in the Drafting Settings dialog box. Select the Polar Snap radio button in the Snap type and style area, and enter 5' as the Polar distance. Select OK to close the Drafting Settings dialog box.

Now you begin the drawing, using the **line** command and absolute coordinates to specify point A at 59,59:

- 5. From the Draw toolbar, select Line.
- 6. The line begins at point A. At the Specify first point: prompt, enter 5',5'.

To continue the line to points B, C, D, E, and F, you use polar tracking and polar snap. Note that you will be using architectural units and the surveyor's units of angle measurements.

 To lo locate point B, move the crosshair to the right until the cursor snaps and the tooltip reads Polar: 130'-0"<N 90d E.</li>



#### Review

	Snap and Grid Polar Tracking Object	Snap
	Polar Tracking On (F10)	Object Coop Tanal ing Calling
	Increment angle:	C Track orthogonally only
	N 60d E	Track using all polar angle settings
	New	Polar Angle measurement
	し対任法	<ul> <li></li></ul>
		Eelative to last segment
igure 4.27		
et the Increment Angle to N 60d		

- To locate point C, move the cursor until it snaps and the tooltip reads Polar: 45'-0"<N Od W and select this point.
- To locate point D, move the cursor until it snaps and the tooltip reads Polar: 45'-0"<N 30d W and select this point.
- To locate point E, move the cursor until it snaps and the tooltip reads Polar: 60'-0"<S 90d W and select this point.
- To locate point F, move the cursor until it snaps and the tooltip reads Polar: 60'-0"<S 60d W and select this point.
- **12.** To close the line on starting point A, right-click and select Close in the pop-up window.
- **13** Save your drawing by selecting Save from the Standard toolbar, naming it site.dwg.

#### SUMMARY

Selecting the proper units and precision gives AutoCAD the power and versatility to be used in a wide variety of occupations. Remember you are not limited to one setting of units or precision in a drawing, but may change them at any time without affecting objects previously created.

Template drawings, if properly used, can save you a considerable amount of time. Including common settings, title blocks, and even geometry in a template drawing not only improves drawing standardization but also increases productivity.

It is important that you become proficient with the coordinate entry methods introduced in this unit. Without a mastery of these concepts it is impossible to create difficult drawings accurately. It's very important that the AutoCAD user never "guess" the appropriate points. What may look fine on the AutoCAD screen will probably result in an inaccurate drawing when plotted.

#### REVIEW

- 1. List four things the type of units selected affects in your drawing.
- 2. List and briefly describe the five types of measurement report formats available.
- 3. Describe the relationship between display precision and accuracy of AutoCAD.
- 4. Explain template drawing and how it is similar to other drawings.
- 5. Create an illustration to explain the absolute coordinate entry method for locating points in a drawing.
- 6. Create an illustration to explain the absolute polar tracking method for locating points in a drawing.

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- 7. Create an illustration to explain the relative polar tracking method for locating points in a drawing.
- 8. Create an illustration to explain the polar snap method for locating points in a drawing.
- 9. Explain the advantages of using the polar snap and polar tracking method for locating points as opposed to using the Snap, Grid, and Ortho modes.
- **10.** Explain how a drawing that looks correct on the screen, or even on a plot, may not be accurate.

#### **EXERCISES**

Prepare detail drawings for the Model Glider shown in Figure 4.28. (See also color insert Figure 1.)

- 1. Set the units to Architectural, precision of  $\frac{1}{32}$ .
- 2. Create a detail drawing for each part required to produce the Model Glider. Use Snap, Grid, Ortho, and coordinate entry methods to produce an accurate drawing.



part of the Model Glider.