

ENGINEERING GRAPHICS WITH **AutoCAD® 2023**



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Engineering Graphics with AutoCAD® 2023

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Preface

This text teaches technical drawing and uses AutoCAD 2023 as its drawing instrument. Although it follows the general format of many technical drawing texts and presents much of the same material about drawing conventions and practices, the emphasis is on creating accurate, clear drawings. For example, the text shows how to locate dimensions on a drawing so that they completely define the object in accordance with ASME Y14.5-2009 national standards, but the presentation centers on the AutoCAD's **Dimensions** panel and its associated tools and options. The standards and conventions are presented and their applications are shown with the use of AutoCAD 2023. This integrated teaching concept is followed throughout the text.

Most chapters include drawing problems. The drawing problems are varied in scope and are open-ended, which means that there are several correct solutions. This is intended to encourage student creativity and increase their problem-solving abilities.

Chapters 1 through 3 cover tools on the **Draw** and **Modify** panels of the **Home** tab of AutoCAD's ribbon, and other commands needed to set up and start drawings. The text starts with simple **Line** commands and proceeds through geometric constructions. The final sections of Chapter 3 describe how to bisect a line and how to draw a hyperbola, a parabola, a helix, and an ogee curve. Redrawing many of the classic geometric shapes will help students learn how to use the **Draw** and **Modify** panels and other associated commands with accuracy and creativity.

Chapter 4 presents freehand sketching. Simply stated, there is still an important place for sketching in technical drawing. Many design ideas start as freehand sketches and are then developed on a computer. This chapter now includes extensive exercise problems associated with visual orientation.

Chapter 5 presents orthographic views. Students are shown how to draw three views of an object using AutoCAD 2023. The discussion includes projection theory, hidden lines, compound lines, oblique surfaces, rounded surfaces, holes, irregular surfaces, castings, and thin-walled objects. The chapter ends with several intersection problems. These problems serve as a good way to pull together orthographic views and projection theory. Several new, more difficult, exercise problems have been added to this edition. The chapter also includes an explanation of the differences between first- and third-angle projections as defined by ANSI and ISO conventions. Appropriate exercise problems help reinforce the understanding of the differences between the two standards.

Chapter 6 presents sectional views and introduces the **Hatch** and **Gradient** commands. The chapter includes multiple, broken-out, and partial sectional views and shows how to draw an S-break for a hollow cylinder.

Chapter 7 covers auxiliary views and shows how to use the **Snap**, **Rotate** command to create axes aligned with slanted surfaces. Secondary auxiliary views are also discussed. Solid modeling greatly simplifies the determination of the true shape of a line or plane, but a few examples of secondary auxiliary views help students refine their understanding of orthographic views and, eventually, the application of user coordinate systems (UCSs).

Chapter 8 shows how to dimension both two-dimensional shapes and orthographic views. The **Dimension** tools and their associated commands are demonstrated, and examples of how to use the **Dimension Styles** tool are included. The commands are presented as needed to create required dimensions. The conventions demonstrated are in compliance with ANSI Y14.5-2009.

Chapter 9 introduces tolerances. The chapter shows how to draw dimensions and tolerances using the **Dimension** and **Tolerance** commands, among others. The chapter ends with an explanation of fit types, and shows how to use the tables included in the Appendix to determine the maximum and minimum tolerances for matching holes and shafts.

Chapter 10 discusses the use of geometric tolerances and explains how AutoCAD 2023 can be used to create geometric tolerance symbols directly from dialog boxes. Both profile and positional tolerances are explained. The overall intent of the chapter is to teach students how to make parts fit together. Fixed and floating fastener applications are discussed, and design examples are given for both conditions.

Chapter 11 covers how to draw and design with the use of standard fasteners, including bolts, nuts, machine screws, washers, hexagon heads, square heads, set screws, rivets, and springs. Students are shown how to use the **Wblock** command to create drawings of the individual thread representations and how to use them for different size requirements.

Chapter 12 discusses assembly drawings, detail drawings, and parts lists. Instructions for drawing title blocks, tolerance blocks, release blocks, and revision blocks, and for inserting drawing notes are also included to give students better preparation for industrial practices.

Chapter 13 presents gears, cams, and bearings. The chapter teaches how to design by using gears selected from manufacturers' catalogs and websites. The chapter shows how to select bearings to support gear shafts and how to tolerance holes in support plates to maintain the desired center distances of meshing gears. It also explains how to create a displacement diagram and then draw the appropriate cam profile.

Chapter 14 introduces AutoCAD 3D capabilities. Both parallel (isometric) and perspective grids, as well as the world coordinate system (WCS) and user-defined coordinate systems (UCSs) are demonstrated so students learn the fundamentals of 3D drawings before drawing objects.

Chapter 15 shows how to create three-dimensional solid models. It includes examples of both parallel and perspective grids and using different **Visual Style** options. The chapter shows how to union, subtract, and intersect primitive shapes to create more complex models and orthographic views from those models.

Chapter 16, which is available online, presents two project problems: a milling vise and a tenon jig. These problems can be used for group or individual projects. These projects are intended to help students learn to work in groups and work on large, complex projects. This chapter can be found on the web as a supplement to the Instructor's Manual by registering your book at <https://www.pearson.com/us/higher-education/subject-catalog/download-instructor-resources.html>. Instructors may distribute this URL to students.

Online Instructor Supplementary Materials

Instructor materials are available from Pearson's Instructor Resource Center. Go to <https://www.pearson.com/us/higher-education/subject-catalog/download-instructor-resources.html> to register or to sign in if you already have an account.

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James D. Bethune

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David Byrnes

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You can find Chapter 16 and the Appendix at informit.com/title/9780137929993.
Click the *Downloads* tab to access the PDF file.

Chapter 16 Projects (Online Only)

Appendix (Online Only)

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Appendix (Online Only)

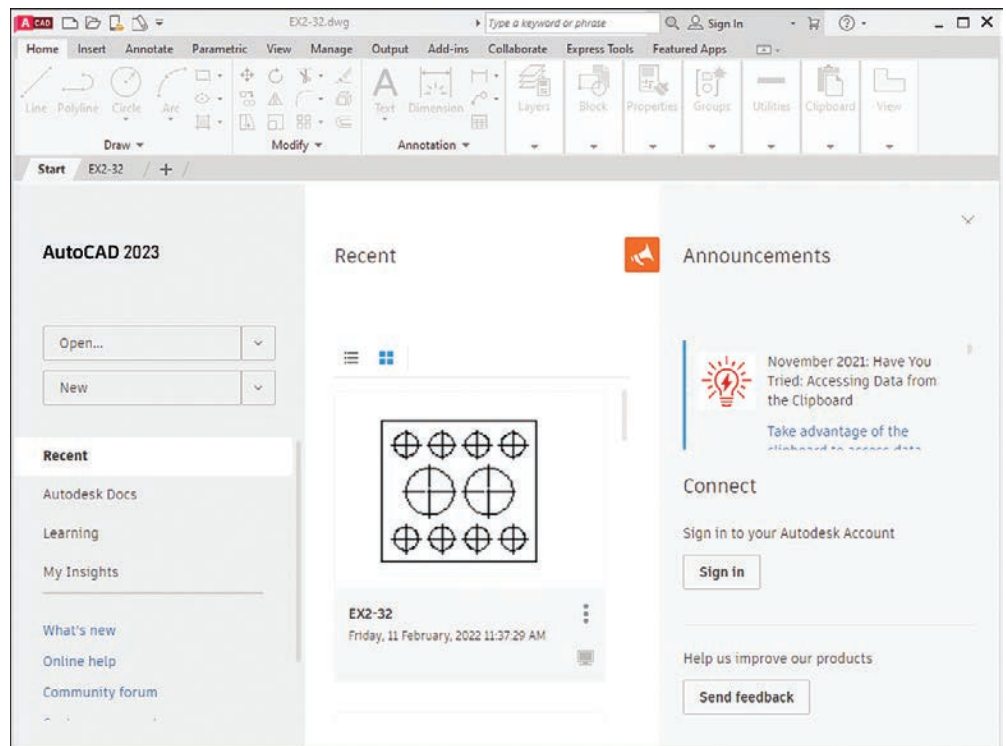
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chapter one

Getting Started

Figure 1-1



1-1 Introduction

This chapter introduces you to AutoCAD 2023. It covers basics such as using the **Application** menu, starting new drawings, making settings and entering data in dialog boxes, and saving your work.

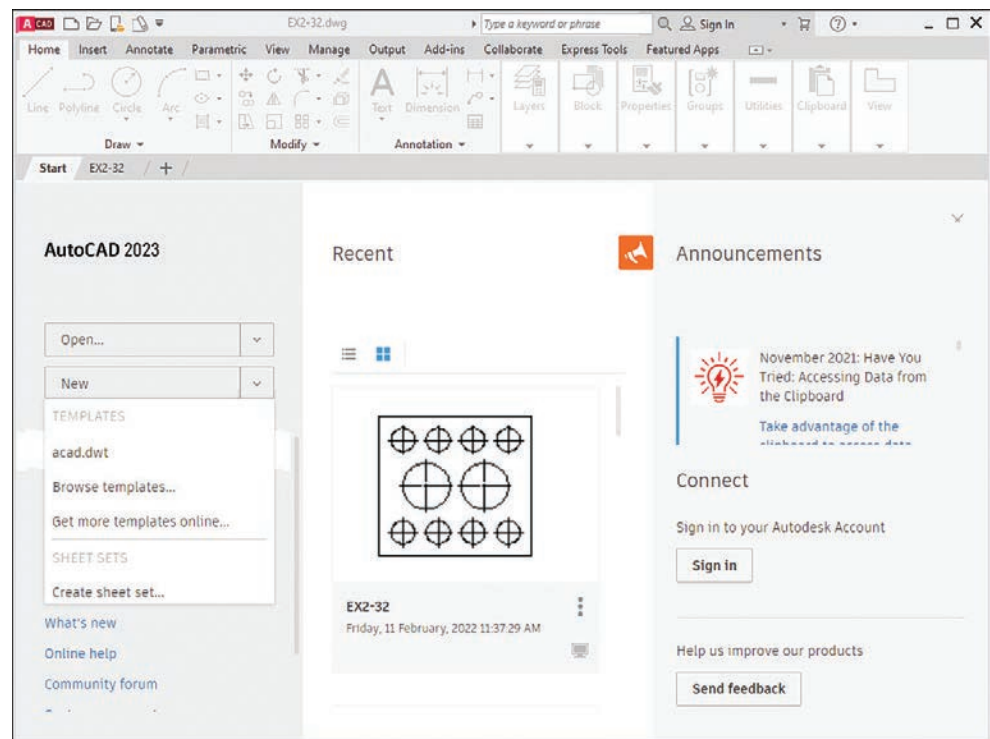
Figure 1-1 shows the initial AutoCAD drawing screen, which appears when the program is first started.

Starting a New Drawing

- 1 Click the down arrow beside the **New** button on AutoCAD's opening screen.

A list of recently used templates appears in the drop-down menu (Figure 1-2). You will use various templates throughout the text, but for a start, you will use the **acad.dwt** template. The **acad.dwt** template defines inches as its primary units. If the **acad.dwt** template does not appear in the drop-down, click **Browse templates...** and select it from there.

Figure 1-2



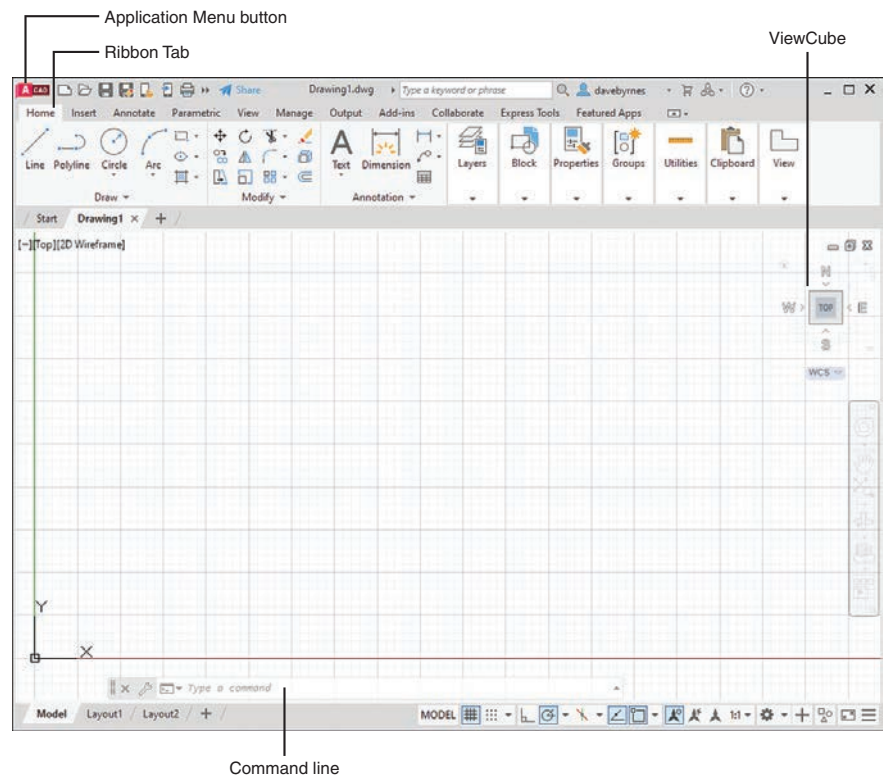
- 2 Click **acad.dwt** on the **Templates** list.

The drawing screen appears (Figure 1-3).

NOTE

The tool panels in the figure have a light-colored background—for printing clarity. Your background may be dark.

Figure 1-3



An Alternative Method to Starting a New Drawing

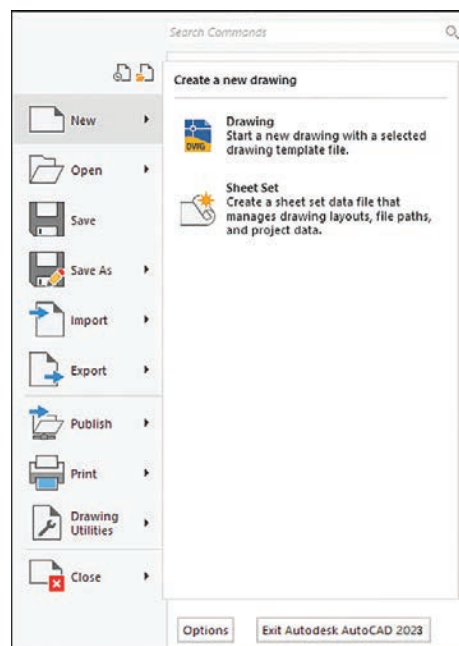
- 1 Click the **Application Menu** button in the upper-left corner of the drawing screen to display the **Application** menu.

A list of drawing commands and utilities appears (Figure 1-4).

- 2 Click **New**.

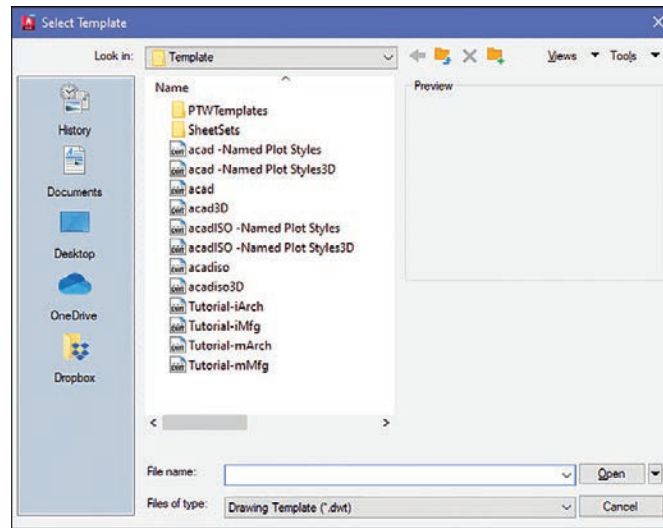
The **Select Template** dialog box appears (Figure 1-5).

Figure 1-4



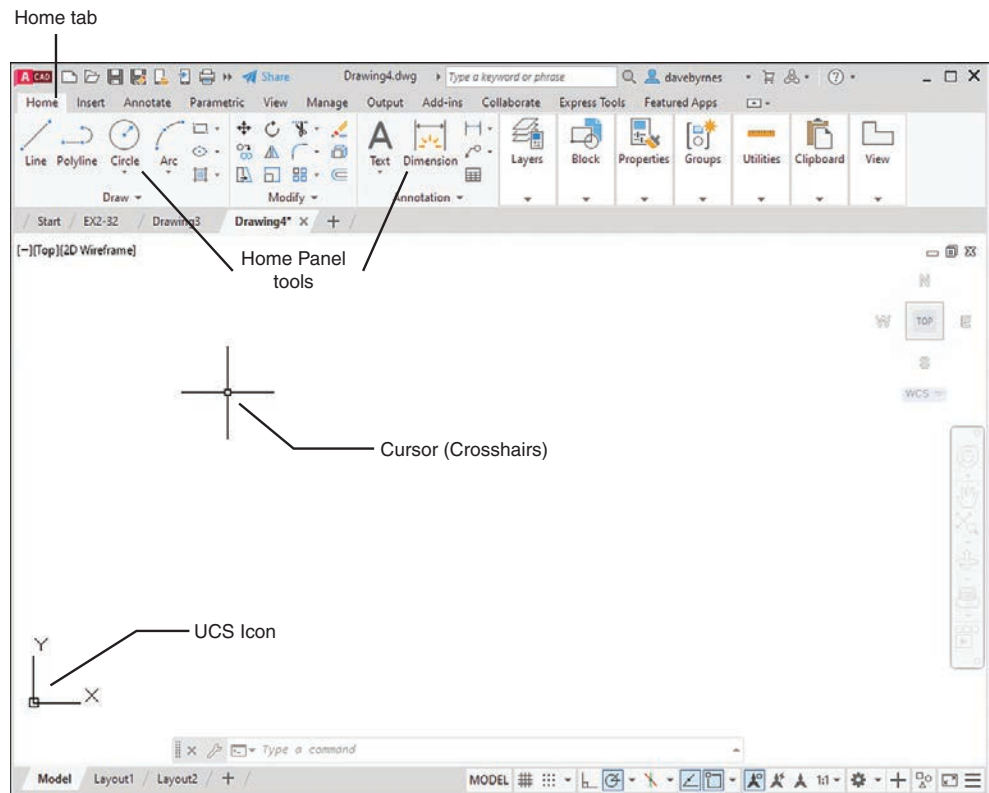
- 3 Select the **acad** template and click **Open**.

Figure 1-5



The AutoCAD drawing screen appears (Figure 1-6). The Ribbon appears at the top of the screen, showing a group of tabs and panels. Select different tabs to access other groups of panels. Panels contain commands.

Figure 1-6



The command line is located at the bottom of the screen, as are other tools (icons) for commands such as **Grid** and **Snap**. Use the command line to enter inputs for the commands, among other uses.

The drawing's name appears at the top of the screen. In Figure 1-4, for example, the drawing name is Drawing1.dwg. This is a default name created by AutoCAD. If a drawing name had been entered, it would appear where the Drawing1.dwg title currently appears.

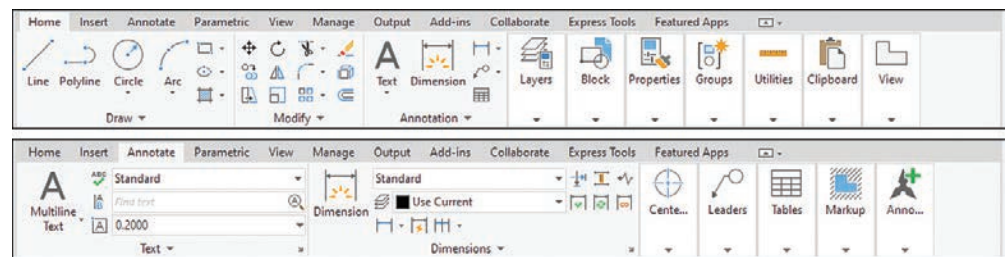
The large open area in the center of the screen is the *drawing area* or *drawing editor*. You create drawings in this area.

The symbol at the bottom-left corner of the drawing area is called the **User Coordinate System (UCS)** icon. It shows the direction of positive X and Y coordinates.

1-2 Tabs and Panels

The headings across the ribbon at the top of the screen (**Home**, **Insert**, etc.) are called *tabs*, and the groups of commands on the tabs are called *panels*. Figure 1-7 shows the **Home** panels and the **Annotate** panels.

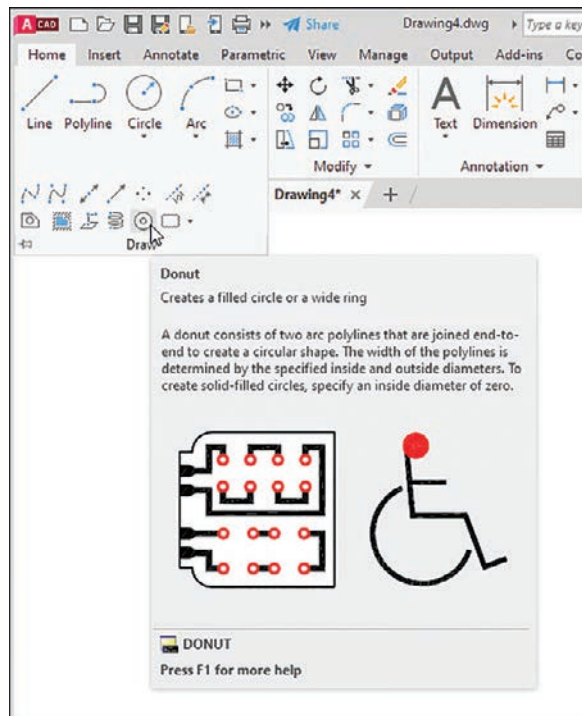
Figure 1-7



Accessing Additional Commands Within a Panel

Each panel shows a group of the most commonly used commands. Additional commands are available by clicking the arrow to the right of the panel's name. Figure 1-8 shows the additional **Draw** commands available.

Figure 1-8



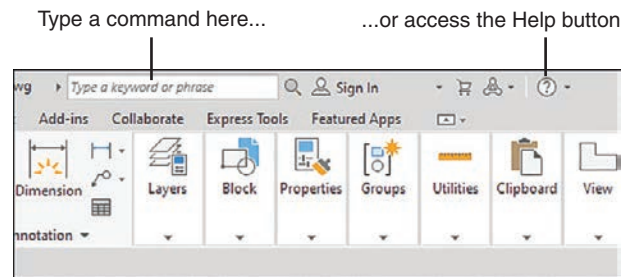
Tooltips for Commands

A *tooltip* is a pop-up help window that appears when the cursor is hovered over a command's icon (Figure 1-8). Initially, when you place the cursor over a command icon but don't click, a tooltip appears, identifying the command. After a few seconds the tooltip expands to further define the command.

Accessing Other Help Information

If you cannot find a command or if you need further instructions for operating a particular command, type a keyword into the text box in the program's title bar, and press **Enter** or click the **Access to Help** button located in the top-right section of the screen (Figure 1-9). The icon for the **Access to Help** tool is a question mark within a circle. The **Help** dialog box appears. Type in the name of what you are seeking and click the magnifying glass icon just to the right of the search box.

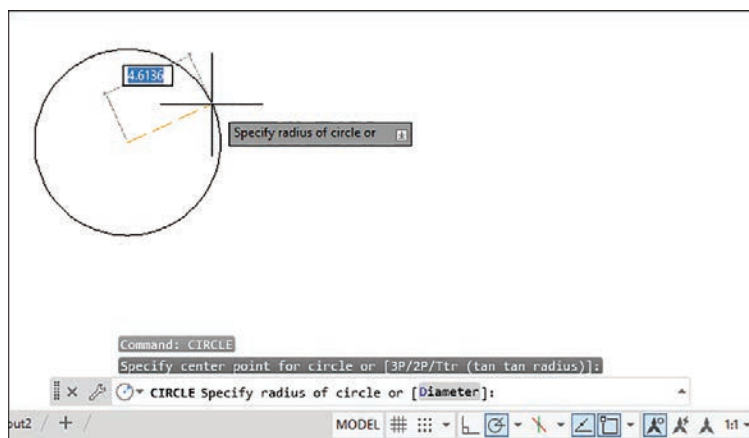
Figure 1-9



1-3 The Command Line Window

The command line window is located at the bottom of the drawing screen. Use it to access commands that do not have their own icons or to select options associated with the command. Figure 1-10 shows a circle. The word **CIRCLE** automatically appears in the command line when you click the **Circle** tool on the **Draw** panel. As presented, the circle will be defined by entering a radius value. Enter the radius value into the box with the blue background before clicking the left mouse button to complete the circle. If the radius value does not appear, press the F12 function key and ensure that the Dynamic Input is **ON**.

Figure 1-10



The command line shows the word Diameter in brackets: [Diameter]. Follow the next steps to use the **Circle** command's **Diameter** option.

Entering a Diameter Value

- 1 Click the **Circle** tool on the Home panel and draw a circle.
- 2 Click the command line box.
- 3 Type **d** and press **Enter**.

The system is now set for a diameter value for the circle.

- 4 Enter a value for the diameter of the circle and press **Enter**.

The options shown at the command line always include one uppercase letter. It may not always be the first letter. Type that letter and press **Enter** to access the option.

Diameter values may also be entered by first clicking the arrowhead next to the **Circle** tool and selecting the **Center, Diameter** option.

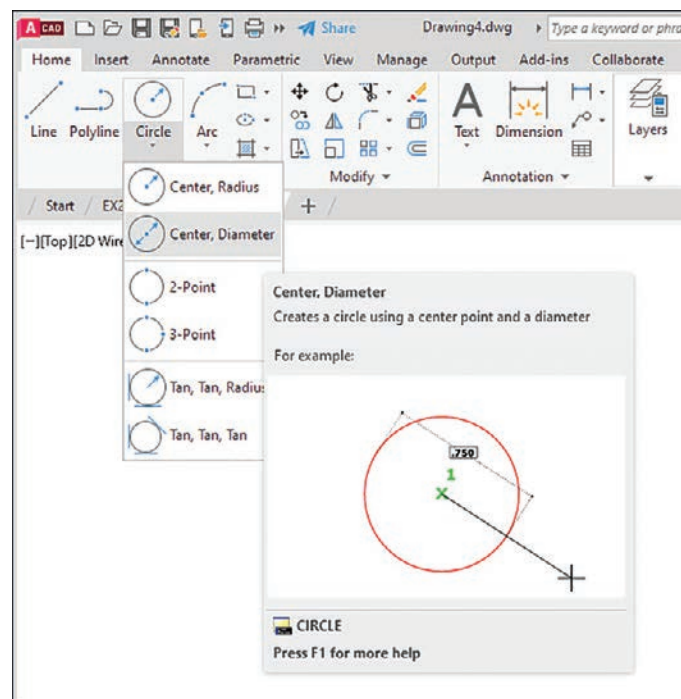
1-4 Command Tools

A *tool button* displays a picture (icon) that represents an AutoCAD command. Most commands have equivalent tool buttons.

Determining the Command That a Tool Button Represents

Figure 1-11 shows the steps to find the name and description of the command that the tool button executes.

Figure 1-11



1 Hover the cursor arrow over the selected tool button.

In the example shown, the **Circle** command tool button with the **Diameter** option is selected.

2 Hold the arrow still without pressing any mouse buttons.

The command name appears in a tooltip. If you continue to keep the cursor arrow on the tool button, an expanded tooltip that further describes the command appears.

1-5 Starting a New Drawing

When you start a new drawing, AutoCAD assigns a drawing name. The drawing units are specified, the drawing limits are modified, if needed, and **Grid** and **Snap** values are defined. The following four sections show you how to start a new drawing.

1-6 Naming a Drawing

You can use any combination of letters and numbers as a file name. Either uppercase or lowercase letters can be used, since AutoCAD file names are not case sensitive. The symbols \$, -, and _ (underscore) can also be used. Other symbols, such as % and *, cannot be used (Figure 1-12).

Figure 1-12

Correct drawing names:

FIRST EK-131-1 PA1-1a

Incorrect drawing names:

100% *. *

To locate a file on a the C: drive:

C:FIRST

All AutoCAD drawing files will automatically have the extension .dwg added to the given file name by default. If you name a drawing **FIRST**, it will appear in the files as **FIRST.dwg**. (A default setting is one that AutoCAD will use unless specifically told to use some other value.)

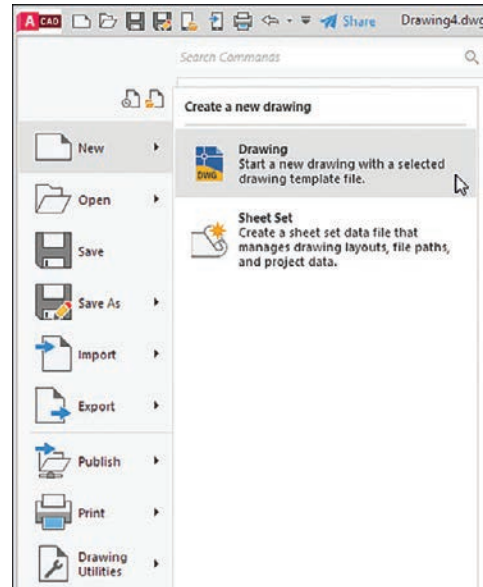
If you want to locate a file on another drive, specify the drive letter followed by a colon in front of the drawing name. For example, in Figure 1-12 **C:FIRST** will locate the drawing file **FIRST** on the C: drive.

Creating a New Drawing

There are three ways to access the **Create New Drawing** dialog box that is used to name a new drawing:

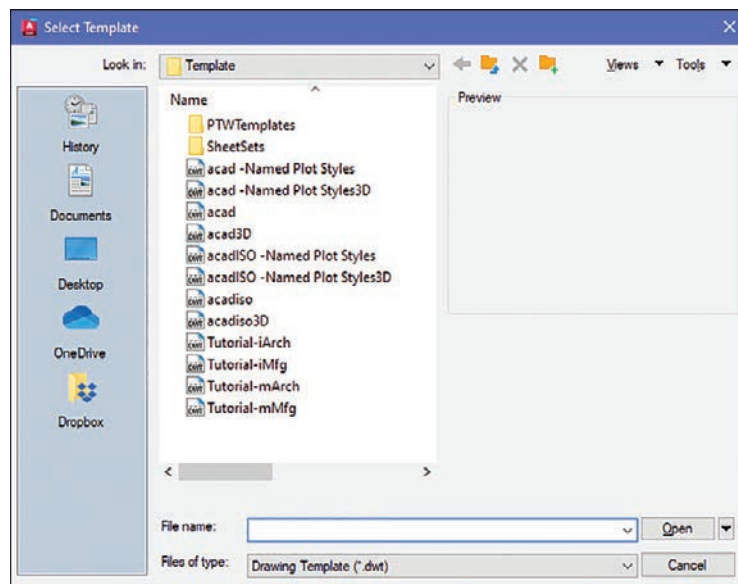
- Select **New** from the **Application** menu (Figure 1-13).
- Type the word **new** at a command prompt.
- Hold down the **Ctrl** key and press **N**.

Figure 1-13



Any of these methods will open the **Select Template** dialog box (Figure 1-14). The **acad** template will set up a drawing with inch values and ANSI style dimensions. The **acadiso** template will set up a drawing with millimeter values and ISO-style dimensions.

Figure 1-14



Saving a New Drawing File

The first time you use one of the **Save** tools to save your drawing, you must give your drawing a name (Figure 1-15). When you click **Save** for the first time in a new drawing, the **Save Drawing As** dialog box appears (Figure 1-16). Select a folder in which to save your work and enter a file name in the text box.

Figure 1-15

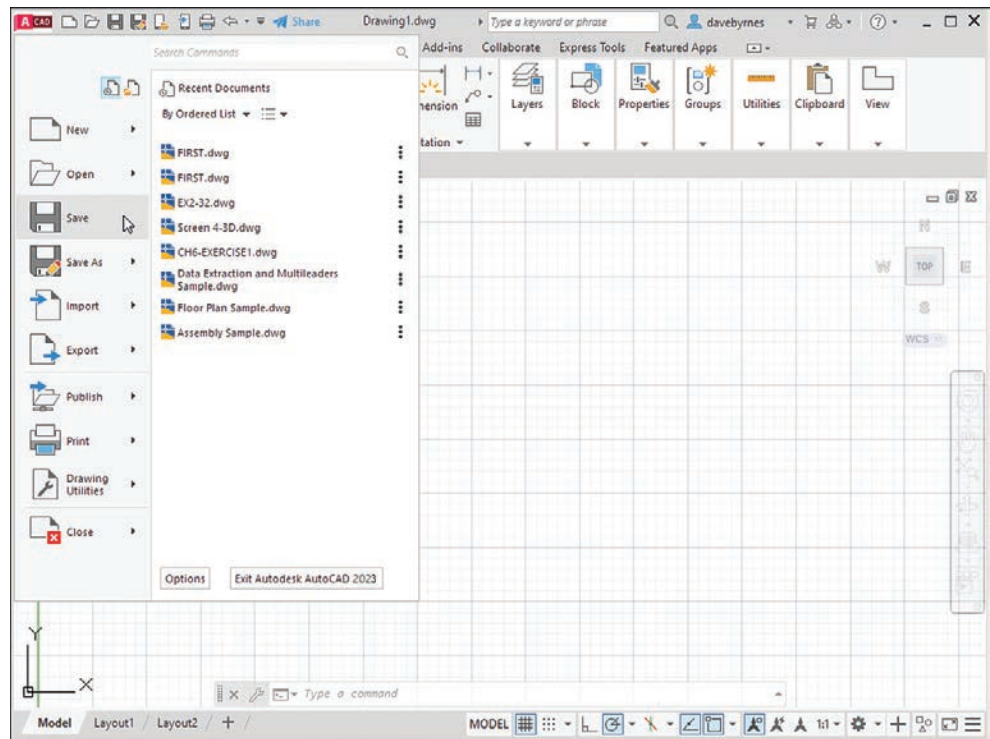
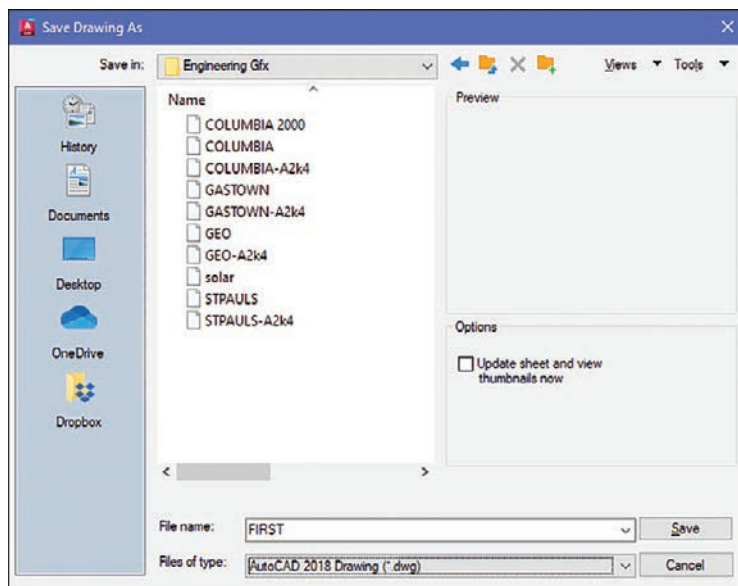


Figure 1-16



It's a good idea to save your work frequently. AutoCAD can be configured to save your drawings automatically, but it's a much better process to actively save your work. After you've created your drawing file in the **Save Drawing As** dialog box, using the **Save** command creates a backup version (*filename.bak*) and updates your saved file.

To save your work after you've given it a name, click the **Save** button in the **Quick Access Toolbar** at the top of the screen, or you can use the standard Windows shortcut: **Ctrl+S**.

- 1** Click the large **Application Menu** button in the upper-left corner of the screen.
- 2** Click **Save**.

Since you have not yet named and saved this drawing file, the **Save Drawing As** dialog box appears (Figure 1-16).

The **Save Drawing As** dialog box lists all existing drawings. Click on the thumbnail option to change the list to thumbnail drawings.

3 Enter the drawing name.

In this example, the drawing name **FIRST** was used.

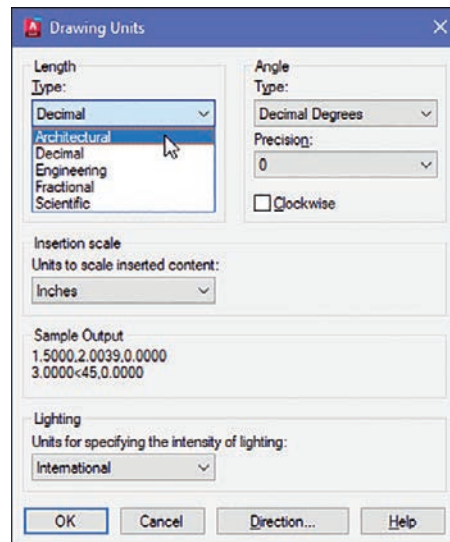
4 Click **Save**.

The name of the drawing appears at the top of the screen.

1-7 Drawing Units

AutoCAD 2023's **Drawing Units** dialog box allows for either English or metric units to be used as default values; however, AutoCAD can work in any of five different unit systems: scientific, decimal, engineering, architectural, or fractional. The default system is the decimal system, and it is used with either English values (inches) or metric values (millimeters). See Figure 1-17.

Figure 1-17



Access the Drawing Units dialog box by first opening the **Application** menu and then selecting **Drawing Utilities**.

Specifying or Changing the Drawing Units

1 Select **Drawing Utilities** in the **Application** menu.

2 Select **Units** (Figure 1-18).

The **Drawing Units** dialog box appears (Figure 1-19).

Figure 1-18

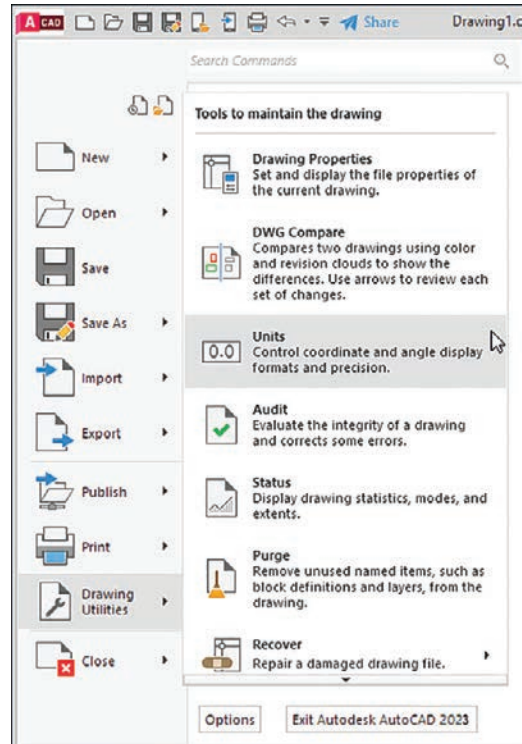
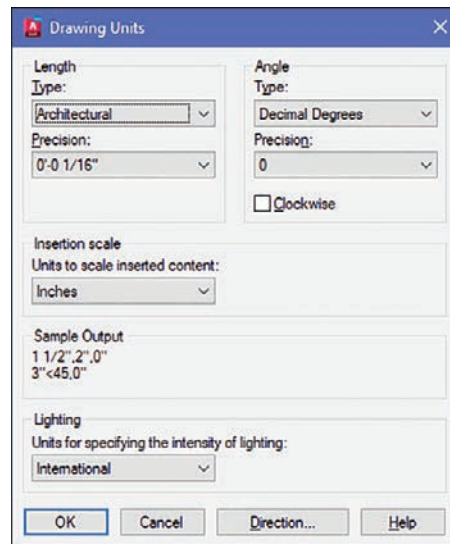


Figure 1-19



- 3 In the **Length** area, select architectural units by clicking the arrow to the right of the **Type** text box.

A list of the five unit options cascades down.

- 4 Select **Architectural**.

Note that the **Sample Output** section, located slightly below the center of the **Drawing Units** dialog box, shows fractional inches.

- 5 Repeat the procedure and set the drawing units back to **Decimal**.

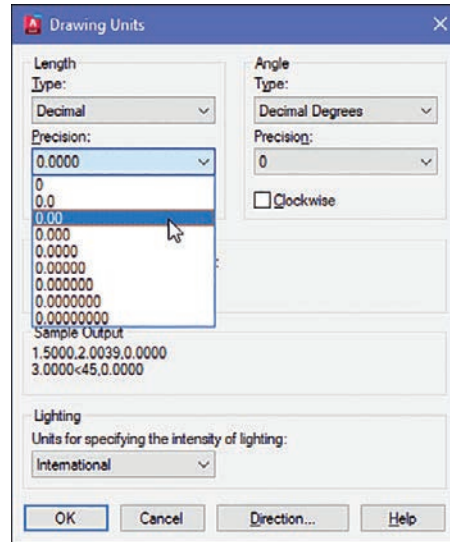
Specifying or Changing the Precision of the Units System

Unit values can be expressed with decimal places from zero to eight or in inches from 0 to 1/256 inch.

- 1 Access the **Drawing Units** dialog box as explained previously.
- 2 In the **Length** area, click the arrow to the right of the current precision value display box below the word **Precision**.

A drop-down list of the possible decimal precision values cascades from the box (Figure 1-20).

Figure 1-20



- 3 Select **0.00**.

The value 0.00 appears in the **Precision** box.

- 4 Click **OK**.

The original drawing screen appears.

Specifying or Changing the Angle Units Value

You can specify angles in one of five different units: **Decimal Degrees**, **Degrees/Minutes/Seconds**, **Gradians**, **Radians**, or **Surveyor** units. **Decimal Degrees** is the default value.

Change the angle units in the **Angle** area by selecting the desired units in the drop-down menu under **Type**. The precision of the angle units is changed in the same way as for linear units.

1-8 Drawing Limits

You can use **drawing limits** to set the boundaries of a drawing. The drawing boundaries are usually set to match the size of a sheet of drawing paper. This means that when the drawing is plotted and a hard copy is made, it will fit on the drawing paper.

Figure 1-21 shows a list of standard flat-size drawing sheets for engineering applications, Figure 1-22 shows standard metric sizes, and Figure 1-23 shows standard architectural sizes.

Figure 1-21

| Standard Drawing Sheet Sizes—Inches | |
|-------------------------------------|-----------------|
| A | 8.5×11 |
| B | 11×17 |
| C | 17×22 |
| D | 22×34 |
| E | 34×44 |

Figure 1-22

| Standard Drawing Sheet Sizes—Millimeters | |
|--|-------------------|
| A4 | 210×297 |
| A3 | 297×420 |
| A2 | 420×594 |
| A1 | 594×841 |
| A0 | 841×1189 |

Figure 1-23

| Standard Drawing Sheet Sizes—Architectural USA | |
|---|----------------|
| A | 9×12 |
| B | 12×18 |
| C | 18×24 |
| D | 24×36 |
| E | 36×48 |

A standard $8.5" \times 11"$ letter-size sheet of paper as used by most printers is referred to as an *A-size* sheet of drawing paper.

NOTE

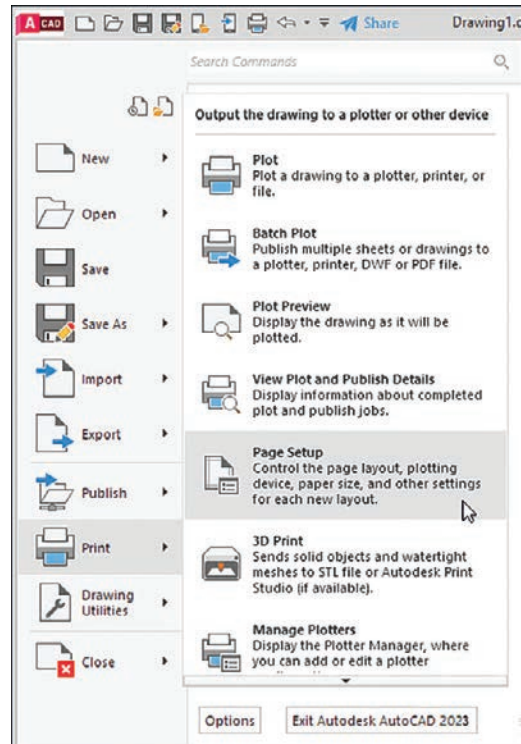
A sheet of paper can be sized to match standard sheet sizes by the capabilities of the printer or plotter. Many printers and plotters have built-in scaling features, and some list standard sheet sizes that can be applied to a drawing.

Aligning the Drawing Limits with a Standard A3 (Metric) Paper Size

- 1 Click the **Application Menu** button in the upper-left corner of the screen.
- 2 Click **Print**, then click **Page Setup** (Figure 1-24).

The **Page Setup Manager** dialog box appears.

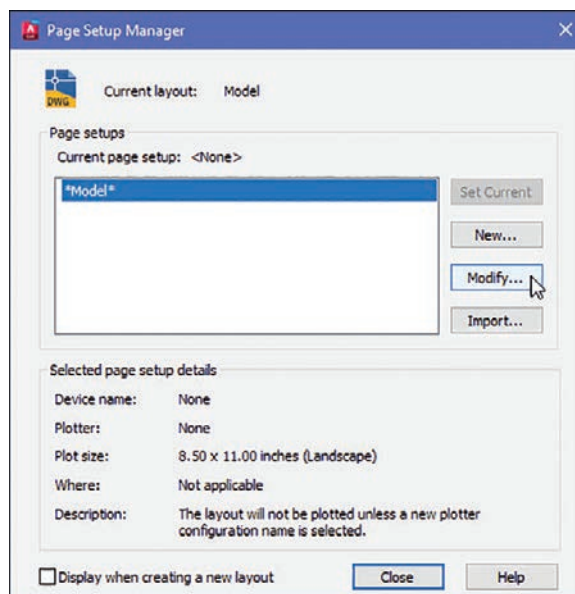
Figure 1-24



- 3 Click **Modify...** (Figure 1-25).

The **Page Setup - Model** dialog box appears.

Figure 1-25



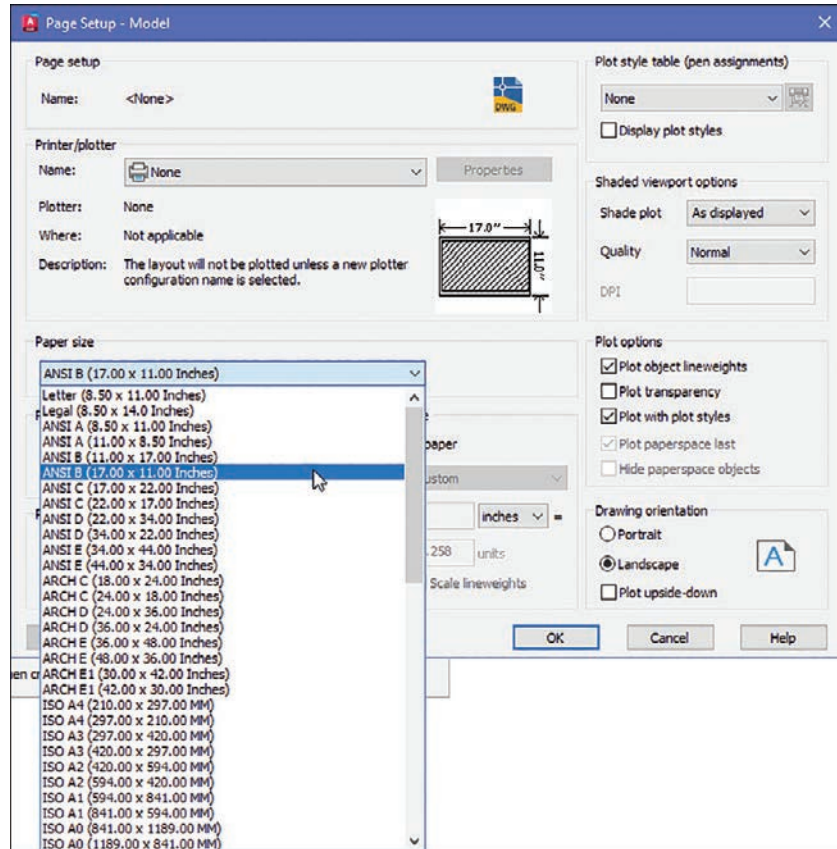
- 4 Click the arrow to the right of the **Paper size** box.

A drop-down list of available paper sizes appears (Figure 1-26).

- 5 Select the **ISO A3 (420.00 x 297.00)** size.

The dimensions in the preview box in the **Printer/plotter** area of the **Page Setup-Model** dialog box change to the selected values.

Figure 1-26



- 6 Click **OK**.

The drawing screen is now sized to the 420.00 × 297.00 ISO A3 dimensions.

NOTE

The sheet size may also be set with the **Limits** command. Type **Limits** at a command prompt and define the drawing limits by specifying the lower-left corner of the drawing as **0.00,0.00** (which is the default setting) and the upper-right corner as needed. If the new limits exceed the current screen limits, type **zoom** at a command prompt, then type **a** for **Zoom All**. The new drawing limits are matched to the screen size. The default sheet size for the **acad** template is 8.5 × 11 (ANSI A), and for the **acadiso** template the default is 210 × 297 (ISO A4).

1-9 Grid and Snap

The **Grid** command is used to place a grid background on the drawing screen. This background grid is helpful for establishing visual reference points for sizing and for locating points and lines. The grid may appear as lines or dots. You can specify the type of grid in the **Drafting Settings** dialog box.

NOTE

A graph paper-style grid background is used in most figures in this book.

The **Snap** command limits the movement of the cursor to predefined points on the screen. For example, if the **Snap** command values are set to match the **Grid** values, the cursor will snap from intersection to intersection (or dot to dot) on the grid.

The default **Grid** and **Snap** setting for the **acad** template is **.50** inch, and the default setting for **Grid** and **Snap** for the **acadiso** template is **10** millimeters.

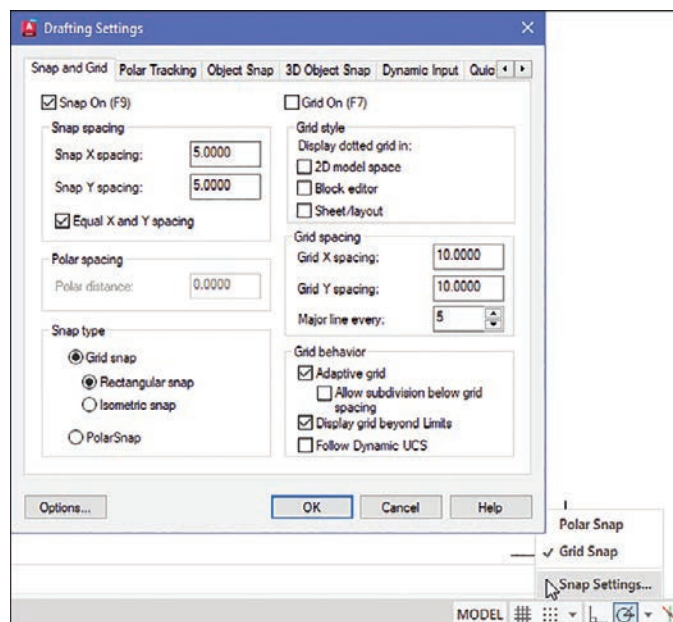
NOTE

The **Grid** function can be toggled off and on with the **F7** key, and the **Snap** function can be toggled with the **F9** key.

Setting the Grid and Snap Values

- 1 Start a new drawing and select the **acadiso** template (where values are in millimeters).
- 2 Right-click the **Snap** tool located at the bottom of the screen and click **Snap Settings** (Figure 1-27).

Figure 1-27



The **Drafting Settings** dialog box appears (Figure 1-27). If it is not already selected, click the **Snap and Grid** tab.

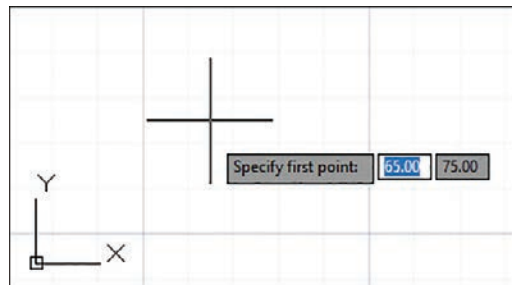
- 3** Click the **Grid On** and **Snap On** checkboxes. A check mark appears in each of the boxes.
- 4** Place the cursor in the **Snap X spacing** text box to the right of the given value under the **Snap On** heading.
A vertical flashing cursor appears.
- 5** Backspace out the existing value and type in **5**.
- 6** Click the **Snap Y spacing** box.

The Y spacing automatically equals the X spacing value. You can create rectangular grid spacing by specifying different X and Y spacing values.

- 7** Select the **Grid X spacing** text box under the **Grid spacing** heading.
- 8** Backspace out the existing value and type in **10** if needed.
- 9** Click the **Grid Y spacing** box to make the X and Y values equal.
- 10** Click **OK**.

Figure 1-28 shows the result. Since the **Snap** values have been set to exactly half of the **Grid** values, the cursor can be located either directly on grid intersections or halfway between them.

Figure 1-28



You can turn the grid on and off either by double-clicking the **Grid** icon at the bottom of the screen or by pressing the **F7** key on the keyboard. Turn Snap on and off by double-clicking the **Snap** icon at the bottom of the screen or by pressing the **F9** key on the keyboard. You can also turn **Grid** and **Snap** off and on by clicking their respective buttons on the status bar located at the bottom of the screen.

1-10 Drawing Problem

Set up a drawing that uses millimeter dimensions and the following parameters:

Sheet size = **297,420 (A3)**

Grid = **10** spacing

Snap = **5** spacing

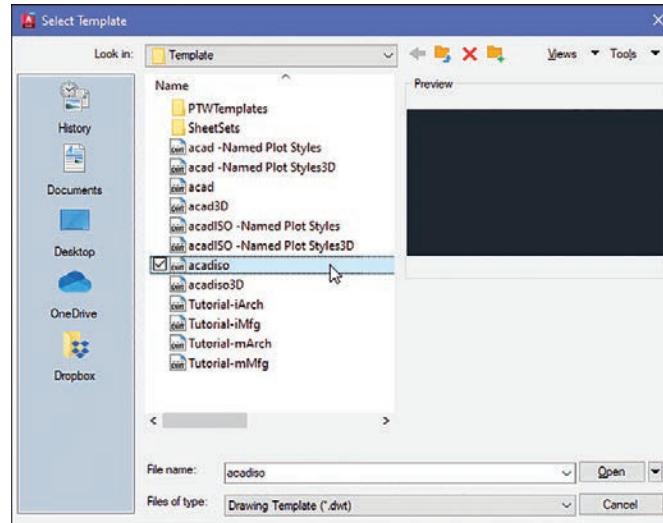
Whole-number precision

Specifying the Drawing Units

- 1 Click the **Application Menu** button in the upper-left corner of the drawing screen and then select **New**, then **Drawing**.

The **Select Template** dialog box appears (Figure 1-29).

Figure 1-29



- 2 Select the **acadiso** template and click **Open**.

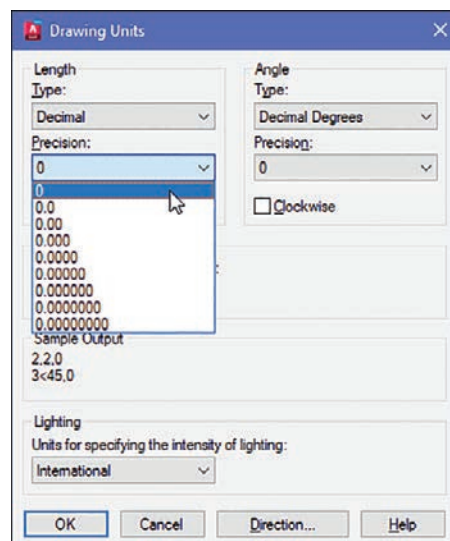
Defining the Units Precision

- 1 Click the **Application Menu** button in the upper-left corner of the drawing screen and then select **Drawing Utilities**, then **Units**.

The **Drawing Units** dialog box appears (Figure 1-30). In this example, only whole numbers will be used, so the **0** option is selected.

- 2 Select the **0** precision and click **OK**.

Figure 1-30



Setting the Sheet Size

The default values for an **acadiso** template are 210 × 297, but this drawing problem calls for 297 × 420, an A3 sheet size.

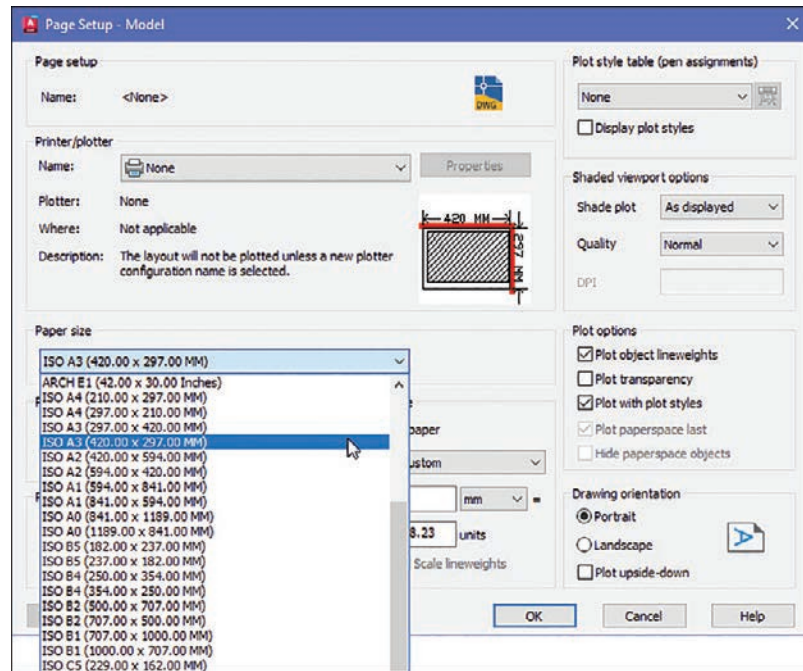
- 1 Open the **Application** menu and select **Print**, then **Page Setup**.

The **Page Setup Manager** dialog box appears.

- 2 Click **Modify**.

The **Page Setup - Model** dialog box appears (Figure 1-31).

Figure 1-31



- 3 Scroll down the available **Paper size** options and select the **ISO A3 (420.00 x 297.00)** option.

- 4 Click **OK**.

Setting Grid and Snap Values

- 1 Right-click the **Grid** button at the bottom of the screen.

- 2 Click **Grid Settings**.

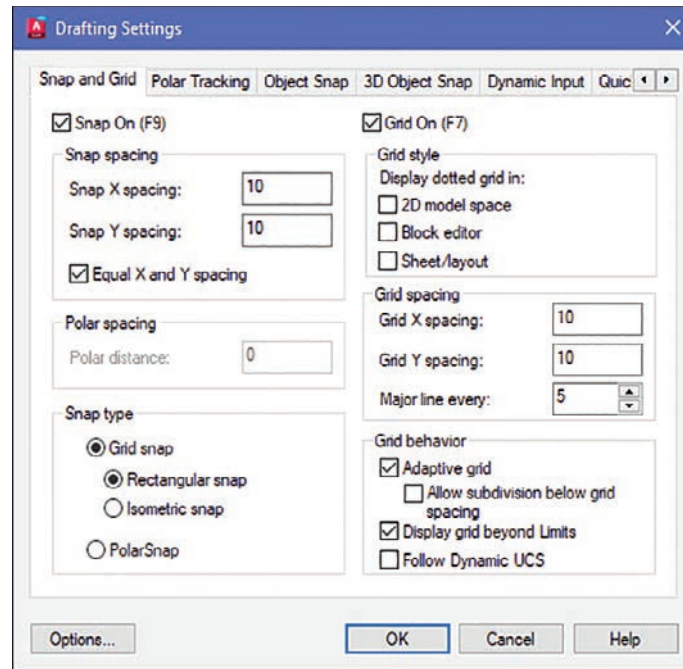
The **Drafting Settings** dialog box appears (Figure 1-32).

- 3 Select **Grid On** and **Snap On** and set the snap spacing to **5** and the grid spacing to **10**.

- 4 Enter **Zoom** at the command prompt, type **A**, and press **Enter**.

The screen is now ready for starting a drawing using millimeter values.

Figure 1-32



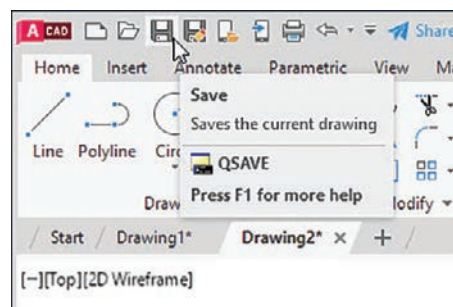
1-11 Save and Save As

Use the **Save** command to save your work. If you start a new drawing, the first time you click **Save**, the **Save As** command displays the **Save Drawing As** dialog box. You can select **Save As** at any time if you want to save your drawing using a different name or in a different location, but most of the time you will use the **Save** command to simply save your work.

Using the Save and Save As Commands

- 1 Click the **Save** button on the **Quick Access Toolbar** at the top of the screen, above the Home panel (Figure 1-33).

Figure 1-33

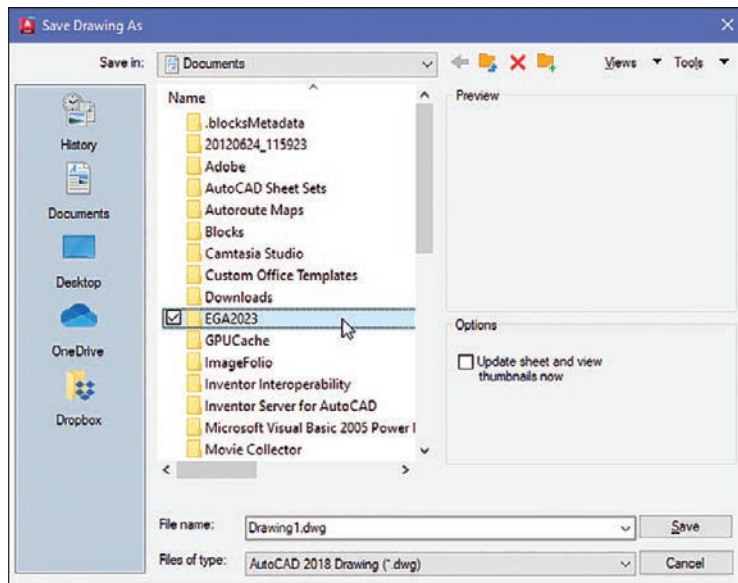


NOTE

The small group of tool buttons to the right of the **Application Menu** button and above the ribbon is called the **Quick Access Toolbar**. You will find frequently used commands here, including New, Open, Save, and Print. You can customize the Quick Access Toolbar to add your own frequently used commands.

Because this drawing has not yet been saved, the **Save Drawing As** dialog box appears (Figure 1-34). In this example, the file name **Drawing1.dwg** appears. This is the default name that was created automatically when the new drawing was opened.

Figure 1-34

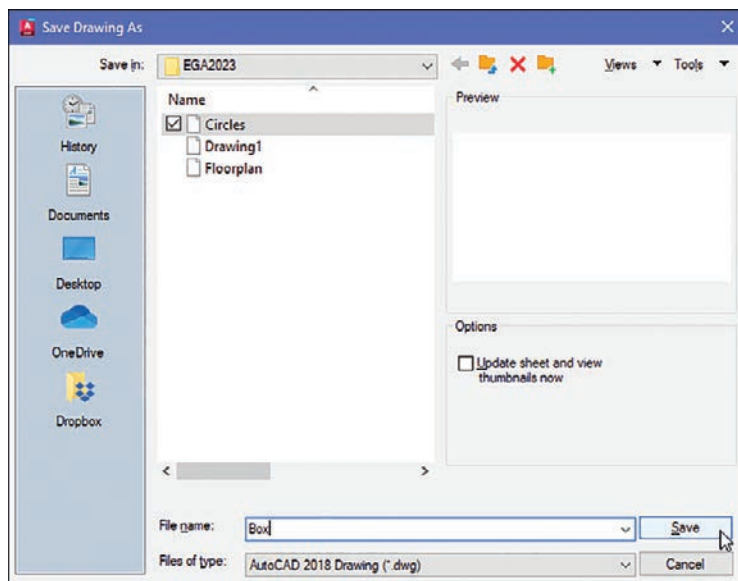


- 2 Create a new folder where you can save your work.

Figure 1-34 shows a folder named **EGA2023** created under **Documents**.

- 3 Save the drawing in the folder **EGA2023** located in the **Documents** folder and enter the name **Box** (Figure 1-35).

Figure 1-35



NOTE

The **Open** option can also be accessed by pressing **Ctrl+O**.

- 4 Click **Save**.

1-12 Open

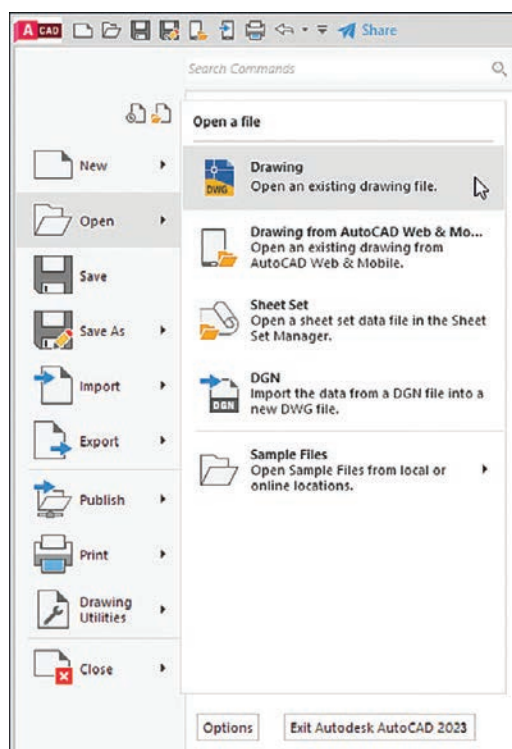
Use the **Open** command to call up an existing drawing so that you can continue working on it.

Using Open

Access existing drawings using **Open** and the **Select File** dialog box.

- 1 On the **Application** menu, click **Open** (Figure 1-36), then click **Drawing**.

Figure 1-36



The **Select File** dialog box appears (Figure 1-37).

- 2 Click the **Views** option at the top of the **Select File** dialog box and click the **Thumbnails** and **Preview** options.

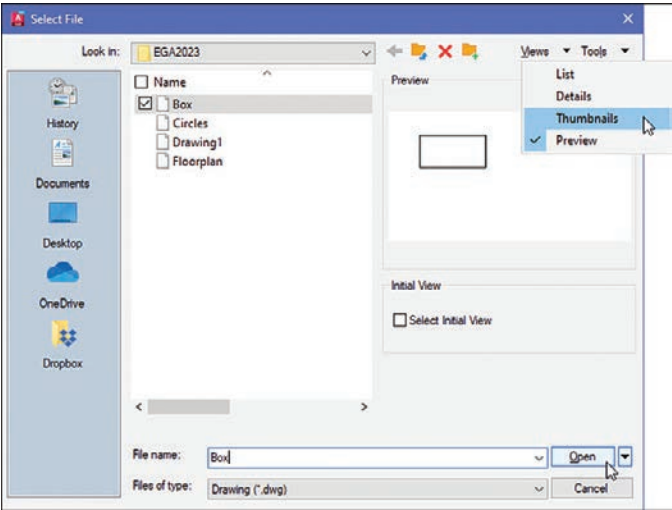
Thumbnails of the drawing files appear.

- 3 Click the desired file.

A preview appears.

- 4 Click **Open**.

Figure 1-37



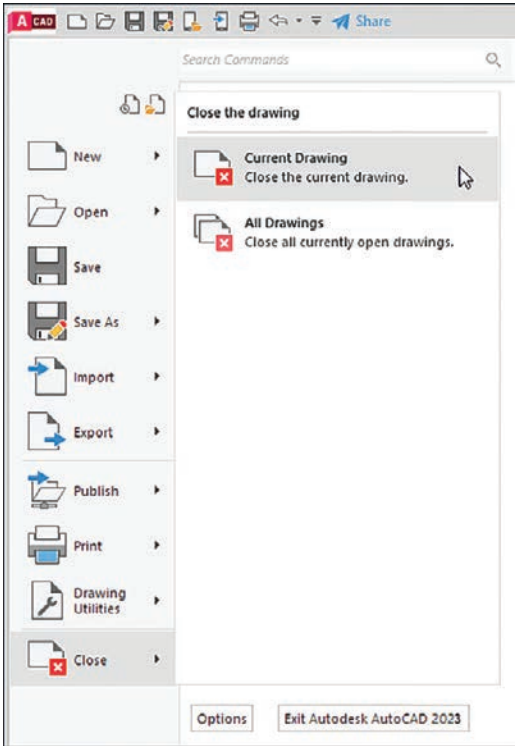
1-13 Close

The **Close** command allows you to close the current drawing.

- 1 On the **Application** menu, click **Close** and then click **Current Drawing** (Figure 1-38).

The system exits the AutoCAD program.

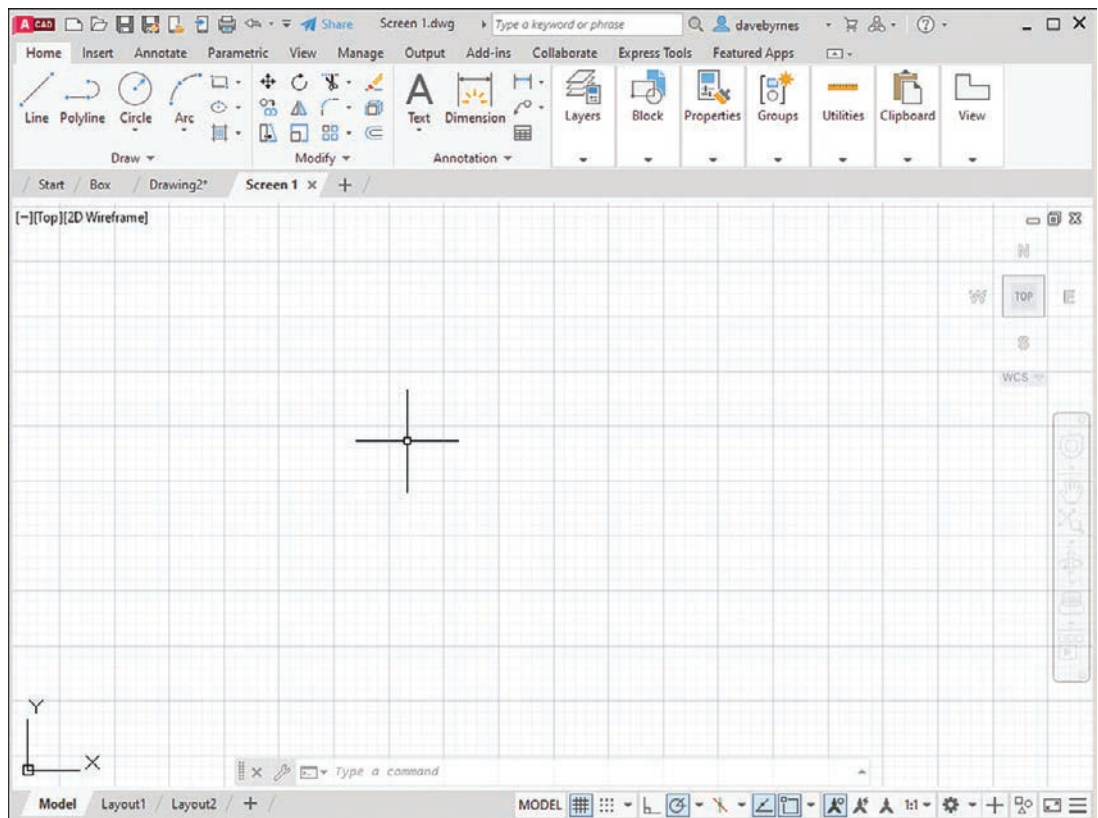
Figure 1-38



1-14 EXERCISE PROBLEMS

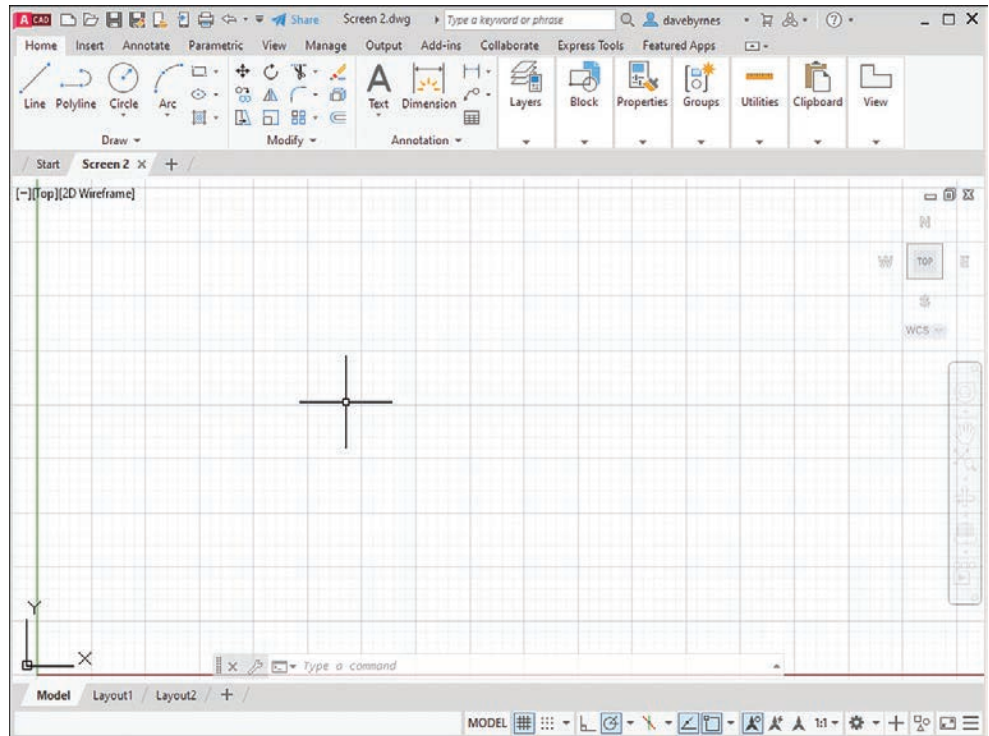
EX1-1

Create a drawing screen as shown in Figure EX1-1. Select the **acadiso** template, turn on the **Grid** and **Snap** functions, and set the grid spacing to **10** and snap spacing to **5**. Set the sheet size to **ISO A3 (297.00 x 420.00)**. Name the drawing **Screen 1**.



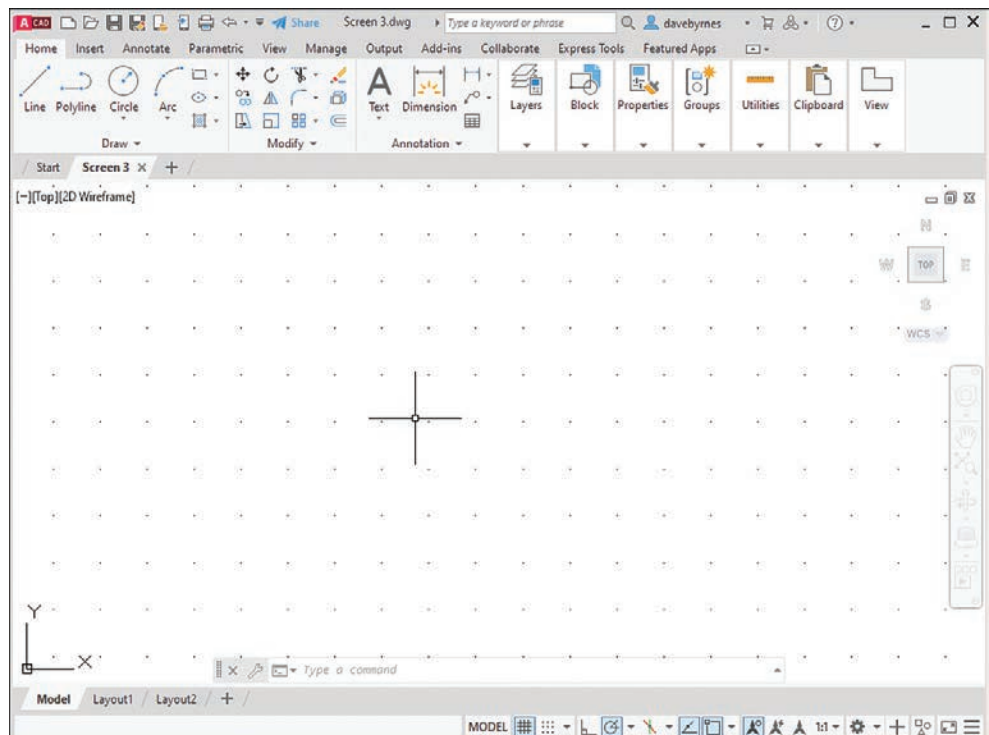
EX1-2

Create a drawing screen as shown in Figure EX1-2. Select the **acad** template, turn on the **Grid** and **Snap** functions, and set the grid spacing to **0.50** and snap spacing to **0.25**. Locate the origin in the lower-left corner of the drawing screen. Name the drawing **Screen 2**.



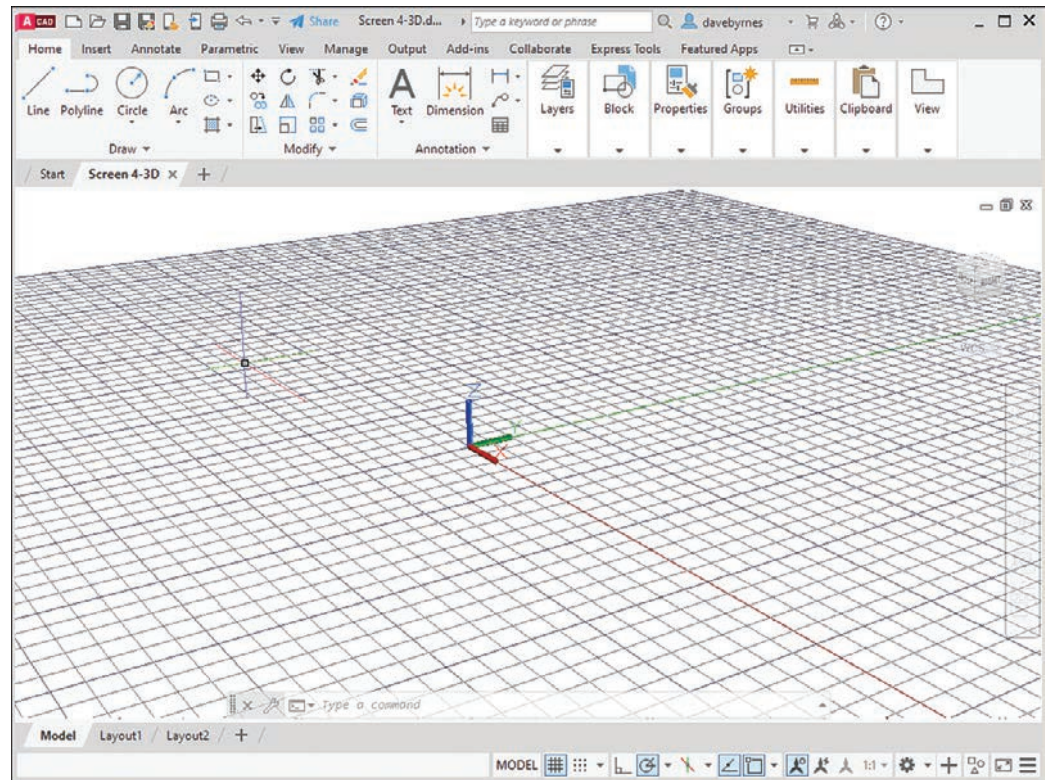
EX1-3

Create a drawing screen as shown in Figure EX1-3. Select the **acadiso** template, turn on the **Grid** and **Snap** functions, and set the grid spacing to **50** and snap spacing to **10**. Set the grid background to **dotted**. Name the drawing **Screen 3**.



EX1-4

Create a drawing screen as shown in Figure EX1-4. Select the **acadiso3D** template, turn on the **Grid** and **Snap** functions, and set the grid spacing to **20** and snap spacing to **5**. Name the drawing **Screen 4-3D**.



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2

chapter two

Fundamentals of 2D Construction

2-1 Introduction

This chapter demonstrates how to work with AutoCAD commands. You can execute AutoCAD commands by using command tools and a series of prompts. The prompts appear in the command line window and ask the user for a selection or numeric input so that a command sequence can be completed. This chapter uses the **Line** command to show the various input and prompt sequences that are typical with AutoCAD commands.

Most of the commands contained in the **Draw** and **Modify** panels are demonstrated in this chapter. Enough commands are presented for you to be able to create simple 2D shapes. Chapter 3 presents more advanced applications for these commands and introduces some new commands as well.

2-2 Line—Random Points

Use the **Line** command to draw a straight line between two defined points.

There are four ways to define the length and location of a line:

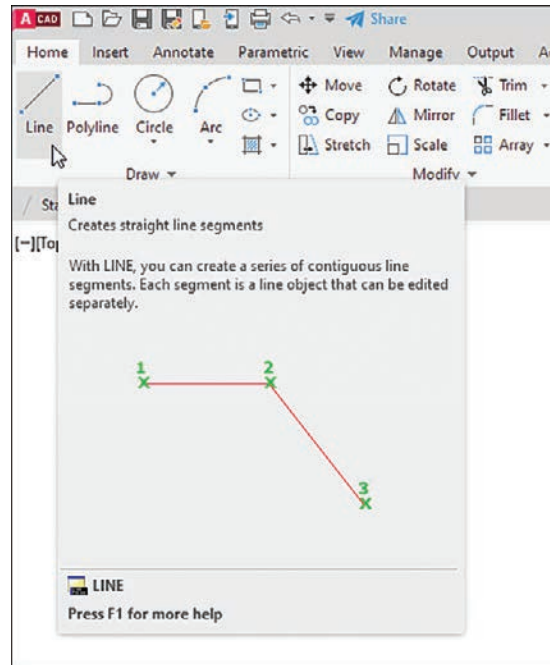
- Randomly select points
- Set a specific value for the **Snap** function and select points with the **Snap** spacing values
- Enter the coordinate values for the starting point and endpoint
- Use relative inputs and specify the starting point, the length, and the direction of the line

Randomly Selecting Points

- 1 Start a drawing using the **acadiso** template and select the **Line** tool on the **Draw** panel on the **Home** tab (Figure 2-1).

You can also access the **Line** command by typing the word **line** or the letter **L** in response to a command prompt.

Figure 2-1



The following command sequence appears in the command line window:

Command: `_line` Specify first point:

- 2 Place the cursor anywhere on the drawing screen and press the left mouse button.

AutoCAD responds:

Specify next point or [Undo]:

As you move the cursor from point to point, dynamic input appears on the screen (Figure 2-2). Dynamic input includes an absolute angle box and a length change box. Dynamic input is discussed in Section 2-5.

NOTE

If you do not see dynamic input on the screen, press the **F12** function key to turn it on.

- 3 Pick another random point on the screen.

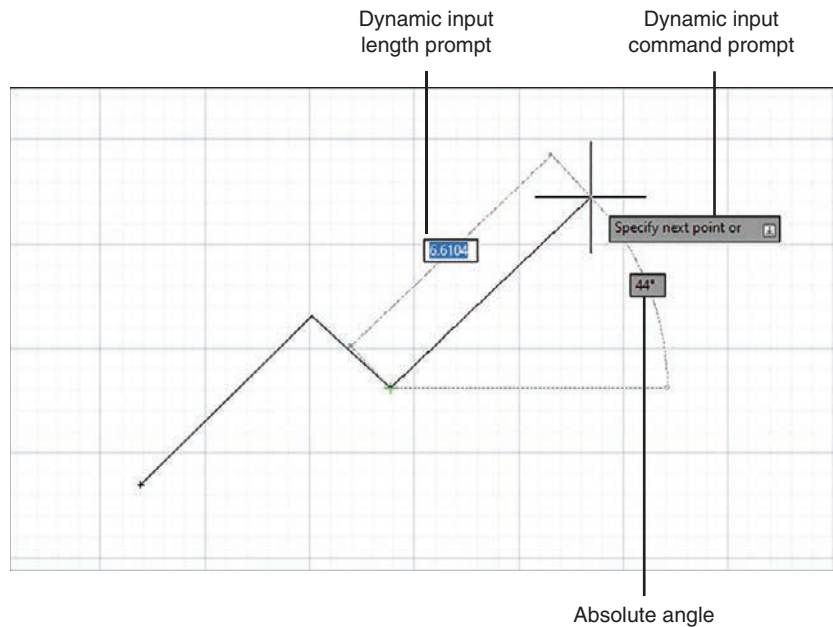
AutoCAD responds:

Specify next point or [Undo]:

AutoCAD keeps asking for another point until you press either the **Enter** key or the right mouse button. When you press the right mouse button (that is, right-click), the right-click menu appears.

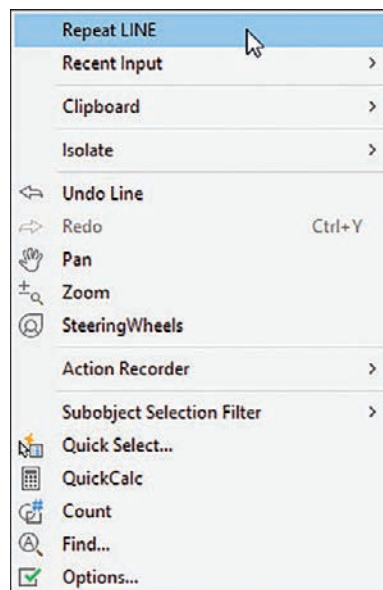
- 4 Press the right mouse button and click **Enter** to end the **Line** command sequence.

Figure 2-2



You can restart the **Line** command by pressing the right mouse button immediately after selecting **Enter**. A right-click menu appears (Figure 2-3). Select **Repeat LINE** to restart the **Line** command.

Figure 2-3



Exiting a Command Sequence

If, as you work, you need to exit a command sequence, press the **Esc** key, and you return to a command prompt. The **Esc** key allows you to exit almost any AutoCAD command.

Creating a Closed Area

- 1 Select the **Line** tool from the **Draw** panel.

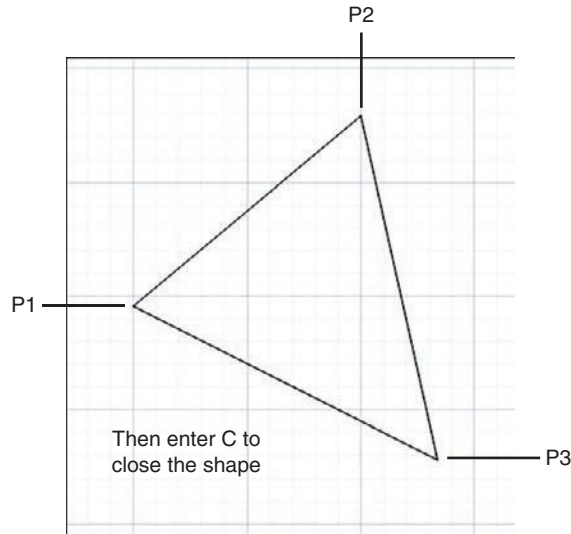
AutoCAD responds:

Command: `_line` Specify first point:

- 2 Click a random point (such as P1 in Figure 2-4).

Specify next point or [Undo]:

Figure 2-4



- 3 Click a second random point.

Specify next point or [Undo]:

- 4 Click a third random point.

Specify next point or [Close Undo]:

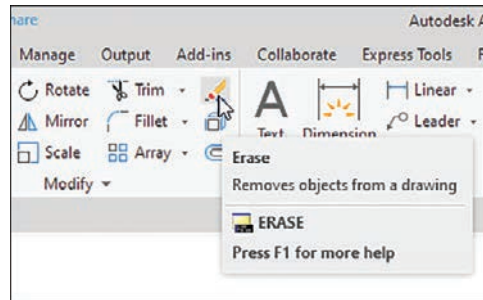
- 5 Type **c** and then press **Enter** to activate the **Close** option.

AutoCAD draws a line from the third point to the starting point, creating a closed area. You can also create an enclosed area by using the **Object Snap Endpoint** option. As you move the cursor toward a line, a box appears at the line's endpoint. Clicking within the endpoint box attaches the line to that endpoint. Right-click the **Object Snap** icon at the bottom of the screen for a list of other **Object Snap** options.

2-3 Erase

You can erase any line by using the **Erase** command. There are two ways to erase lines: Select individual lines or use a selection box to choose a group of lines. The **Erase** tool is located on the **Modify** panel on the **Home** tab (Figure 2-5).

Figure 2-5



Erasing Individual Lines

- 1 Click the **Erase** tool on the **Modify** panel.

The following prompt appears in the command line window:

Command: `_erase`

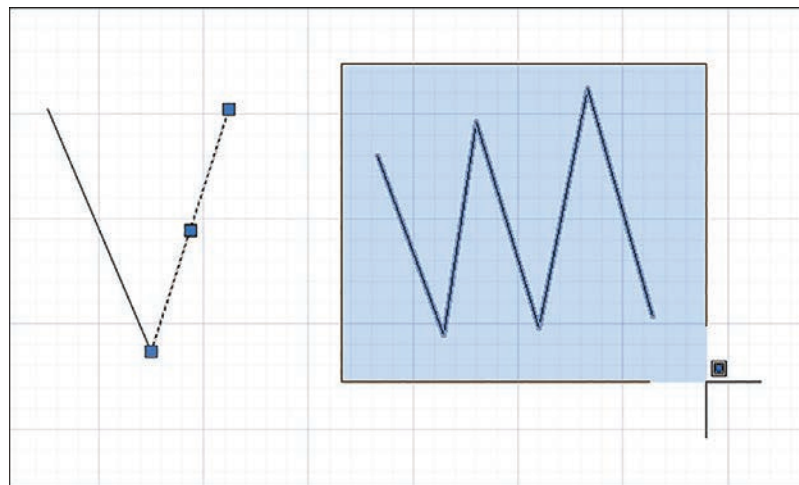
Select objects:

The normal crosshairs are replaced by a rectangular selection box called the *pickbox*. Any time you see the pickbox, AutoCAD expects you to select an entity—in this example, a line.

If you change your mind and do not want to erase anything, select the next command that you want to use, and the **Erase** command is terminated.

- 2 Select the two open lines by placing the rectangular cursor on each line, one at a time, and pressing the left mouse button. The left side of Figure 2-6 shows a single object selected.

Figure 2-6



The lines are highlighted and display grips at their midpoints and endpoints. The highlight indicates that the line has been selected. The selection is confirmed by a change in the prompt:

Select objects: 1 found

Select objects:

- 3 Right-click or press the **Enter** key to complete the **Erase** sequence.

The two lines disappear from the screen.

Erasing a Group of Lines

- 1 Click the **Erase** tool.

Command: `_erase`

Select objects:

- 2 Place the rectangular select cursor above and to the left of the lines to be erased and click the left mouse button (Figure 2-6). Do not hold the mouse button down.
- 3 Move the mouse, and a shaded window drags from the selected first point.

The right side of Figure 2-6 shows a window selection.

- 4 When all the lines to be erased are completely within the window, press the left mouse button.

All the lines completely within the window are selected and change to a faint hue. The number of lines selected is indicated in the command line box.

- 5 Right-click or press the **Enter** key to complete the command sequence.

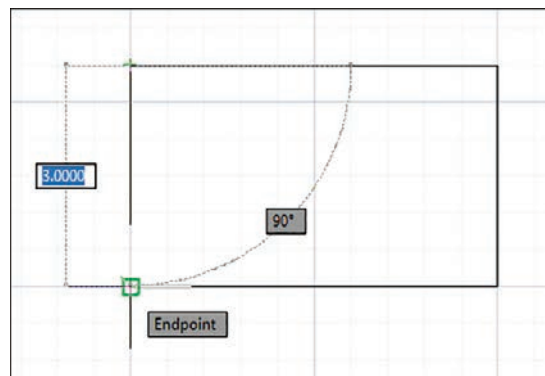
The lines disappear from the screen. You can use the **Undo** tool to return all the lines, if needed.

If you create the **Erase** window from right to left, any line even partially within the **Erase** window is erased. If the left mouse button is held down and dragged across the object, an irregularly shaped window is formed.

2-4 Line—Snap Points

You can draw lines by using calibrated snap spacing. This technique is similar to drawing points randomly, but because the cursor is limited to specific points, the lengths of lines can be determined accurately (Figure 2-7).

Figure 2-7



Drawing Problem

- 1 Turn on both the **Grid** and **Snap** functions and set both grid spacing and snap spacing to **.5**. Zoom the grid in or out to fit the screen.

NOTE

See Section 1-9 for instructions on how to use the **Grid** and **Snap** functions.

- 2 Select **Line** from the **Draw** panel.

Command: `_line` Specify first point:

- 3 Click a grid point.

Specify next point or [Undo]:

- 4 Move the cursor horizontally to the right **10** grid spaces and press the left mouse button.

Because the snap spacing has been set to **.5**, 10 spaces equal 5 units. Watch the dynamic input readings as you move the cursor. The X value should increase by 5, and the Y value should stay the same.

Specify next point or [Undo]:

- 5 Move the cursor vertically **6** spaces and press the left mouse button.

Specify next point or [Close Undo]:

- 6 Move the cursor horizontally to the left **10** spaces and press the left mouse button.

Specify next point or [Close Undo]:

- 7 Type **c** and press **Enter** or place the endpoint of the left vertical line on the starting point of the first line. Right-click and select **Enter**.

You can save or erase the drawing as desired.

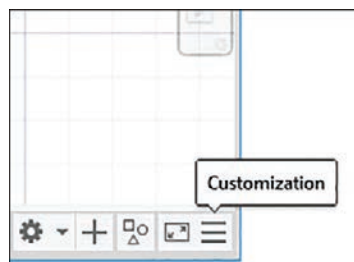
2-5 Line—Dynamic Input

Using the **Dynamic Input** mode allows you to add both angular and distance values to a drawing through the use of screen prompts.

By default, AutoCAD 2023 does not display the **Dynamic Input** button on the status bar. Use the following steps to turn it on:

- 1 Click the **Customization** button at the right end of the status bar (Figure 2-8).

Figure 2-8



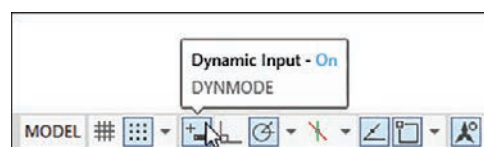
- 2 Select **Dynamic Input** from the pop-up menu to place a check mark next to it and turn it on (Figure 2-9).

The **Dynamic Input** button now appears on the status bar (Figure 2-10).

Figure 2-9



Figure 2-10



- 3** Click anywhere in the drawing area to close the Customization menu.

Dynamic Input mode is enabled when the status bar button is highlighted (Figure 2-10).

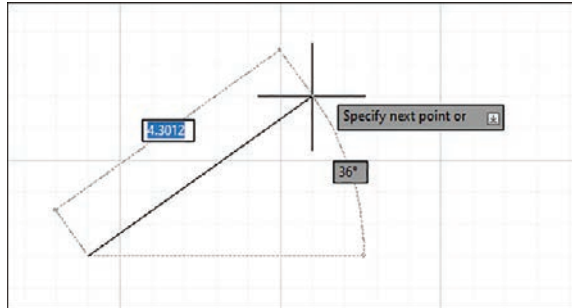
Creating Lines Using Dynamic Input

This example was created using inch values.

- 1** Click the **Line** command tool on the **Draw** panel and select a starting point.

Figure 2-11 shows the dynamic input for the next point.

Figure 2-11



- 2** Move the cursor to the approximate location of the second line point.

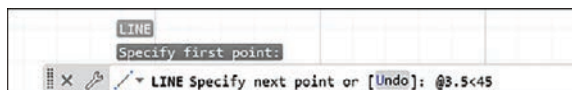
The distance and angular values for this new point are displayed in dynamic input labels. Do not click the mouse.

Angular values assume that 0° is a horizontal line to the right of the first line point.

- 3a** Move the cursor to the command line window and type in the desired distance and angle value.

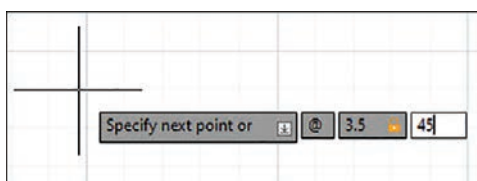
Use the format `@Distance<angle`. In this example, the input `@3.5<45` was entered (Figure 2-12).

Figure 2-12



- 3b** You can also enter the distance and angle for the line by typing the values starting with @. The values appear in the dynamic input boxes on the screen. When the < symbol is entered, a lock icon appears next to the distance value (Figure 2-13). Press **Enter** when the values are entered.

Figure 2-13



- 4 Press **Enter**.

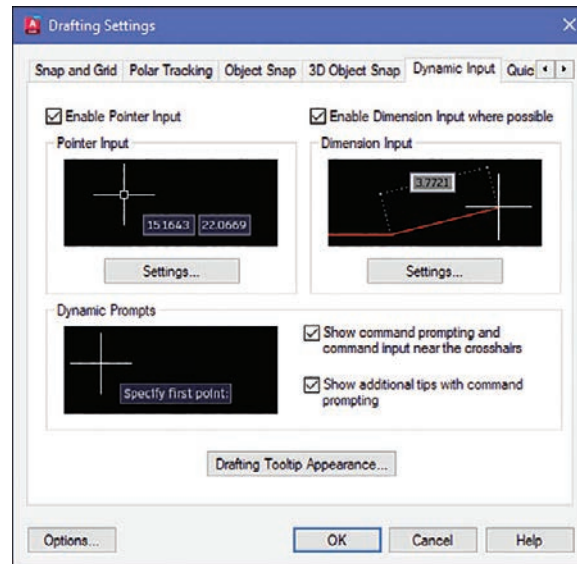
The angular value is applied to the line, and the dynamic input label prepares for the next line point input.

- 5 Complete the desired shape and press **Enter**.

Accessing Dynamic Input Settings

Use the **Drafting Settings** dialog box to access dynamic input settings (Figure 2-14).

Figure 2-14



- 1 Right-click the **Dynamic Input** tool at the bottom of the screen and select **Dynamic Input Settings**.

The **Drafting Settings** dialog box appears.

The **Dynamic Input** tab of this dialog box controls the pointer input, dimension input, and dynamic prompts aspects of dynamic input. The default values are used throughout the text.

2-6 Construction Line

The **Construction Line (Xline)** command is used to draw lines of infinite length. Construction lines are very helpful during the initial layout of a drawing. They can be trimmed as needed during the creation of a drawing.

NOTE

If you trim a construction line so it now has an endpoint, the remaining part of the construction line becomes a **ray**. You can create rays directly using the **Ray** command, which is on the expanded **Draw** panel.

- 1 Select the **Construction Line** tool from the expanded **Draw** panel (Figure 2-15).

AutoCAD prompts:

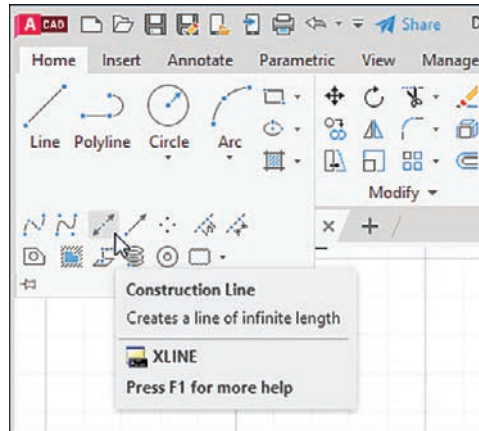
Command: `_xline` Specify a point or [Hor Ver Ang Bisect Offset]:

Specify a point is the default option.

NOTE

The **Construction Line** tool is located on a flyout from the **Draw** panel. Click the arrow next to the word **Draw** to access the flyout.

Figure 2-15



2 Select or define a starting point.

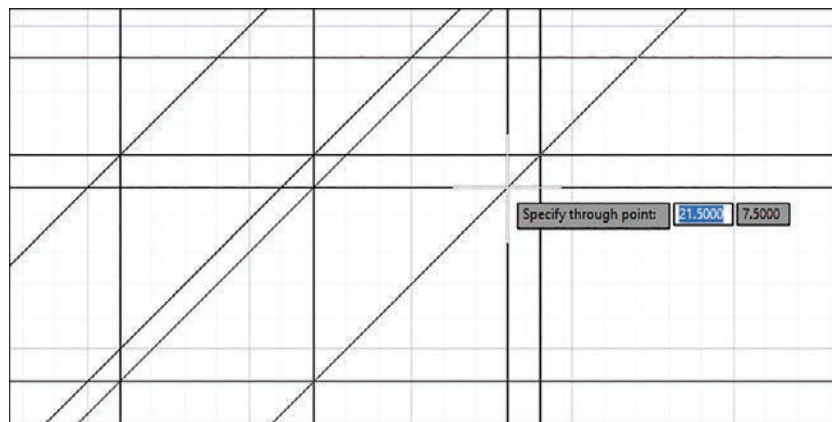
You can position the direction of the line by moving the cursor or by using dynamic input, as defined in Section 2-5, to define the line's direction.

3 Select or use dynamic input to define a through point.

Specify through point:

A line pivots about the designated throughpoint and extends an infinite length in both directions through the cursor. You can position the angle of the line by moving the cursor (Figure 2-16).

Figure 2-16



- 4** Select or define a through point.

An infinitely long line is drawn through the two designated points.

Specify through point:

- 5** Press **Enter** to end the **Construction Line** command sequence.

You can reactivate the sequence and draw another construction line by pressing the **Enter** key a second time.

Using Other Construction Line Commands: Hor, Ver, and Ang

The lines shown in Figure 2-16 were created by using the **Hor** (horizontal), **Ver** (vertical), and **Ang** (angular) options. A grid was created, and **Snap** was turned on so that the lines could be drawn through known points. Figure 2-16 was created as follows:

- 1** Set up the drawing screen with grid and snap spacings of **.5**.

- 2** Select **Construction Line** in the expanded **Draw** panel.

AutoCAD prompts:

Command: `_ xline Specify a point or [Hor Ver Ang Bisect Offset]:`

- 3** Type **H** and press **Enter**.

A horizontal line appears through the cursor.

Specify through point:

- 4** Pick or define a point on the drawing screen.

A horizontal line appears through the point. As you move the mouse, another horizontal line appears through the cursor.

Specify through point:

- 5** Pick or define a second point.

Specify through point:

- 6** Pick or define a third point.

Specify through point:

- 7** Right-click, right-click again, and select **Repeat XLINE**.

Command: `_ xline Specify a point or [Hor Ver Ang Bisect Offset]:`

- 8** Type **V** and press **Enter**.

Specify through point:

- 9** Draw vertical lines, right-click twice, and select the **Repeat XLINE** option.

Command: `_ xline Specify a point or [Hor Ver Ang Bisect Offset]:`

- 10** Type **A** and press **Enter**.

Enter angle of xline (0) or [Reference]:

- 11** Type **45** and press **Enter**.

Specify through point:

An infinite line at 45° appears through the crosshairs.

- 12** Draw **45°** lines.

Specify through point:

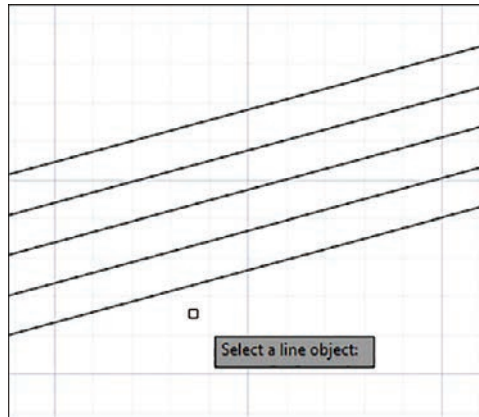
- 13** Press **Enter**.

Your drawing should look approximately like Figure 2-16.

Using Another Construction Line Option: Offset

The **Offset** option allows you to draw a construction line parallel to an existing construction line, regardless of the line's orientation, at a predefined distance (Figure 2-17).

Figure 2-17



- 1** Select **Construction Line** in the expanded **Draw** panel.

Command: `_xline` Specify a point or [Hor Ver Ang Bisect Offset]:

- 2** Draw a line approximately **15°** to the horizontal.

AutoCAD prompts:

Specify through point:

- 3** Double-click the **Enter** key to restart the **Construction Line** command sequence.

Command: `_xline` Specify a point or [Hor Ver Ang Bisect Offset]:

- 4** Type **O** and press **Enter**.

The **O** appears in the dynamic input box.

Offset distance or through <Through>:

- 5** Type **.5** and press **Enter**.

Select a line object:

- 6** Select the line.

Pick side to offset:

- 7** Select a point above the line.

Select a line object:

- 8** Select the line just created by the **Offset** option.

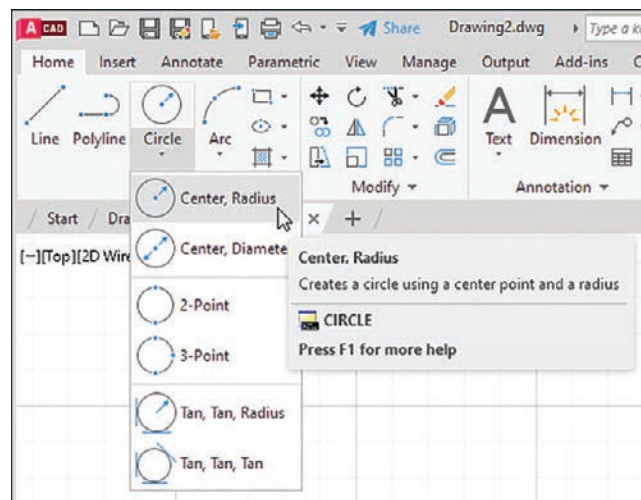
Pick side to offset:

- 9 Again select a point above the selected line.
Select a line object:
- 10 Draw several more lines below the original line.
Select a line object:
- 11 Right-click and select **Enter** to end the **Construction Line Offset** sequence and return to the command prompt.

2-7 Circle

The **Circle** tool button is on the **Draw** panel. A circle can be defined by a center point and either a radius or a diameter, by two or three points on the diameter, or by the tangents to either two existing lines or arcs and a radius value or to three objects. Each of these command options has a separate tool button on the **Draw** panel in the **Circle** tool button's drop-down (Figure 2-18).

Figure 2-18



Drawing a Circle—Radius

- 1 Click **Circle** on the **Draw** panel.
Command: `_circle` Specify a center point for circle or [3P 2P Ttr (tan tan radius)]:
- 2 Pick a point or use dynamic input to define a center point.
Specify radius of circle or [Diameter]:
- 3 Type **3.50** and press **Enter**.
The radius value appears in the dynamic input box.

Drawing a Circle—Diameter

- 1 Click the down arrow on the **Circle** tool button and then click **Center, Diameter** in the drop-down menu.

Command: `_circle` Specify a center point for circle or [3P 2P Ttr (tan tan radius)]:

- 2 Pick a point or define a center point.

Specify radius of circle or [Diameter]:

- 3 Type **8.00** and press **Enter**.

Drawing a Circle—Two Points

- 1 Click the down arrow on the **Circle** tool button and then click **2-Point** in the drop-down menu.

Command: `_circle` Specify a center point for circle or [3P 2P Ttr (tan tan radius)]: `_2p` Specify first end point of circle's diameter:

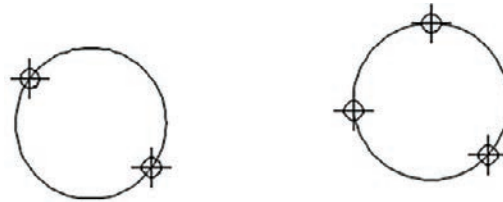
- 2 Select or define a first point.

Select second end point of circle's diameter:

- 3 Select or define a second point.

The circle is automatically drawn through the first and second points, as shown on the left side of Figure 2-19.

Figure 2-19



Drawing a Circle—Three Points

- 1 Click the down arrow on the **Circle** tool button and then click **3-Point** on the drop-down menu.

Command: `_circle` Specify a center point for circle or [3P 2P Ttr (tan tan radius)]: `_3p` Specify first point on circle:

- 2 Pick or define a first point.

Specify second point on circle:

- 3 Pick or define a second point.

Specify third point on circle:

- 4 Pick or define a third point.

The circle is automatically drawn through the three points, as shown on the right side of Figure 2-19.

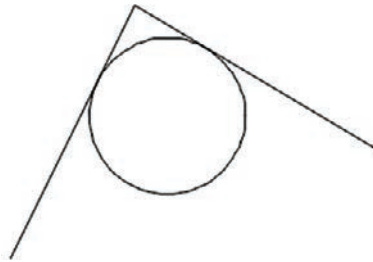
Drawing a Circle—Tangent Tangent Radius

The **tangent tangent radius** option allows you to draw a circle tangent to two entities and then specify a radius. Figure 2-20 shows a circle drawn tangent to two lines.

- 1 Click the down arrow on the **Circle** tool button and then click **Tan, Tan, Radius** on the drop-down menu.

Command: `_circle` Specify a center point for circle or [3P 2P Ttr (tan tan radius)]: `_ttr` Specify point on object for first tangent of circle:

Figure 2-20



- 2 Select an entity.

Specify point on object for second tangent of circle:

- 3 Select the other entity.

Specify radius of circle <value>:

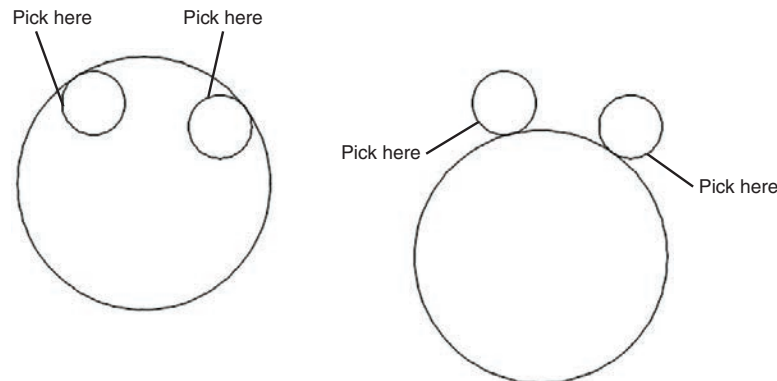
- 4 Type a radius value and press **Enter**.

Drawing a circle using the **Tan, Tan, Tan** option is similar to the steps for **Tan, Tan, Radius**. Simply select three objects—lines, arcs, or circles—rather than two objects and a radius.

Quadrant-Sensitive Applications

The **Circle** command's **TTR** option is quadrant sensitive—that is, the final location of the tangent circle depends on the location of the tangent spec points. In Figure 2-21, the **TTR** option was used to draw a large circle tangent to two smaller circles, and in both examples, the larger tangent circle was created by using the same radius. Note the difference in results depending on where on the smaller circles the tangent points were chosen.

Figure 2-21



2-8 Circle Centerline

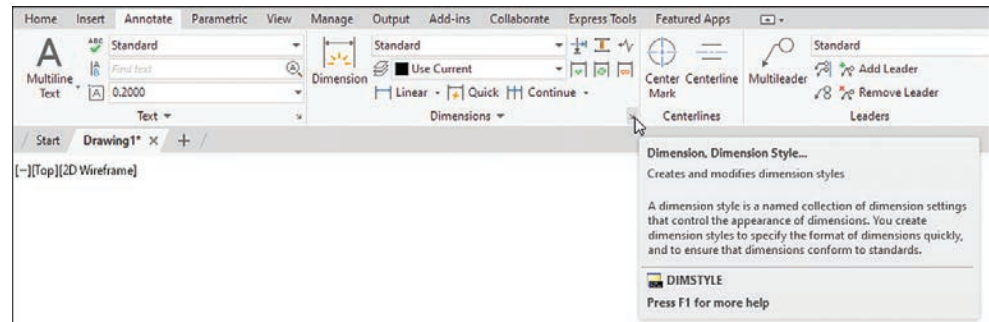
Standard drawing convention includes centerlines with all circles. Circles usually represent holes. The circle's center point locates the hole and, during manufacture, serves as the location point for a drill.

Creating Center Marks

- 1 Click the **Annotate** tab on the ribbon.

A new set of ribbon panels appears (Figure 2-22).

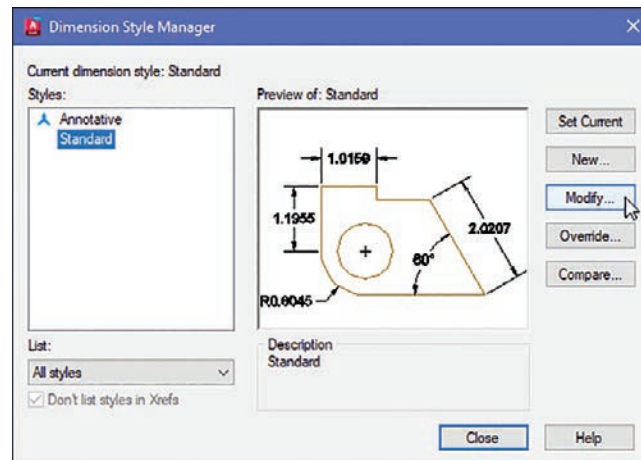
Figure 2-22



- 2 Click the arrow in the lower-right corner of the **Dimensions** panel.

The **Dimension Style Manager** dialog box appears (Figure 2-23).

Figure 2-23

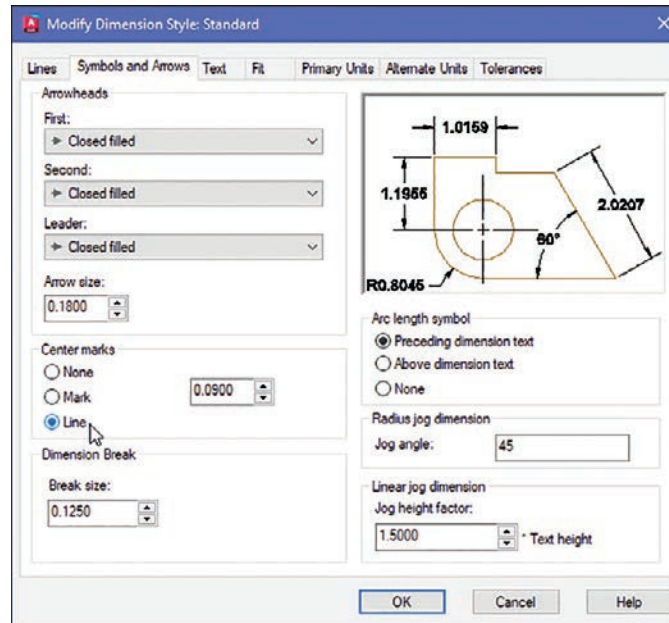


- 3 In the **Dimension Style Manager** dialog box, click **Modify**.

The **Modify Dimension Style: Standard** dialog box appears.

- 4 Select the **Symbols and Arrows** tab, then in the **Center marks** area, click the **Line** option (Figure 2-24).

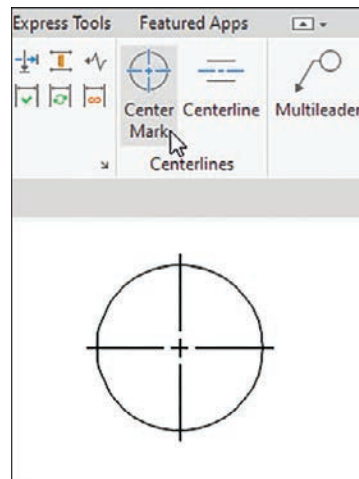
Figure 2-24



The preview screen displays centerlines.

- 5 Click **OK** and then **Close** to return to the drawing screen.
- 6 Click the **Center Mark** tool on the **Centerlines** panel, and then click the given circle, as shown in Figure 2-25.

Figure 2-25



Centerlines appear on the circle.

2-9 Polyline

A *polyline* is a linear object made from a series of individual, connected line or arc segments that act as a single entity. Polylines can be used to generate curves and splines and can also be used in three-dimensional applications to produce solid objects.

Drawing a Polyline

- 1 Select **Polyline** from the **Draw** panel.

AutoCAD prompts:

Command: `_pline`

Specify start point:

- 2 Pick or define a starting point.

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

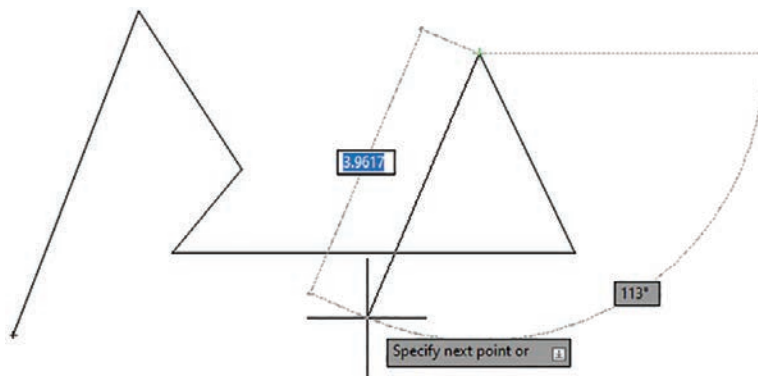
- 3 Pick or define a second point.

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

- 4 Pick or define several more points (Figure 2-26).

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

Figure 2-26



- 5 Right-click and then select **Enter**.

Verifying That a Polyline Is a Single Entity

Figure 2-26 shows a polyline.

- 1 Select **Erase** from the **Modify** panel or enter the word **erase**.

AutoCAD prompts:

Select objects:

- 2 Pick any one of the line segments in the polyline.

Select objects:

The entire polyline, not just the individual line segment, is selected because in AutoCAD, the polyline is a single entity.

- 3 Press **Enter**.

The entire polyline disappears.

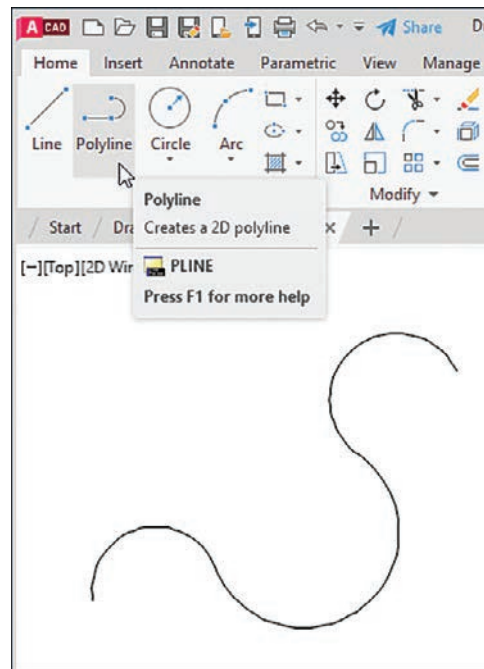
- 4 Click **Undo** on the **Quick Access Toolbar**.

The object reappears.

Drawing a Polyline Arc

Figure 2-27 shows a polyline drawn using the **Arc** option.

Figure 2-27



- 1 Select **Polyline** from the **Draw** panel.

AutoCAD prompts:

Specify start point:

- 2 Pick or define a starting point.

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

- 3 Type **A** and press **Enter**.

Specify endpoint of arc or [Angle CEnter Direction Halfwidth Line Radius Second pt Undo Width]:

- 4 Pick or define another point.

Specify endpoint of arc or [Angle CEnter Direction Halfwidth Line Radius Second pt Undo Width]:

- 5 Pick or define another point.

- 6 Right-click, and then select **Enter**.

Other Options with a Polyline Arc

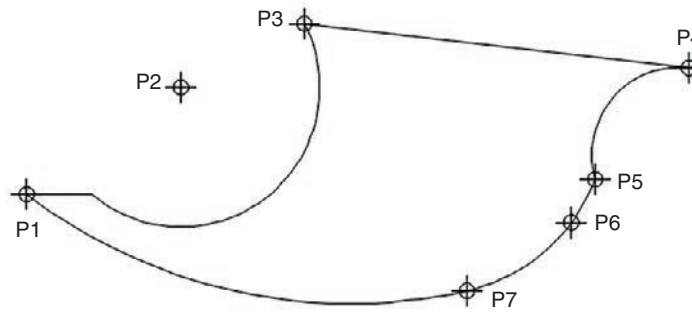
Figure 2-28 shows examples of polyline arc options.

- 1 Select **Polyline** from the **Draw** panel.

Command: `_pline`

Specify start point:

Figure 2-28



- 2** Pick or define a starting point (such as P1 in Figure 2-28).
Specify next point or [Arc Halfwidth Length Undo Width]:
- 3** Draw a short horizontal line segment.
Specify next point or [Arc Close Halfwidth Length Undo Width]:
- 4** Type **A** and press **Enter**.
Specify endpoint of arc or [Angle CEnter CClose Direction Halfwidth Line Radius Second pt Undo Width]:
- 5** Type **CE** and press **Enter**.
CE activates the **Center** option. You can now define an arc that will be part of the polyline by defining the arc's center point and its angle, chord length, or endpoint.
Specify center point of arc:
- 6** Pick or define a center point (such as P2 in Figure 2-28).
Specify endpoint of arc or [Angle Width]:
- 7** Pick or define an endpoint (such as P3 in Figure 2-28).
Specify endpoint of arc or [Angle CEnter Direction Halfwidth Line Radius Second pt Undo Width]:
- 8** Type **L** and press **Enter**.
The **Line** option is used to draw straight-line segments.
Specify next point or [Arc Close Halfwidth Length Undo Width]:
- 9** Pick or define an endpoint (such as P4 in Figure 2-28).
Specify next point or [Arc Close Halfwidth Length Undo Width]:
- 10** Type **A** and press **Enter**.
Specify endpoint of arc or [Angle CEnter Direction Halfwidth Line Radius Second pt Undo Width]:
- 11** Type **R** and press **Enter**.
Specify radius of arc:
- 12** Type **1.5** or another value and press **Enter**.
Specify endpoint of arc or [Angle]:
- 13** Pick or define an endpoint (such as P5 in Figure 2-28).
Specify endpoint of arc or [Angle CEnter CClose Direction Halfwidth Line Radius Second pt Undo Width]:

- 14** Type **S** and press **Enter**.

Specify second point on arc:

- 15** Pick or define a point (such as P6 in Figure 2-28).

Specify endpoint of arc:

- 16** Select or define an endpoint (such as P7 in Figure 2-28).

Specify endpoint of arc or [Angle CEnter CClose Direction Halfwidth Line Radius Second pt Undo Width]:

- 17** Type **CL** and press **Enter**.

The **CL** (close) option joins the last point drawn to the first point of the polyline by use of an arc.

Drawing Different Line Thicknesses

- 1** Select **Polyline** from the **Draw** panel.

Specify start point:

- 2** Select or define a starting point.

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

- 3** Type **W** and press **Enter**.

The **Width** option defines the width of a line. The **Halfwidth** option is used to define half of the width of a line.

Specify starting width <0.0000>:

- 4** Type **1.00** and press **Enter**.

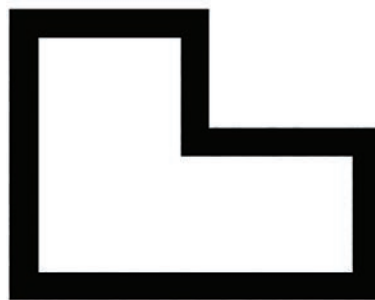
Specify ending width <1.0000>:

- 5** Press **Enter**.

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

- 6** Draw several line segments (Figure 2-29).

Figure 2-29



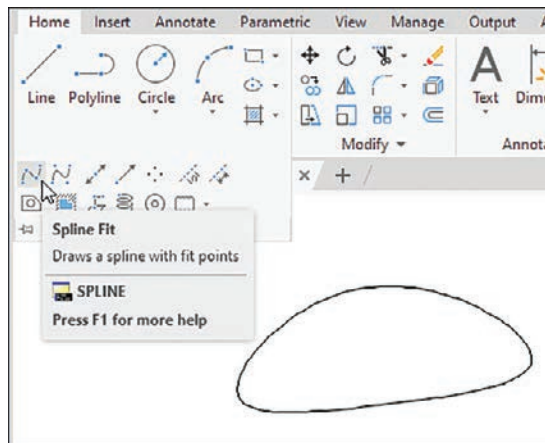
2-10 Spline

A *spline* is a non-radial curved line created through (*fit spline*) or near (*control vertex spline*) a series of points. If the curve forms an enclosed area, it is called a *closed spline*. Curved lines that do not enclose an area are called *open splines*. See Figure 2-30.

NOTE

There are two ways of drawing splines: using either the **Fit** or **Control Vertex** tool buttons on the expanded **Draw** panel. Fit splines pass through the points you pick onscreen. Control vertex splines are created using inflection points that you pick.

Figure 2-30



- 1** Select **Spline Fit** from the **Draw** panel. AutoCAD prompts:
 Command: `_spline`
 Specify first point or [Method Knots Object]:
- 2** Pick or define a starting point.
 Specify next point:
- 3** Pick or define a second point.
 Specify next point or [Close Fit tolerance] <start tangent>:
- 4** Select or define two more points.
 Specify next point or [Arc Close Halfwidth Length Undo Width]:
- 5** Type **C** and press **Enter** (Figure 2-30).

2-11 Ellipse

The **Ellipse** command has two options. These options allow you to define an ellipse by using the lengths of its major and minor axes or by picking a center point and specifying axis lengths. You can also draw elliptical arcs using the third option on the **Ellipse** tool button.

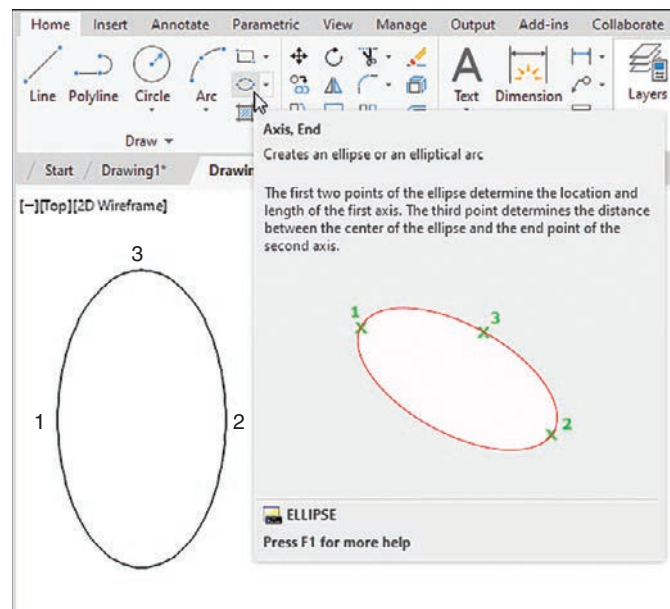
Drawing an Ellipse—Axis Endpoint

- 1** On the **Draw** panel, click the down arrow next to the **Ellipse** tool and then click **Axis, End**.
 AutoCAD prompts:
 Specify axis endpoint of ellipse or [Arc Center]:

- 2** Pick or define a starting point for one of the axes.
Specify other endpoint of axis:
- 3** Pick or define an endpoint that defines the length of the axis.
Specify distance to other axis or [Rotation]:
- 4** Pick or define a point that defines half of the length of the other axis.

The distance specified in step 4 is the radius of the axis. In the example shown in the extended tooltip (Figure 2-31), points 1 and 2 define the major axes, and point 3 defines the minor axis. The sample ellipse drawn in Figure 2-31 shows an object in which points 1 and 2 were used to define the minor axes, and point 3 was used to define the major axis.

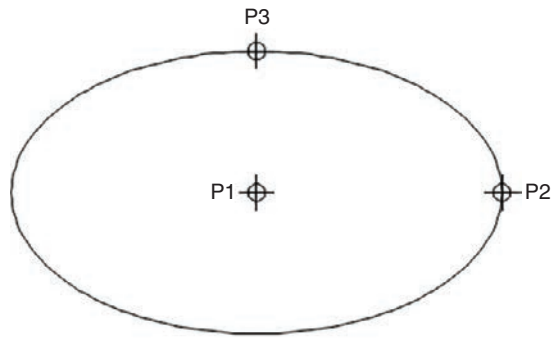
Figure 2-31



Drawing an Ellipse—Center

- 1** On the **Draw** panel, click the down arrow next to the **Ellipse** tool and then click **Center**.
AutoCAD prompts:
Specify axis endpoint of ellipse or [Arc Center]:
Specify center of ellipse:
- 2** Pick or define the center point of the ellipse (such as P1 in Figure 2-32).
Specify axis endpoint:
- 3** Pick or define one of the endpoints of one of the axes (such as P2 in Figure 2-32).
The distance between the center point and the endpoint is equal to the radius of the axis.
Specify distance to other axis or [Rotation]:

Figure 2-32



- 4 Pick or define a point that defines half of the length of the other axis (such as P3 in Figure 2-32).

Drawing an Elliptical Arc

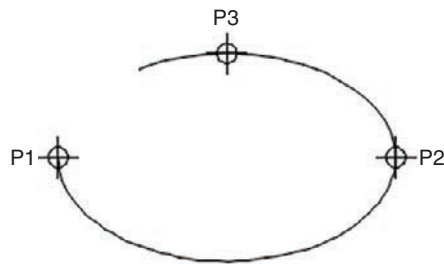
- 1 On the **Draw** panel, click the down arrow next to the **Ellipse** tool button and then click **Elliptical Arc**.

Specify axis endpoint of elliptical arc or [Center]:

- 2 Select or define a point (such as P1 on Figure 2-33).

Specify other endpoint of axis:

Figure 2-33



- 3 Select or define a point (such as P2 in Figure 2-33).

The distance between points 1 and 2 defines the length of the major axis.

Specify distance to other axis or [Rotation]:

- 4 Select or define a point that defines the minor axis (such as P3 in Figure 2-33).

Specify start angle or [Parameter]:

- 5 Type **0** and press **Enter**.

Specify end angle or [Parameter Included angle]:

- 6 Type **315** and press **Enter**.

2-12 Rectangle

Use the **Rectang** command to draw rectangular polylines. If you select one of the lines that constitute the rectangle, the entire rectangle is highlighted because it's a single object. The **Explode** command, whose tool button is located on the **Modify** panel, can reduce any block to its individual elements. If a rectangle is exploded, it is changed from a single polyline object to four individual straight lines.

Drawing a Rectangle

- 1 Select the **Rectangle** tool from the **Draw** panel.

AutoCAD prompts:

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

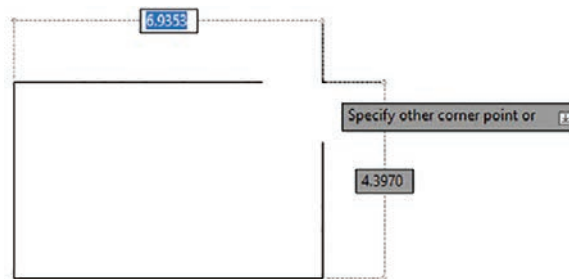
- 2 Pick or define a starting point.

Specify other corner point:

- 3 Pick or define a point.

The two points define the length and width of the rectangle. (Figure 2-34).

Figure 2-34



Once you have picked the first corner, you can also choose an option to enter the dimensions, the area, or a non-orthogonal rotation.

Exploding a Rectangle

- 1 Select **Explode** from the **Modify** panel (Figure 2-35).

Select object:

- 2 Select the rectangle.

Select object:

- 3 Press **Enter**.

There is no visible change in the rectangle, but it is now composed of four individual straight lines. As a test, use the **Erase** command to remove one of the lines.

Figure 2-35

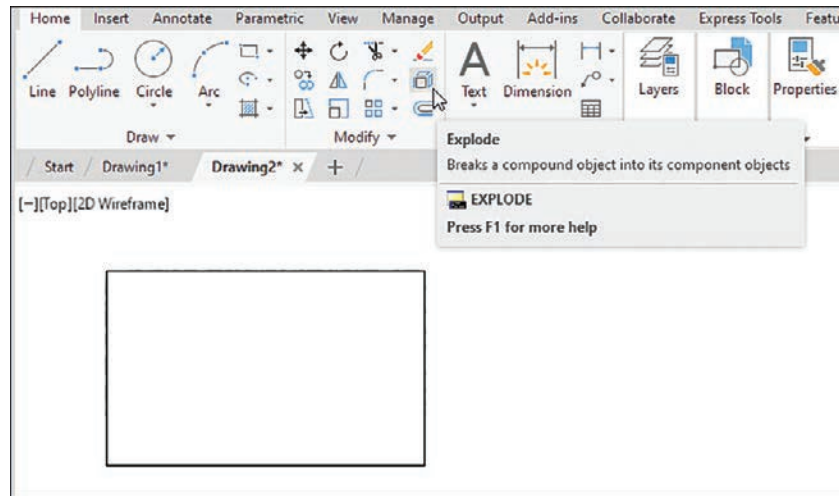


Figure 2-35

2-13 Polygon

A *polygon* is a closed polyline object bounded by straight lines. The **Polygon** command draws only regular polygons, in which all sides and angles are equal. A regular polygon with four equal sides is a square. This example shows how to draw a six-sided polygon, or hexagon.

Drawing a Polygon—Center Point

The **Polygon** and **Rectangle** commands share a tool button in the **Draw** panel. If the tool button shows a rectangle, click the down arrow beside the button and select **Polygon**.

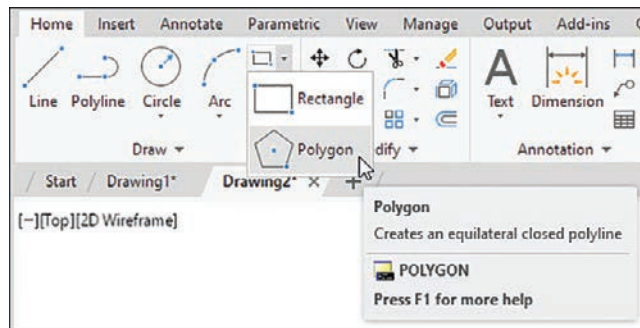
- 1 Select **Polygon** from the **Draw** panel (Figure 2-36).

AutoCAD prompts:

Command: `_polygon`

Enter number of sides <4>:

Figure 2-36



- 2 Type **6** and press **Enter**.

Specify center of polygon or [Edge]:

- 3 Select or define a center point.

Enter an option [Inscribed in circle Circumscribed about circle]:

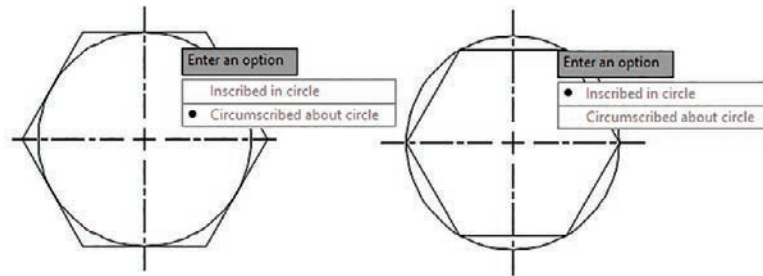
- 4 Type **C** and press **Enter**.

Specify radius of circle:

- 5 Type a value and press **Enter**.

The **Circumscribe** option was selected here because the diameter of the designated circle will equal the distance across the flats of the hexagon. The distance across the flats is often used to specify hexagon head bolt sizes and the wrench sizes used to fit them. The **Inscribe** option creates a polygon where the radius specifies the distance between the corners (see Figure 2-37).

Figure 2-37



Drawing a Polygon—Edge Distance

- 1 Select the **Polygon** tool from the **Draw** panel.

AutoCAD prompts:

Command: `_polygon`

Enter number of sides <4>:

- 2 Type **6** and press **Enter**.

Specify center of polygon or [Edge]:

- 3 Type **E** and press **Enter**.

Specify first endpoint of edge:

- 4 Pick or define a point.

Specify second endpoint of edge:

- 5 Pick or define a point.

2-14 Point

The **Point** command creates point objects on a drawing. You can set the appearance and size of point objects in the **Point Style** dialog box. By default, a point object is dimensionless and always appears as a single pixel, regardless of zoom magnification. If you select the **Point** tool from the **Draw** menu, you can place multiple point objects until you press **Esc** to exit the command.

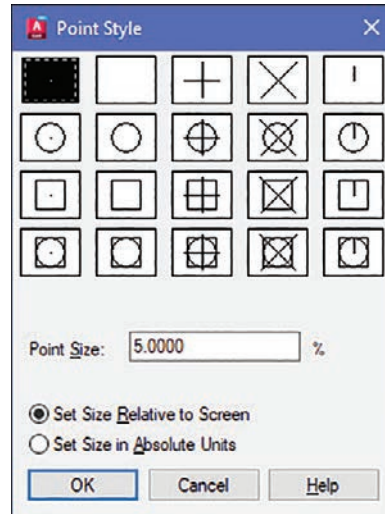
Changing the Shape of a Point

Start with the cursor at the command prompt. Do not click the **Point** tool.

- 1 Type **Ptype** on the command line and press **Enter**.

The **Point Style** dialog box appears, displaying 20 different visual appearances for point objects (Figure 2-38).

Figure 2-38



- 2 Click the icon in the middle of the top row and click **OK**.
- 3 On the expanded **Draw** panel, click **Multiple Points** and add several points to the screen.
- 4 Press the **Esc** key to exit the **Point** tool.

Changing the Size of a Point

- 1 Type **Ptype** at a command prompt and press **Enter**.
- 2 In the **Point Size** edit box, type **2.0** and press **Enter**.

If you change the point type or point size value, all existing and new point objects take on the new values.

2-15 Text

You can add drawing text with the **Multiline Text** tool found on the **Annotate** panel of the **Home** tab.

Creating Multiline Text

Use the **Multiline Text** command to enter drawing text. First, define the area in which the text is to be entered. Enter text in the **Multiline Text Editor**, just as you would with a word processing program. You can move text objects to different locations on the drawing screen by using the **Move** command.

- 1 Select the **Multiline Text** tool button from the **Annotate** panel.

AutoCAD prompts:

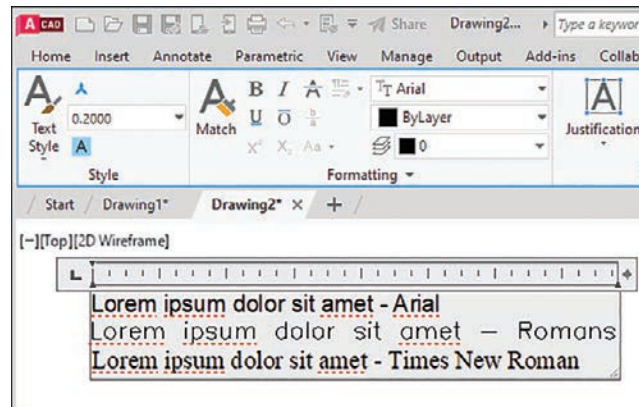
Specify first corner:

- 2 Pick or define a point.

The area that you are about to define is for the entire text entry, but it can be modified. Text can be created with different fonts (Figure 2-39).

Specify opposite corner or [Height Justify Line spacing Rotation Style Width]:

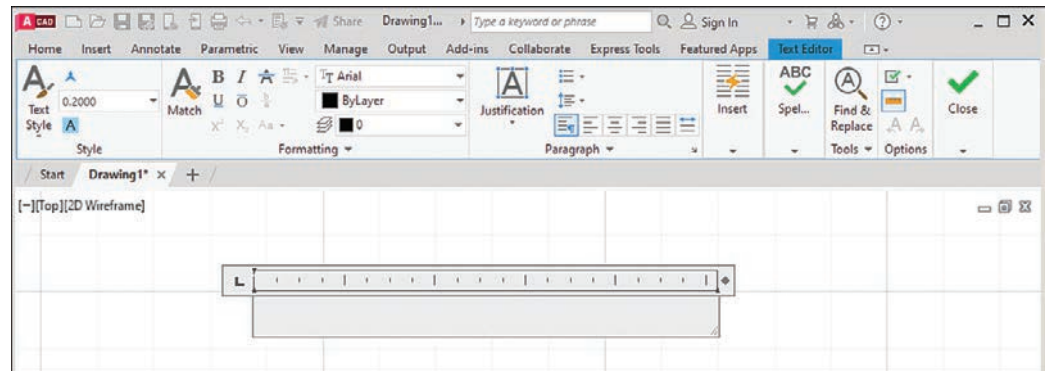
Figure 2-39



- 3 Define the other corner of the area.

The **Multiline Text Editor** window appears in the drawing area, and the **Text Editor** tab opens at the right end of the ribbon (Figure 2-40).

Figure 2-40



- 4 Type in your text and then click the screen outside the text box.

The text will be located on the drawing screen within the specified area.

The Multiline Text Editor

You can use the **Multiline Text Editor** to change the text height, color, font style, and justification.

Accessing the Multiline Text Editor

The **Multiline Text** tool is available in the **Text** panel of the **Annotate** tab.

- 1 Click the **Annotate** tab.
- 2 Click the **Multiline Text** tool.
- 3 Define an area for text.

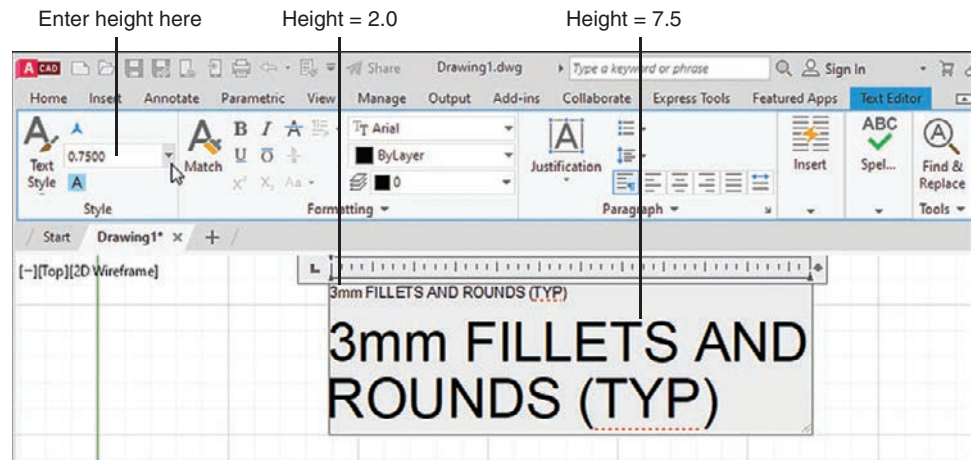
The **Text Editor** tab now appears at the end of the ribbon (Figure 2-40).

The **Text Editor** tab and its panels are not normally visible. It appears only in concert with the **Multiline Text** command.

Changing Text Height

- 1 Click the **Multiline Text** tool and type the desired text.
- 2 Highlight the text in the text editor.
- 3 In the **Style** panel of the **Text Editor** tab, click and highlight the text height box and type in a new text height (Figure 2-41).

Figure 2-41



- 4 Press **Enter**.

Changing the Text Font

- 1 Click the **Multiline Text** tool and type the desired text.
- 2 Highlight the text.
- 3 Access the drop-down list of fonts in the **Formatting** panel of the **Text Editor** tab by clicking the arrow on the right side of the drop-down and then scroll down the list and select the **Times New Roman** font.

The font of the selected text changes to Times New Roman (Figure 2-42).

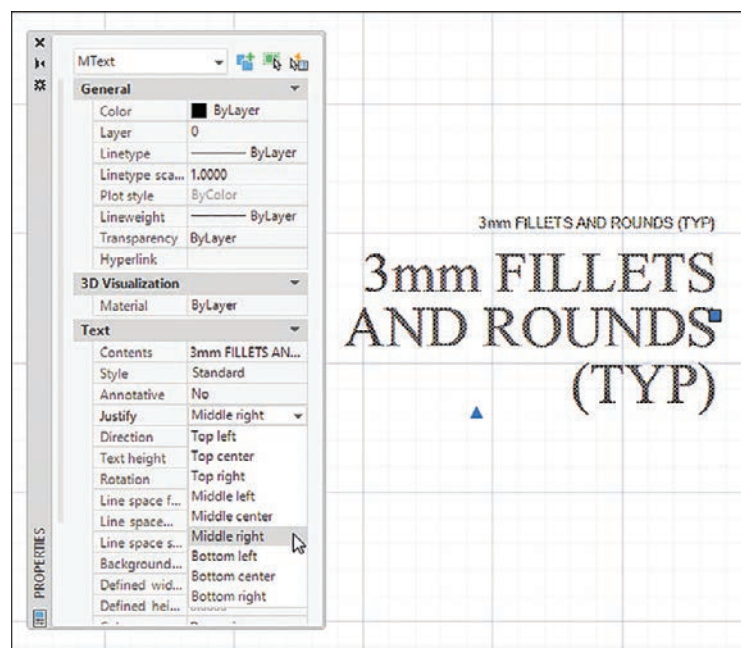
The screenshot shows the Microsoft Word interface. The 'Font' menu is open, displaying a list of fonts including Times New Roman, Arial, Calibri, and others. The text box in the foreground contains the text '3mm FILLETS AND ROUNDS (TYP)' in a large, bold, serif font.

Justifying Text

Justifying Text to the Right

- 1 Use a window to select the lines of text shown in Figure 2-42 (that is, select them so that they appear in a selection window).
- 2 With the text selected, right-click and select the **Properties** (not **Quick Properties**) option.

Figure 2-43

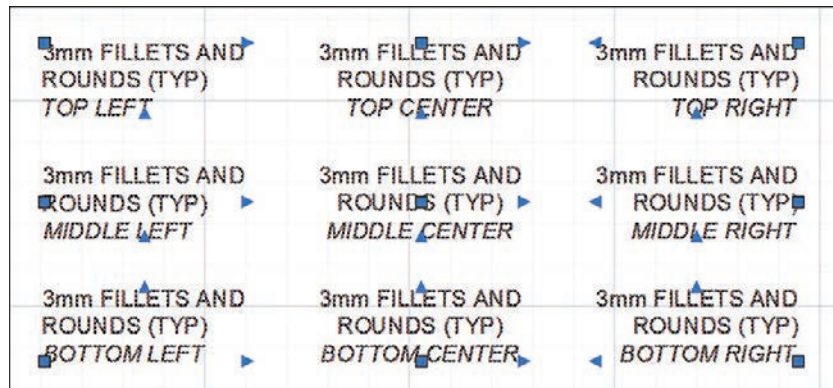


- 3 In the **Text** section of the **Properties** palette, click the **Justify** line and then click the arrow on the right side of the box.
- 4 Scroll down the drop-down list and click **Middle right** (Figure 2-43).

The text justifies to the right. Note that the justification is dynamic—that is, as you scroll, the lines of text change.

Figure 2-43 shows a list of other possible justifications available. Figure 2-44 shows examples of the nine possible justifications shown in the drop-down list.

Figure 2-44



Using the Symbol Options

In AutoCAD, you can type some basic characters to display three common symbols used in drawing text:

- %%C = Ø (diameter symbol)
- %%P = ± (plus/minus symbol)
- %%D = ° (degree symbol)

As you type a symbol with one of these options, it begins by appearing just as you type it (in the %% form) in the **Multiline Text Editor**, but when you type the third character, it changes to the actual symbol.

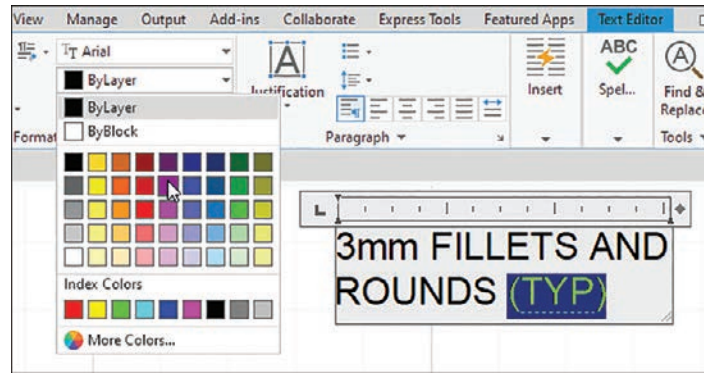
You can also create these symbols by using the **Windows Character** chart, as follows:

- Alt+0216** = Ø
- Alt+0177** = ±
- Alt+0176** = °

Changing Text Color

You can change the color of a text object by using the **Color** option located on the **Text Editor** tab (Figure 2-45). Click the arrow to the right of the **Color** box, and a chart of available colors appears.

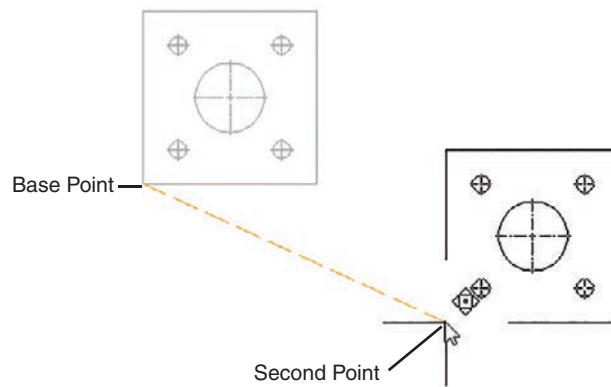
Figure 2-45



2-16 Move

The **Move** command relocates selected objects to a new location or locations on a drawing (Figure 2-46).

Figure 2-46



Moving an Object

- 1 Select the **Move** tool button from the **Modify** panel on the **Home** tab.

Select objects:

- 2 Window the entire object.

Select objects:

- 3 Press **Enter** or right-click.

Specify base point or [Displacement]:

- 4 Pick or define a base point.

You can select any point. Snap points are usually used as base points because they can define precise displacement distances and accurate new locations.

Specify second point or <use first point as displacement>:

- 5 Pick or define a second point, which is the displacement point (that is, the new location of the objects relative to the base point).

The object moves relative to the first point to its new location, relative to the second point.

2-17 Copy

The **Copy** command makes an exact duplicate of an existing object. You can use **Copy** to create more than one copy without reactivating the command. **Copy** is similar to **Move** in that you pick a base point and a second point, but **Copy** leaves the original object.

Copying an Object

- 1 Select **Copy** from the **Modify** panel.

Select objects:

- 2 Window the entire object.

Select objects:

- 3 Press **Enter** or right-click.

Specify base point:

- 4 Pick or define a base point.

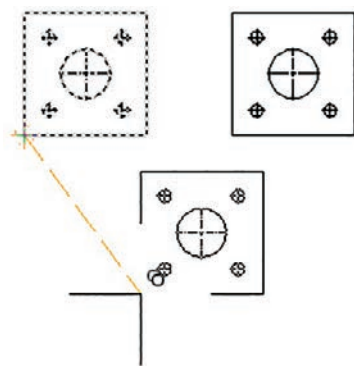
In the example shown in Figure 2-46, the lower-left corner of the object was selected as the base point.

Specify second point of displacement or <use first point as displacement>:

- 5 Pick or define a second point, which is the displacement point.

The original object remains in its original location, and a new object appears relative to the displacement point (Figure 2-47).

Figure 2-47



- 6 Right-click and select **Enter** to create the copy and end the command.

Making Multiple Copies

AutoCAD continues to make copies of a selected object until all of the copied objects are made.

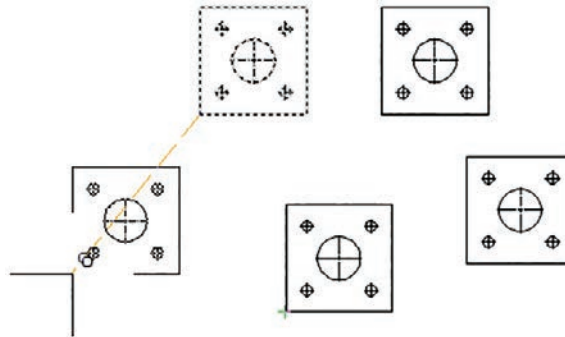
- 1 Make as many copies as you like.

After you make each copy, you see this prompt:

Select a second displacement point:

- 2 Move the cursor to a new displacement point or right-click and select **Enter** to confirm that the objects have been copied (Figure 2-48).

Figure 2-48



2-18 Offset

- 1 Select **Offset** from the **Modify** panel.

Specify offset distance or [Through Erase Layer] <Through>:

- 2 Specify the offset distance by typing a value and then press **Enter**.

In this example (Figure 2-49), a distance of **.25** was selected.

Select object to offset or <exit>:

Figure 2-49



- 3 Select an object.

Specify point on side to offset:

- 4 Select a point on the side of the object you want to offset.

Select object to offset or <exit>:

You can repeat the process by double-clicking the right mouse button and selecting the **Repeat Offset** option.

2-19 Mirror

The **Mirror** command is very useful when drawing symmetrical objects because only half of the object needs be drawn. The second half can be created by using the **Mirror** command.

- 1 Select **Mirror** from the **Modify** panel.

Select objects:

- 2 Window the object.

Select objects:

- 3 Press **Enter** or right-click.

Specify first point of mirror line:

- 4 Select a point on the mirror line.

Specify second point of mirror line:

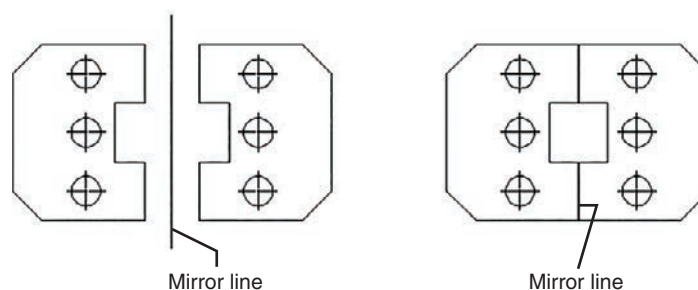
- 5 Select a second point on the mirror line.

Delete source object? [Yes No]<N>:

- 6 Press **Enter** or right-click.

Any line can be used as a mirror line, including lines within an object. Figure 2-50 shows an object mirrored about one of its edge lines.

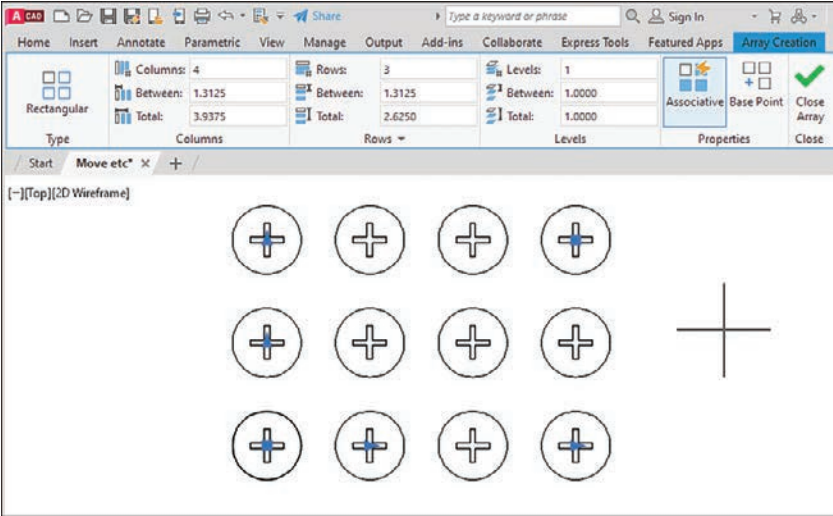
Figure 2-50



2-20 Array

An array is a group of objects arranged in a regular pattern. You can put objects in arrays using the **Rectangular Array**, **Path Array**, and **Polar Array** commands. Figure 2-51 shows a rectangular array.

Figure 2-51

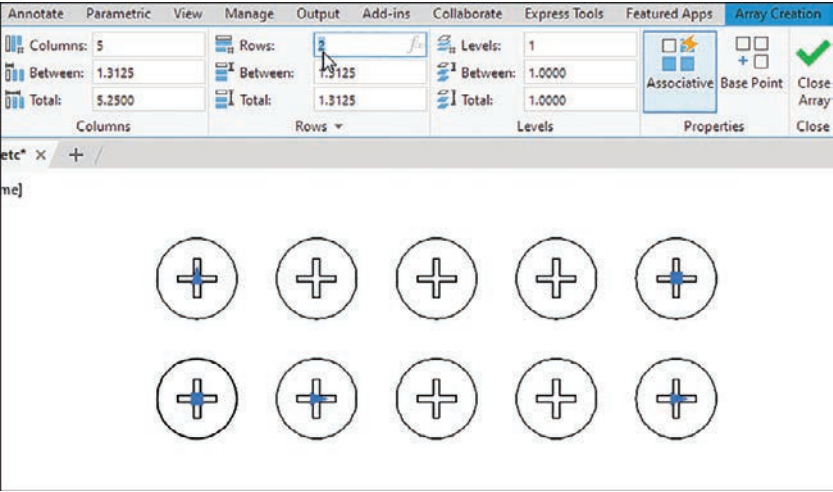


Using the Rectangular Array Option

- 1 Select **Rectangular Array** from the **Modify** panel.
- 2 Select the object to be arrayed and right-click.

The screen changes. The array creation panels appear on the **Array Creation** tab (Figure 2-51). The default setting is for a 4-by-3 array. Edit the default array by entering values in the panels on the **Array Creation** tab. In the example in Figure 2-52, a 5-by-2 array was created.

Figure 2-52



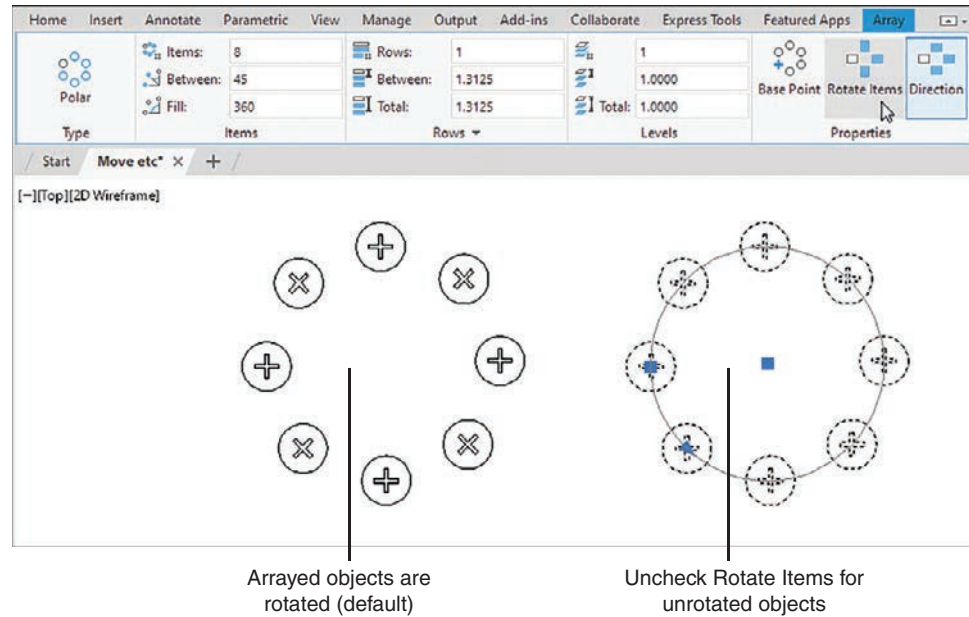
- 3 Enter the array values.
- 4 Right-click and select the **Enter** option.

Using the Polar Array Option

With the **Polar Array** option, objects are arrayed in a circular pattern around a center point (Figure 2-53).

- 1 Select the **Polar Array** tool from the **Modify** panel.

Figure 2-53



- 2 Select the object to be arrayed and right-click.

Specify center point of Array or [Base point Axis of rotation]:

- 3 Select a center point for the array.

The screen changes. The array creation panels appear on the **Array Creation** tab (Figure 2-51). The default setting is for six items in the array. The default array can be edited by using the panels on the **Array Creation** tab. In this example, eight items were selected, and the **Direction** tool was activated. Arrayed items can be rotated (the default) or unrotated.

- 4 Enter the array values.
- 5 Right-click and select the **Enter** option.

Using the Path Array Option

Use **Path Array** when you want to create multiple objects along a defined path. The path can be a line, circle, polyline, spline, or any linear or curved object. You can also create 3D path arrays by selecting a helix (see Chapter 3) or a 3D polyline as a path. 3D polylines are not covered in this book.

- 1 Select the **Path Array** tool from the **Modify** panel.
- 2 Select the object to be arrayed and right-click.

Specify center point of Array or [Base point Axis of rotation]:

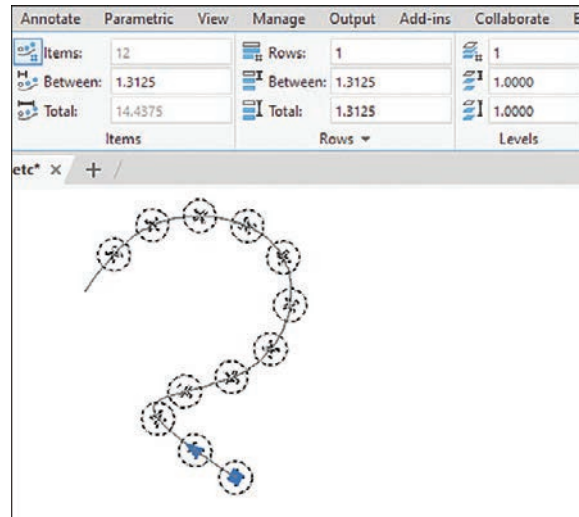
- 3 Select the path object.

Select path curve:

The **Array Creation** tab appears at the end of the ribbon.

- 4 Enter values for the array numbers and distances, and indicate whether the object should be aligned to the path as it is arrayed.
- 5 Right-click and select **Enter** (Figure 2-54).

Figure 2-54

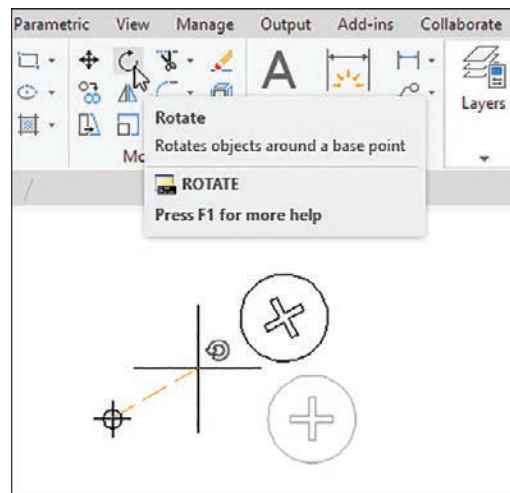


2-21 Rotate

The **Rotate** command rotates an object about a specified base point (Figure 2-55).

By default in AutoCAD, **0°** is to the east, or three o'clock on a clock face. Angles in the counterclockwise direction are positive angles.

Figure 2-55



Rotating an Object

- 1 Select **Rotate** from the **Modify** panel.

Select objects:

- 2 Select the object or objects to be rotated.

Select objects:

- 3 Press **Enter** or right-click.

Specify base point:

- 4 Pick or define a base point.

The base point may be anywhere on the screen.

Specify rotation angle or [Reference]:

- 5 Type a value and press **Enter**.

The object rotates about the base point by the specified angle in the counterclockwise direction. To rotate the object in a clockwise direction, enter a negative value.

2-22 Trim

Use the **Trim** command to remove parts of objects that are crossed by other objects. By default, starting the command selects all objects that can be trimmed. You can hover the cursor over parts of lines or other objects and click to remove them. **Trim** is a very important AutoCAD command and is used frequently.

Using Trim

- 1 Select **Trim** from the **Modify** panel.

Select cutting edges:

Select objects:

- 2 Move the cursor over a part of any object you want to remove and click when you see the red X over the object.

- 3 Press **Enter** or right-click.

Select object to trim or [Project Edge Undo]:

As shown in Figure 2-56, the left portion of the circle, and the portion of the line within the circle, are deleted.