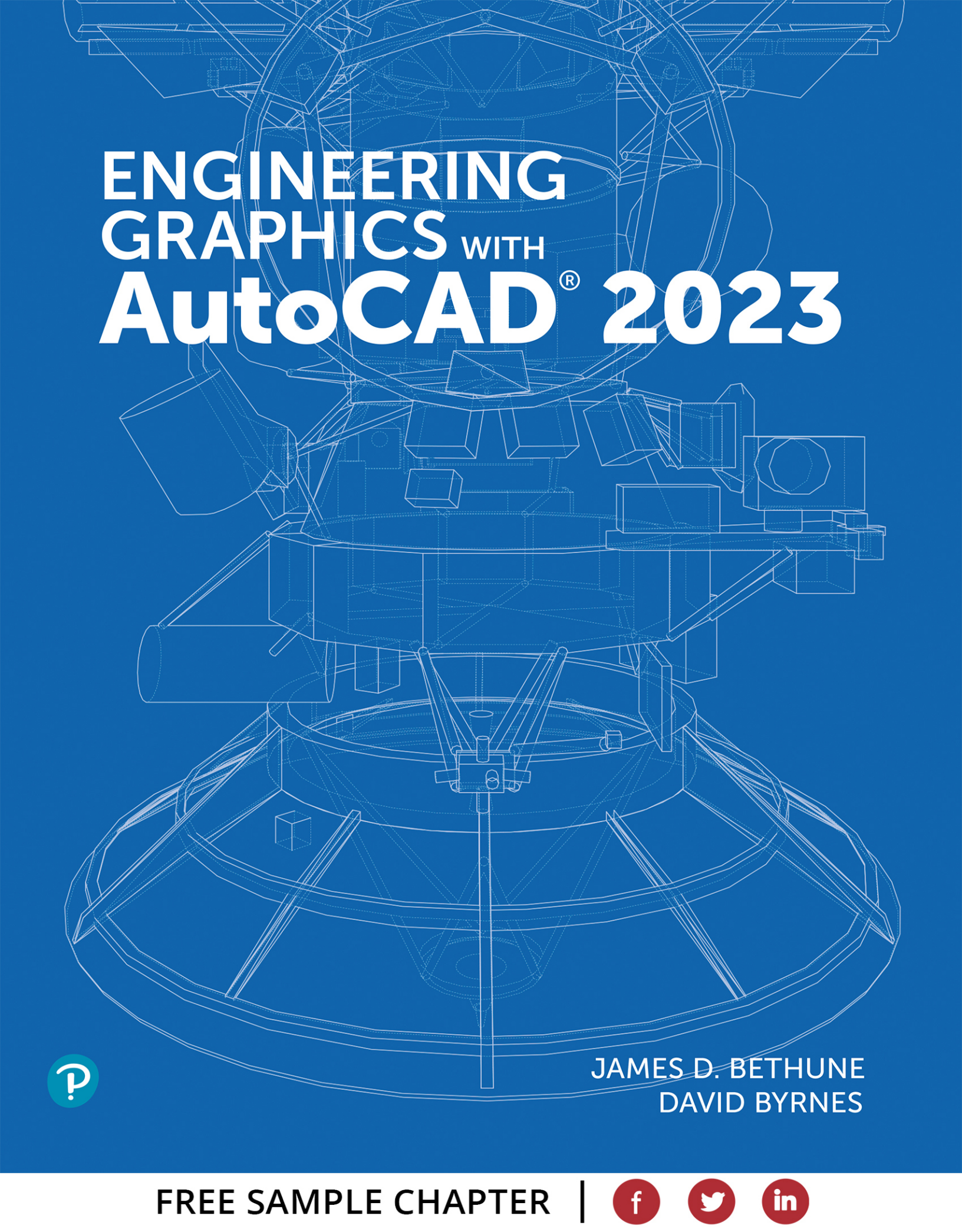

ENGINEERING GRAPHICS WITH AutoCAD® 2023

JAMES D. BETHUNE
DAVID BYRNES

FREE SAMPLE CHAPTER |

Engineering Graphics with AutoCAD® 2023

James D. Bethune

David Byrnes

Pearson

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

My thanks to Anshul Sharma, Patrice Rutledge, Kitty Wilson, and Tonya Simpson at Pearson; also thanks to Jim Bethune for creating such a fine edition to update.

David Byrnes

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Brief Contents

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Contents

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

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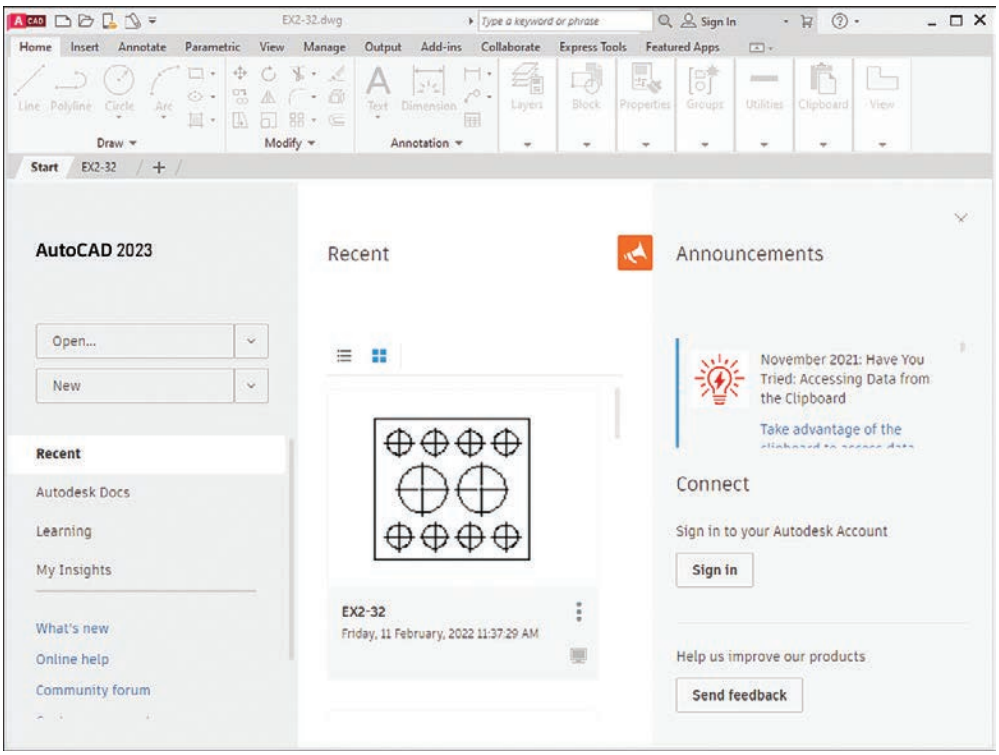
chapter one

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.

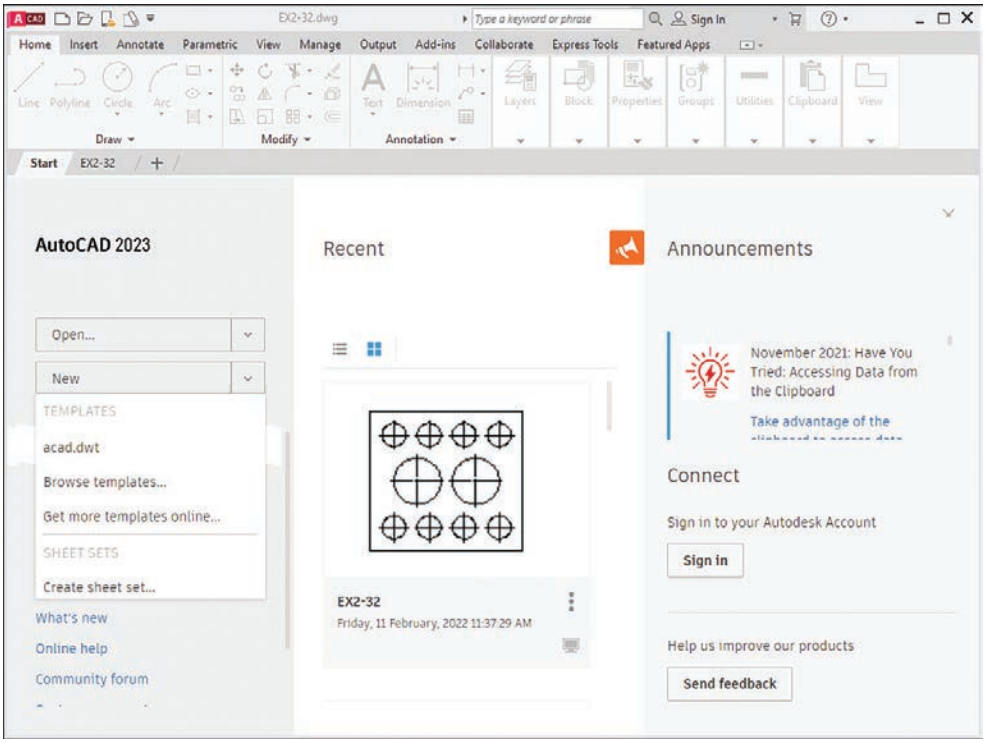
Figure 1-1 shows the initial AutoCAD drawing screen, which appears when the program is first started.

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



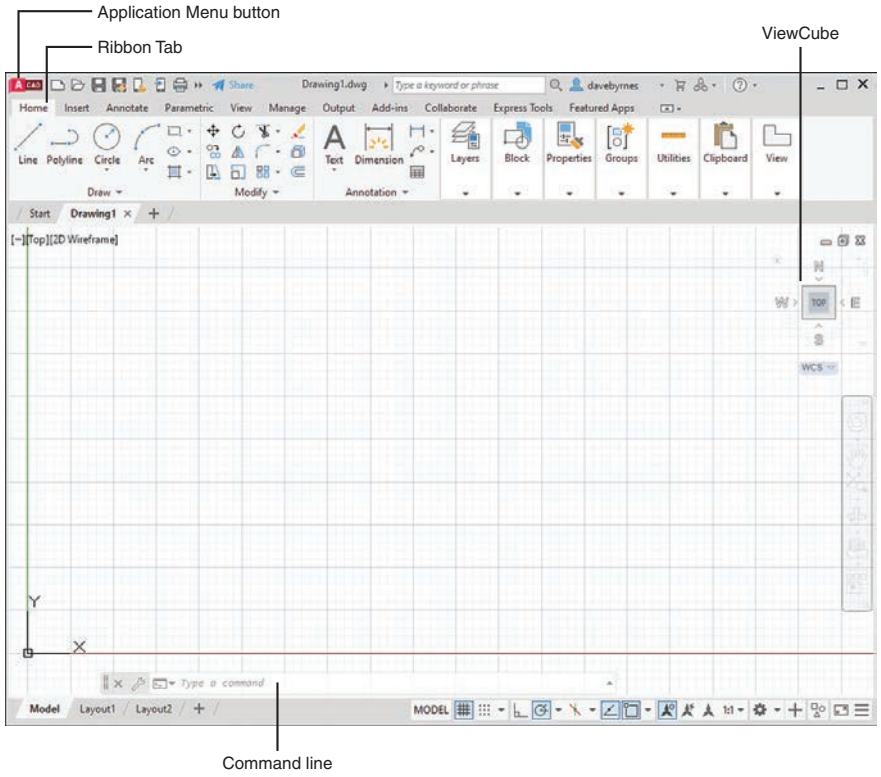
2 Click **acad.dwt** on the **Templates** list.

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

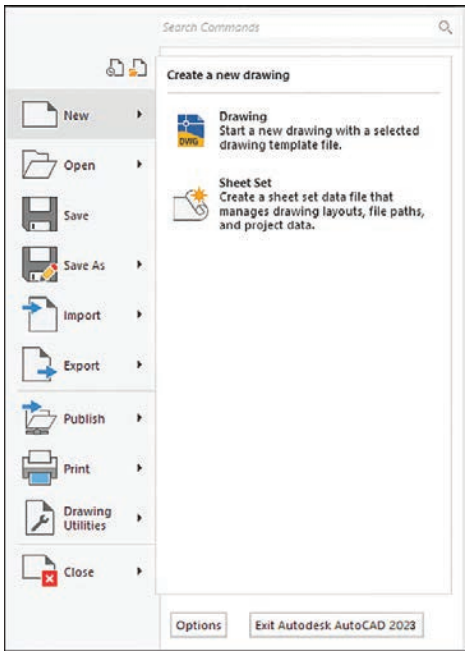
1. Click the **Application Menu** button in the upper-left corner of the drawing screen to display the **Application** menu.

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

2. Click **New**.

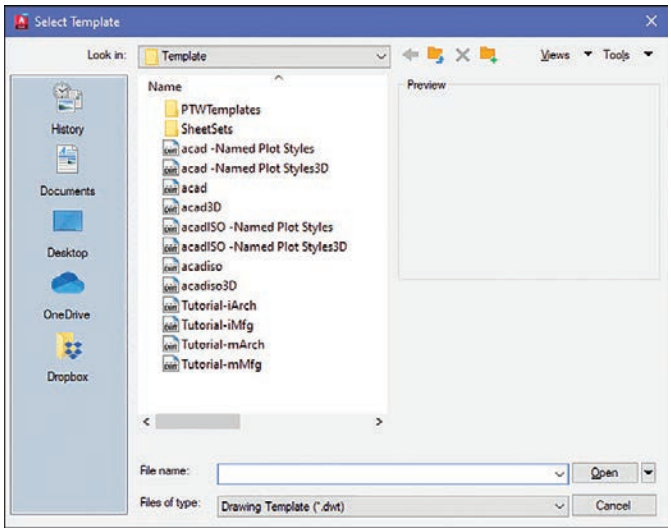
   The **Select Template** dialog box appears (Figure 1-5).

Figure 1-4



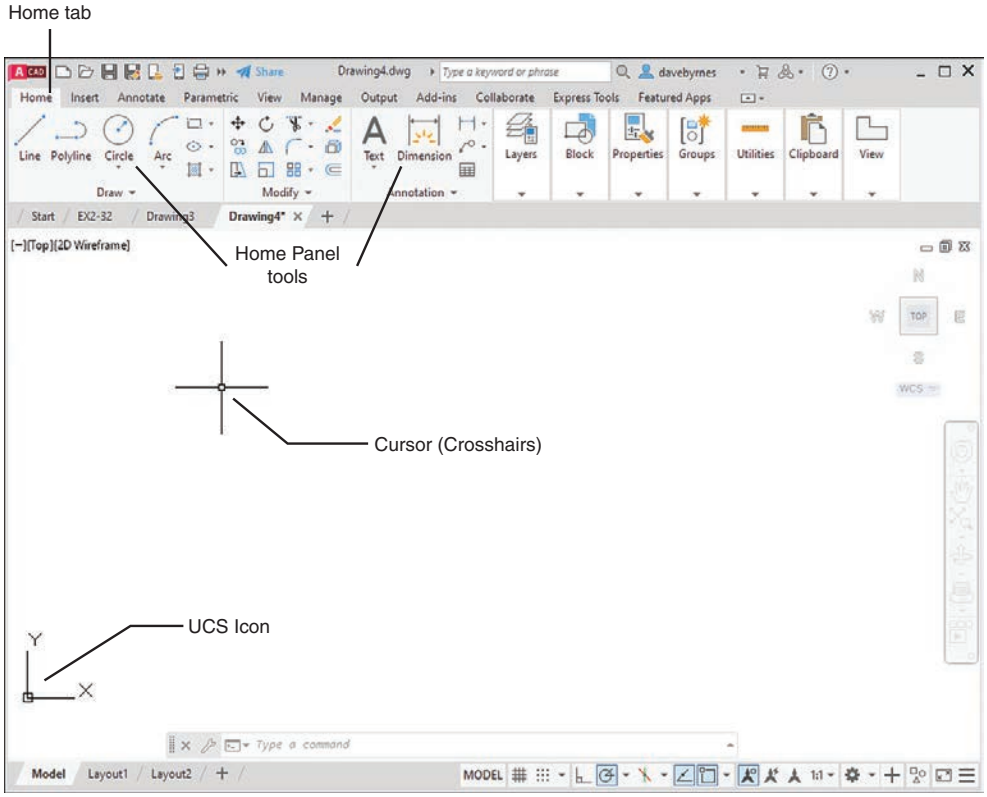
3 Select the **acad** template and click **Open**.

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.

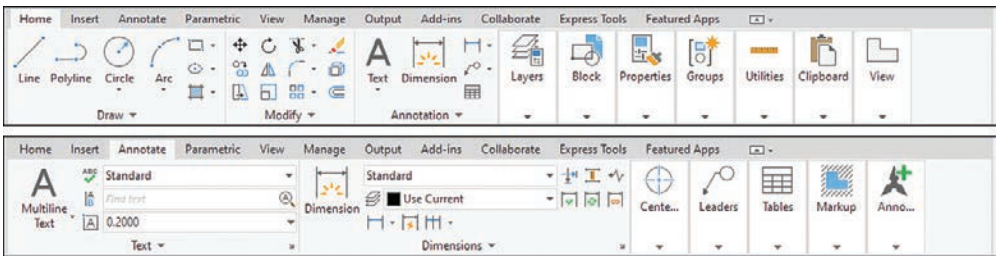
The large open area in the center of the screen is the *drawing area* or *drawing editor*. You create drawings in this area.

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.

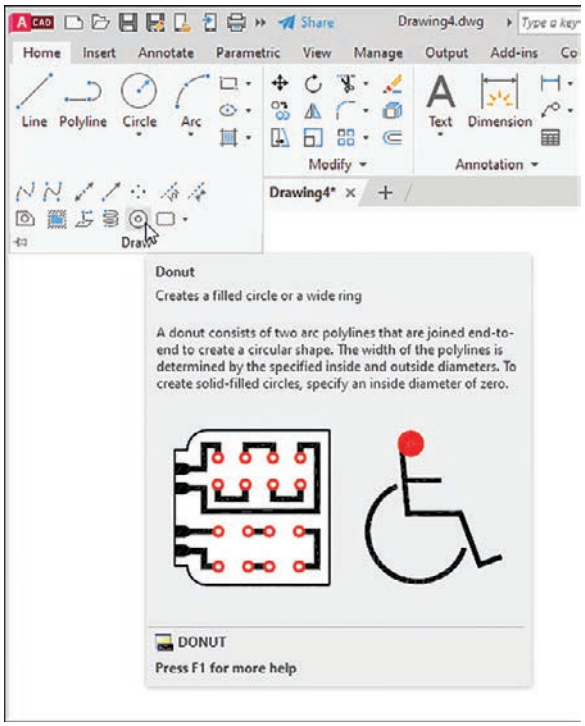
Figure 1-7



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



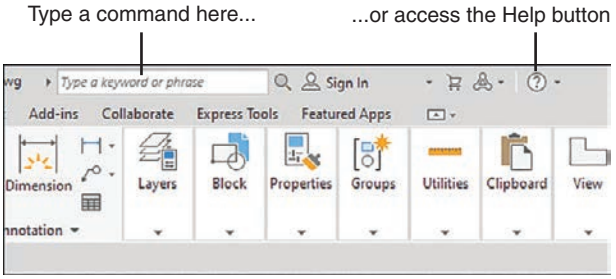
Tooltips for Commands

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.

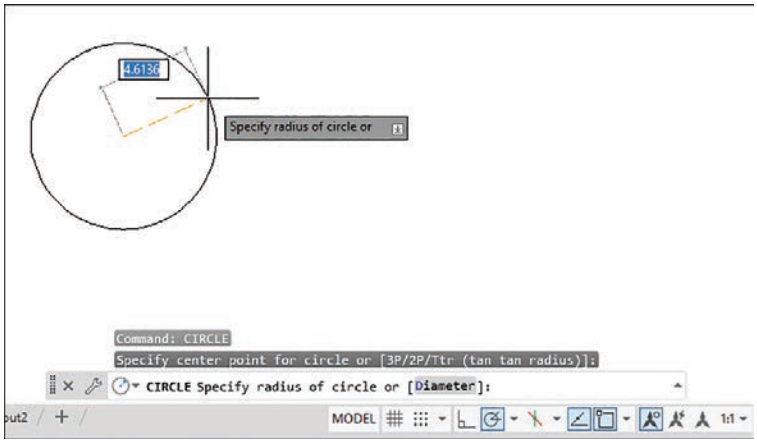
Figure 1-9



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.

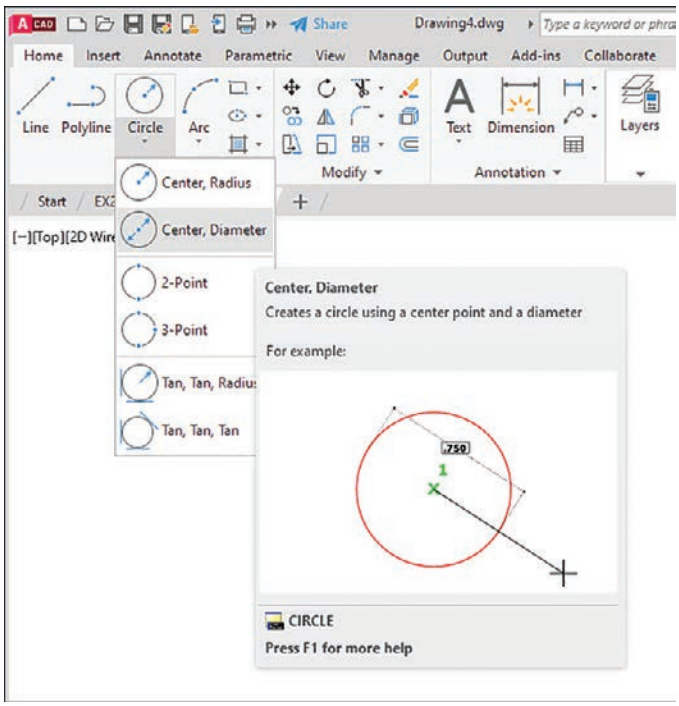
1-4 Command Tools

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.

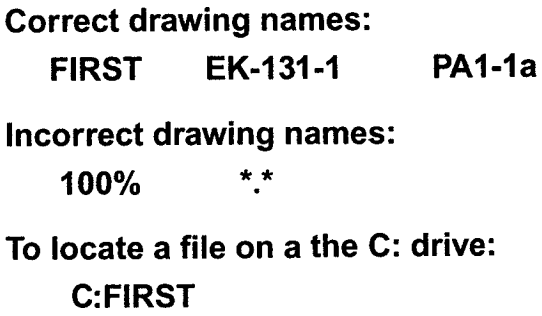
1-5 Starting a New Drawing

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



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.)

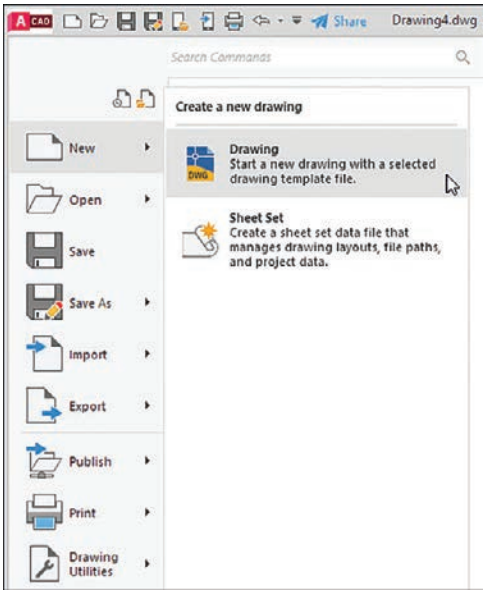
If you want to locate a file on another drive, specify the drive letter followed by a colon in front of the drawing name. For example, in Figure 1-12 **C:FIRST** will locate the drawing file **FIRST** on the C: drive.

Creating a New Drawing

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

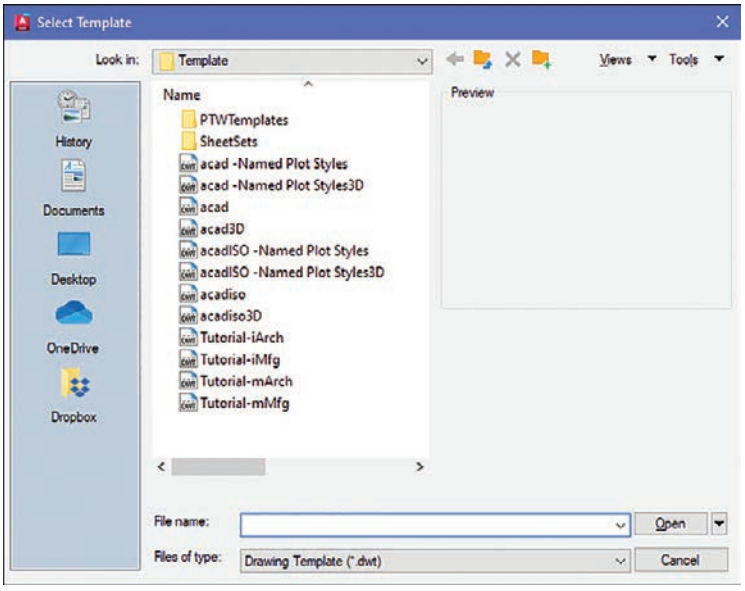
- Select **New** from the **Application** menu (Figure 1-13).
- Type the word **new** at a command prompt.
- Hold down the **Ctrl** key and press **N**.

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

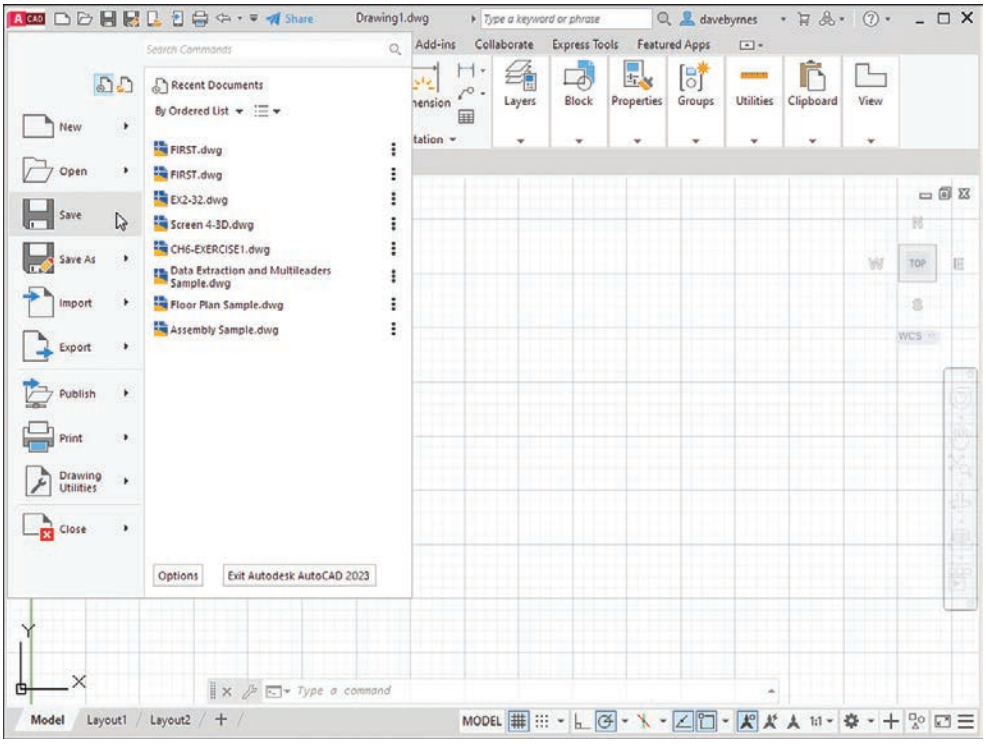
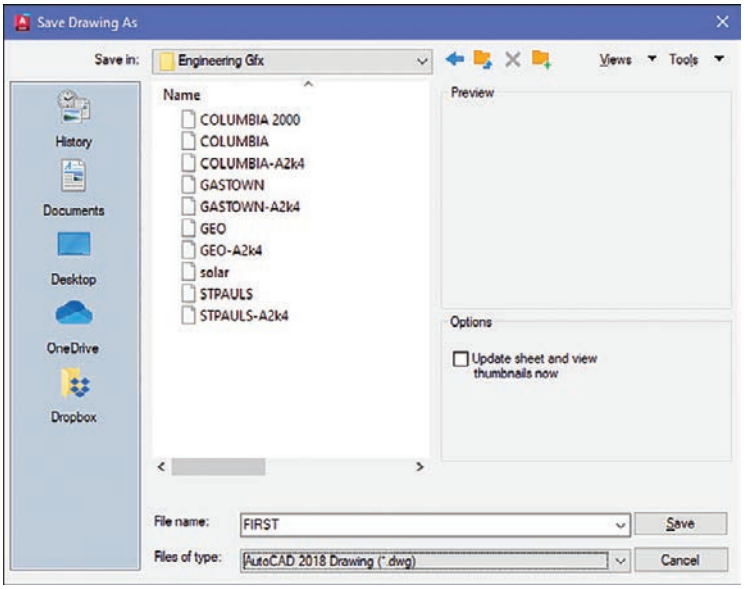


Figure 1-16



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.

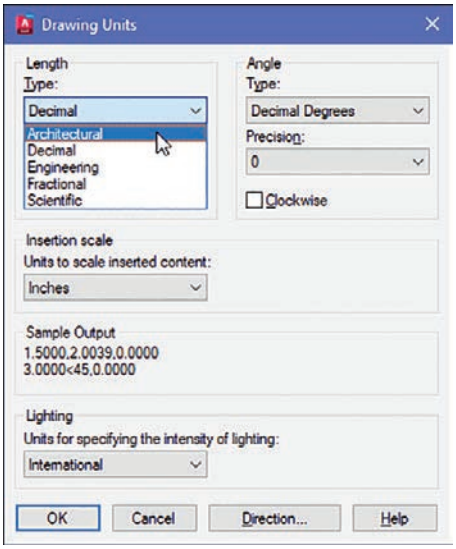
3. Enter the drawing name.
   In this example, the drawing name **FIRST** was used.
4. Click **Save**.

   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

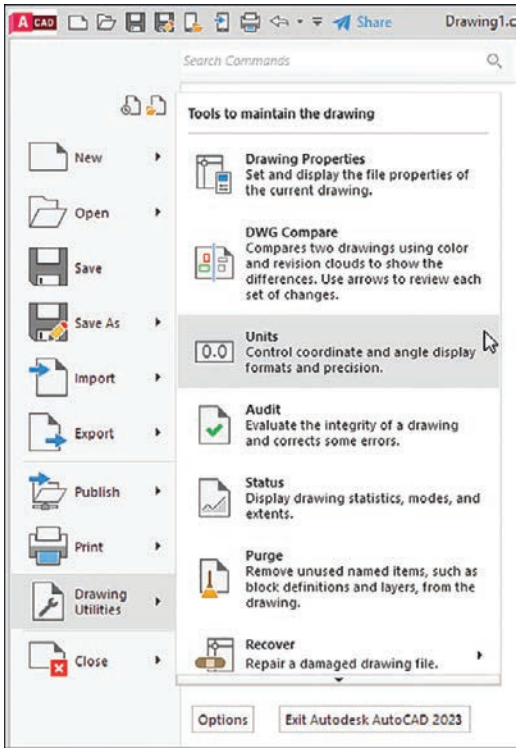
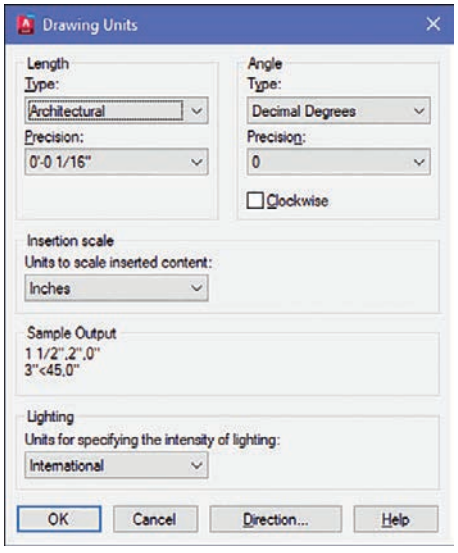


Figure 1-19



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**.

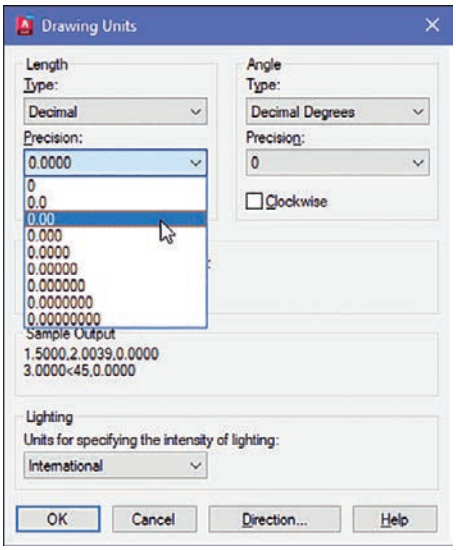
Specifying or Changing the Precision of the Units System

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

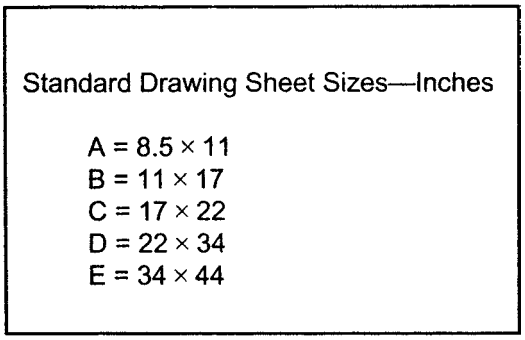


Figure 1-22

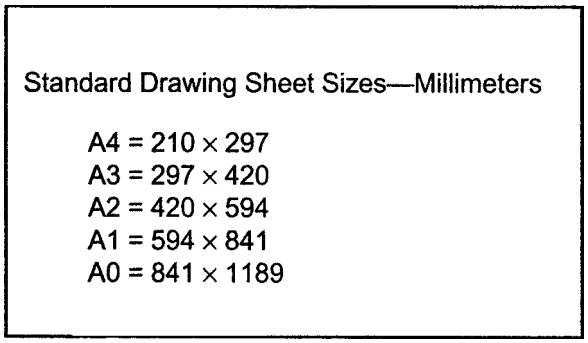
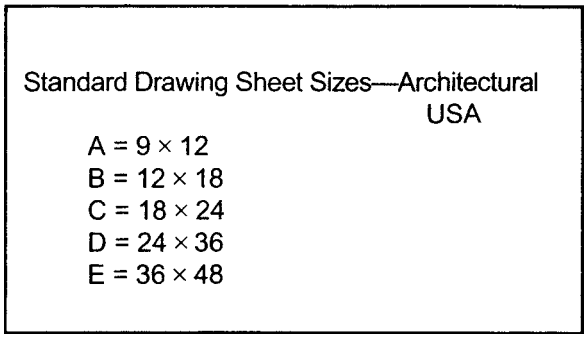


Figure 1-23



A standard 8.5″ × 11″ letter-size sheet of paper as used by most printers is referred to as an *A-size* sheet of drawing paper.

NOTE

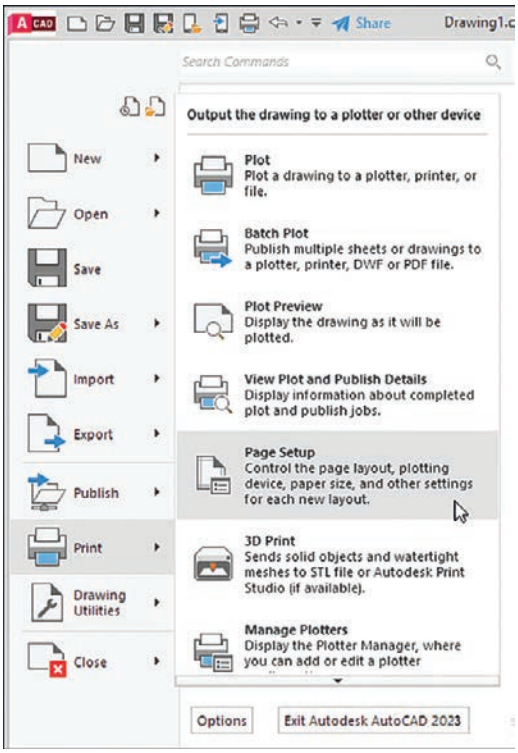
A sheet of paper can be sized to match standard sheet sizes by the capabilities of the printer or plotter. Many printers and plotters have built-in scaling features, and some list standard sheet sizes that can be applied to a drawing.

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.

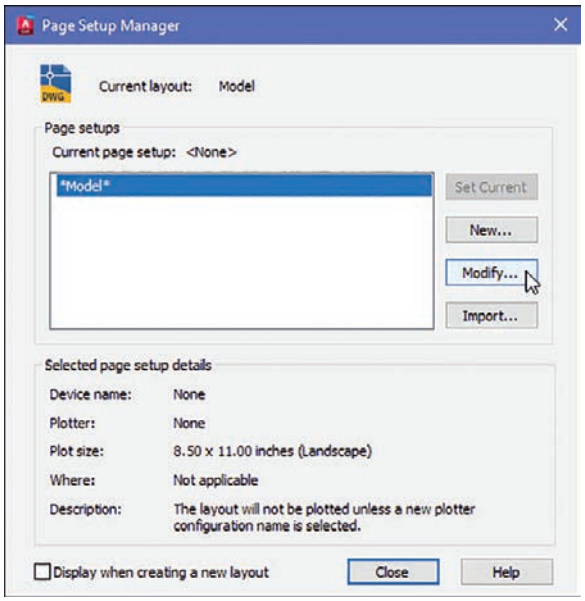
Figure 1-24



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

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

Figure 1-25



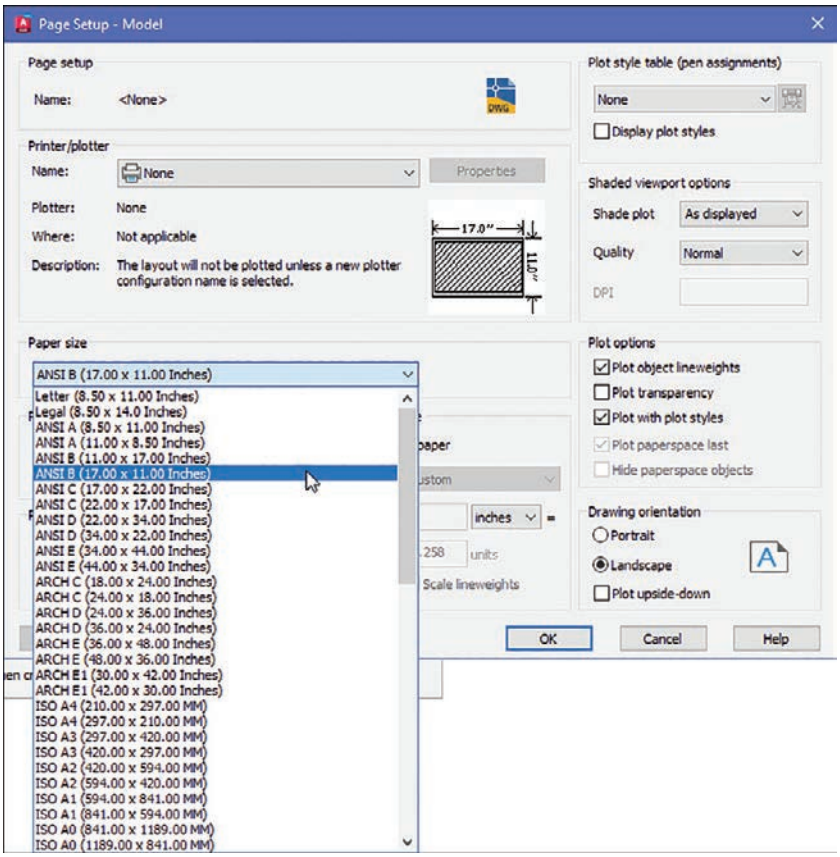
4. Click the arrow to the right of the **Paper size** box.

   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.

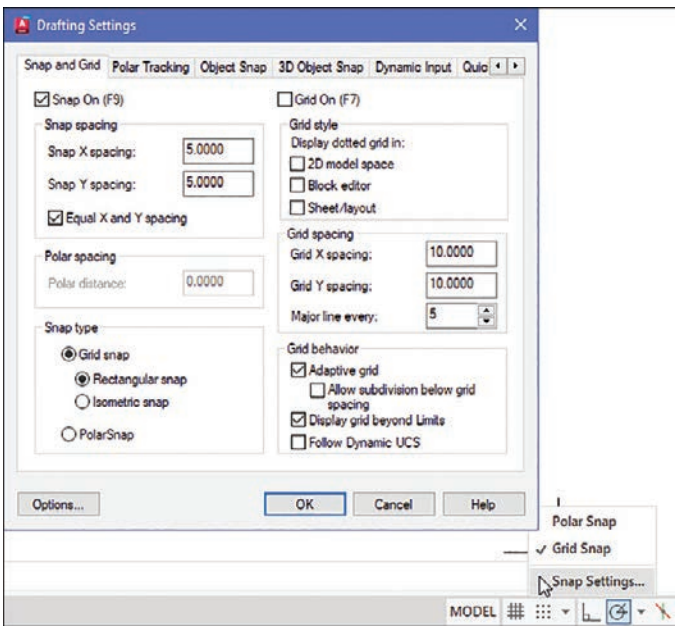
NOTE

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.

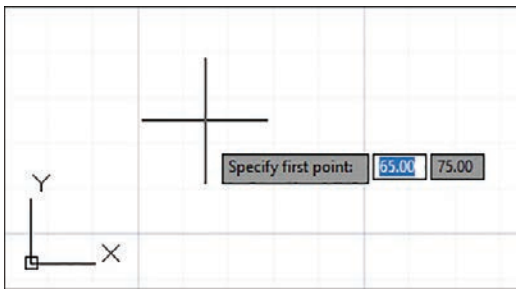
3. Click the **Grid On** and **Snap On** checkboxes. A check mark appears in each of the boxes.
4. Place the cursor in the **Snap X spacing** text box to the right of the given value under the **Snap On** heading.
   A vertical flashing cursor appears.
5. Backspace out the existing value and type in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

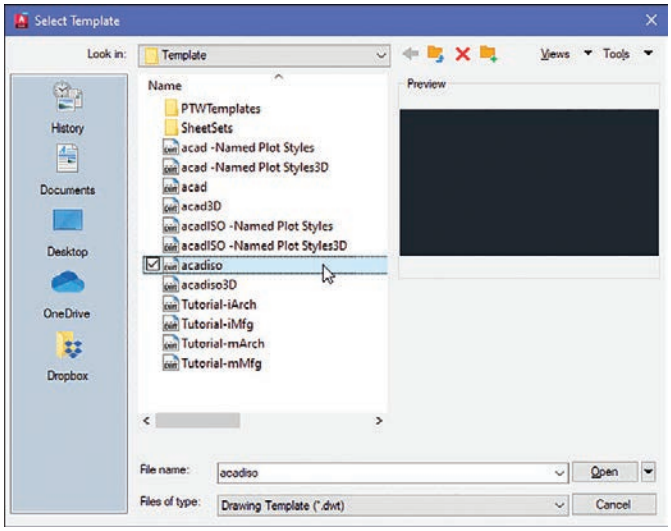
Whole-number precision

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**.

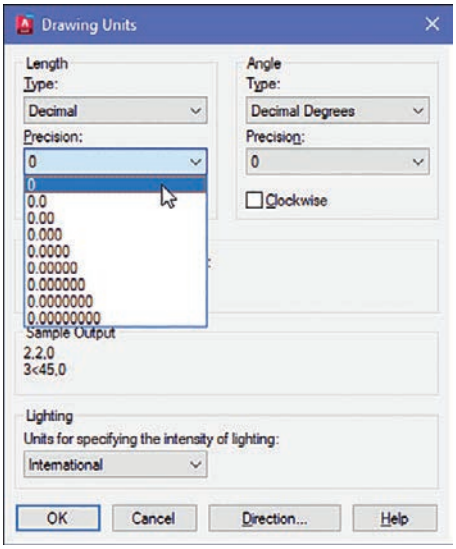
Defining the Units Precision

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.

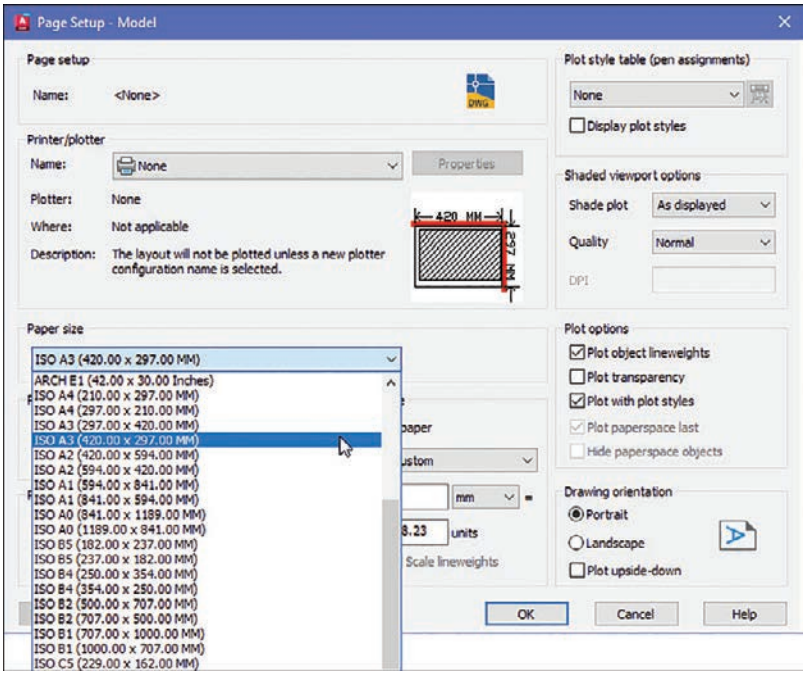
1. Open the **Application** menu and select **Print**, then **Page Setup**.

   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**.

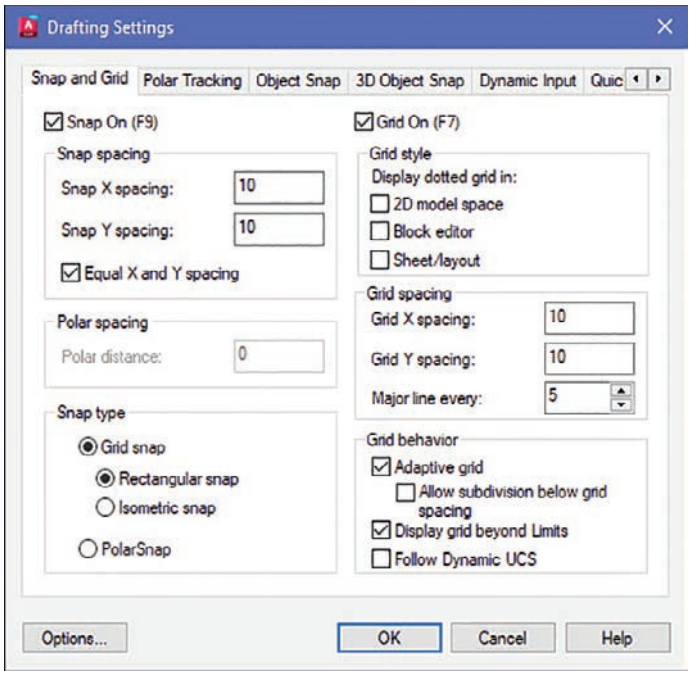
   The **Drafting Settings** dialog box appears (Figure 1-32).

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



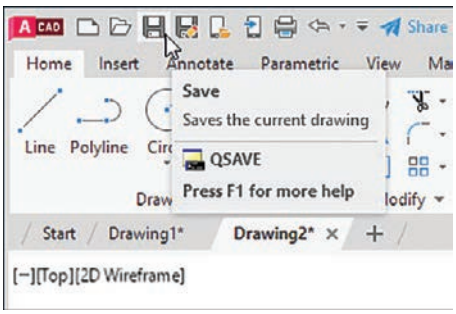
1-11 Save and Save As

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

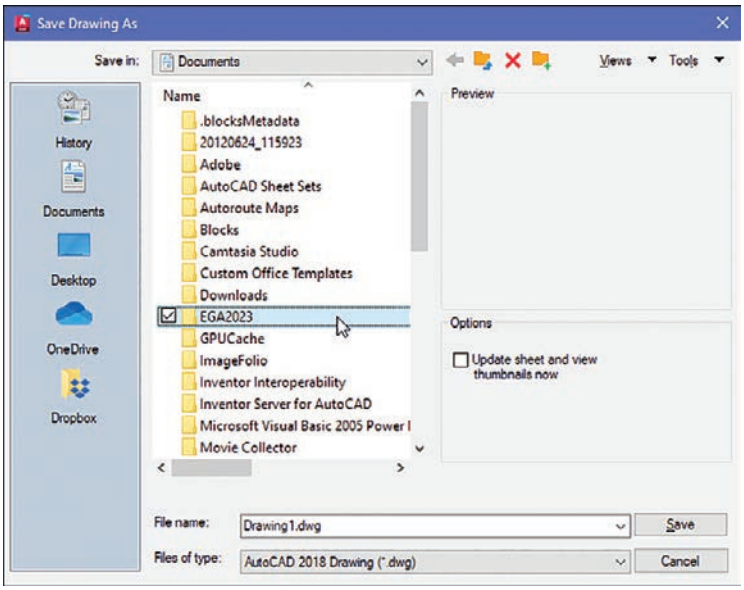


NOTE

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

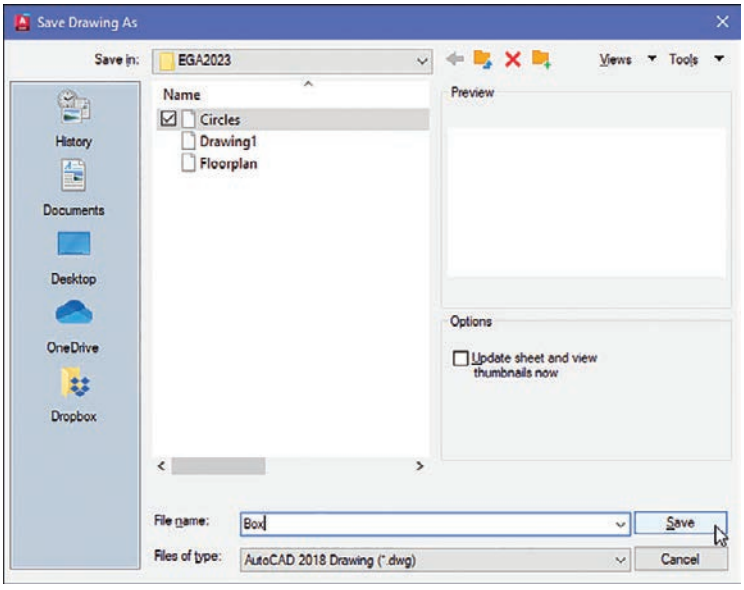


2. Create a new folder where you can save your work.

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

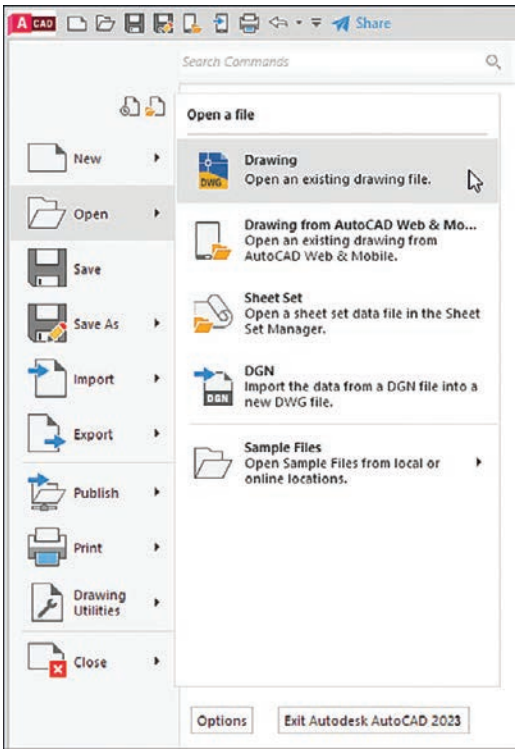
Use the **Open** command to call up an existing drawing so that you can continue working on it.

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.

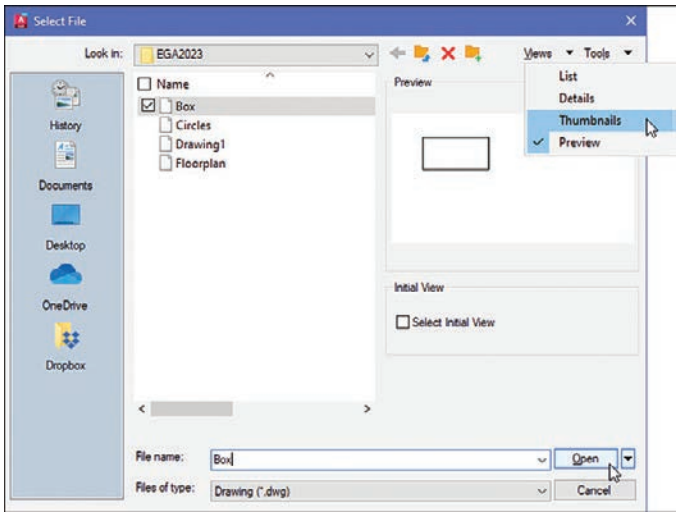
Thumbnails of the drawing files appear.

3 Click the desired file.

A preview appears.

4 Click **Open**.

Figure 1-37



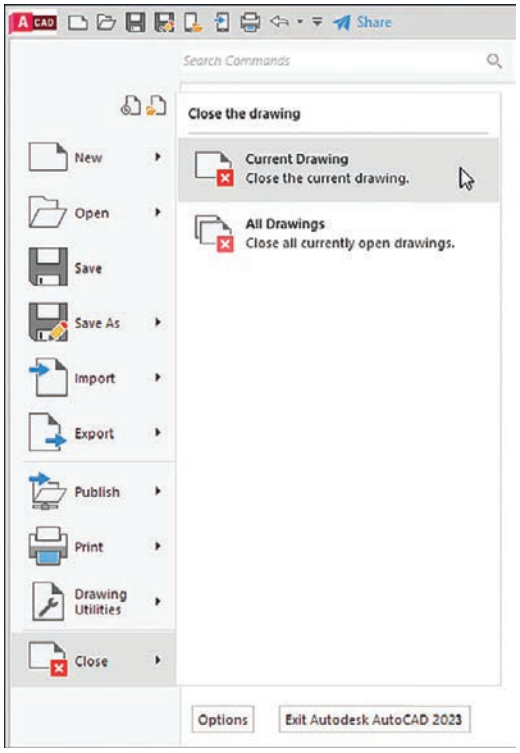
1-13 Close

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.

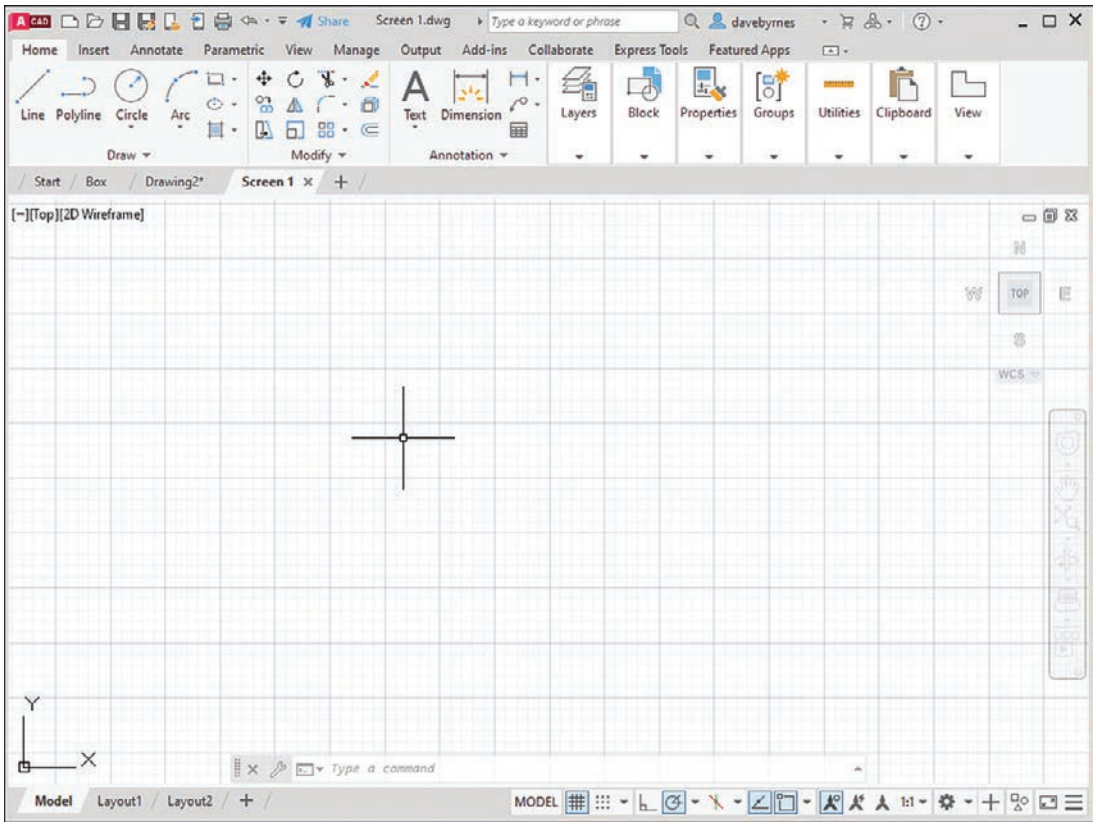
Figure 1-38



1-14 EXERCISE PROBLEMS

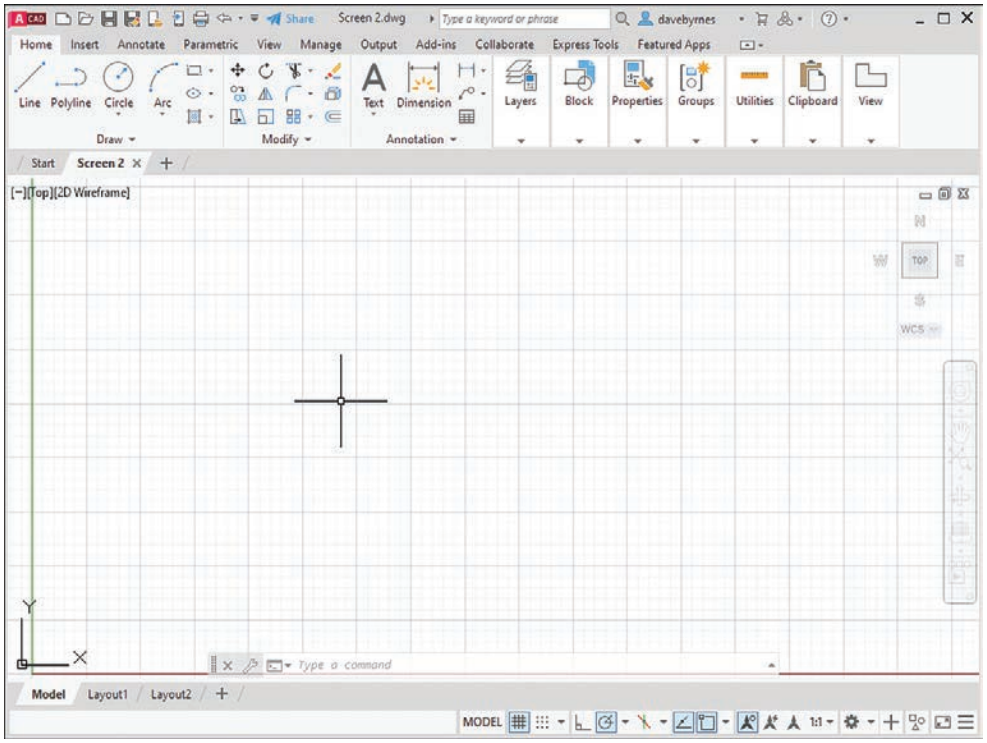
EX1-1

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**.



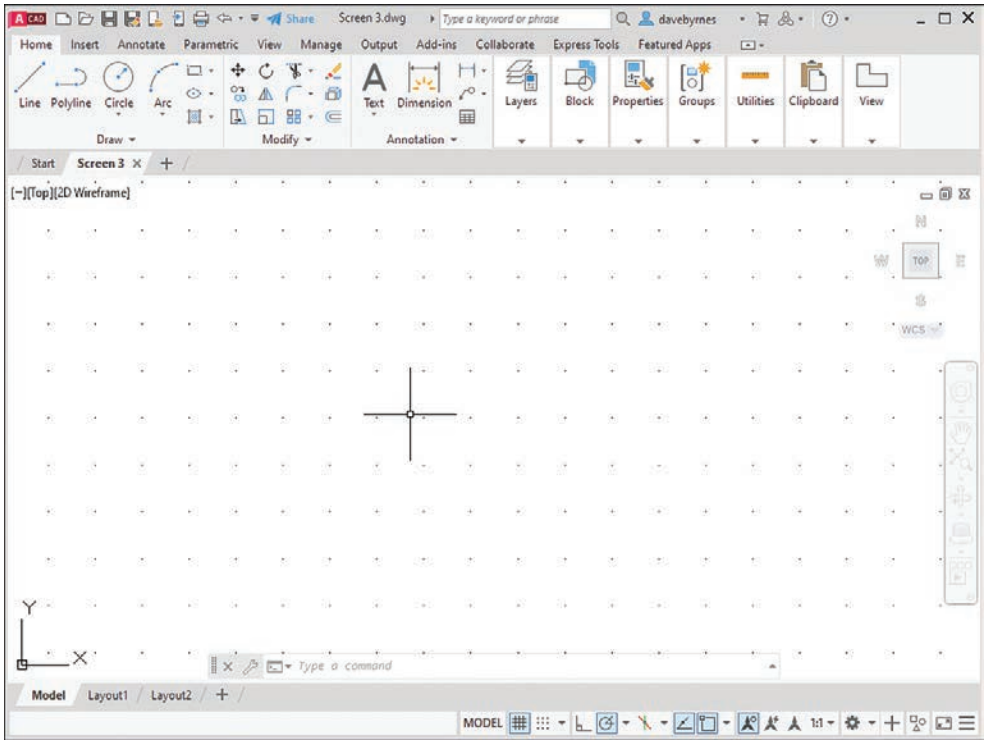
EX1-2

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**.



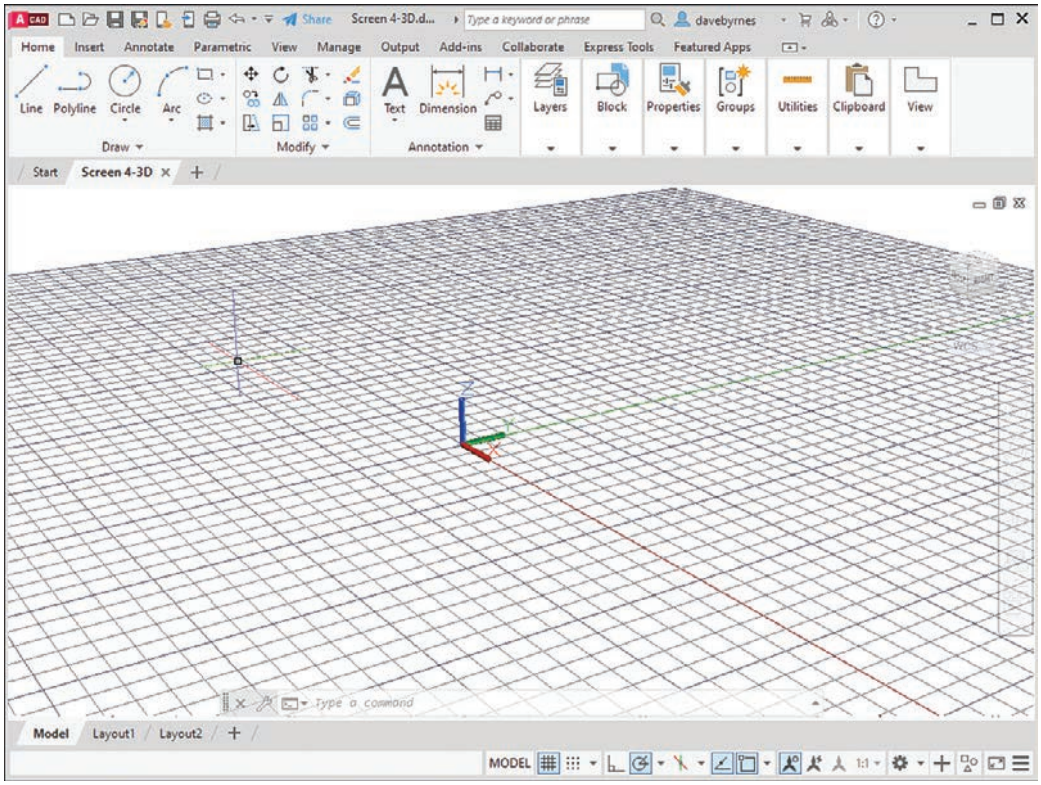
EX1-3

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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