Applying
AutoCAD® 2010

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Engineering Graphics

Applying AutoCAD
2010 Edition

Wohlers

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Contents

Wohlers • *Applying AutoCAD® 2010*

Wohlers: Applying
AutoCAD® 2010

Front Matter

What's New in 2010

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1

What's New in 2010

AutoCAD 2010 introduces new features along with continued enhancements to existing capabilities. You will notice the following differences in *Applying AutoCAD 2010:*

- Further refinements to the user interface. The arrangement of tabs, panels, and buttons on the **Ribbon** has improved, allowing for faster, more intuitive modeling–particularly in the 3D Modeling workspace.

- An entirely new **3D object**, the **3D mesh**, which allows users to create models with smooth, flowing geometry in 3D space. When mesh modeling is combined with surface modeling and solid modeling, AutoCAD users now have more power and freedom to design in three dimensions than ever before.

- New tools for working with solid primitives and mesh primitives. Subobject filters and gizmo tools make it much easier to select and modify any face, edge, or vertex of a primitive.

- Dynamic blocks are easier to author and edit, with new procedures and on-screen icons.

About the Author

For more than two decades, Terry Wohlers has focused his education, research, and practice on design and manufacturing. He has authored more than 350 books, articles, reports, and technical papers on engineering and manufacturing automation. He has presented to thousands of engineers and managers and has been a keynote speaker at major industry events in Asia, Europe, the Middle East, the Americas, and South Africa.

Wohlers is a renowned CAD educator. He developed and taught the first graduate and undergraduate courses on CAD at Colorado State University in 1983. Since then, he has taught many courses and given countless lectures on CAD and related subjects. In 1986, he founded Wohlers Associates, Inc., an independent consulting firm that provides organizations with technical and strategic advice on new developments and trends in CAD/prototyping, 3D printing, prototyping, and additive manufacturing.

In May 2004, Wohlers received an Honorary Doctoral Degree of Mechanical Engineering from Central University of Technology, Free State (Bloemfontein, South Africa).

Chapter **1**

Exploring AutoCAD

Objectives

- Start AutoCAD Vocabulary
- Preview drawing files
- Open drawing files
- View details on an AutoCAD drawing
- Exit AutoCAD

Vocabulary

button
Command line
dialog box
double-clicking
drawing area
icon
resolution
status bar
thumbnail

AutoCAD provides countless methods and tools for producing, viewing, and editing two-dimensional drawings and three-dimensional models. The software permits designers, drafters, engineers, and others to create, revise, model, and document industrial parts and assemblies for prototyping, mold-making, and manufacturing. Around the world, organizations also use AutoCAD for the design of maps, buildings, bridges, factories, and just about every other product imaginable, ranging from car parts and stereo equipment to snow skis and cellular phones.

Starting AutoCAD

The first step in using AutoCAD is to learn how to start the software and open a drawing file.

1. Start AutoCAD by **double-clicking** the **AutoCAD 2010 icon** found on the Windows desktop. An icon is a picture on the screen that identifies and activates a tool.

> **HINT**
>
> Double-clicking means to position the pointer on the item and press the left button on the pointing device twice very quickly. If the **AutoCAD 2010** icon is not present, pick the Windows **Start** button, move the pointer to **Programs** (or to **All Programs**), move it to **Autodesk**, then to **AutoCAD 2010**, and pick the **AutoCAD 2010** option.

2. If the **New Features Workshop dialog box** appears, as shown in Fig. 1-1, select the round button next to **No, don't show me this again** and pick the **OK** button. (A dialog box provides information and permits you to make selections and enter information. In this case, the dialog box asks you if you would like to view the **New Features Workshop**.)
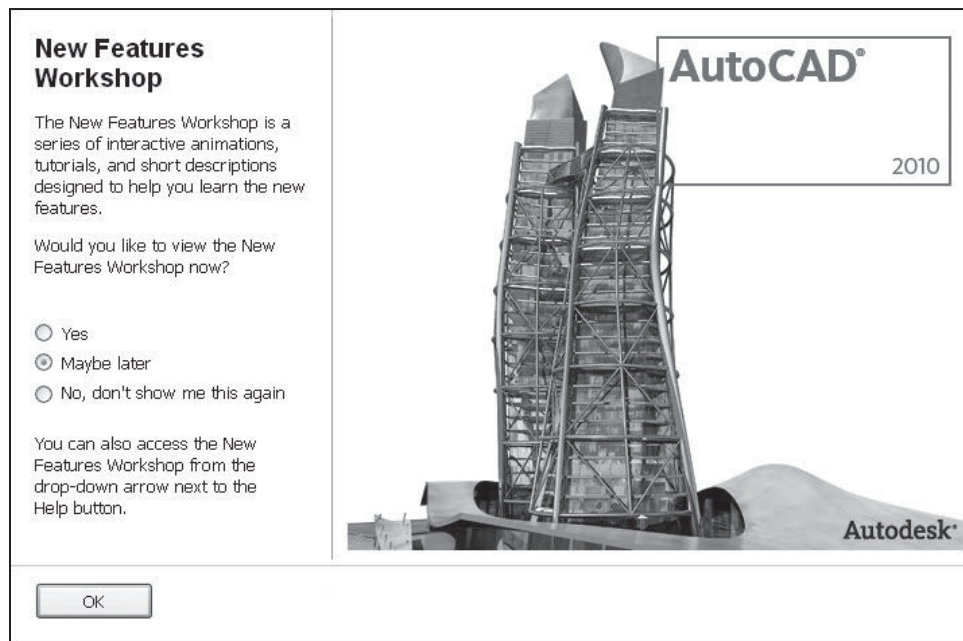
Fig. 1-1

The **New Features Workshop** is a series of tutorials that help you learn the new features of AutoCAD 2010. It is always available to you in the **AutoCAD 2010 Help** window. You'll learn more about getting help in **Chapter 8**.

> **NOTE**
>
> The exact sequence for starting AutoCAD 2010 may vary depending on how the software was left by the last person to use it. If windows do not appear as described, skip to the following step.
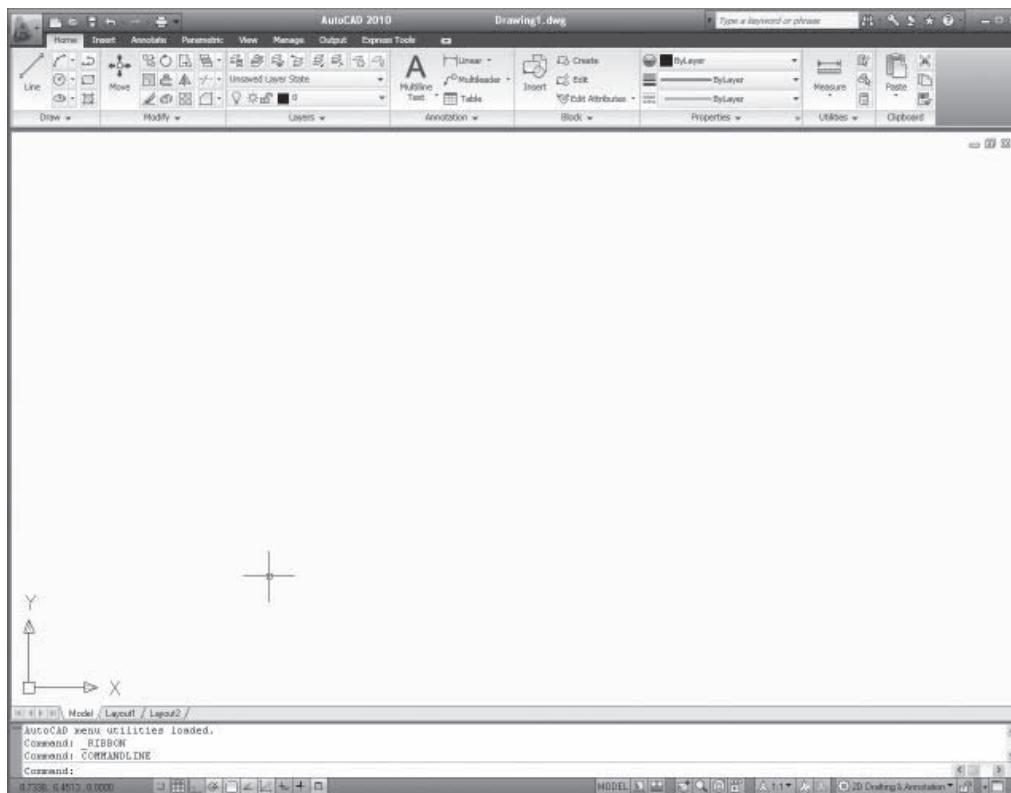
Fig. 1-2

3. If the **Startup** dialog box appears, click the **Cancel** button.

The AutoCAD window appears, as shown in Fig. 1-2. User commands and settings appear along the top and bottom of the AutoCAD window. The large, blank, center portion of the window is the **drawing area**.

> **NOTE**
>
> The exact configuration of the AutoCAD window may vary depending on the screen **resolution** and how it was left by the last person to use the software. The display hardware can cause the appearance of screen elements to vary. Resolution is the number of pixels per inch on a display system; this determines the amount of detail that you can see on the screen.

Previewing Drawing Files

Notice the small toolbar in the title bar, just to the right of the large red "A" in the upper left corner of the screen. This is the **Quick Access** toolbar.

1. On the **Quick Access** toolbar, click on the **Open** button. (The **Open** button is the small square with a picture of an open file folder.)

Buttons in AutoCAD are the small areas that activate commands or functions. Buttons are identified by an icon, or picture.

This displays the **Select File** dialog box, shown in Fig. 1-3. In this case, the dialog box enables you to open a drawing file, as well as select other options.

Fig. 1-3

Wohlers: Applying
AutoCAD® 2010

I. Groundwork

1. Exploring AutoCAD

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7
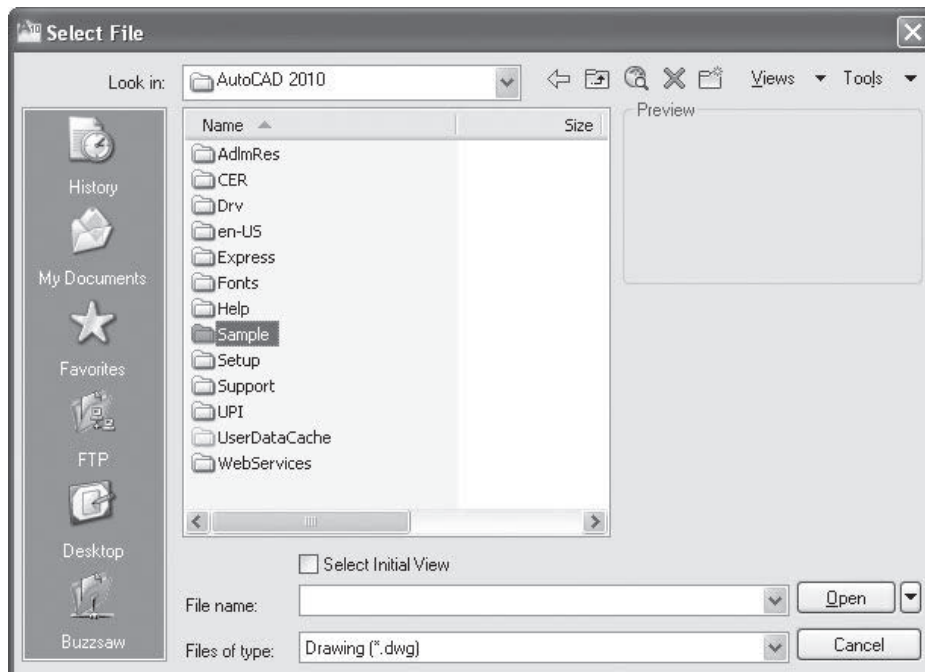
2. Navigate inside the **Select File** dialog box and find the **AutoCAD 2010** folder.

3. Find and double-click the AutoCAD folder named **Sample**.

4. Find and double-click the folder named **Mechanical Sample**.

This displays all of the files and folders contained in the Mechanical Sample folder.

5. To preview a file, rest the pointer on any of the files in the list. If nothing displays in the **Preview** area of the dialog box, single-click the file. (Your computer may be configured for either of these two methods.)

Notice that a small picture of the drawing, or **thumbnail**, appears at the right in the area labeled **Preview**. This gives you a quick view of the graphic content of the selected file.

6. Preview each of the remaining drawing files by resting the pointer or single-clicking on each of them.

Opening a Drawing File

7. Double-click the file named **Mechanical-Multileaders.dwg,** or single-click it and pick the **Open** button. (The **DWG** extension may not be shown as part of the file name. This depends on the configuration of your computer.)

A drawing of a wellhead piping design appears. Notice the tabs at the bottom of the AutoCAD drawing area, as shown in Fig. 1-4. These are the **Model** and **Layout** tabs. In this drawing, the **Layout** tab has been renamed.
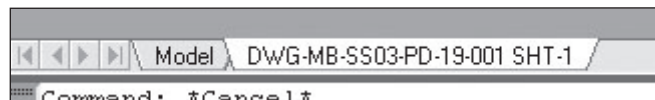
⟨ **F i g .   1 - 4** ⟩

8. Pick the **Model** tab.

This displays model view of the drawing. This is the view for most drafting and design work in AutoCAD.

9. Pick the **Layout** tab, labeled **DWG-MB-SS03-PD-19-001 SHT-1**.

This displays the drawing as it would appear on a printed or plotted sheet.

Viewing Details

AutoCAD offers powerful tools for viewing details on a drawing. For example, the zoom function allows you to take a closer look at very small sections of the drawing.

STATUS BAR

1. On the bar at the bottom of the AutoCAD window, pick the **Zoom** button.

The **Zoom** button displays a picture of a magnifying glass.

> **HINT**
>
> Notice the button printed to the left of Step 1. This has been provided as a visual aid to help you locate and select the button. The screen location of the button appears below the button for reference. The **status bar** is the bar across the bottom of the AutoCAD screen that displays the location of the pointer and many of the most commonly used buttons.
>
> Buttons like this will appear in the margin throughout the chapter in which they are introduced and in subsequent chapters. They will also appear occasionally if the button has not been used recently, to refresh your memory.

In the lower left area of the window, you will see AutoCAD's **Command line**, which allows you to key in commands and information. AutoCAD also displays important information in this area. Notice that it is now asking you to specify the first corner of the window.

Imagine a small window surrounding the title block of the drawing, as shown in Fig. 1-5. This will become the zoom window.

2. Using the pointing device, pick a point (by pressing the left button) to specify one of the four corners of the imaginary window. See Fig. 1-5 for the location of the window.

3. Move the pointing device to the opposite corner to create the window and pick another point.

AutoCAD zooms in to fill the drawing area with the area represented by the zoom window.

4. Move the pointing device slowly around the drawing. Notice that each item in the drawing becomes highlighted as the cursor touches it.
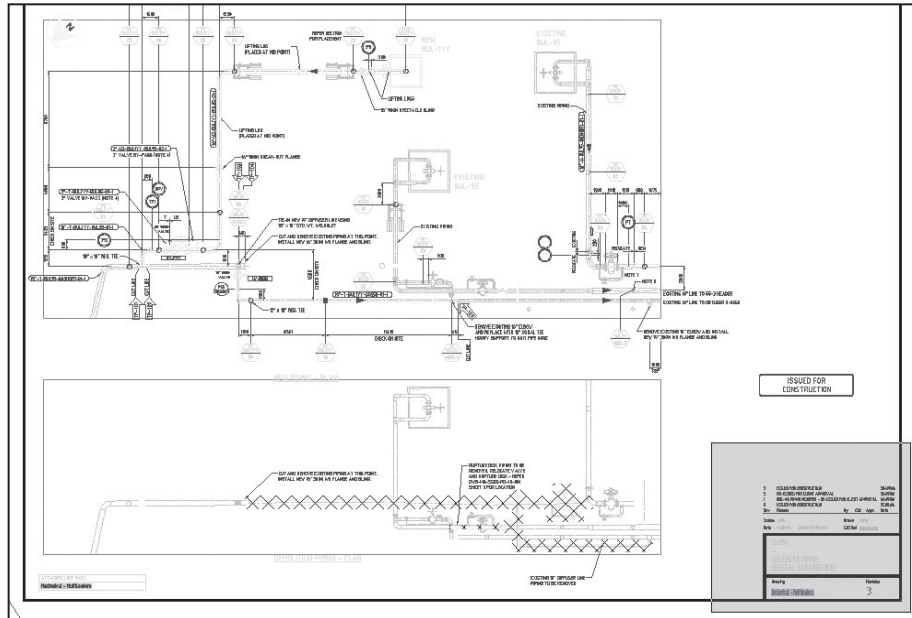
Fig. 1-5

Exiting AutoCAD

1. Click on the **Command** line, type the **CLOSE** command using upper- or lowercase letters, and press **ENTER**.

When you press **ENTER**, AutoCAD asks whether you want to save your changes.

2. Pick the **No** button.

3. In the upper right corner of the AutoCAD window, pick the small box containing an **x**.

NOTE

It is very important to exit AutoCAD at the end of each chapter. The appearance of some of the buttons may change as you work, depending on your selections. Exiting AutoCAD resets the buttons to their default settings.

Chapter 1 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What are five examples of products or industries that use AutoCAD?

2. How do you start AutoCAD?

3. What is the process for previewing AutoCAD drawing files prior to opening one?

4. How is it possible to view small details on a complex AutoCAD drawing?

5. How do you exit AutoCAD?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Explore the concept behind screen resolution. What is the impact of screen resolution? How can you change your computer's resolution?

2. In AutoCAD's **Select File** dialog box, explore a second method for previewing thumbnails of AutoCAD drawing files.

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Start AutoCAD.

2. Preview the drawing files located in AutoCAD's **Sample** folder.

3. Open the drawing file named **Mechanical - Text and Tables.dwg** located in the **Mechanical Sample** subfolder in the **Sample** folder. Pick the **Model** tab. Then pick the **Layout** tab labeled **SK-M-DR-001-RA**. Pick the **Quick View Layouts** button to the right of the **Layout** button and select **Model**.

Review & Activities Chapter 1

4. Open the drawing file named **db_samp.dwg** in the **Database Connectivity** subfolder. Pick the **Zoom** button on the status bar and make a window to take a closer look at different areas of the floor plan and other details in the drawing.

5. Exit AutoCAD. Do not save your changes.

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Start AutoCAD. Open the drawing file named **chroma.dwg** in the **AutoCAD 2010/Support** folder. Then open the drawing file named **attrib.dwg** in the **AutoCAD 2010/ActiveX/ExtAttr** folder. Pick the **Quick View Drawings** button on the status bar and notice the thumbnails of both open drawings. Rest the pointing device over a drawing thumbnail and notice smaller thumbnails of the **Model** and **Layout** views of the drawing appear. Experiment with switching between drawings and between views in this way. Exit AutoCAD without saving.

2. Start AutoCAD. Using AutoCAD's preview feature, find and open each of the following files in AutoCAD's **Sample/DesignCenter** folder:

   Home - Space Planner.dwg

   Landscaping.dwg

   Plant Process.dwg

   Explore the drawings. What is the purpose of these files? Explain how knowing that these files exist can help you as you use AutoCAD in the future. Exit AutoCAD without saving.

Chapter 2

User Interface

Objectives

- Identify the parts that make up the AutoCAD user interface and describe the function of each part
- Become familiar with AutoCAD's **Ribbon** and learn its display options
- Navigate AutoCAD's **Menu Browser**
- Describe the functions of the **Quick Access** toolbar, the **InfoCenter**, the **Command Line** window, and the status bar
- Introduce the Display tab on the Options dialog box

Vocabulary

context-sensitive
crosshairs
panel
pull-down menu

AutoCAD's user interface includes many parts that are important to the efficient operation of the software. For example, the proper positioning and grouping of buttons improves the speed with which you create, edit, and dimension drawings. Productive users of AutoCAD employ the panels, toolbars, palettes, dialog boxes, and keystrokes to enter commands and interact with AutoCAD. Each of these elements is a part of the AutoCAD user interface.

In this chapter, you will be introduced to the different elements of the user interface. In later chapters, you will explore each element of the user interface further.

Overview of the User Interface

1. Start AutoCAD.

2. If the Startup dialog box appears, click the **Cancel** button.

> **NOTE**
>
> As you learned in Chapter 1, the exact configuration of the AutoCAD window may vary depending on the screen resolution and how it was left by the last person to use the software. If windows, buttons, and menus are not present as described in the steps of this chapter, that's okay. You will learn how to display and position them in this chapter and later in the book.

The AutoCAD window appears, as shown in Fig. 2-1. Notice the locations of the title bar, the Ribbon, the Command Line window, and the status bar.
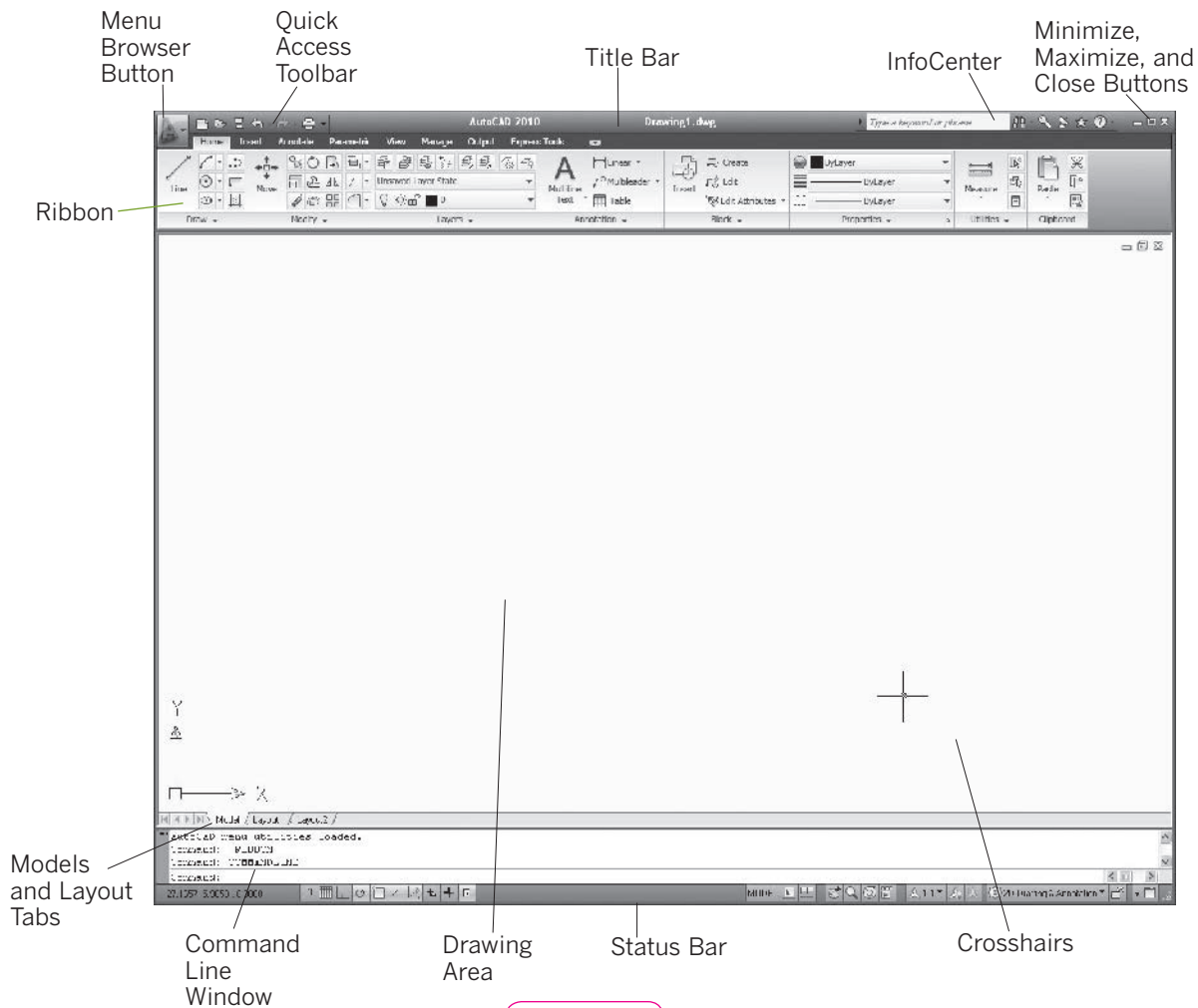
Fig. 2-1

The rest of the screen is the drawing area. As you move the pointing device in the drawing area, notice that the pointer becomes a special cursor called the **crosshairs**. You will use the crosshairs to pick points and select objects in the drawing area. When you move outside the drawing area, the normal Windows pointer appears. You will learn more about the features in the drawing area in future chapters.

The Ribbon

On the top of the AutoCAD window, just below the title bar, is a wide area called the **Ribbon**. It provides easy access to many AutoCAD tools with tabs, panels, and buttons. In AutoCAD, a **panel** is a grouping of buttons for similar commands. In the **Home** tab of the **Ribbon**, the **Draw**, **Modify**, **Layers**, **Annotation**, **Block**, **Properties**, **Utilities,** and **Clipboard** panels are displayed.

1. Move the pointer to the word **Insert** in the tab area above the **Ribbon** and single-click on the **Insert** tab.

A different collection of panels appears on the **Ribbon**.

2. Select each of the other tabs and notice the panels that appear; then return to the **Home** tab.

Notice that several of the panel title bars contain a small arrow pointing down and to the right. This means that when you select them, they will display additional buttons and commands.

3. Click inside the **Draw** panel title bar.

Three additional rows of buttons appear in the expanded **Draw** panel.

4. Move the pointer out of the expanded **Draw** panel and into the drawing area.

The expanded part of the **Draw** panel disappears.

5. Click inside the **Draw** panel title bar again.
6. Move the pointer to the lower left corner of the expanded panel and select the pin icon.

7. Move the pointer out of the expanded **Draw** panel and into the drawing area.

Notice the expanded **Draw** panel remains visible, as shown in Fig. 2-2. AutoCAD allows you to "pin" the expanded panel open. When you no longer need the panel expanded, you can unpin it.

8. Click on the pin icon and move the pointer out of the expanded **Draw** panel.

The expanded part of the **Draw** panel disappears.
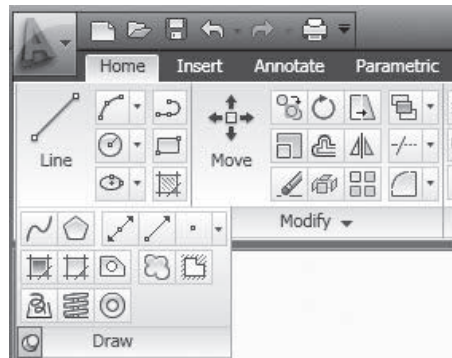
Fig. 2-2

To the right of the **Ribbon** tabs is a button with a down-facing arrow. This is the **Collapse** button.

9. Select the **Collapse** button.

The **Ribbon** is minimized to the panel titles only. The buttons are now hidden.

10. Move the pointer over the **Layers** panel title.

The buttons for the **Layers** panel appear.

11. Select the **Collapse** button again.

The **Ribbon** is further minimized to display only the tabs.

12. Move the pointer over the **Home** tab and single click.

The full **Ribbon** appears, with buttons and panel titles.

13. Move the pointer off of the **Ribbon** into the drawing area.

The **Ribbon** disappears.

14. Select the **Collapse** button once again to return to the **Show Full Ribbon** display mode.

The Menu Browser

Now direct your attention to the big red "A" in the upper left corner of the AutoCAD window. This large button is called the **Menu Browser** button.

   1. Single-click on the **Menu Browser** button.

A pull-down menu appears, as shown in Fig. 2-3. A **pull-down menu** is a vertical list of commands that appears when a button or menu item is selected. This pull-down menu is called the **Menu Browser**. The **Menu Browser** is **context-sensitive**, which means it changes depending on what command the user has selected. In this case, since we have not opened a file, AutoCAD displays options for opening, saving, and other file management commands.
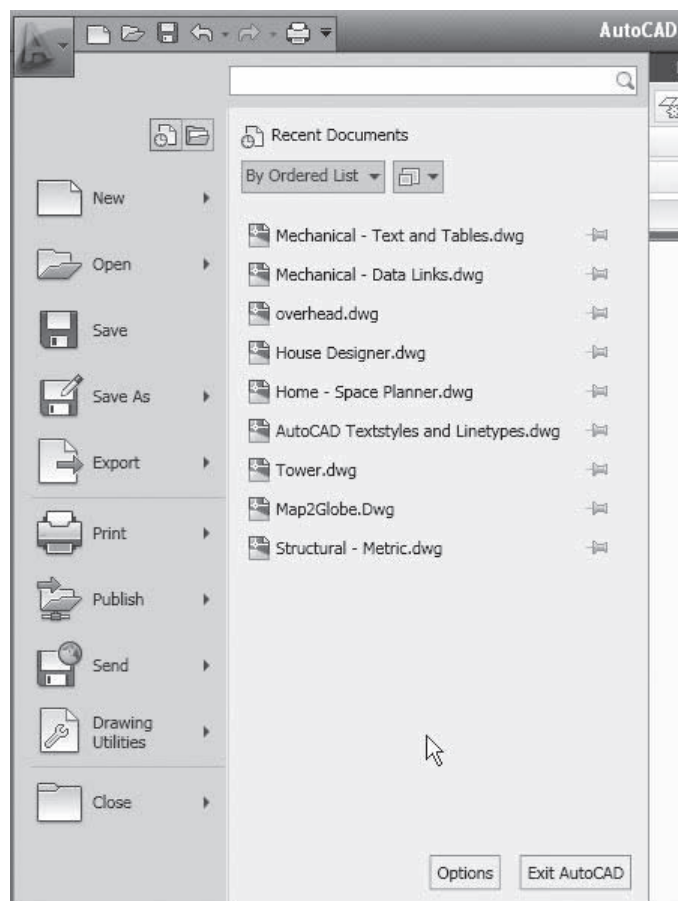
Fig. 2-3

2. Move and rest the pointer over each of the menu items in the **Menu Browser**, and notice the different groupings of commands that appear to the right.

3. Move the pointer into the drawing area and single-click to close the **Menu Browser**.

Other Features

The Title Bar

In addition to housing the **Menu Browser** button, the title bar contains the **Quick Access** toolbar and the **InfoCenter**. You were introduced to the **Quick Access** toolbar in Chapter 1. It contains buttons for the most common functions: New file, Open file, Save, Undo, Redo, and Print. The InfoCenter allows you to search for help by typing into the search field.

Command Line Window

Notice the **Command Line** window near the bottom of the screen. As you learned in Chapter 1, you can enter AutoCAD commands at the **Command** line. However, in many cases, it is faster and easier to use buttons on the **Ribbon** and on toolbars, or to use dynamic input. You will learn more about the **Command** line and dynamic input in Chapter 4.

Status Bar

The **Status Bar** at the bottom of the screen shows the coordinates of the screen crosshairs, the status of various AutoCAD modes and settings, the **Quick View** buttons, drawing annotation tools, and several other important commands. The more you work with AutoCAD, the more you will appreciate the availability of the buttons on the status bar.

Display Options

1. Single-click the red "A" to display the **Menu Browser** and select the **Options** button near the bottom of the **Menu Browser**.
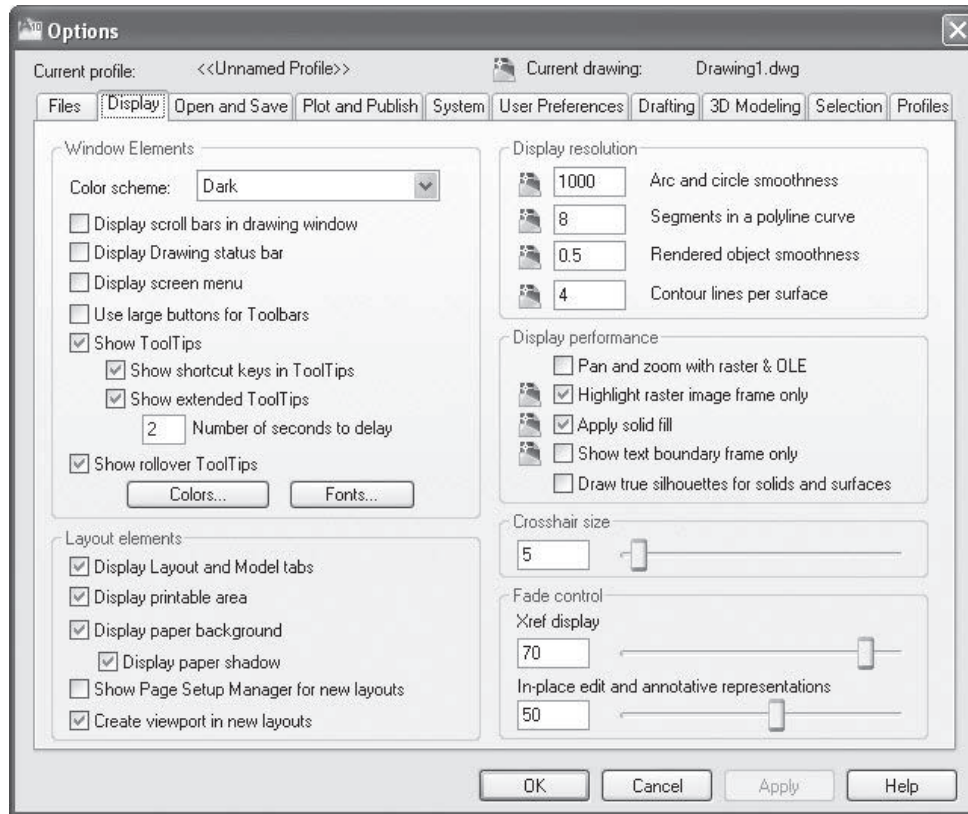
Fig. 2-4

The **Options** dialog box appears.

   2. Select the **Display** tab in the **Options** dialog box.

The **Display** tab is shown in Fig. 2-4. In this dialog box you can choose to show, hide, and resize the different elements of the user interface. For example, you can choose to display or hide scroll bars. Another useful tool in the **Display** tab is the **Colors...** button, where you can change the color of many elements of the user interface. Some AutoCAD users prefer a dark drawing area over a light one, or choose to personalize parts of the user interface with color.

   3. Pick the **Cancel** button to close the **Options** dialog box.

   4. Exit AutoCAD without saving.

Chapter 2   Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. How do you access the **Menu Browser**?
2. Describe how to display all of the buttons of the **Draw** panel on the **Ribbon**. How can you keep them displayed while you work elsewhere in the drawing?
3. What is the quickest way to open a file in AutoCAD?
4. How do you hide all the buttons on the **Ribbon**?
5. Write the names of items A through J in Fig. 2-5.
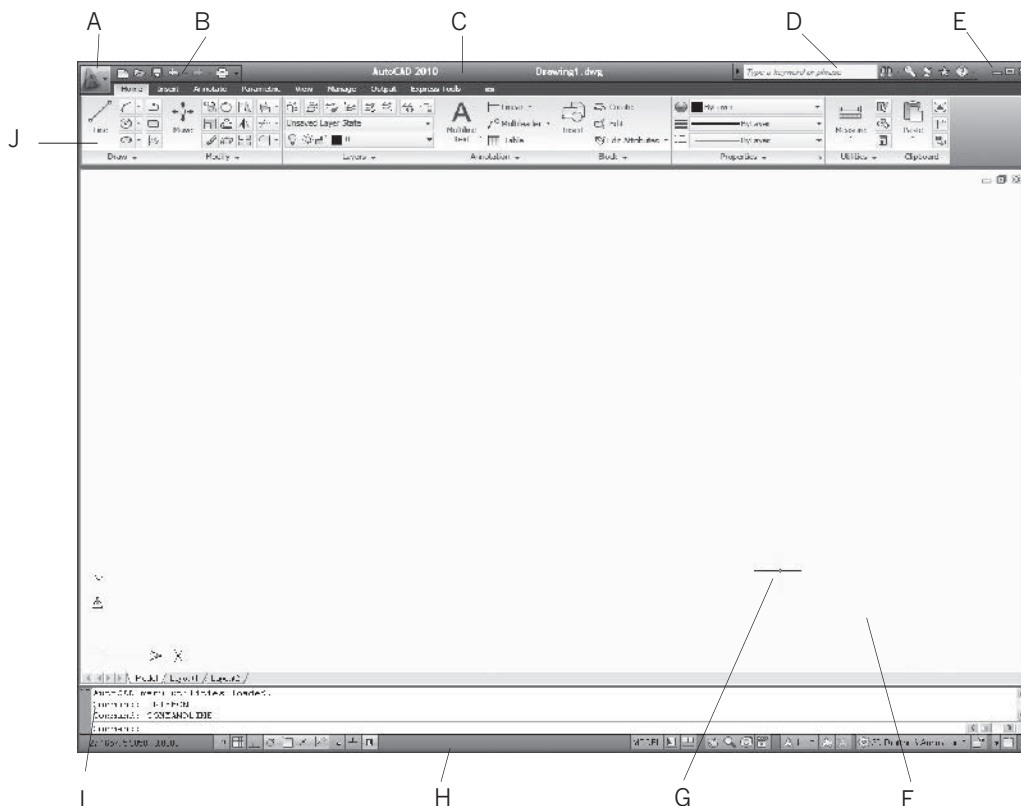
Fig. 2-5

Continued

Chapter 2 **Review & Activities**

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. AutoCAD provides ways to change the appearance and colors of the AutoCAD window. For example, if you prefer to work on a blue background, you can change the drawing area to blue. Find out how to customize AutoCAD's appearance and colors. Then write a short paragraph explaining how to customize the display. Tell why it might sometimes be necessary to do so.

2. The AutoCAD window includes two sets of Windows-standard **Maximize**, **Minimize**, and **Close** buttons. Explain why there are two sets. What is the function of each set?

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Navigate to AutoCAD's **Sample/Mechanical Sample** folder and open and view each of the following drawings. Where possible, use the **Zoom** button to look closely at the drawings. Close each of the drawings when you are finished.

   Mechanical - Data Links.dwg

   Mechanical - Multileaders.dwg

   Mechanical - Text and Tables.dwg

2. Arrange the panels of the **Ribbon** so that your AutoCAD screen looks similar to the one in Fig. 2-6.
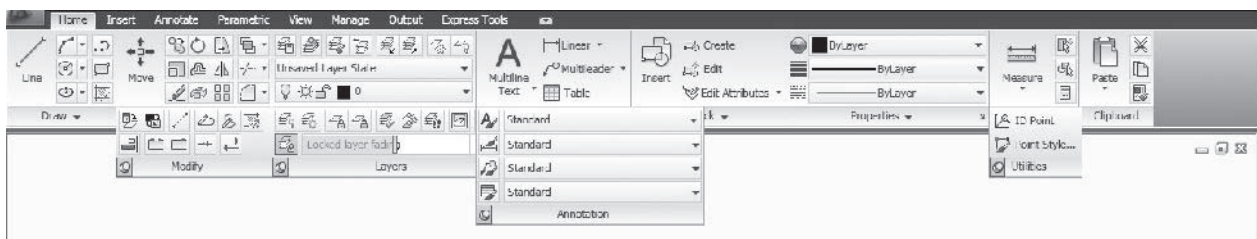
Fig. 2-6

Review & Activities Chapter 2

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Open the file **db_samp.dwg**, which is located in the **Sample/Database Connectivity** folder. Use the **Zoom Window** button to zoom in on a small area near the center of the drawing. Notice that scroll bars are not available in the default window display. Click on the **Menu Browser** button and select the **Options** button. In the **Options** dialog box, pick the **Display** tab. Then check the box next to **Display scroll bars in drawing window** and pick **OK**. The scroll bars are now displayed. Use the horizontal and vertical scroll bars to navigate the drawing. Pick and drag the movable box within the scroll bars to move quickly around the drawing. Experiment to find other ways to navigate. What other ways does AutoCAD provide? Close the drawing without saving.

2. The following files are located in AutoCAD's **Sample/Dynamic Blocks** folder: **Architectural - Metric.dwg, Civil - Metric.dwg,** and **Multileader Tools.dwg**. Open all three drawings. Then click on the **View** tab of the **Ribbon** and focus on the **Windows** panel. Experiment with the following buttons on the **Windows** panel: **Tile Horizontally**, **Tile Vertically**, and **Cascade**. These are some of the ways you can arrange the AutoCAD window when you have more than one drawing open. Describe a situation where it would be preferable to cascade drawings. When would tiling be preferable? When you are finished, quit AutoCAD without saving.

Chapter **3**

Workspaces, Toolbars, and Palettes

Objectives

- Become familiar with **2D Drafting & Annotation**, **3D Modeling**, and **AutoCAD Classic** workspaces
- Switch between workspaces
- Display and reorganize docked and floating toolbars
- Describe the functions of palettes

Vocabulary

cascading menu
docked toolbar
floating toolbar
Menu Bar
palette
toolbar
workspace

A **workspace** is a complete user interface, which includes the location and appearance of the drawing area and all menus, buttons, and groupings of buttons. Think of a workspace as a unique, specific arrangement of AutoCAD's features and capabilities that can be retrieved at any time. It is easy to switch between workspaces, modify workspaces, and create custom workspaces to match the specific requirements of a user or a particular task.

AutoCAD 2010 features more than one kind of workspace. This allows users to match the workspace to the type of work they are performing or their personal preference. In this chapter, you will be introduced to three workspaces. As you will see, the user interface can be quite different between one workspace and another.

Over the years, AutoCAD has developed several different methods for grouping buttons. The **Ribbon** is the most recent way to group buttons. Before the **Ribbon**, there were palettes, and before that, toolbars. You'll learn about toolbars and palettes in this chapter.

> **NOTE** Depending on how the software was left by the last person to use it, windows, toolbars, or palettes may or may not be present as described in the steps in this chapter. That's okay. You will learn how to display and position them in this chapter. For now, skip to the next step.

Workspaces

1. Start AutoCAD.

2. If the **Startup** dialog box appears, click the **Cancel** button.

The 2D Drafting & Annotation Workspace

3. Click on the **Workspace Switching** button on the right side of the status bar, as shown in Fig. 3-1.

4. If **2D Drafting & Annotation** is not selected in the **Workspaces** menu, select it now.
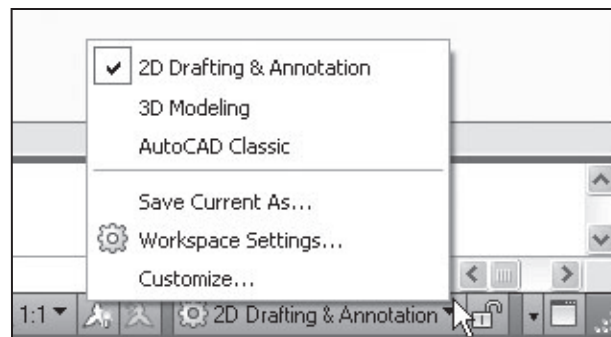
AutoCAD displays the **2D Drafting & Annotation** workspace. This is AutoCAD's default workspace. It is probably the workspace you have been using through the first two chapters of the book. Fig. 2-1 on page 11 describes the parts that make up the **2D Drafting & Annotation** workspace.

The 3D Modeling Workspace

5. On the status bar, pick the **Workspace Switching** button and select the **3D Modeling** workspace, as shown in Fig. 3-2.

Menu
Browser
Button

Quick
Access
Toolbar

Title Bar

InfoCenter

Maximize,
Minimize, and
Close Buttons

Ribbon

Drawing
Area

Crosshairs

Command
Line
Window

Tool
Palettes
Window

Status Bar
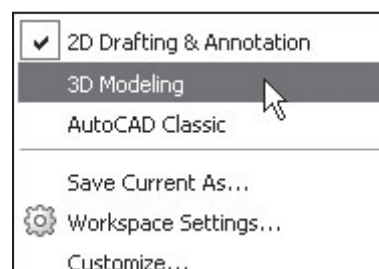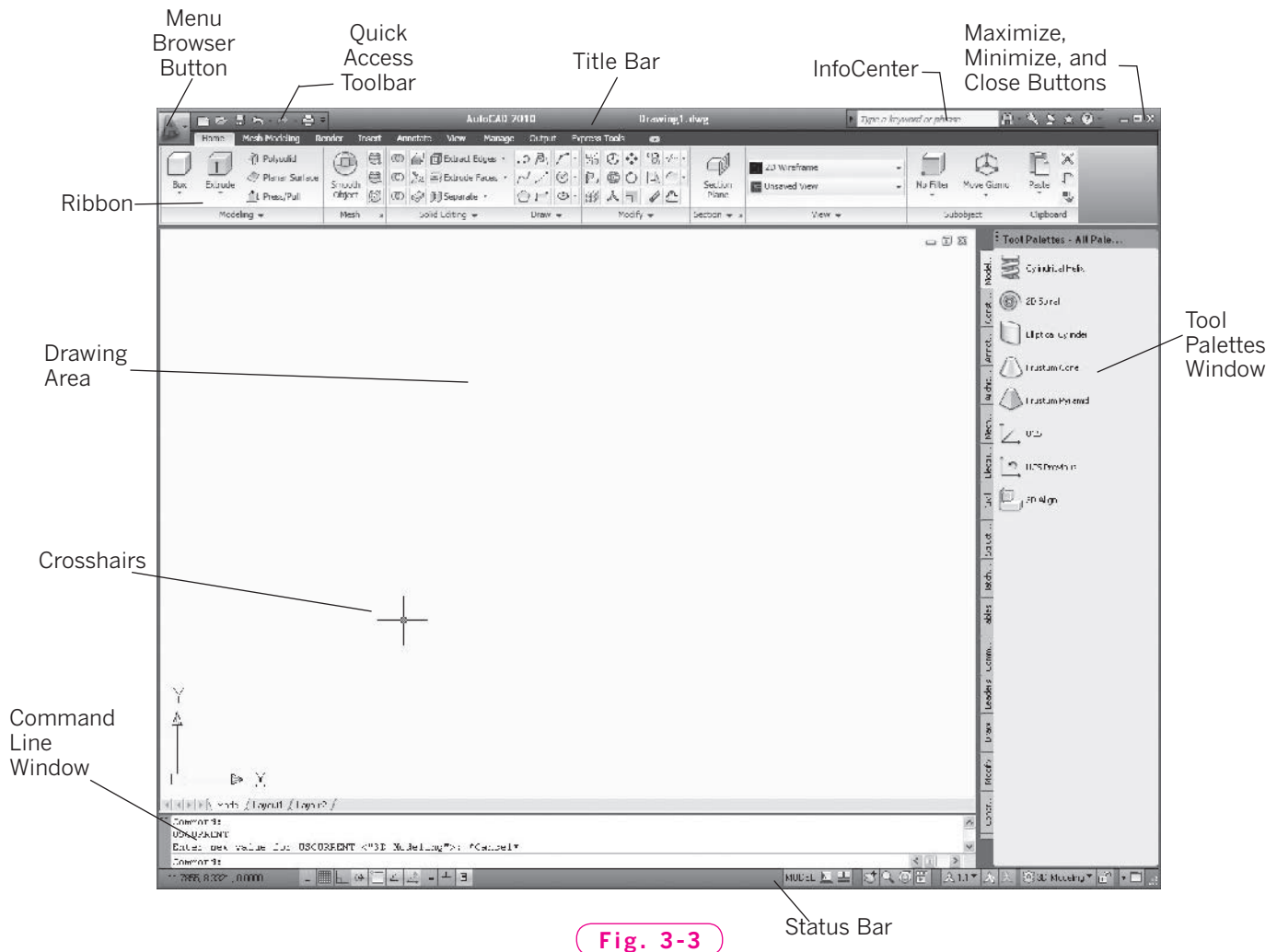
Fig. 3-3

AutoCAD displays the **3D Modeling** workspace. Fig. 3-3 describes the parts that make up the **3D Modeling** workspace.

Notice the differences between the workspaces. The **3D Modeling** workspace contains the **Ribbon** along the top of the screen, but the tabs and panels on the **Ribbon** are different. It also includes the **Tool Palettes** window along the right side of the screen.

The AutoCAD Classic Workspace

6. On the status bar, pick the **Workspace Switching** button and select the **AutoCAD Classic** workspace, as shown in Fig. 3-4.

AutoCAD displays the **AutoCAD Classic** workspace. Fig. 3-5 describes the parts that make up the **AutoCAD Classic** workspace.

2D Drafting & Annotation

✓ 3D Modeling

AutoCAD Classic

Save Current As...

Workspace Settings...

Customize...
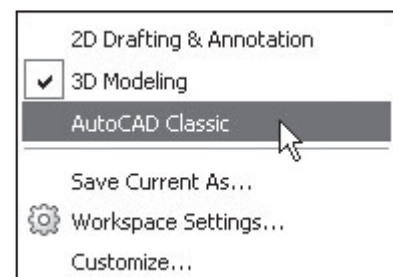
Fig. 3-4
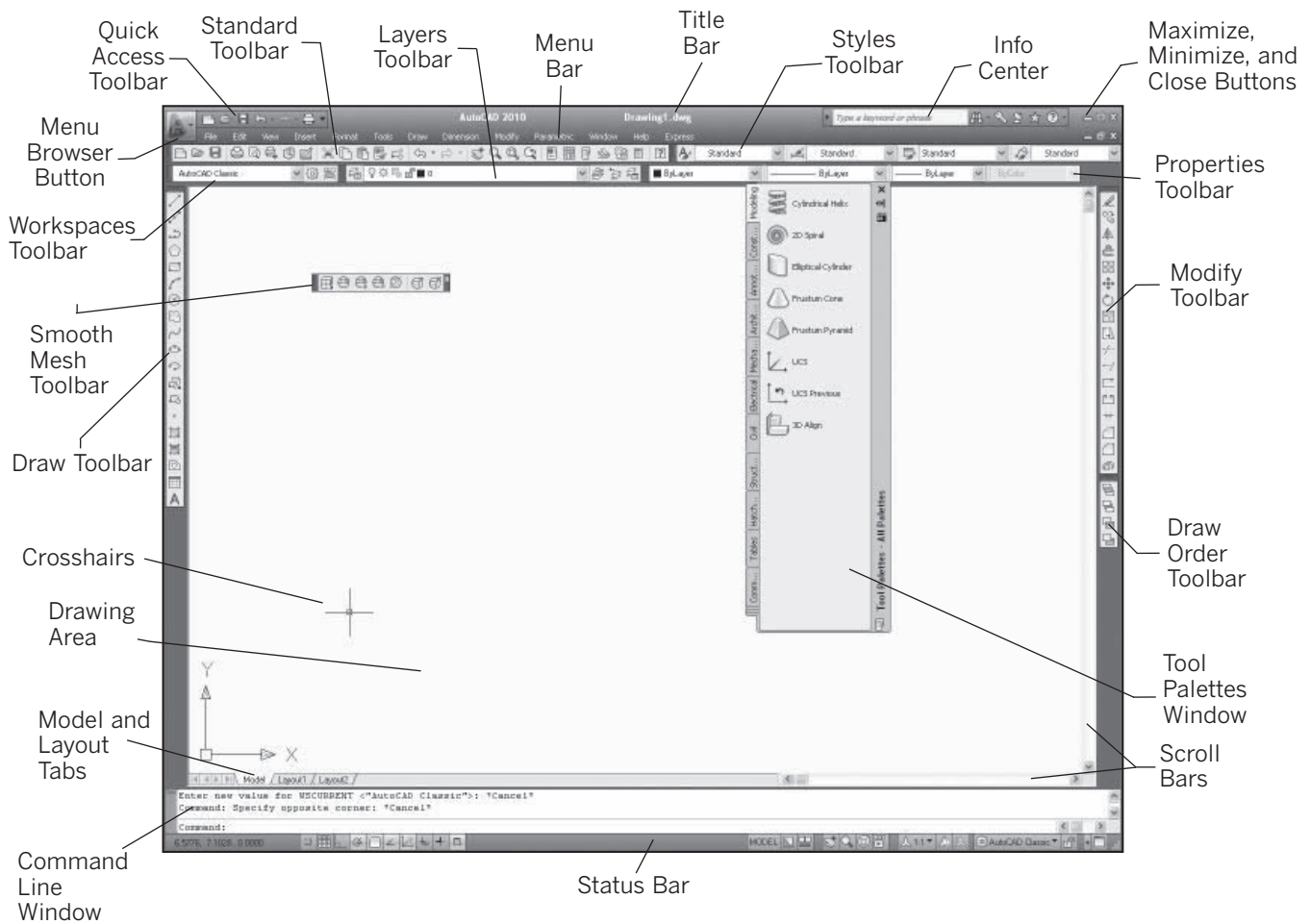
Fig. 3-5

Again, notice the differences between the workspaces. The **AutoCAD Classic** workspace contains a row of menu items below the title bar, called the **Menu Bar**, and two rows of docked toolbars below that, as well as docked toolbars on both sides of the drawing area. The **Tool Palettes** window is floating inside the drawing area.

It is possible to modify existing workspaces, rename workspaces, and create new workspaces. You can display the **Menu Bar** and the scroll bars in any workspace, even though they are hidden in the default settings of the **2D Drafting & Annotation** and **3D Modeling** workspaces. You can rearrange the elements on the screen to your liking, then save that arrangement as a workspace. As you learn AutoCAD, keep this in mind.

Toolbars

A **toolbar** is a thin strip that contains a collection of buttons. The **AutoCAD Classic** workspace includes eight toolbars along the top, left, and right of the AutoCAD window. Under the menu bar (**File**, **Edit**, **View**, etc.), you should see five toolbars in two rows docked across the top of the screen. A **docked toolbar** is one that appears to be part of the AutoCAD screen's border, outside of the drawing area.

Floating Toolbars

Let's focus on displaying and positioning toolbars. A toolbar that is not docked is called a **floating toolbar**.

1. Rest the pointer on any one of the docked toolbars and right-click.

This displays a list of toolbars that AutoCAD makes available to you, as shown in Fig. 3-6. By right-clicking on any docked or floating toolbar on the screen, you can display this list of toolbars. The ones that are checked are the toolbars that are present on the screen.

To display a toolbar from the list, click its name.

2. Click the word **Lights** to display the **Lights** toolbar.

3. Close the floating **Lights** toolbar by picking the **x** (Close button) in the upper right corner of the toolbar.

4. If any other floating toolbars are present, close them by picking the **x** in the upper right corner of each toolbar.

It is easy to move toolbars to a new location. As you work with AutoCAD, you should move floating toolbars to a location that does not interfere with the drawing.

5. Open the **Render** toolbar.

6. Move the **Render** toolbar by clicking and dragging the left or right border of the toolbar.

3D Navigation
CAD Standards
Camera Adjustment
Dimension
Dimensional Constraints
✔ Draw
✔ Draw Order
Find Text
Geometric Constraint
Inquiry
Insert
✔ Layers
Layers II
Layouts
Lights
Mapping
Measurement Tools
Modeling
✔ Modify
Modify II
Multileader
Object Snap
Orbit
Parametric
✔ Properties
Refedit
Reference
Render
✔ Smooth Mesh
Smooth Mesh Primitives
Solid Editing
✔ Standard
Standard Annotation
✔ Styles
Text
UCS
UCS II
View
Viewports
Visual Styles
Walk and Fly
Web
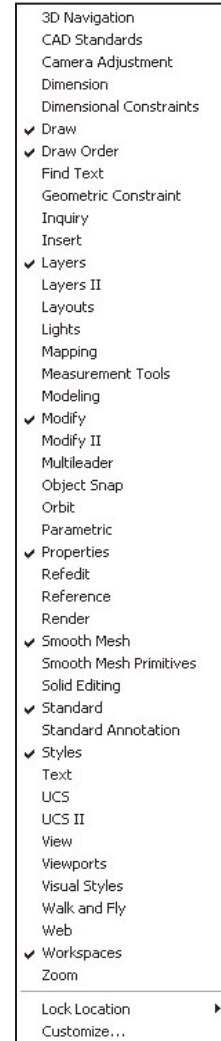✔ Workspaces
Zoom

Lock Location ▶
Customize…

Fig. 3-6

NOTE AutoCAD allows users to lock toolbars in place. If a toolbar does not move when you try to drag it, right-click any open toolbar, go to the bottom of the pop-up menu, and rest the cursor on **Lock Location**. From the cascading menu, pick the checkmark next to **Floating Toolbars/Panels** to unlock the toolbar's position. A **cascading menu** is a second menu that appears when you point at a menu item containing a small arrow pointing to the right.

Changing a Toolbar's Shape

AutoCAD makes it easy to change the shape of toolbars. You might want to change the shape of a toolbar, for example, to make it fit better in the area of the drawing in which you are working.

1. Move the pointer to the bottom edge of the **Render** toolbar, slowly and carefully positioning it until a double arrow appears. With the double arrow visible, click and drag downward until the toolbar changes to a vertical shape and then release the button.

2. Move the pointer to the right edge of the **Render** toolbar, positioning it until a double arrow appears. With the double arrow visible, click and drag to the right until the toolbar changes back to a horizontal shape.

3. Close the **Render** toolbar.

Docked Toolbars

The AutoCAD Classic workspace has many docked toolbars. The **Standard** and **Styles** toolbars fill the first row of toolbars at the top of the screen. The second row includes the **Workspaces**, **Layers**, and **Properties** toolbars. The **Draw** toolbar is docked vertically on the left side of the screen. Finally, the **Modify** and **Draw Order** toolbars are docked on the right side of the screen.

1. Move the pointer to the **Standard** toolbar and click and drag the outer left edge (border) of the toolbar, moving it into the drawing area. The toolbar is undocked and becomes a floating toolbar.

2. Repeat Step 1 for the remaining four toolbars that are docked along the top of the AutoCAD window.

3. Close all five of these floating toolbars.

4. Right-click in any of the three remaining toolbars to display the list of toolbars and select **Layers** to display the **Layers** toolbar.

5. Repeat Step 4 to display the **Workspaces**, **Properties**, **Standard**, and **Styles** toolbars.

6. Click and drag the **Standard** toolbar and carefully dock it under the menu bar (**File**, **Edit**, **View**, etc.).

The toolbar locks into place.

7. Click and drag the other four toolbars and carefully dock them in place. Refer to Fig. 3-5 on page 23 if you are unsure where they belong.

8. If the **Draw**, **Modify**, and **Draw Order** toolbars are not present and docked in their default locations, position them now as shown in Fig. 3-5.

You should now see eight docked toolbars on the screen.

> **NOTE**
>
> This textbook assumes that these eight toolbars are present in the **AutoCAD Classic** workspace in a configuration similar to the one in Fig. 3-5. Instructions are written to work with AutoCAD's default values at installation. It assumes that the screen color, crosshairs, and other elements have not been customized.

Palettes

A **palette** is a special toolbar that contains a large grouping of buttons, tools, or information. Palettes eliminate the need to display a lot of toolbars and reduce the clutter on your computer screen. Like toolbars, palettes can be docked or floating in the drawing area.

1. In the **Workspaces** toolbar, pick the down arrow to the right of **AutoCAD Classic** and click **3D Modeling**, as shown in Fig. 3-7.

In the **3D Modeling** workspace, the **Tool Palettes** window is docked on the right side of the screen. It provides access to many useful tools for creating objects and drawings in AutoCAD. It is a long, vertically oriented window with tabs along its left edge. Fig. 3-8 shows the Tool Palettes window.

2. If the **Tool Palettes** window is not present on the screen, select the **View** tab above the Ribbon, then select the **Tool Palettes** button on the **Palettes** panel.

3. On the status bar, pick the **Workspace Switching** button and select the **AutoCAD Classic** workspace.
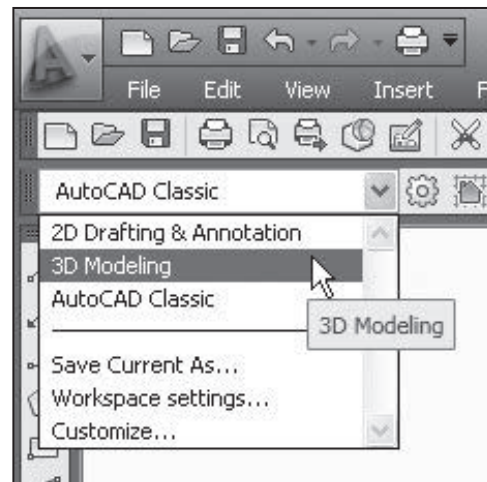
Fig. 3-7

In the **AutoCAD Classic** workspace, the **Tool Palettes** window and the **Smooth Mesh** toolbar may be floating.

4. If the **Smooth Mesh** toolbar is open, close it.

5. If the **Tool Palettes** window is open, close it by picking the **x** located in the upper right corner.

6. From the **Tools** pull-down menu, move the pointer to the **Palettes** option, and notice the different options available in the cascading menu, as shown in Fig. 3-9.

As you can see, the **Ribbon** is also a palette.

7. Select **Tool Palettes** from the cascading menu.

This causes the **Tool Palettes** window to display. Pressing the **CTRL** and **3** keys together toggles the display of the **Tool Palettes** window on and off.

8. Click on the left border of the **Command Line** window and drag it into the drawing area.

As you can see, the **Command Line** window behaves like a palette. Pressing the **CTRL** and **9** keys together toggles the display of the **Command Line** window on and off.

9. Return the **Command Line** window to its docked position by dragging it to the bottom of the AutoCAD screen.

Auto-Hide

AutoCAD's **Auto-hide** feature increases the size of the drawing area by hiding a palette, except for its title bar, until you place the cursor over the title bar.

Fig. 3-8

Fig. 3-9

1. Rest the pointer over the vertical title bar of the **Tool Palettes** window and right-click.
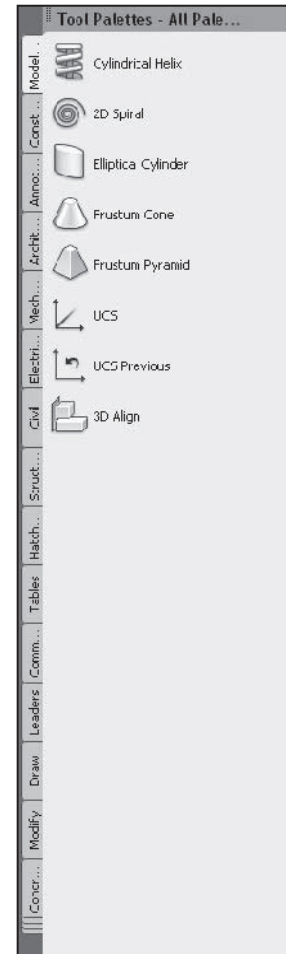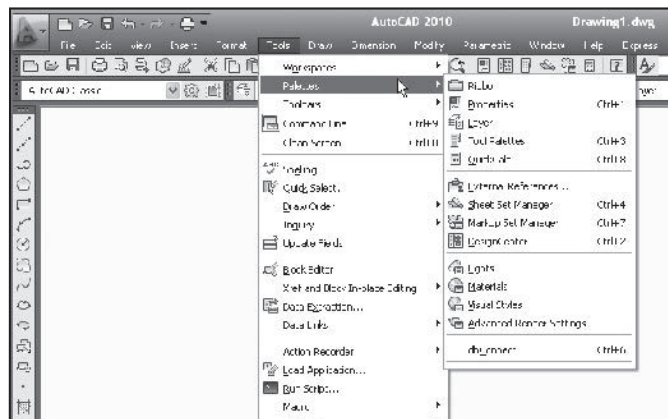
This displays a menu of options.

2. Select **Auto-hide**.

3. Move the pointer away from the **Tool Palettes** window.

The window disappears, except for its title bar.

4. Move the pointer to the title bar.

The window reappears.

5. Move the pointer away from the window and its title bar.

The window disappears. As you can see, **Auto-hide** helps conserve screen space, which is especially helpful when the AutoCAD window becomes busy.

6. Rest the pointer on the title bar of the **Tool Palettes** window, right-click, and uncheck **Auto-hide**.

Transparency

Transparency is another feature that helps free up space on the screen.

1. Rest the pointer inside the **Tool Palettes** vertical title bar, right-click, and select the **Transparency…** item.

2. Click and drag the **General** slider in the dialog box to the far left, click and drag the **Rollover** slider to the middle position, and pick the **OK** button.

This causes the **Tool Palettes** window to become nearly transparent. When you move the pointer over it, it becomes semi-transparent.

3. From the **Quick Access** toolbar, pick the **Open** button and open the file named **db_samp.dwg** in the **Sample/Database Connectivity** folder.

You should be able to see the drawing through the **Tool Palettes** window.

4. Right-click inside the **Tool Palettes** title bar, select **Transparency…**, move the **General** slider to the middle position, and pick **OK**.

The **Tool Palettes** window becomes less transparent.

5. Right-click inside the **Tool Palettes** title bar, select **Transparency…**, move the slider to the far right, and pick **OK**.

The **Tool Palettes** window is no longer transparent. Auto-hide and transparency work with all palettes and with the **Command Line** window.

6. Exit AutoCAD without saving.

Chapter 3 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Define workspace in your own words.
2. Briefly describe the differences between the **2D Drafting & Annotation**, **3D Modeling**, and **AutoCAD Classic** workspaces.
3. What is a cascading menu?
4. Describe how to display a toolbar that is not currently displayed on the screen.
5. How do you move a toolbar?
6. How do you make a floating toolbar disappear?
7. Explain how to make the **Tool Palettes** window semi-transparent.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. AutoCAD allows users to modify, customize, and create new workspaces. Name two or three different tasks that might best be performed in different workspaces. What would you name each workspace?
2. Workspaces in AutoCAD are similar to physical workspaces in your home or office. What "tools" do you use on the dining room table for a large meal? How would those tools change if you were to use the dining room table to do homework or pay bills?

● APPLYING AUTOCAD SKILLS

1. Starting in the **3D Modeling** workspace, rearrange the AutoCAD window so that it is similar to the one in Fig. 3-10 on page 30.
2. Starting in the **2D Drafting & Annotation** workspace, rearrange the AutoCAD window so that it is similar to the one in Fig. 3-11 on page 30.

Continued

Chapter 3 **Review & Activities**
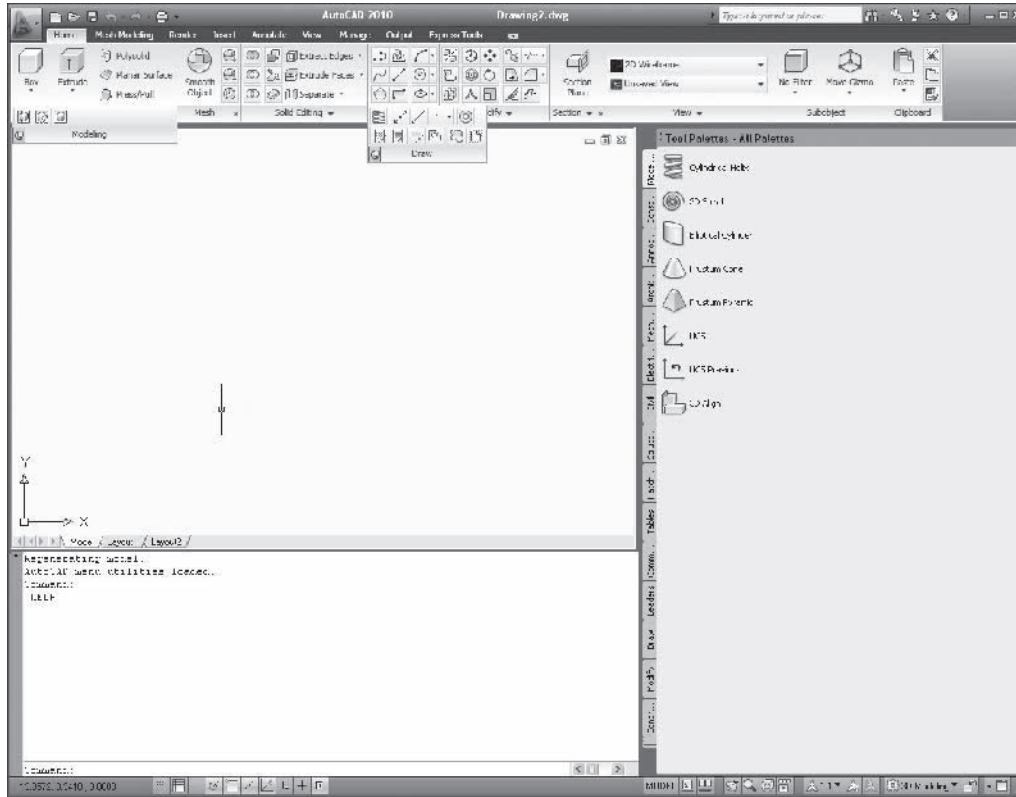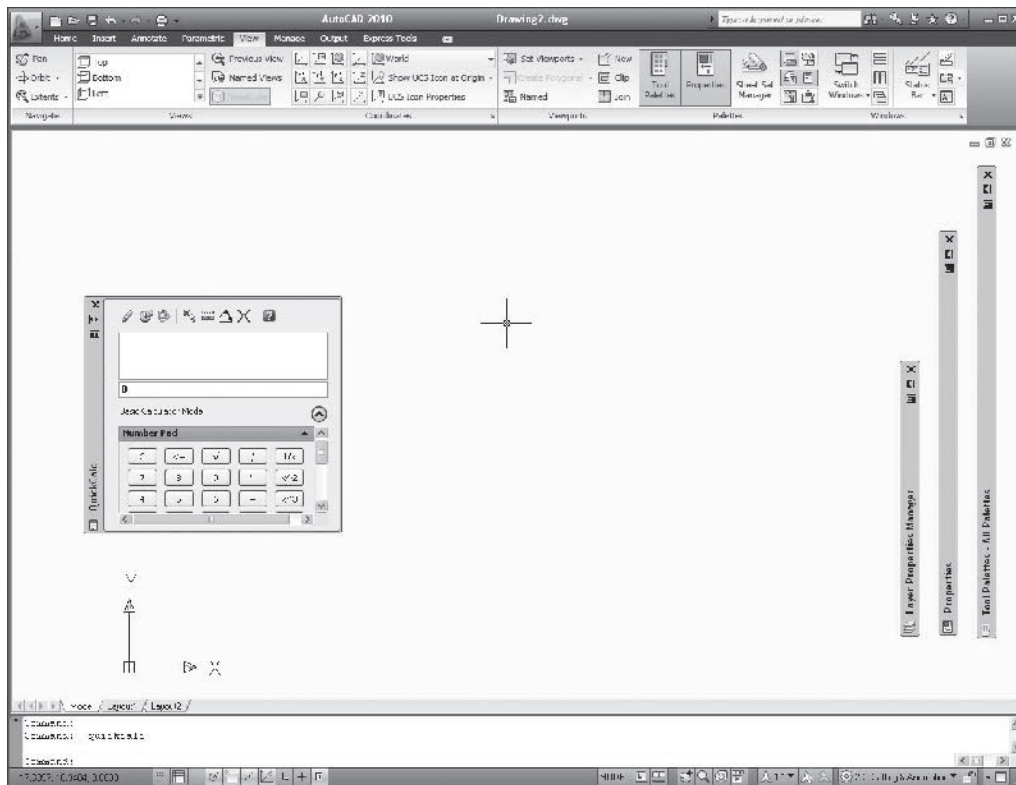
Review & Activities Chapter 3

3. Starting in the **AutoCAD Classic** workspace, rearrange the AutoCAD window so that it is similar to the one in Fig. 3-12.
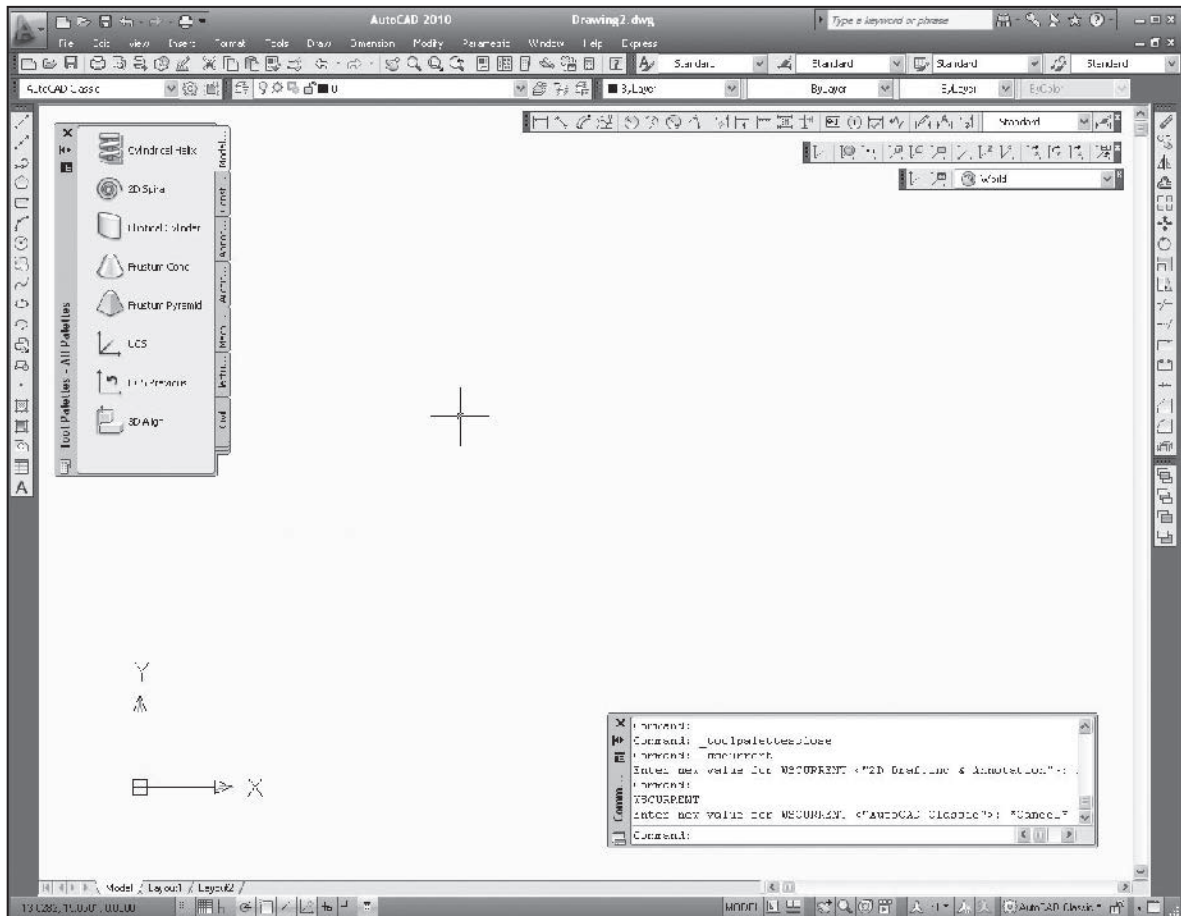
Fig. 3-12

4. Starting in the **AutoCAD Classic** workspace, right-click in any toolbar to display the toolbar list and uncheck a toolbar. Repeat until all toolbars are unchecked. Notice that the drawing area has increased, but you have lost the convenience of toolbar command selection. Click the **Tools** pull-down menu at the top of the screen, move the pointer over **Workspaces**, and single-click **AutoCAD Classic**. This restores the toolbars to their default positions in the **AutoCAD Classic** workspace.

Chapter 3 **Review & Activities**

● USING PROBLEM-SOLVING SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Start AutoCAD. Switch to the **3D Modeling** workspace. Modify the workspace by stretching the **Command Line** window upward and undocking and stretching the **Tool Palettes** window to the left. The drawing area should cover about a quarter of the screen. Pick the **Workspace Switching** button on the status bar, select **Save Current As...**, and save the workspace as **Workspace1**.

2. Delete the workspace you created in problem 1 by picking the **Workspace Switching** button and selecting the **Customize...** option. This opens the **Customize User Interface** dialog box. Right click on **Workspace1** and pick **Delete**. Close the dialog box.

3. Starting in the **AutoCAD Classic** workspace, open all of the toolbars provided in AutoCAD. Dock them along the sides, top, and bottom of the drawing area. Using the tooltips, investigate the buttons that make up the toolbars. (Tooltips are the words that appear when the pointer rests on the button.) What two problems might you encounter if you left all the toolbars open on the screen? When you are finished, select **AutoCAD Classic** in the **Workspaces** toolbar at the top left of the screen, and notice that the additional toolbars have all been closed automatically.

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35

Chapter 4

Command Entry

Objectives

- Create, save, and open an AutoCAD drawing file
- Prepare AutoCAD for drawing in 2D
- Enter commands using the keyboard and buttons
- Apply shortcut methods to enter commands
- Reenter commands

Vocabulary

command aliases
dynamic input
extended tooltips
rubber-band effect
shortcut menu
tooltips

AutoCAD offers several methods of entering and repeating commands. When entering them frequently, as you will practice in this chapter, you will want to use methods that minimize the time it takes. Command aliases and other shortcuts permit you to enter commands with as little effort as possible. If you take advantage of these time-saving features of AutoCAD, you will require less time to complete a drawing. Consequently, you become more attractive to prospective employers looking for highly productive individuals.

In the steps that follow, you will practice entering commands by doing some basic operations such as creating a file and drawing lines. You will apply the concepts you learn in this chapter nearly every time you use AutoCAD.

Creating and Saving a Drawing File

1. Start AutoCAD.

The AutoCAD window appears.

It is important that you start (or restart) AutoCAD at the beginning of each chapter, unless instructed otherwise. This ensures that the buttons on the **Ribbon** and in the toolbars are set to their default settings.

We will begin this chapter in the **2D Drafting & Annotation** workspace.

2. Click on the **Workspace Switching** button on the status bar, then select **2D Drafting & Annotation** from the menu.

> **NOTE**
>
> It is important that you select the workspace as instructed, even if that workspace already appears to be displayed. This resets the screen arrangement to the selected workspace.

QUICK ACCESS

3. Pick the **Save** button from the **Quick Access** toolbar.

The **Save Drawing As** dialog box appears, as shown in Fig. 4-1. If you use other Windows software, the dialog box should look familiar to you.

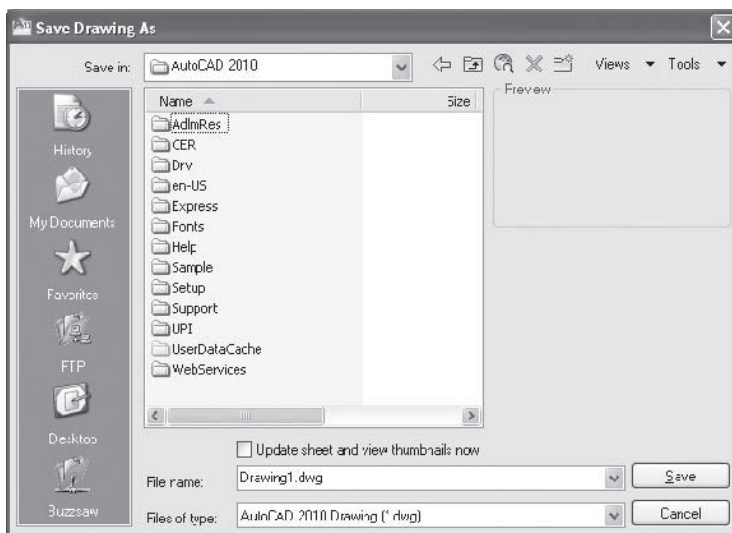**4.** Navigate inside the **Save Drawing As** dialog box and find the **AutoCAD 2010** folder.

Fig. 4-1

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37

5. Pick the **Create New Folder** button (located in the upper right area of the dialog box).

This creates a folder named **New Folder**.

6. Right-click the new folder, select **Rename** from the menu, type your first and last name, and press **ENTER**.

7. Double-click this new folder to make it current.

8. In the **File name** text box, highlight the name of the drawing file (e.g., **Drawing1.dwg**) by double-clicking it, and enter **stuff**.

9. Pick the **Save** button in the dialog box to create a file named **stuff.dwg**.

> **NOTE**
>
> If you do not see the file extension (**dwg**) after the names of drawing files, extension display may be turned off on your computer. Your work with AutoCAD will be much easier if you turn on the extension display.

Notice that the drawing file name now appears in the title bar of the AutoCAD window.

10. Pick the **x** in the upper right corner of the drawing area to close the stuff.dwg file. Do not pick the **x** in the upper right corner of the AutoCAD screen. This will close AutoCAD.

The **stuff.dwg** file disappears, and the AutoCAD screen becomes gray because no drawing file is open. Notice also that the **Ribbon**, status bar, and **Command Line** window disappear. An abbreviated version of the **Quick Access** toolbar and a smaller version of the **Menu Browser** button are visible.

Opening a Drawing File

If AutoCAD is already open, you can open a file easily by using the **Open** button.

1. Pick the **Open** button on the **Quick Access** toolbar.

The **Select File** dialog box appears. AutoCAD defaults to the folder you named after yourself.

QUICK ACCESS

2. Double-click **stuff.dwg** in the selection window.

You have reopened the **stuff.dwg** file. Currently, the drawing area is blank.

Entering Commands

AutoCAD allows you to draw two-dimensional (2D) and three-dimensional (3D) objects. For the next several chapters, you will learn how to enter commands, select objects, and navigate the user interface in the 2D drawing environment.

For the purpose of learning to enter commands, we will use the **LINE** command. You will learn more about using various AutoCAD commands in the chapters that follow.

Using Dynamic Input

Entering commands at the **Command** line may force your attention away from the drawing you are creating. **Dynamic input** allows you to enter commands at the cursor instead of moving the pointing device to the **Command** line. This allows you to keep your focus on the drawing area.

STATUS BAR

1. Find the **Dynamic Input** button on the status bar, and toggle it on (blue) if it is not on already.

2. With the pointer anywhere in the drawing area, type **LINE** using upper- or lowercase letters.

Notice that the command appears near the crosshairs as you type.

3. Press **ENTER**.

AutoCAD asks you to specify the first point and shows the coordinates (current position) of the crosshairs. You will learn more about coordinates in Chapter 7.

4. Pick a point in the lower left portion of the drawing area.

AutoCAD's dynamic input prompt now requests the next point. It also displays the distance and angle of the crosshairs from the last point, as shown in Fig. 4-2.
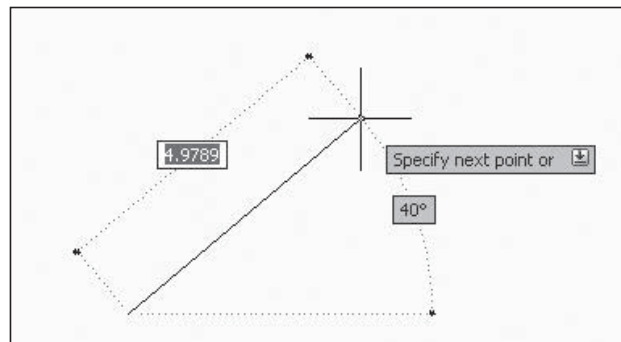

Fig. 4-2

5. Move the crosshairs on the screen and notice how the distance and angle values change dynamically.

6. Instead of picking the next point, type **5** and press the **TAB** key.

Notice that the value you typed appears in the distance box of the dynamic input. Also, a padlock appears to indicate that the value of **5** is locked into place.

7. Type **45** and press **ENTER**.

AutoCAD creates a line segment that is 5 units long at an angle of 45°.

8. Produce the second line segment by picking a point below and to the right of the previous point.

9. Produce the third line segment to create a triangle by moving the crosshairs over the first point and clicking.

> **NOTE**
> You may notice that temporary dotted lines and yellow symbols appear on the screen as you complete the triangle in Step 9. These are produced by the AutoCAD features known as object snap and AutoTrack™. You will use these features extensively in the chapters that follow. For now, you can ignore them.

10. Press **ENTER** to terminate the **LINE** command.

> **NOTE**
> If the entire triangle is not visible in the drawing area, use the mouse wheel, the scroll bars, or other methods of panning and zooming to bring the entire triangle into the drawing area. You will learn more about panning and zooming in Chapters 13 and 14.

Dynamic input can be toggled on and off by clicking the **Dynamic Input** button on the status bar. You will use dynamic input more in the chapters that follow.

Using the Command Line

In earlier versions of AutoCAD, the only way to enter a command was to enter the command name on the **Command** line. This method is still available, although other methods have been added in recent years to provide more efficiency.

STATUS BAR

1. Single-click the **Dynamic Input** button on the status bar to toggle dynamic input off.

2. Type **LINE** and press the **ENTER** key.

In the **Command Line** window, notice that AutoCAD asks you to specify the first point.

3. Pick a point anywhere in the drawing area.

AutoCAD requests the next point.

4. Pick a second point anywhere on the screen.

AutoCAD requests another point.

5. Move the crosshairs away from the last point and notice the stretching effect of the line segment.

This is known as the **rubber-band effect**.

6. Produce the second and third line segments to create a triangle, and then press **ENTER** to terminate the **LINE** command.

STATUS BAR

7. Click on the **Dynamic Input** button on the status bar to reactivate dynamic input.

NOTE You can use the **Command** line method of entry even when dynamic input is active. Simply single-click at the **Command** line before typing the command.

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41

Picking Buttons on the Ribbon

It can be faster to pick a button than to enter a command at the keyboard or from a pull-down menu.

1. Rest the pointer on the **Line** button located in the **Draw** panel of the **Ribbon**, but do not pick the button.
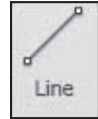
DRAW

As discussed briefly in Chapter 1, a small window with a brief explanation appears after about one second. Then, after a delay of a few seconds, a larger window with a detailed explanation appears. These windows, called **tooltips**, help you understand the purpose of the button. The larger tooltips are called **extended tooltips**. You can turn tooltips and extended tooltips on or off in the **Display** tab of the **Options** dialog box.

2. Slowly position the pointer on top of other buttons and read the information that AutoCAD displays in each tooltip and extended tooltips.

3. Pick the **Line** button and move the pointer into the drawing area.

You have just entered the **LINE** command. You can see that AutoCAD is again asking where you want the line to begin.

4. Pick a point anywhere in the drawing area; then pick a second point to form a line.

5. Press the **ENTER** key to terminate the **LINE** command.

6. Pick another button (any one of them) from the **Draw** toolbar.

7. Press the **ESC** key to cancel the last entry at the **Command** line.

HINT As you work with AutoCAD, you will be entering commands by picking buttons, selecting them from menus, or entering them at the keyboard. Occasionally you might accidentally select the wrong command or make a keyboarding error. It is easy to correct such mistakes.

If you catch a keyboarding error before you press **ENTER**…	use the backspace key to delete the incorrect character(s).
If you select the wrong button or menu item…	press the **ESC** key to clear the **Command** line.
If you key in the wrong command…	press the **ESC** key to clear the **Command** line.
If you accidentally pick a point or object on the screen…	press the **ESC** key.

Command Aliases

AutoCAD permits you to issue commands by entering just the first few characters of the command. Command abbreviations such as these are called **command aliases**.

1. Type **L** in upper- or lowercase and press **ENTER**.

This activates the **LINE** command. You can also enter other commands using aliases. Some of the more common command aliases are listed in Table 4-1.

2. Press the **ESC** key to cancel the **LINE** command.

3. Enter each of the commands listed in Table 4-1 using the alias method. For now, press **ESC** to cancel each command.

You will learn more about these commands in future chapters.

4. Reenter the **LINE** command using the command alias method and create another line.

5. Press **ENTER** to terminate the **LINE** command.

Table 4-1. Common Command Aliases

Command	Entry
ARC	A
CIRCLE	C
COPY	CO
DONUT	DO
ELLIPSE	EL
ERASE	E
LINE	L
MOVE	M
PAN	P
POLYGON	POL
SCALE	SC
ZOOM	Z

Even when you do not know a command alias, AutoCAD can help you enter the command without typing the entire command. We will use the **POLYGON** command as an example.

6. Type **PO** in upper- or lowercase letters, but do not press **ENTER**.

7. Press the **TAB** key repeatedly until the **POLYGON** command is displayed on the screen.

AutoCAD cycles through an alphabetical list of all commands that begin with the letters **PO**.

8. Press **ENTER** to enter the **POLYGON** command.

9. Press **ESC** to terminate the command.

Command aliases are defined in the AutoCAD file **acad.pgp**. You can view this file or create additional command aliases by modifying it using a text editor. The file is located in the following folder:

C:\Documents and Settings\username\Application Data\ Autodesk\AutoCAD 2010\R18.0\enu\Support

Substitute your user name for "username" in the file path above.

Reentering the Last Command

After you have entered a command, you have several choices for repeating it.

Line

DRAW

1. Enter the **LINE** command and create a line segment anywhere in the drawing area. Press **ENTER** to end the command.
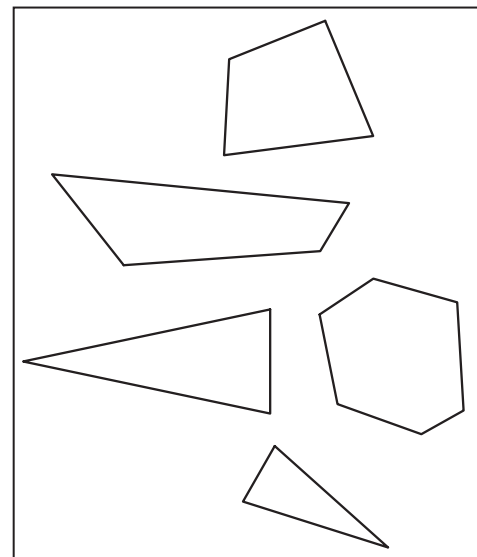2. Press the spacebar or **ENTER**.

This enters the last-entered command, in this case the **LINE** command. You will find that this saves time compared to other methods of entering the command.

3. Create a line segment and then press **ENTER** to end the **LINE** command.
4. Press the right button on the pointing device.

Right-clicking in AutoCAD displays a context-sensitive **shortcut menu**. The top item on the menu, when selected, repeats the last command you entered.

5. Pick **Repeat Line**.
6. Draw two of the polygons shown in Fig. 4-3.

The **Repeat** option is context-sensitive. This means that the software remembers the last command you used.

Fig. 4-3

HINT

After you have entered at least two line segments, the prompt changes to include the **Close** option. This option allows you to close a polygon accurately and automatically. It also terminates the **LINE** command. To choose the **Close** option, type the letter **C** and press **ENTER**. As you can see, you can use shortcuts or aliases to choose command options as well as to enter commands.

7. Draw the remaining polygons shown in Fig. 4-3 using the **LINE** command. Be sure to use one of the shortcut methods of reentering the last command.

8. Enter the **LINE** command and draw two connecting line segments, but do not terminate the command.

9. In reply to **Specify next point or [Close/Undo]**, enter **U** for **Undo**.

AutoCAD backs up one segment, undoing it so that you can recreate it.

QUICK ACCESS

10. Practice the **Undo** option with additional polygons.

11. Pick the **Save** button from the **Quick Access** toolbar to save your work.

As you work with the different methods of entering commands, you may prefer one method over another. Picking the **Line** button from the **Ribbon** may or may not be faster than entering **L** at the keyboard. Keep in mind that there is no right or wrong method of entering commands. Experienced users of AutoCAD use a combination of methods.

12. Exit AutoCAD.

Chapter 4 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Describe the purpose of the **Open** button.
2. When a file has not yet been saved for the first time, what appears when you pick either **Save** or **Save As...** from the **Menu Browser**?
3. What appears when you rest the pointer on a button on the **Ribbon?**
4. What is the fastest and simplest method of reentering the previously entered command?
5. How can you enter commands such as **LINE**, **CIRCLE**, and **ERASE** quickly at the keyboard?
6. What is a fast method of closing a polygon when you are using the **LINE** command?
7. Explain the use of the **LINE Undo** option.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Discuss the advantages and disadvantages of having more than one way to enter a command. Which methods of entering commands are most efficient? Collect opinions from other AutoCAD users and consider ergonomic factors.
2. Describe a way to open a drawing file in AutoCAD when AutoCAD is not currently running on the computer.

Continued

Chapter 4 **Review & Activities**

<div style="background-color:magenta; color:white; display:inline-block; padding:4px 12px;">● **APPLYING AUTOCAD SKILLS**</div>

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create a new drawing file named **prb4-1.dwg** and store it in the folder named after yourself. Create the bookcase shown in Fig. 4-4. For practice, enter the **LINE** command in various ways: using dynamic input, using the **Command** line, or picking a button. Use the various shortcut methods. Save the drawing.
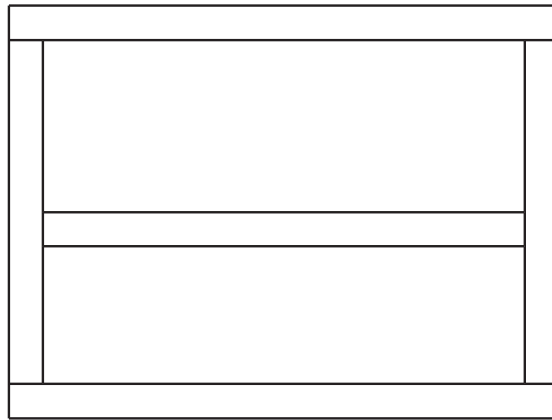
Fig. 4-4

2. Create a new drawing file named **prb4-2.dwg** and store it in the folder named after yourself. Draw the concrete block shown in Fig. 4-5 using the most efficient method(s) of entering the **LINE** command.
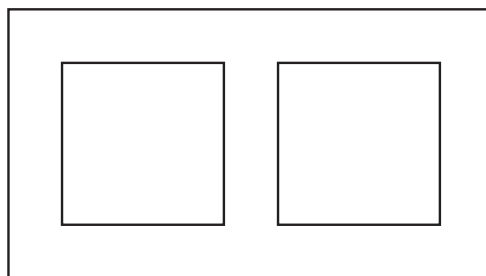
Fig. 4-5

Review & Activities Chapter 4

3. Create a new drawing file named **prb4-3.dwg** and store it in the folder
named after yourself. Draw the simple house elevation drawing shown
in Fig. 4-6 by picking the **Line** button from the **Draw** panel of the **Ribbon**.
When reentering the command, use a shortcut method.

Fig. 4-6

4. Create a new drawing file named **prb4-4.dwg** and store it in the folder
named after yourself. Draw the front and side views of the sawhorse
shown in Fig. 4-7 by entering the **LINE** command's alias at the keyboard.
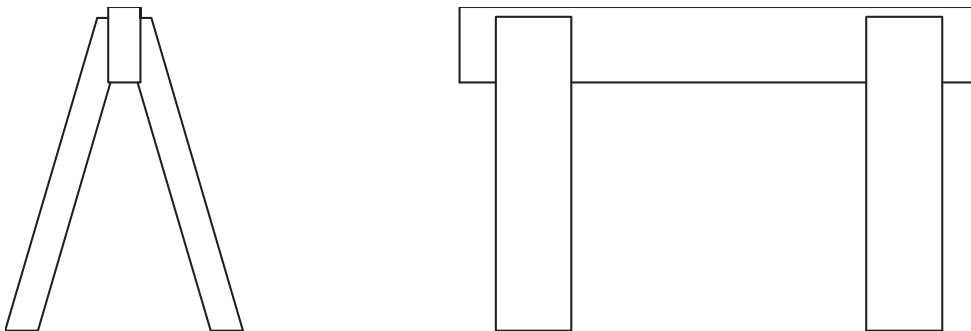Be sure to use shortcut methods when reentering the command.

Fig. 4-7

*Courtesy of Joseph K. Yabu, Ph.D., San Jose State University*

Continued

Chapter 4 **Review & Activities**

● USING PROBLEM-SOLVING SKILLS

Complete the following problems using problem-solving skills and your knowledge of AutoCAD.

1. A boat trailer manufacturer needs a new design for the rollers that facilitate raising a boat onto the trailer. The engineering division of your company, which designs the rollers, has proposed the design shown in Fig. 4-8. Create a drawing to be submitted to the trailer manufacturer for approval. Notice that the drawing is dimensioned. Experiment with the buttons on the status bar and see if you can figure out a way to draw the roller approximately to size. Do not dimension the drawing. Save the drawing as **roller1.dwg**.
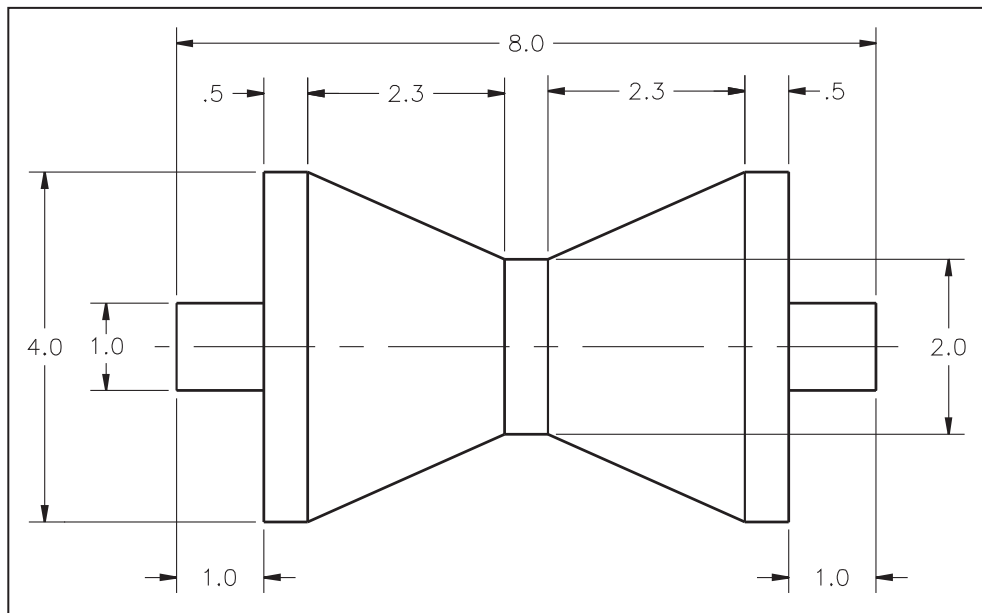
Fig. 4-8

2. The trailer manufacturer accepted the roller design with the following modification: the 4-inch overall diameter is to be increased to 5 inches. All other dimensions are to remain the same. Open **roller1.dwg** and make the change. Save the drawing as **roller2.dwg**.

Chapter 5

Basic Objects

Objectives

- Draw two-dimensional views of holes, cylinders, and rounded and polygonal features
- Create curved objects such as circles, arcs, ellipses, and donuts
- Create rectangles and other types of regular polygons

Vocabulary

concentric
control panels
diameter
donuts
ellipse
radial
radius
regular polygon
tangent
2D representation

Most products, buildings, and maps contain holes, rounded features, and other shapes that are circular in nature. Consequently, when drawing them with AutoCAD, you must use commands that produce circular and radial features. A **radial** feature is one in which every point is the same distance from an imaginary center point. Fig. 5-1 on page 48 shows a race car engine that contains many examples of circular and radial features. Technical drawings also include rectangles and other polygonal shapes, as shown in Fig. 5-1 and other drawings in this book.
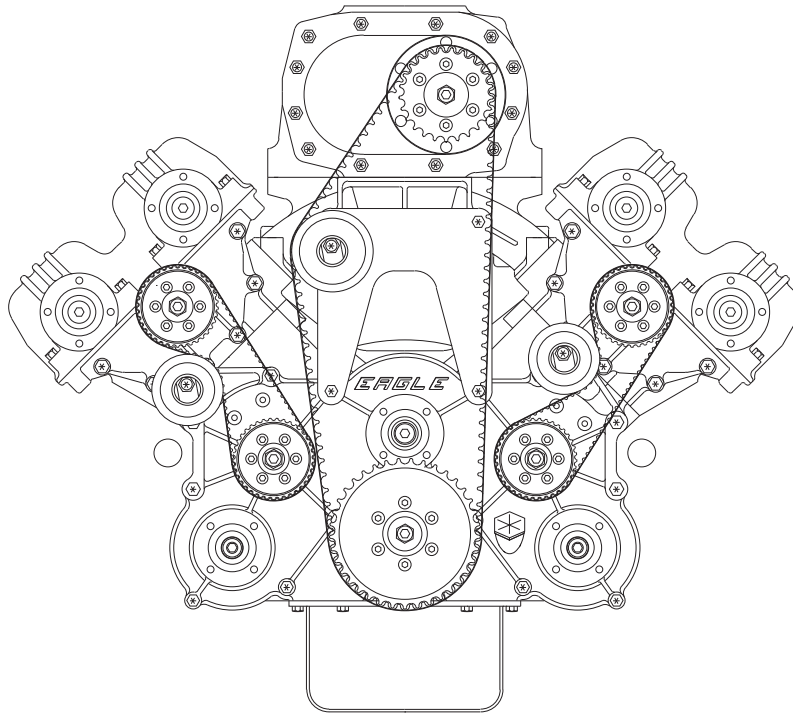
Creating Circles

AutoCAD makes it easy to draw two-dimensional (2D) representations of holes, cylinders, and other round shapes. A **2D representation** is a single profile view of an object seen typically from the top, front, or side.

1. Start AutoCAD. If the AutoCAD window does not appear and you instead see the **Startup** dialog box, click the **Cancel** button in the dialog box.

> **NOTE**
> Remember to exit and restart AutoCAD if it was loaded prior to Step 1 to reset all of the buttons to their default appearance.

2. Pick the **Workspace Switching** button on the status bar and select **2D Drafting & Annotation**, even if it already appears as the current workspace.

3. Pick the **Save** button from the **Quick Access** toolbar, or select **Save As...** from the **Menu Browser.**

This causes the **Save Drawing As** dialog box to appear.

4. Open the folder with your name and highlight **Drawing1.dwg**, located to the right of **File name**.

5. Type **engine** in upper- or lowercase letters and pick **Save** or press **ENTER**.

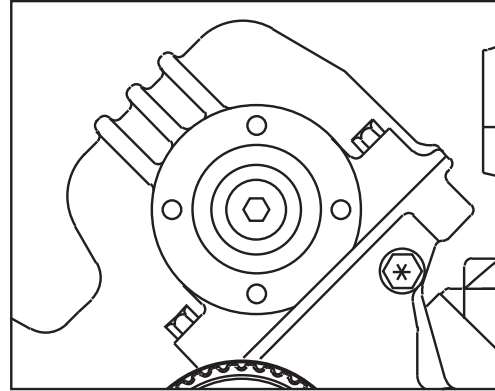This creates and stores a new drawing file named **engine.dwg** in your named folder.

The detail drawing in Fig. 5-2 shows one of the engine's cylinder heads. Let's create the center portion of the cylinder head. (For now, disregard the bolt at the center of the head. You will add it later in this chapter.)

First, turn your attention to the **Ribbon**. In the **2D Drafting & Annotation** workspace, the **Ribbon** contains **control panels**, or groupings, of buttons specifically for creating two-dimensional drawings.

6. Pick the **Circle** button in the **Draw** panel of the **Ribbon**, or enter **C** (the command alias for **CIRCLE**) at the keyboard.

DRAW
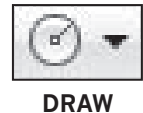
Notice the instruction on the **Command** line: **Specify center point for circle**. Other options are in brackets. These same instructions are displayed on the dynamic input prompt. Other options are displayed if you press the down arrow key.

7. Use the pointing device to pick a center point near the center of the drawing area.

8. Move the crosshairs and notice that you can "drag" the radius of the circle.

> **NOTE**
>
> If the radius of the circle does not change as you move the crosshairs, a feature called **DRAGMODE** may be turned off. To turn it back on, enter **DRAGMODE** and then **A** (for **Auto**).

Notice that AutoCAD is requesting the radius or diameter of the circle. The **radius** of a circle is the length of a line extending from its center to one side of the circle. The **diameter** of a circle is the length of a line that extends from one side of a circle to the other and passes through its center.
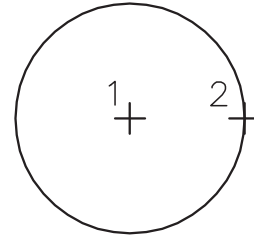
9. Pick a point a short distance from the center, as shown in Fig. 5-3.

Fig. 5-3

This completes the command and forms the innermost circular feature of the cylinder head.

Concentric Circles

Now let's draw a concentric circle to represent the next feature. **Concentric** circles are circles that share a common center.

1. Reenter the **CIRCLE** command.

DRAW

> **HINT**
>
> Use one of the shortcuts to reenter the command.

2. Move the crosshairs to the center of the circle you created.

A small yellow circle appears at the center of the circle. This is the **Center** object snap. Object snaps are like magnetic points that lock onto a specific point on an object easily and accurately. You will learn more about object snaps in Chapter 10.

3. While the small yellow circle is present, pick a point.

AutoCAD snaps to the center point of the existing circle.

4. Drag the radius into place to create a second circle, as shown in Fig. 5-4. Review the detail of the cylinder head to approximate the size of the circle.
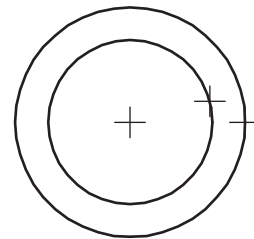
5. Repeat Steps 1 through 4 to place the remaining concentric circles, as shown in the detail of the cylinder head in Fig. 5-2 (page 49).

Fig. 5-4

DRAW

6. Reenter the **CIRCLE** command and create the four small circles to represent bolt holes. Approximate their size and location.
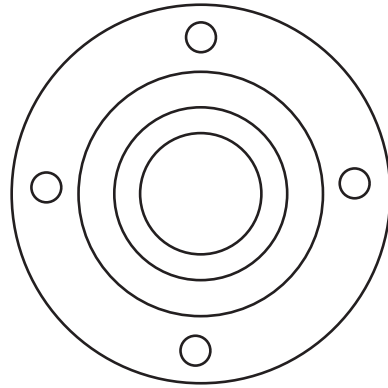
Fig. 5-5

Your drawing should now look similar to the one in Fig. 5-5.

7. Save the drawing file.

Tangent Circles

Circles that touch at a single point are said to be **tangent**. The **Ttr** option of the **CIRCLE** command allows you to create a circle with a specified radius that is tangent to two other circles or arcs.

1. Select the **QNew** button from the **Quick Access** toolbar to begin a new drawing file.

This displays the **Select template** dialog box. This dialog box permits you to select from a number of template files. A template file contains pre-established settings and values that speed the creation of new drawings. You will learn more about them later in the book.

2. Select the **acad.dwt** file from the list of options and pick the **Open** button.

3. Create two circles and position them relatively close to one another.

4. Pick the down-facing arrow to the right of the **Circle** button, and select the **Tan, Tan, Radius** button from the pull-down menu.

This enters the **Tan, Tan, Radius** option of the Circle command. Another way to enter this option is to reenter the **Circle** command and type **T** at the keyboard to select the **Ttr (tan, tan, radius)** option.

5. Read the instructions on the **Command** line and then move the crosshairs over one of the two circles.

A small yellow symbol appears, showing that the center of the crosshairs is tangent to the circle.

QUICK ACCESS

DRAW

> **NOTE**
>
> If the yellow symbol does not appear, pick the **Object Snap** button on the status bar to turn object snap on and then try Step 5 again.

6. Pick a point on the first circle.

7. Move the crosshairs to the other circle and pick a point.

8. For the radius of the new circle, enter **2**.

AutoCAD creates a new circle that is tangent to the two circles. It's okay if the circle extends off the screen. If you see the message **Circle does not exist** on the **Command** line, repeat Steps 4 through 8, but use a larger radius value.

Creating Arcs

The person who created the engine drawing shown in Fig. 5-1 used arcs to produce the radial features on some of the individual parts. In AutoCAD, the **ARC** command is often used to produce arcs.
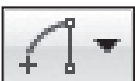
DRAW

1. Pick the **Arc** button on the **Draw** panel of the **Ribbon** or enter **A** at the keyboard.

This activates the **ARC** command. The default option for the **ARC** command is **3-Point.**

2. Pick three consecutive points anywhere on the screen.

3. Reenter the **ARC** command and produce two additional arcs using the same method.

Options for Creating Arcs

AutoCAD offers several options for creating arcs. The option you choose depends on the information you know about the arc. Suppose you want to create an arc for an engine part and you know the start point, the endpoint, and the radius of the arc.

DRAW

1. Pick the down-facing arrow to the right of the **Arc** button and select the **Start, End, Radius** button from the pull-down menu.

2. Pick a start point, then pick an end point a short distance away.

3. Move the crosshairs and watch the arc form dynamically on the screen.

4. Enter **3** for the **Radius**.

An arc forms on the screen.

You can also create this arc using the **3-Point Arc** command: before selecting the second point, enter **E** for **End**, and before selecting the third point, enter **R** for **Radius**.

Suppose you don't know the radius of the arc or where the endpoint should be. However, you know where the center point of the arc should be, and you know how long the arc must be.

5. Pick the down-facing arrow to the right of the **Arc Start, End, Radius** button and select the **Start, Center, Angle** button from the pull-down menu.

6. Pick a start point, then pick a center point for the arc.

7. Enter a number (positive or negative) up to 360. (The number specifies the angle in degrees.)

DRAW
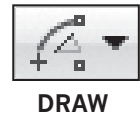
An arc forms on the screen. You can also create this arc using the 3-Point Arc command and the Command line prompt by entering C for Center before selecting the second point, and A for Angle before selecting the third point.

8. Experiment with the other options on your own.

As you can see, AutoCAD's options allow you to create an arc accurately using the information that is available to you. Understanding these options can save time when you need to produce accurate engineering drawings that contain circular or radial features.

Series of Tangent Arcs

In some cases, you may need to draw a series of arcs that are connected to one another. The **Continue** option of the **ARC** command provides an efficient way to draw such a series.

The line in Fig. 5-6 is really a series of tangent arcs. This line was developed for a map. It represents a stretch of Highway 34 through Rocky Mountain National Park, west of Estes Park, Colorado. Let's reproduce the map line to practice creating tangent arcs.
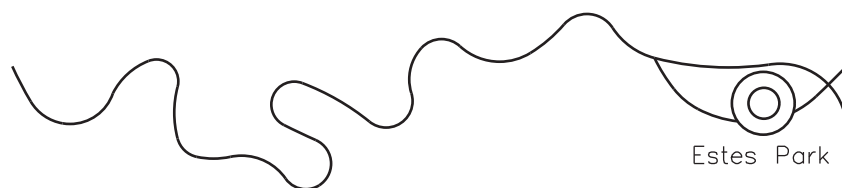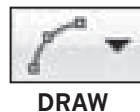

Estes Park

Fig. 5-6

1. Enter the **ARC** command and create an arc using the **3-Point** option.

2. Reenter the **ARC** command by pressing the spacebar or **ENTER**; press the spacebar or **ENTER** a second time.

DRAW

Notice how another arc segment develops as you move the crosshairs.

3. Create the next arc segment.

Notice that the new arc is tangent to the first one. (The point of tangency is the point at which the two arcs join.)

4. Repeat Steps 2 and 3 until you have finished the road.

5. Create the concentric circles that mark Estes Park, but omit the text.

You can also enter the **Continue** option of the **Arc** command by selecting the **Continue** button on the **Arc** pull-down menu.

Drawing Ellipses

The **ELLIPSE** command enables you to create mathematically correct ellipses. An **ellipse** is a regular oval shape that has two centers of equal radius. Ellipses are used to construct shapes that become a part of engineering drawings. For example, the race car engine includes a partial ellipse, as shown in Fig. 5-7.
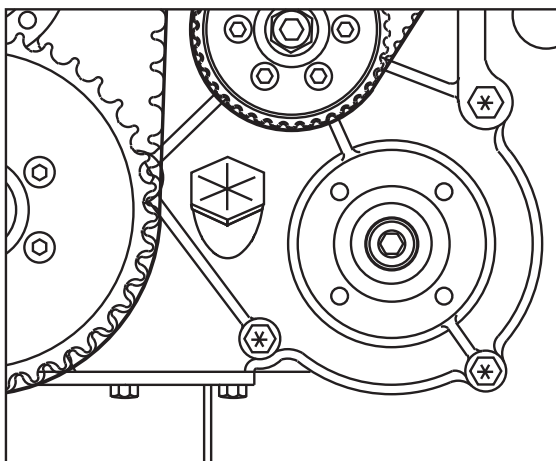
Fig. 5-7

DRAW

1. Pick the down-facing arrow to the right of the **Ellipse** button on the **Ribbon** and select the **Axis, End** button from the pull-down menu. Or, enter the **ELLIPSE** command at the keyboard.

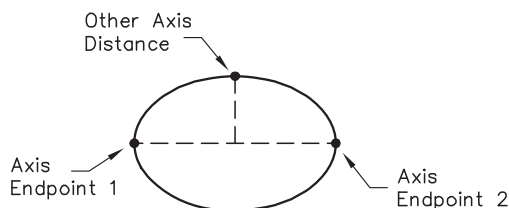**2.** Pick a point for an endpoint of the first axis, as shown in Fig. 5-8.

Fig. 5-8

3. Pick a second point for the other endpoint of the first axis directly to the right or left of the first point.

As noted in the previous chapter, temporary dotted lines appear when you move the crosshairs over existing objects on the screen. This feature allows you to align and create new points from the existing objects. Take advantage of the dotted horizontal line that appears when creating axis endpoint 2.

4. Move the crosshairs and watch the ellipse develop. Pick a point or enter a numeric value, such as **.3**, and the ellipse will appear.

5. Experiment with the **Center** option on your own. How might this option be useful?

Elliptical Arcs

Many parts that include elliptical shapes do not use the entire ellipse. To draw these shapes, you can use the **ELLIPSE Arc** option.

1. Pick the down-facing arrow to the right of the **Ellipse Axis, End** button and select the **Elliptical Arc** button.

2. Pick three points similar to those shown in Fig. 5-8.

DRAW

A temporary ellipse forms.

3. In reply to **Specify start angle**, pick a point anywhere on the ellipse.

4. As you slowly move the crosshairs counterclockwise, notice that an elliptical arc forms.

5. Pick a point to form an elliptical arc.

Donuts

The **DONUT** command allows you to create thick-walled or solid circles, known in AutoCAD as **donuts**. Drafters commonly use donuts to represent features on a machine part, architectural drawing, or map.

1. Click inside the **Draw** panel title bar on the **Ribbon** to display the expanded panel, then select the **Donut** button.

DRAW

2. Specify an inside diameter of **.4** …

3. … and an outside diameter of **.75**.

The outline of a small donut locks onto the crosshairs and is ready to be dragged and positioned by its center.

4. Place the donut anywhere in the drawing by picking a point.

5. Move the crosshairs away from the new solid-filled donut and notice the **Command** line at the bottom of the screen.

6. Place several additional donuts in the drawing.

7. Press **ENTER** to terminate the command.

Rectangles

One of the most basic shapes used by drafters and designers is the rectangle. You can create rectangles using the **LINE** command, but doing so has some disadvantages. For example, you would have to take the time to make sure that the corner angles are exactly 90°. Also, each line segment would be a separate object. Therefore, AutoCAD provides the **RECTANG** command, which allows you to create a rectangle with perfect corners and as a single object. Clicking anywhere on a rectangle created with the **RECTANG** command selects the entire rectangle.

1. On the **Draw** panel of the **Ribbon**, pick the **Rectangle** button.

DRAW

As you can see, AutoCAD is asking for the first corner of the rectangle. Other options appear in brackets at the **Command** line.

2. In reply to **Specify first corner point**, pick a point at any location.

3. Move the pointing device in any direction and notice that a rectangle begins to form.

4. Pick a second point at any location to create the rectangle.

5. Create a second rectangle. Since the **RECTANG** command was just entered, reenter it by pressing the spacebar or **ENTER**, or right-click and pick **Repeat RECTANG** from the shortcut menu.

6. Create a third rectangle.

7. Close the drawing without saving it.

Polygons

The **POLYGON** command enables you to create regular polygons with 3 to 1024 sides. A **regular polygon** is one with sides of equal length. Using the **POLYGON** command, let's insert the bolt head into the engine drawing you started earlier in this chapter.

1. Open the **engine.dwg** file by picking the **Open** button from the abbreviated **Quick Access** toolbar.

2. On the expanded **Draw** panel of the **Ribbon**, pick the **Polygon** button, or enter the **POL** alias.

DRAW

Notice the **<4>** at the end of the AutoCAD prompt. This is the default value, meaning that if you were to press **ENTER** now, AutoCAD would enter **4** in reply to **Enter number of sides**. To represent the bolt head shown in Fig. 5-2 on page 49, you will need a hexagon (six-sided polygon).

3. Enter **6**.

AutoCAD now needs to know if you want to define an edge of the polygon or select a center point. Let's specify a center.

4. Move the pointing device over any of the larger circles in the drawing. When the small yellow symbol appears at the center of the cylinder head, pick a point to select the center of the circle as the center point of the hexagon. (The **Object Snap** button on the status bar must be depressed.)

AutoCAD allows you to create a polygon by inscribing it inside a circle of a specified diameter or by circumscribing it around the specified circle. Figure 5-9 shows the difference.

PENTAGON INSCRIBED
INSIDE A CIRCLE

PENTAGON CIRCUMSCRIBED
AROUND THE SAME CIRCLE
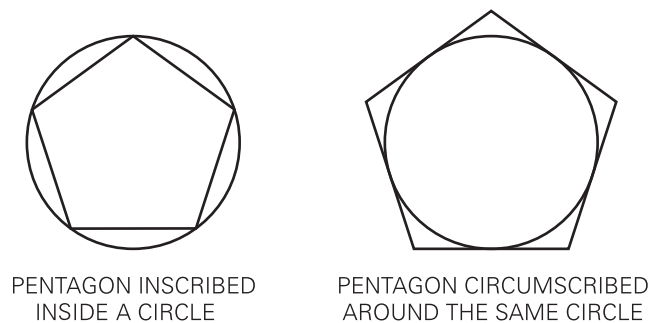
Fig. 5-9

5. Press **ENTER** to select the **I** (**Inscribed**) default value.

AutoCAD now wants to know the radius of the circle within which the polygon will appear.

6. With the pointing device, move the crosshairs from the center of the polygon and notice that a hexagon begins to form.

7. Pick a point to create the hexagonal bolt head at an appropriate size relative to the cylinder head. Refer to Fig. 5-2, if necessary.
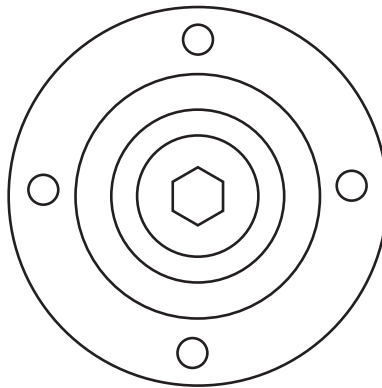
The drawing of the cylinder head component is now complete. Your drawing should look similar to the one in Fig. 5-10. Note that the relative diameters of the circles and the size of the bolt head relative to the circles may be different on your drawing because we did not use exact sizes in this drawing.

QUICK ACCESS

8. Save your work and exit AutoCAD.

Fig. 5-10

HINT Back up your files regularly. The few seconds required to make a backup copy may save you hours of work. Experienced users back up faithfully because they know the consequences of not doing so.

Chapter 5 Review & Activities

• REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Briefly describe the following methods of producing circles.

   a. 2 points

   b. 3 points

   c. tan tan radius

2. In which **Ribbon** control panel are the **Arc** and **Circle** buttons found?

3. What function does the **Arc Continue** option serve?

4. Explain the purpose of the **DONUT** command.

5. When you create a polygon using the **Inscribed in circle** option, does the polygon appear inside or outside the imaginary circle?

6. What are the practical differences between creating a rectangle using the **LINE** command and creating it using the **RECTANG** command? Why might you choose one command instead of the other?

7. In addition to the **LINE** and **RECTANG** commands, which other command is capable of creating a rectangle? When might you choose this method?

• CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Review the information in this chapter about specifying an angle in degrees. How might you be able to create an arc in a clockwise direction? Try your method to see if it works, and then write a paragraph describing the method you used.

2. Experiment further with the **ARC** command. Is it possible to create a single arc that has a noncircular curve using the options available for this command? (A noncircular curve is one in which not all the points are exactly the same distance from a common center point.) Explain your answer.

Continued

Chapter 5 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Complete the drawings shown in Figs. 5-11 and 5-12. Don't worry about text matter or exact shapes, sizes, or locations, but do try to make your drawings look similar to the ones in the figures. Save the drawings as **prb5-1a.dwg** and **prb5-1b.dwg**.
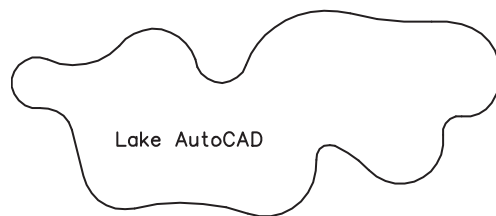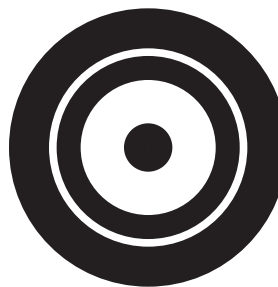
Lake AutoCAD

Fig. 5-11

Fig. 5-12

2. Use the **LINE**, **POLYGON**, **CIRCLE**, and **ARC** commands to create the hex bolt shown in Fig. 5-13. Save the drawing as **prb5-2.dwg**.
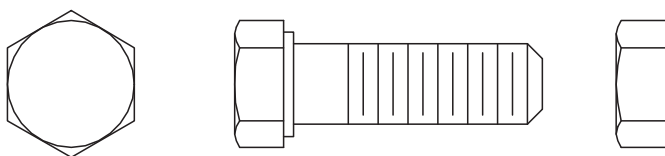
Fig. 5-13

Courtesy of Joseph K. Yabu, Ph.D., San Jose State University

Review & Activities Chapter 5

3. Create the object shown in Fig. 5-14 by first drawing a six-sided polygon with the **POLYGON** command. Then draw the six-pointed star using the **LINE** command. Save the drawing as **prb5-3.dwg**.

4. In the same drawing, create the same object again, but draw the star using two three-sided polygons. Save your work.

Fig. 5-14

5. How would you draw a five-pointed star? Draw it in the same drawing file and save your work.

6. Draw a block with a rectangular cavity like the one shown in Fig. 5-15 using the **RECTANG** and **LINE** commands. Save the drawing file as **prb5-6.dwg**.
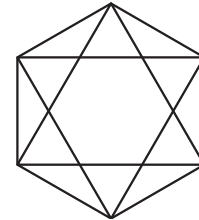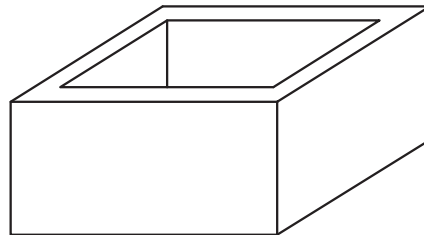

Fig. 5-15

7. The two objects shown in Fig. 5-16 are composed entirely of equal-sided and equal-sized polygons with common edges surrounding a central polygon. Can this be done with equal-sided and equal-sized polygons of any number of sides? Answer this question by trying to draw such objects using the **POLYGON** command with polygons of five, six, seven, and eight sides.
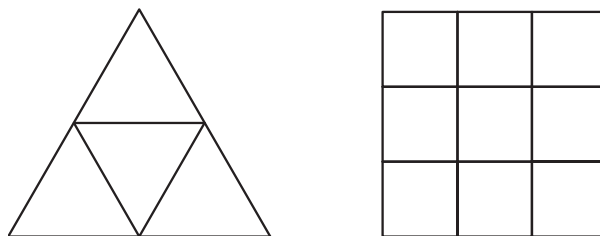

Fig. 5-16

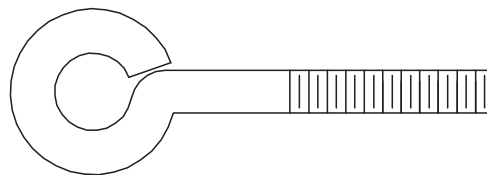*Problems 3–7 courtesy of Gary J. Hordemann, Gonzaga University*

Continued

Chapter 5: Basic Objects   61

Chapter 5 **Review & Activities**

8. Draw the screw heads shown in Fig. 5-17. Save the drawing as **prb5-8.dwg**.

9. Draw the eyebolt shown in Fig. 5-18 using all of the commands you have learned. Save the drawing as **prb5-9.dwg**.

Fig. 5-17

Fig. 5-18
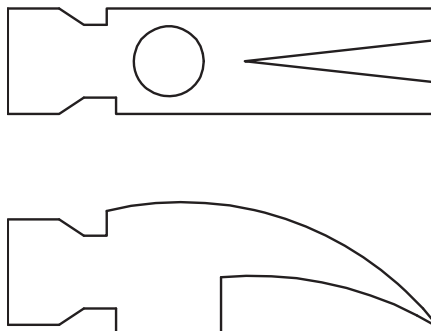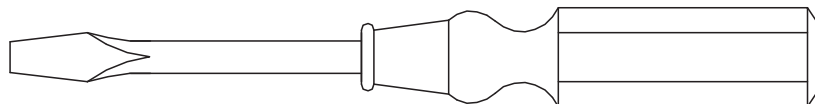
10. Using the **LINE**, **ARC**, and **CIRCLE** commands, draw the two views of a hammer head shown in Fig. 5-19. Save the drawing as **prb5-10.dwg**.

Fig. 5-19

11. Use the **ARC** and **LINE** commands to create the screwdriver shown in Fig. 5-20. Save the drawing as **prb5-11.dwg**.

Fig. 5-20

Problems 9–11 courtesy of Joseph K. Yabu, Ph.D., San Jose State University

Review & Activities Chapter 5

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Draw the ski-lift rocker arm shown in Fig. 5-21 for a design study by a gondola manufacturer. Do not include dimensions. Save your drawing as **rockerarm.dwg**.

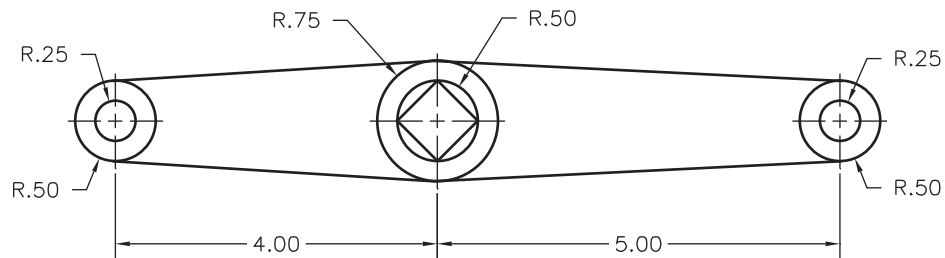Fig. 5-21

2. The traditional mantle clock body drawing shown in Fig. 5-22 is necessary for the reconstruction of antique and collectible clocks. Draw the clock body. Do not include dimensions. Save your drawing as **mantle.dwg**.
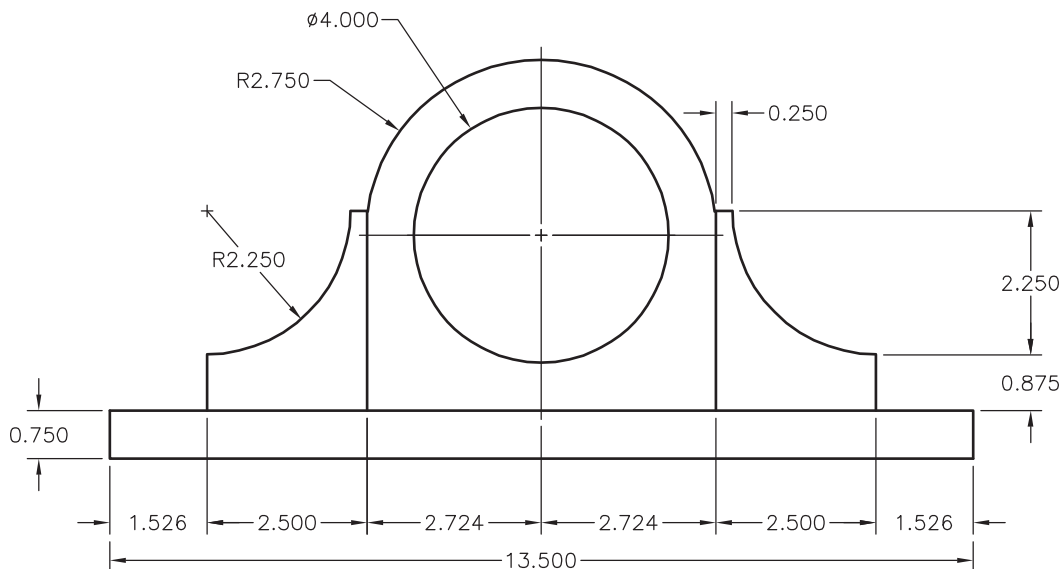

Fig. 5-22

66   Wohlers: Applying
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Shop Coordinator

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AutoCAD at Work

3D Print Shop Coordinator

With more manufacturers accepting 3D data to drive computer-controlled equipment, drafters and engineers are moving rapidly toward designing in 3D. Designing in 3D has an important advantage over traditional methods. The 3D files can be sent to a 3D printer or rapid prototyping device to create a physical prototype. This prototype can undergo testing before the manufacturer goes to the trouble and expense of tooling up for the product. Design changes can be made relatively inexpensively at this stage.

Photo Courtesy of Z Corporation

The equipment required to produce a 3D prototype is still quite expensive, although costs are coming down. As a result, many companies choose to outsource their prototyping needs. They create the 3D CAD files and send them to a 3D print shop for processing.

Types of 3D Printing

Several types of 3D printing devices are available. In addition to rapid prototyping devices, 3D scanners and freeform modeling systems are now being used. Many 3D print shops invest in several different types of printing devices. This gives the shop the flexibility to serve a wide range of customers, but it requires a skilled coordinator.

Coordinating a Print Shop

To coordinate a 3D print shop successfully, you will need to be familiar not only with the equipment in your shop, but also with the various CAD software programs used to create the data used by these machines. You will need to know which machines can handle which software. The best way to prepare for this career is to take drafting and modeling classes offered at area high schools, community colleges, and vocational schools. You will also need management experience and the ability to work well both with customers and with technical personnel.

▶ Career Activities

1. Find out if there are any 3D print shops in your area. What services do they offer?

2. Research on the Internet to learn more about the various types of 3D printing and the capabilities of each.

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I. Groundwork

6. Object Selection

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67

Chapter 6

Object Selection

Objectives

- Identify AutoCAD objects
- Select objects to create a selection set
- Add and remove objects to and from selection sets
- Erase and restore objects
- Resize objects parametrically

Vocabulary

entity
grips
noun/verb selection
object
parametric shape editing
pickbox
Quick Properties panel
selection previewing
selection set
selection window
verb/noun selection

When using AutoCAD, you will find yourself selecting objects so that you can move, copy, erase, or perform some other operation. Because drawings can become very dense with lines, dimensions, and text, it is important to use the most efficient methods of selecting these objects. Usually, the quickest way to move or copy an object is to click and drag. This is true whether you are using an inexpensive shareware program on a low-cost computer or sophisticated CAD software on an expensive computer. This chapter covers this and alternative methods of selecting objects and performing basic editing operations, such as resizing objects and erasing.

Objects

An **object**, also called an **entity**, is an individual predefined element in AutoCAD. The smallest element that you can add to or erase from a drawing is an object.

The following list gives examples of object types in AutoCAD.

3Dface	leader	solid
3Dsolid	line	spline
arc	mline	text
attribute	multiline text	tolerance
body	point	trace
circle	polyline	vertex
dimension	ray	viewport
ellipse	region	xline
image	shape	

You will learn more about these objects and how to use them in future chapters.

Selecting Objects

Basic object selection in AutoCAD is straightforward—you select the object by picking it with the left button on the pointing device. However, AutoCAD also offers more advanced object selection techniques. This chapter illustrates many of these techniques.

DRAW

DRAW

1. Start AutoCAD, then pick the **Workspace Switching** button on the status bar and select (or re-select) the **2D Drafting & Annotation** workspace.

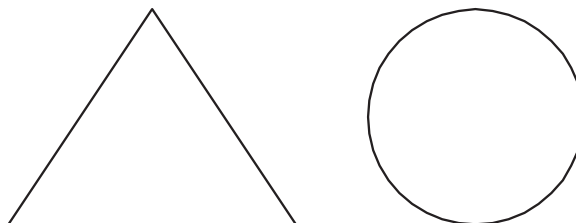2. Using the **LINE** and **CIRCLE** commands, draw the triangle and circle shown in Fig. 6-1.

Fig. 6-1

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69

NOTE

In this book, all new drawings use Imperial (English) units unless otherwise specified.

3. Enter the **ERASE** command by picking the **Erase** button from the **Modify** panel on the **Ribbon** or by entering **E** at the keyboard.

MODIFY

The AutoCAD prompt changes to **Select objects**. Notice that the crosshairs have changed to a small box called the **pickbox**. The pickbox is used to pick objects. At this point, you can also create a window to select points.

4. Pick a point anywhere to the lower left of the triangle.

5. Move the pointing device to the right and notice that a blue box forms.

This box defines the **selection window**.

6. Move the pointing device so that the triangle fits completely inside the window and pick a point.

The objects to be erased are highlighted with broken lines, as shown in Fig. 6-2. The highlighted objects make up the **selection set**.
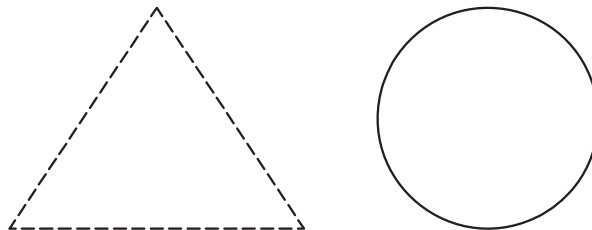

F i g .   6 - 2

7. Press **ENTER** to make the triangle disappear. (This also terminates the **ERASE** command.)

Restoring Erased Objects

What if you have erased an object by mistake and you want to restore it? Let's restore the triangle.

8. Pick the **Undo** button on the **Quick Access** toolbar.

QUICK ACCESS

The triangle reappears. The **Undo** button conforms to standard Windows functionality, so you may be familiar with it from other Windows applications. Picking this button repeatedly undoes your previous actions, one at a time, beginning with the most recent.

Using a Crossing Window

The **Crossing** object selection option selects all objects that cross the window boundary as well as those that lie completely within it.

MODIFY

1. Enter the **ERASE** command.
2. Type **C**, press **ENTER**, and pick a point to the left of the triangle.
3. Move the crosshairs to the right to form a green box and notice that the box is made up of broken lines.
4. Form the box so that it crosses over at least one side of the triangle and press the pick (left) button in reply to **Specify opposite corner**.

Notice that AutoCAD selected those objects that cross over the box as well as any objects that lie completely within it.

> **HINT**
>
> You can also select objects using a crossing window without formally entering the **Crossing** option. When the **Select objects** prompt appears, simply pick two points to form a window, being sure to pick the *right-most* point first.

Polygonal Selection Options

The **WPolygon** and **CPolygon** object selection options are similar to the **Window** and **Crossing** options. However, they offer more flexibility because they are not restricted to the rectangular window shape used by the **Window** and **Crossing** options.

MODIFY

1. Fill the screen with several objects, such as lines, circles, arcs, rectangles, polygons, ellipses, and donuts.
2. Enter the **ERASE** command.
3. In reply to **Select objects**, enter **WP** for **WPolygon** (**W** is for **Window**.)
4. Pick a series of points that form a polygon of any shape around one or more objects. As you create the polygon, notice that the area inside the polygon is blue.
5. Press **ENTER** to close the polygon.

NOTE When forming the polygon, AutoCAD automatically connects the last point to the first point.

AutoCAD selects those objects that lie entirely within the polygon.

6. Press **ENTER** to erase the objects.

The **CPolygon** option is similar to **WPolygon**.

7. Enter the **ERASE** command or press the spacebar.

8. In response to **Select objects**, enter **CP** for **CPolygon**. (**C** is for **Crossing**.)

9. Pick a series of points to form a polygon. Make part of the polygon cross over at least one object and press **ENTER**.

MODIFY

As you create the polygon, notice that it is made up of broken lines, and that the area inside the polygon (the selection window) is green. The broken lines and green selection window indicate that the **Crossing** option is in effect. Solid lines around a blue selection window indicate a normal (not crossing) **Window** selection.

AutoCAD selects those objects that cross the polygon, as well as those that lie completely within it. As you can see, **CPolygon** is similar to the **Crossing** option, whereas **WPolygon** is similar to the basic **Window** option.

Using a Fence

The **Fence** option is similar to the **CPolygon** option, except that you do not close a fence as you do a polygon. When you select objects using the **Fence** option, AutoCAD looks for objects that touch the fence. Objects "inside" the fence are not selected unless they actually touch the fence.

1. Enter **ERASE**.

2. Enter **F** for **Fence**.

3. Draw a line that crosses over one or more objects and press **ENTER**.

4. Press **ENTER** to complete the erasure.

MODIFY

Selecting Single Objects

Suppose you want to select a single object, such as a line segment.

1. Recreate the triangle and circle you drew previously.

2. Pick the **Erase** button or enter **E** at the keyboard.

3. Using the pickbox, pick two of the line segments in the triangle.

MODIFY

NOTE

AutoCAD automatically highlights objects as the pickbox rolls over them. This is called **selection previewing**. This feature is helpful when you need to select individual objects in a crowded area of the drawing. It allows you to see in advance which object will be selected when you click the button on the pointing device.

4. Press **ENTER** to complete the command and to make the selected objects disappear.

HINT

If you are using a pointing device that has more than one button, pressing one of them, usually the one on the right, is the same as pressing the **ENTER** key. This is normally faster than pressing **ENTER** at the keyboard.

Selecting the Last Object Drawn

The **Last** object selection option automatically selects the last object you drew. This option works whenever the **Select objects** prompt is present.

QUICK ACCESS

1. Pick the **Undo** button to restore the line segments.

Notice that because you erased both line segments in a single **ERASE** operation, you only have to pick the **Undo** button once to make both lines reappear.

MODIFY

2. Enter the **ERASE** command.

3. Type **L** (for **Last**) and press **ENTER**.

AutoCAD highlights the last object you drew.

NOTE

The last object you drew may be different from the last object you selected in the drawing. If you want to reselect the last object you selected, enter **P** (for **Previous**) instead of **Last**. The **Previous** option is especially useful when you need to reselect an entire group of objects.

4. Press **ENTER**.

If you continue to enter **ERASE** and **Last**, you will erase objects in the reverse order from which you created them. However, it is usually faster to use the **Undo** button for this purpose.

5. Pick the **Undo** button to restore the object you deleted using the **Last** option.

Selecting the Entire Drawing

The **All** object selection option permits you to select all of the objects in the drawing quickly.

1. Enter the **ERASE** command.

2. In reply to **Select objects**, type **All** and press **ENTER**.

AutoCAD selects all objects in the drawing.

3. Press **ENTER** again to erase all of the objects.

4. Pick **Undo** to restore the objects.

Editing Selection Sets

AutoCAD allows you to add and remove objects from a selection set while you are in the process of creating it. For example, if you accidentally pick an object that you did not mean to select, you can remove it without affecting the rest of the selected objects. This is especially helpful when you have many objects in a selection set on a complex drawing.

Removing Objects from a Selection Set

To remove an unwanted object from a selection set, you must activate the **Remove objects** mode.

1. Enter the **ERASE** command.

2. Create a selection window to select the triangle and the circle.

Both the triangle and the circle should now be highlighted. Suppose you have decided not to erase the circle.

3. Type **R** (for **Remove**) and press **ENTER**.

Notice that the AutoCAD prompt changes to **Remove objects**. You can now remove one or more objects from the selection set. The object(s) you remove from the selection set will not be erased.

4. Remove the circle from the selection set by picking it.

Note that it is no longer highlighted.

QUICK ACCESS

5. Press **ENTER** to complete the **ERASE** operation.

6. Pick the **Undo** button to restore the triangle.

Adding to the Selection Set

After you have used the **Remove objects** option to remove objects from a selection set, you can select additional objects to include in the set.

1. Repeat Steps 1 through 4 in the previous section ("Removing Objects from a Selection Set").

2. Instead of pressing **ENTER**, type **A** for **Add** and then press **ENTER**.

Notice that the AutoCAD prompt changes back to **Select objects**.

3. Select the circle.

4. Press **ENTER** to complete the **ERASE** operation.

So you see, you can add and remove objects as you wish until you are ready to perform the operation. The objects selected are indicated by broken lines. These selection procedures work not only with the **ERASE** command, but also with other commands that require object selection, such as **MOVE**, **COPY**, **MIRROR**, **ARRAY**, and many others.

QUICK ACCESS

5. Pick the **Undo** button to restore the objects to the screen.

Using Grips

Another method of selecting objects is to pick them using the pickbox located at the center of the crosshairs. When you pick an object without first entering a command, small boxes called **grips** appear at key points on the object. Using these grips, you can perform several basic operations such as moving, copying, or changing the shape or size of an object.

DRAW

1. Enter the **RECTANG** command and draw a rectangle of any size and at any location.

2. Select any point on the rectangle. (No command should be entered.)

Notice that blue grips appear at each corner of the rectangle. Also notice the floating panel that appears to the right of the rectangle. This is the **Quick Properties panel**. When turned on, **Quick Properties** displays some of the properties of selected objects. For this chapter, when the **Quick Properties** panel appears, close it by picking the **x** in the upper right corner.

Editing Size and Shape with Grips

3. Move the crosshairs until it locks over the lower right grip, but do not pick the grip yet.

AutoCAD displays the current length of the two lines that meet at the lower right grip, as shown in Fig. 6-3.

4. Move the crosshairs to each of the other grips to see the current length of other sides of the rectangle.

5. Pick the lower right grip to select it.

Fig. 6-3

The grip turns red, showing that you have selected it. The length of the right side of the rectangle is now displayed in an editable text box, as shown in Fig. 6-4.

6. Move the crosshairs on the screen and notice how the lengths update dynamically as you move the grip around. See Fig. 6-5.

Fig. 6-4

7. At the keyboard, enter a new length of **3**.

The length of the right side of the rectangle changes to a length of exactly 3 units and the shape is no longer rectangular. This method of editing a CAD object by entering new size values is known as **parametric shape editing**.
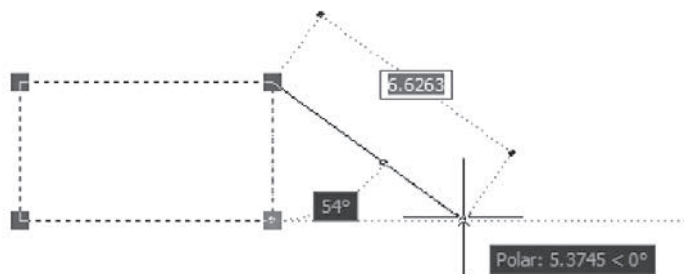

Fig. 6-5

Chapter 6: Object Selection   **73**
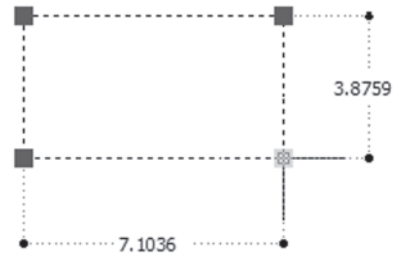
Moving Objects with Grips

1. Select the rectangle again and select any one of the four grips.
2. Press the right button on the pointing device (right-click) to display a shortcut menu.
3. Pick **Move**, drag the rectangle to a new location, and pick a point.
4. Display the shortcut menu again and select one of the other editing options, such as **Rotate**, and complete the operation. Follow the instructions at the AutoCAD prompt.

Copying Objects with Grips

You can also copy objects using grips.

1. Select the circle and pick one of the four grips that lie on it.
2. Move the pointing device and notice that you can adjust the circle's radius.
3. Pick a point to change the circle's radius.
4. Pick the circle's center grip, right-click, and pick **Copy**.
5. Copy the circle three times by picking three points anywhere on the screen.
6. Press **ENTER** to terminate the copying.

Modifying the Grips Feature

For most AutoCAD users, the standard grips characteristics, such as color and size of the grip boxes, are adequate. In some companies or circumstances, however, it may be necessary to adjust the grips so that you can work with them more easily. For example, if your company has customized the background of the drawing area to dark gray, you may want to change the color of unselected grip boxes from blue to yellow so that you can see them more easily. You may need to adjust the size of the grip boxes depending on your display resolution. Grip boxes viewed at 1600×1200 resolution are twice as small as those viewed at 800×600. AutoCAD allows you to make changes such as these using the **Options** dialog box.

1. Pick the **Menu Browser** button; then select the **Options** button at the bottom of the **Menu Browser**.

The **Options** dialog box includes ten tabs with important information and settings in each one.

2. Pick the **Selection** tab.

Information and settings related to the grips feature appear in the right half of the dialog box, as shown in Fig. 6-6.
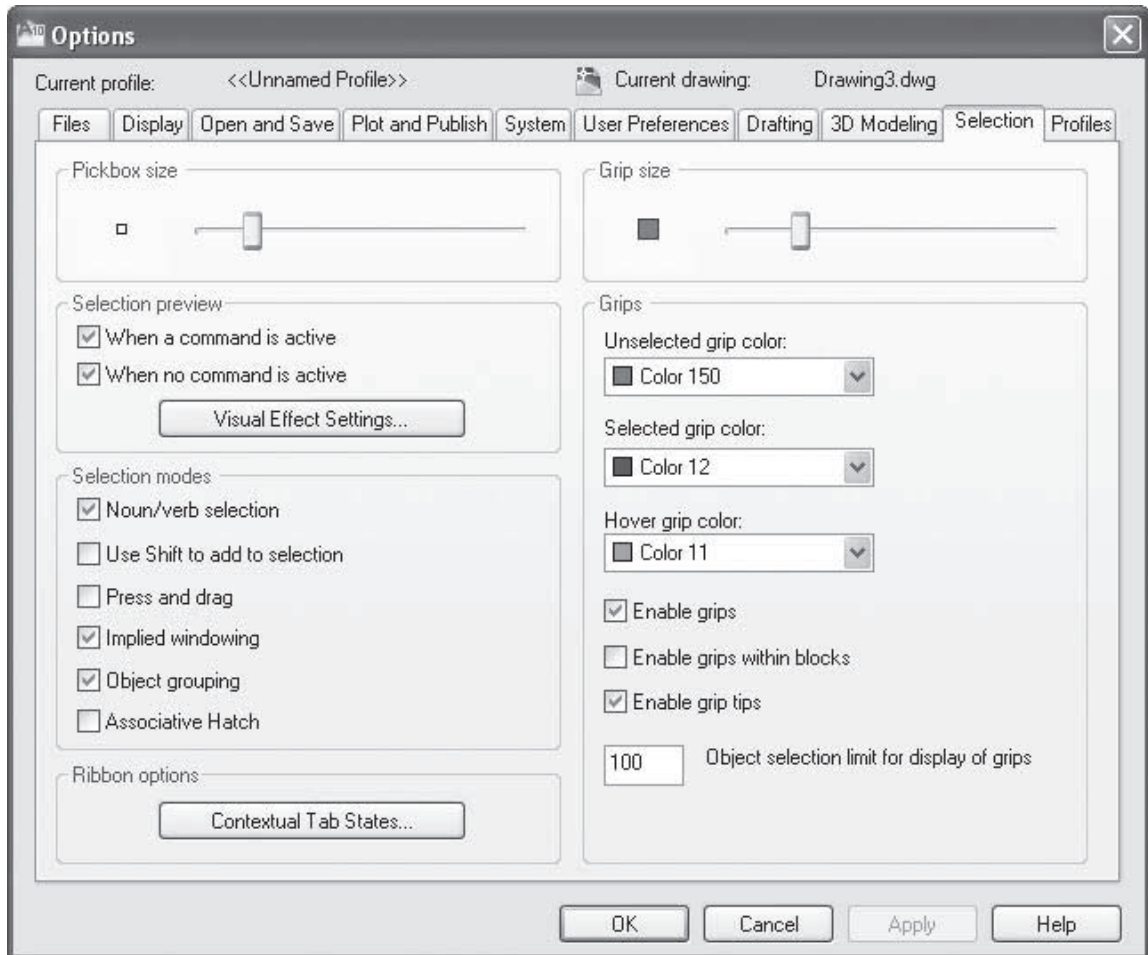
Fig. 6-6

3. In the area labeled **Grip Size**, move the slider bar and watch how it changes the size of the box to the left of it.

4. Increase the size of the grips box and pick the **OK** button.

5. Select one of the objects and notice the increased size of the grips.

6. Pick one of the grips and perform an editing operation.

7. Display the **Options** dialog box and adjust the size of the grips box to its original size, but do not close the dialog box.

The **Selection** tab also permits you to enable and disable the grips feature, as well as control the assignment of grips within blocks. (You will learn more about blocks in Chapter 33.) You can also make changes to the color of selected and unselected grips. For now, do not change these settings.

Using Grips with Commands

The traditional AutoCAD method of entering a command and then selecting the object to be edited is sometimes called **verb/noun selection**. First you tell the software what to do (using a verb such as **ERASE**) and then you select the object to be acted upon (using a noun such as **LINE**).

When you use grips with editing commands, you are using **noun/verb selection**. See Fig. 6-7. First you select the object, and then you tell the software what to do with the object. You may find the grips (noun/verb) method convenient to use, especially if you are familiar with other graphics programs that use this technique. However, either method is acceptable. The method you choose is a matter of personal preference.

MODIFY

1. Select one of the circles.

2. Pick the **Erase** button or enter **E** for **ERASE**.

As you can see, AutoCAD erased the circle without first asking you to select objects. You have just used the noun/verb technique.

3. Pick another circle and press the **Delete** key on the keyboard.

As you can see, pressing the **Delete** key is the same as entering the **ERASE** command.

QUICK ACCESS

4. Pick the **Undo** button from the docked **Standard** toolbar twice to undo the deletions.

5. Exit AutoCAD without saving.
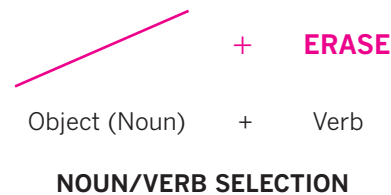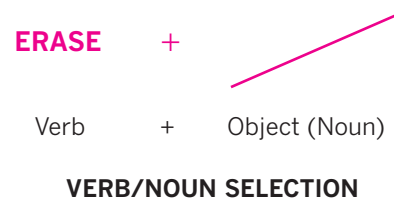
ERASE   +

Verb   +   Object (Noun)

VERB/NOUN SELECTION

+   ERASE

Object (Noun)   +   Verb

NOUN/VERB SELECTION

Fig. 6-7

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6. Object Selection

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79

Chapter 6 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. In AutoCAD, what is an object? Give three examples.

2. After you enter the **ERASE** command, what does AutoCAD ask you to do?

3. Experiment with and describe each of these object selection options.

   a. **Last**

   b. **Previous**

   c. **Crossing**

   d. **Add**

   e. **Remove**

   f. **Undo**

   g. **WPolygon**

   h. **CPolygon**

   i. **Fence**

   j. **All**

4. How do you place a window around one or more objects during object selection?

5. If you erase an object by mistake, how can you restore it?

6. How can you retain part of what has been selected for erasure while remaining in the **ERASE** command?

7. What is the fastest way of erasing the last object you drew?

8. When you use a graphics program, such as AutoCAD, what is usually the quickest way to edit an object?

9. Describe how you would remove an object from a selection set.

10. Explain the primary benefit of using the grips feature.

11. What editing functions become available to you when you use grips?

12. Explain how to copy an object using grips.

13. Explain the difference between the noun/verb and verb/noun selection techniques.

Continued

Chapter 6 **Review & Activities**

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. As you have seen, AutoCAD's grips feature is both easy and convenient to use. With that in mind, discuss possible reasons AutoCAD also includes specific commands that you must enter to perform some of the same functions you can do easily with grips.

2. Describe a situation in which the **Remove** option can be useful for more than just removing an object that was added to a selection set by mistake.

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create a new drawing. Draw a circle and a rectangle as illustrated in Fig. 6-8. Use the **RECTANG** command to create the rectangle. Then enter the **ERASE** command and try to erase one line of the rectangle. At another location in the drawing, create another rectangle, but this time use the **LINE** command. Try to erase one line of the second rectangle. What is the difference? Do not save the drawing.
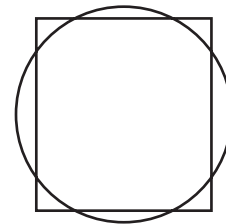
Fig. 6-8

2. Create a new drawing. Use the **LINE** command to draw polygons *a* through *e* as shown in Fig. 6-9. Enter the **ERASE** command and use the various object selection options to accomplish the following operations without pressing **ENTER**.

   • Place a window around polygon *a*.
   • Pick two of polygon *b*'s lines for erasure.
   • Use the **Crossing** option to select polygons *c* and *d*.
   • Remove one line selection from polygon *c* and one from polygon *d*.
   • Pick two lines from polygon *e* for erasure, but then remove one of the lines so it won't be erased.
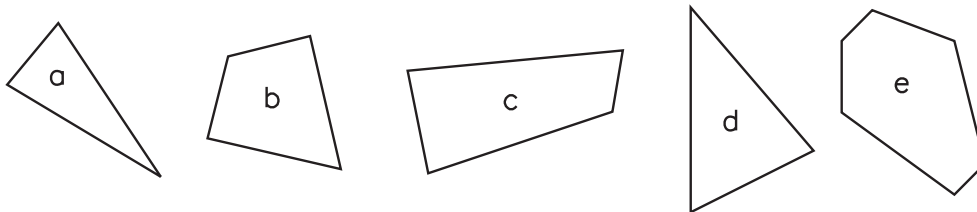
Review & Activities Chapter 6

Fig. 6-9

After you have completed the five operations, press **ENTER**. You should have nine objects (line segments) left on the screen. Save the drawing as **prb6-2.dwg** in the folder with your name, but do not close the file.

3. Select **Save As...** in the **Menu Browser** and enter **prb6-3.dwg** for the file name. Restore the line segments that you deleted in problem 2. Then use grips to rearrange the polygons in order of size from smallest to largest. Save your work.

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. The computer sketch in Fig. 6-10 was sent to the project engineer for comments and has now been routed to your desk with the attached note. Use the **ELLIPSE**, **ARC**, and **LINE** commands to create the sketch of the bushing. Because this is only a rough sketch, you may approximate the dimensions. Make the change requested by the engineering department. Save the drawing as **ch6-bushing.dwg**.

REDUCE WALL THICKNESS BY HALF

Fig. 6-10

Continued

Chapter 6: Object Selection   **79**
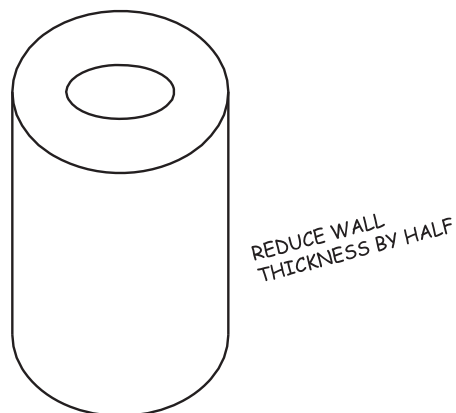
Chapter 6 **Review & Activities**

2. Your architectural firm's customer wants the gable-end roof changed to a hip roof, as shown in Fig. 6-11. Draw the front elevation with both roof lines. Since this is only a roof line change, approximate the size and location of the windows and door. Save the drawing as **ch6-roof.dwg**.
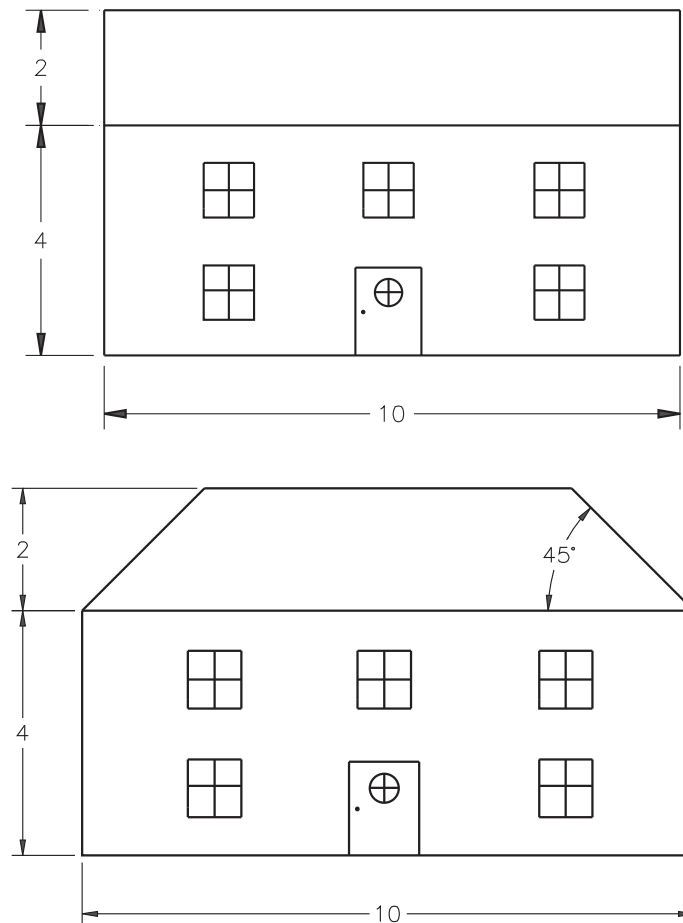

Fig. 6-11

Chapter 7

Entering Coordinates

Objectives

- Describe several methods for entering coordinates
- Locate points and draw objects using the Cartesian coordinate system
- Apply the absolute, relative, polar, and polar tracking methods for entering coordinates

Vocabulary

absolute points
alignment paths
Cartesian coordinate system
coordinate pair
coordinates
origin
polar method
polar tracking method
relative method
world coordinate system (WCS)

Sets of numbers used to describe specific locations on a grid are called **coordinates**. In AutoCAD, coordinates are used to specify locations of objects in a drawing. In previous chapters, when you used the pointing device to pick the endpoints of lines, etc., AutoCAD recorded the coordinates of the points that you picked on the screen.

You can also use the keyboard to enter coordinates. The keyboard method allows you to specify points and draw lines of any specific length and angle. This method also applies to creating arcs, circles, and other object types. By specifying the coordinates of objects, you can achieve the accuracy that is required for engineering drawings.

AutoCAD uses a **Cartesian coordinate system** similar to that used in geometry. In AutoCAD, this system is called the **world coordinate system (WCS)**. In the two-dimensional version of this system, two axes are used. See Fig. 7-1. This is the version used for most drafting work. The X axis is an infinite, imaginary line that runs horizontally. The Y axis is an infinite, imaginary line that runs vertically. Each axis is numbered with sequential positive and negative numbers. Each combination of one *x* value and one *y* value (a **coordinate pair**) specifies a unique point in the coordinate system.
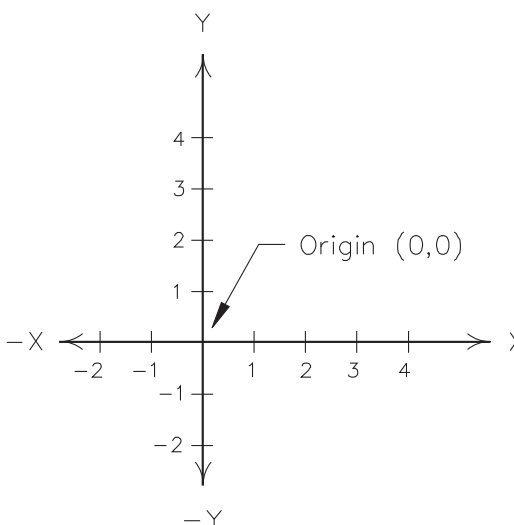
Fig. 7-1

The point at which the axes cross is known as the **origin**. At the origin, the values of *x* and *y* are both zero. The values to the right and above the origin are positive. Those below and to the left of the origin are negative.

In three-dimensional modeling, a third axis—the Z axis—is added at right angles to the other two axes to allow you to specify the third dimension, depth.

Methods of Entering Coordinates

The Cartesian system does not limit you to entering coordinate pairs to specify points. Consider the following ways to specify points using the **LINE** command. Note that dynamic input must be off for the following methods to work as described.

Absolute Method

When you enter specific *x* and *y* values, AutoCAD places the points according to the Cartesian coordinate system. Points entered in this way are considered to be **absolute points**. See Fig. 7-2.
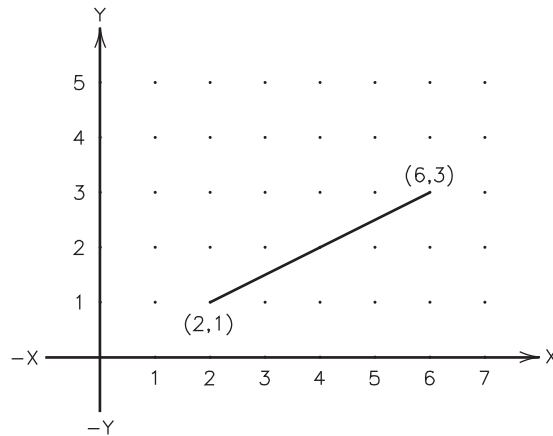
Fig. 7-2

Example:

> **LINE Specify first point: 2,1**
> **Specify next point or [Undo]: 6,3**

This begins the line at absolute point **2,1** and ends it at absolute point **6,3**.

Relative Method

The **relative method** allows you to enter points based on the position of a point that has already been defined. AutoCAD determines the position of the new point relative to a previous point, as shown in Fig. 7-3. Notice the use of the @ symbol in the following example. Using this symbol tells AutoCAD that the following coordinate pair should be read relative to the previous point.

Example:

> **LINE Specify first point: 2,1**
> **Specify next point or [Undo]: @2,0**

This draws a line 2 units in the positive X direction and 0 units in the Y direction from absolute point **2,1**. In other words, the distances **2,0** are relative to the location of the first point.

Fig. 7-3

86 Wohlers: Applying
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I. Groundwork

7. Entering Coordinates

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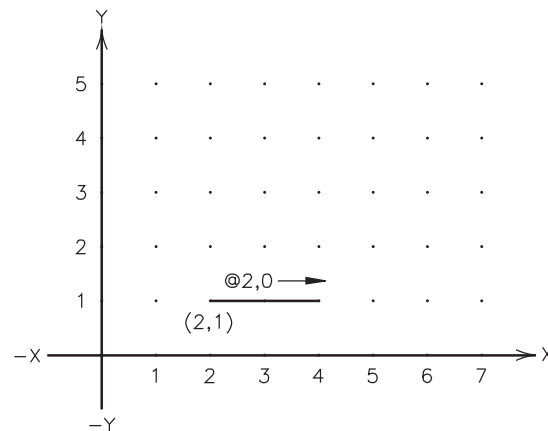
Polar Method

Another form of relative positioning is the **polar method**. Using this method, you can produce lines at precise angles. The area surrounding a point is divided into 360 degrees, as shown in Fig. 7-4. If you specify an angle of 60 degrees, for example, the line extends upward from the last point, as shown in Fig. 7-5. Notice the use of the < symbol to specify the angle.
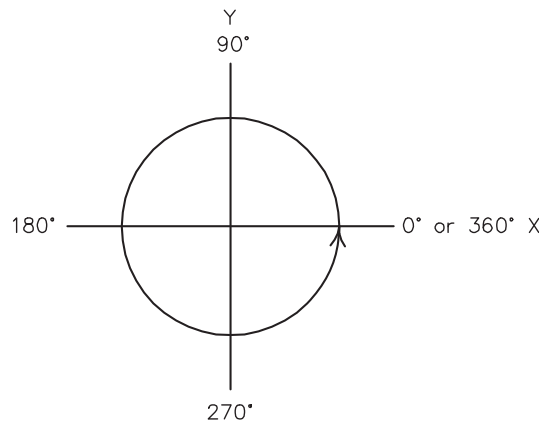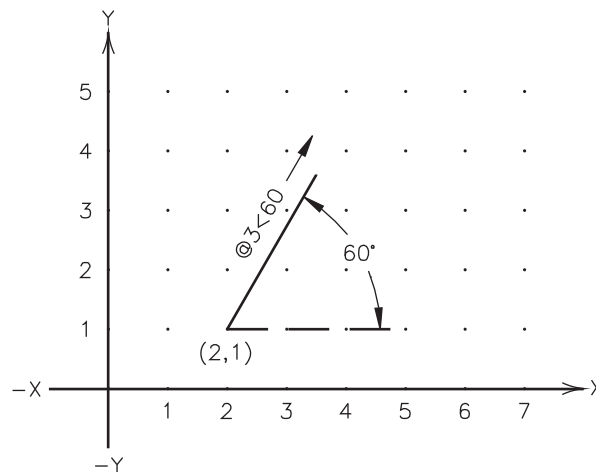

Fig. 7-4

Fig. 7-5

Example:

LINE Specify first point: 2,1
Specify next point or [Undo]: @3<60

This produces a line segment 3 units long at a 60° angle. The line begins at absolute point **2,1** and extends for 3 units at a 60° angle.

Polar Tracking Method

AutoCAD also provides the **polar tracking method**, which is similar to the polar method, except that it's faster and easier. With this method, you can produce lines at precise angles and lengths with minimal keyboard entry. See Fig. 7-6.
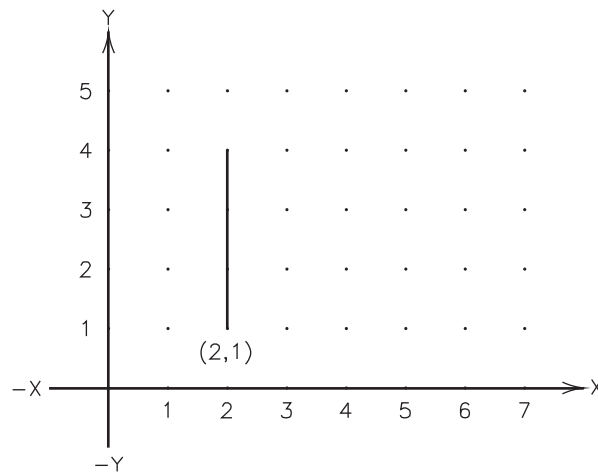
Fig. 7-6

Polar tracking causes AutoCAD to display temporary **alignment paths** at prespecified angles to help you create objects at precise positions and angles. When the cursor follows an alignment path, AutoCAD assumes that the alignment path defines the direction of the line. Polar tracking is on when the **Polar Tracking** button on the status bar is depressed.

Example:

LINE Specify first point: 2,1 Specify next point or [Undo]: Use the 90° alignment path that appears automatically on the screen and enter **3**.

This produces a line segment 3 units long at a 90° angle. The line begins at absolute point **2,1** and extends 3 units upward vertically.

The alignment paths appear at 90° angles by default. You can change the angle of the alignment paths to meet your needs for specific drawing tasks. The following steps guide you through this process.

1. Start AutoCAD, pick the **Workspace Switching** button on the status bar, and select (or re-select) the **2D Drafting & Annotation** workspace.

2. Find the **Polar Tracking** button on the left side of the status bar, right-click, and select **Settings...** from the shortcut menu.
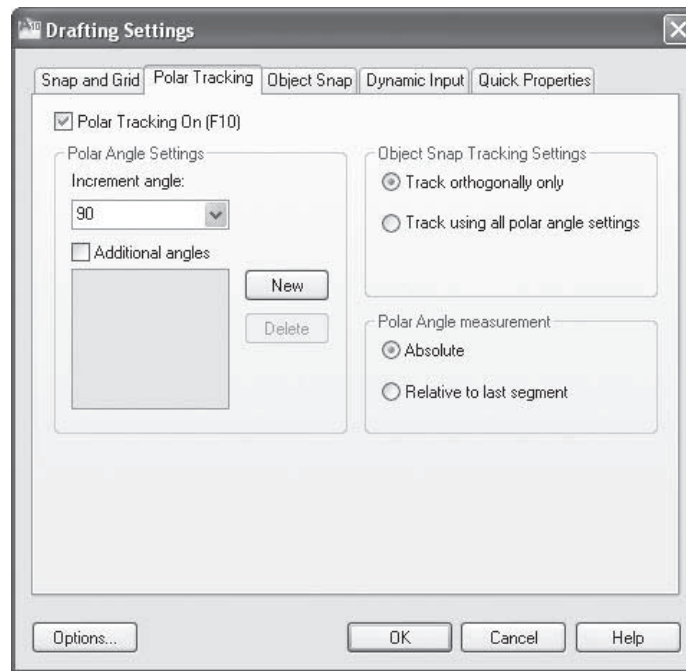

STATUS BAR

Fig. 7-7

The **Drafting Settings** dialog box appears, as shown in Fig. 7-7.

DRAW
Line

3. In the **Polar Angle Settings** area of the dialog box, pick the down arrow and change the increment angle to **30**.

4. Pick the **OK** button to close the dialog box.

5. Enter the **LINE** command and pick a point anywhere on the screen.

6. Move the crosshairs away from and around the point.

At 30° increments around the point, AutoCAD displays an alignment path and a small box showing the distance and angle from the point.

7. Position the crosshairs so that they are at a 60° (1 o'clock) position from the point and enter **2**.

AutoCAD produces a line 2 units long at a 60° angle.

8. Use the **Polar Tracking** feature to produce additional line segments.

9. Erase everything in the drawing file.

10. Display the **Drafting Settings** dialog box, set the polar tracking increment angle to **90**, and pick the **OK** button.

Entering Coordinates with Dynamic Input

When dynamic input is on, entering coordinates at the keyboard produces different results than the methods we have just described. This is because
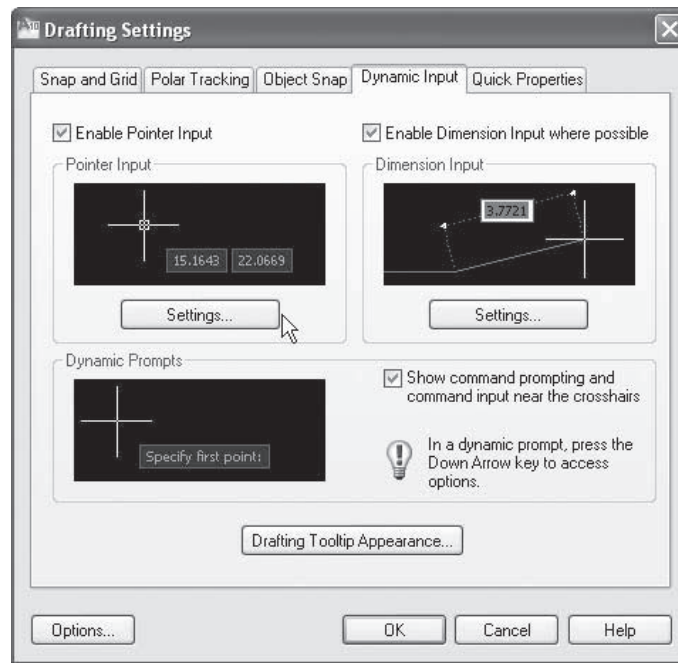
Fig. 7-8

dynamic input uses preset default formats for coordinate entry. Dynamic input is on when the **Dynamic Input** button on the status bar is depressed.

1. Display the **Drafting Settings** dialog box once again and pick the **Dynamic Input** tab.

2. Select the **Settings...** button in the **Pointer Input** area, as shown in Fig. 7-8.

This displays the **Pointer Input Settings** dialog box. As you can see, AutoCAD forces you to choose between polar and Cartesian for the default format and between absolute and relative methods for coordinate entry.

When dynamic input is on, AutoCAD uses these default settings during coordinate entry. Using the @ and < symbols with dynamic input on will produce unpredictable results.

3. Pick the **Cancel** button in the **Pointer Input Settings** dialog box.

4. Pick the **Cancel** button in the **Drafting Settings** dialog box.

Applying Methods of Coordinate Entry

Some methods of coordinate entry work better than others in a given situation. The best way to become familiar with the various methods of coordinate entry, and which to use when, is to practice using them.
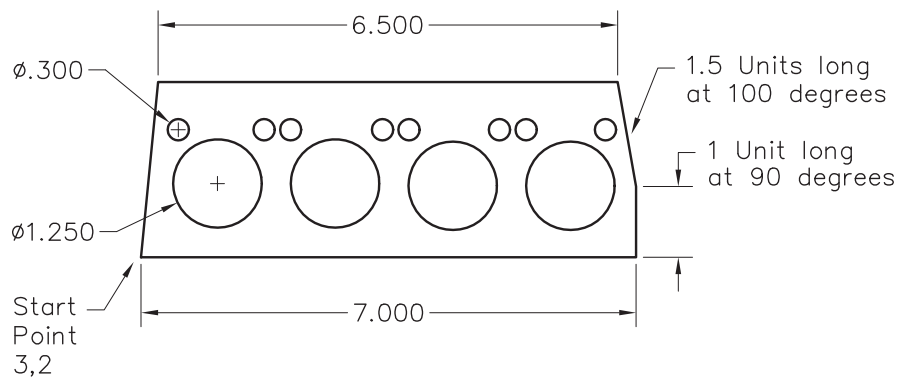
Fig. 7-9

STATUS BAR

1. Turn dynamic input off by releasing the **Dynamic Input** toggle button on the status bar.
2. Enter the **LINE** command.
3. Using the methods presented in the chapter, create the drawing of the gasket shown in Fig. 7-9. Don't worry about the exact locations of the holes, and don't dimension the drawing. However, do make the drawing precisely the size shown. The top and bottom edges of the gasket are perfectly horizontal. (You may need to enter **Z** and **A** to **ZOOM All**. This makes the gasket larger on the screen. You will learn more about **ZOOM All** later in the book.)

> **HINT**
> As you draw the holes in the gasket, keep in mind that the **Ø** (*theta*) symbol indicates that the dimensions are given as diameters.

> **NOTE**
> You can enter negative values for line lengths and angles.
>
> **LINE Specify first point: 5,5**
> **Specify next point or [Undo]: @2<–90**
>
> This polar point specification produces a line segment 2 units long downward vertically from absolute point 5,5. This is the same as entering **@2<270**. Try it and then erase the line.

4. Save your work in a file named **gasket.dwg** and exit AutoCAD.

Chapter 7   Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is the relationship between AutoCAD's world coordinate system and the Cartesian coordinate system?

2. Briefly describe the differences among the absolute, relative, polar, and polar tracking methods of point specification.

3. Explain the advantage of specifying endpoints at the keyboard rather than with the pointing device.

4. What is an advantage of specifying endpoints with the pointing device rather than at the keyboard?

5. What is the effect on the direction of the new line segment when you specify a negative number for polar coordinate entry?

6. In your own words, describe the ways dynamic input might change coordinate entry.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Why may entering absolute points be impractical much of the time when you are completing drawings?

2. Consider the following command sequence:

   LINE Specify first point: 4,3
   Specify next point or [Undo]: 5,1

   What polar coordinates could you enter at the **Specify next point or [Undo]** prompt to achieve exactly the same line?

3. When might the polar tracking feature be useful?

4. When might the default settings of dynamic input be useful?

Continued

Chapter 7 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter. Turn dynamic input off and **ZOOM All** at the beginning of each drawing.

1. Figure 7-10 shows the end view of a structural member for an architectural drawing. Create the rectangle using the **LINE** command and the following keyboard entries.

Fig. 7-10

Specify first point: 1,1

Specify next point or [Undo]: 10.5, 1

Specify next point or [Undo]: @0,5.5

Specify next point or [Close/Undo]: @9.5<180

Specify next point or [Close/Undo]: c

Draw a line from the intersection at the lower left corner of the rectangle to the upper right corner of the rectangle. Use coordinate pairs to place the endpoints of the diagonal line exactly at the corners of the triangle. The new line defines the rectangle's diagonal distance, or the hypotenuse of each of the two triangles you have created. Then draw a line from the lower right corner to the upper left corner of the rectangle to create the other diagonal. Save the drawing as **prb7-1.dwg**.

2. Draw the $3^1/_2''$ disk shown in Fig. 7-11 with dynamic input on. If necessary, temporarily change the default settings for dynamic input in the **Pointer Input Settings** dialog box. Save the drawing as **prb7-2.dwg**.
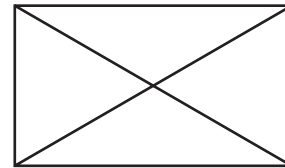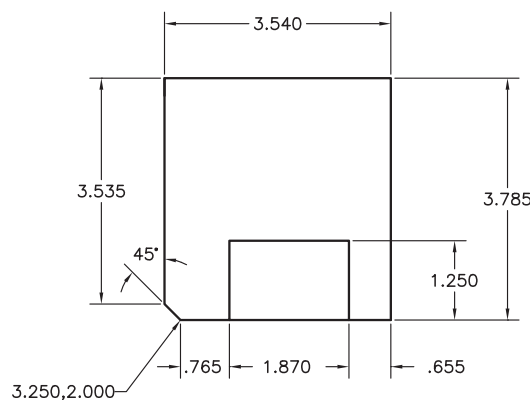

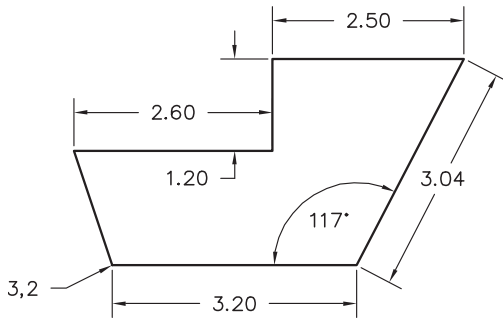**Fig. 7-11**

Review & Activities Chapter 7
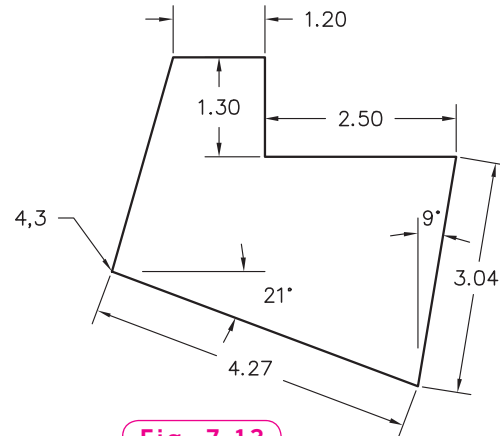

Fig. 7-12

Fig. 7-13

3–4. List on a separate sheet of paper exactly what you would enter when using the **LINE** command to produce the drawings of sheet metal parts shown in Figs. 7-12 and 7-13. Then step through the sequence in AutoCAD. Try to use four methods—absolute, relative, polar, and polar tracking—to enter the points. Note that the horizontal lines in the drawings are perfectly horizontal. Be sure to **ZOOM All** at the beginning of each drawing. Save the drawings as **prb7-3.dwg** and **prb7-4.dwg**.

5. Draw the front view of the spacer plate shown in Fig. 7-14. Start the drawing at the lower left corner by entering a first point using absolute coordinates of **3,3**. Determine and use the best methods of coordinate entry to draw the spacer plate. Save the drawing as **prb7-5.dwg**.
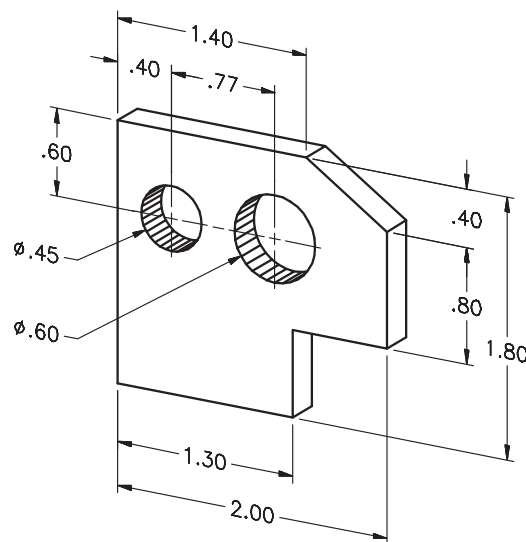

Fig. 7-14

Problem 5 courtesy of Gary J. Hordemann, Gonzaga University

Continued

Chapter 7 **Review & Activities**

6. Draw the front view of the locking end cap shown in Fig. 7-15. Start the drawing at the center of the object using absolute coordinates of **5,5**. Use absolute coordinates to locate the centers of the four holes, and use polar tracking with the **RECTANG** command to draw the rectangular piece. Save the drawing as **prb7-6.dwg**.
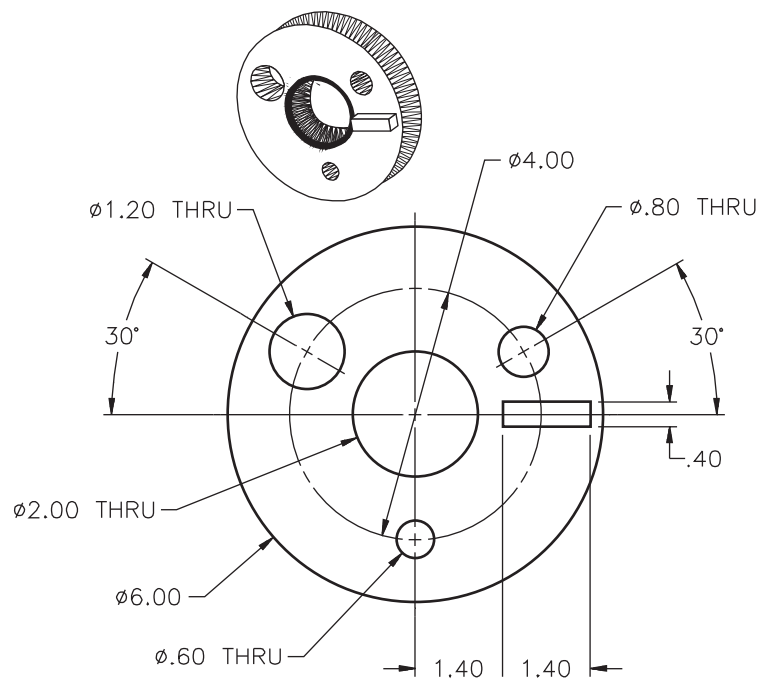

Fig. 7-15

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Draw the shim shown in Fig. 7-16 with dynamic input off. Decide on the best method of coordinate entry for each point, based on the dimensions shown and your knowledge of the various methods. Position the drawing so that the lower left corner of the shim is at absolute coordinates **1,1**. Do not include the dimensions. Save the drawing as **ch7-shim.dwg**.

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7. Entering Coordinates

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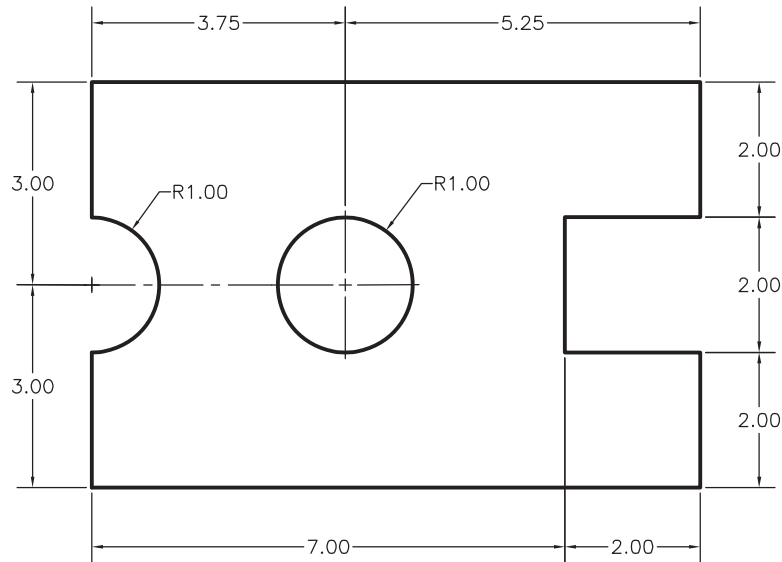Companies, 2010

95

Review & Activities Chapter 7

Fig. 7-16

2. Draw the front view of the slider block shown in Fig. 7-17. Assume the spacing between dots is .125. Start the drawing at the lower left corner using absolute coordinates of **1,3**. Use the **RECTANG** command to draw the rectangular slot. Use relative polar coordinates to locate the endpoints of the lines, and absolute rectangular coordinates to locate the centers of the three holes. Save the drawing as **ch7-block.dwg**.
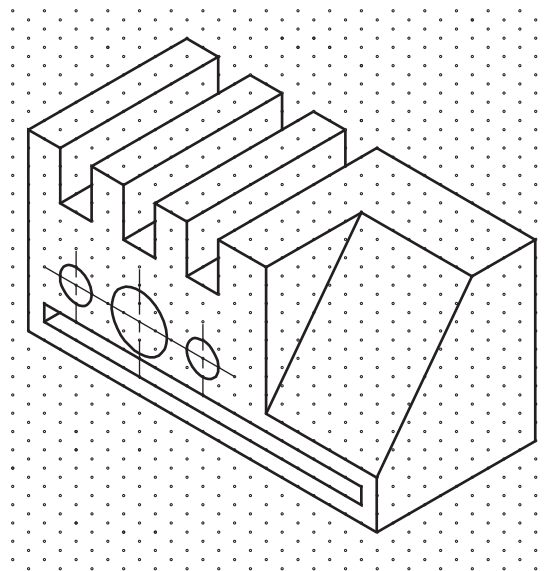
Fig. 7-17

<div style="text-align:right">

C h a p t e r   **8**

</div>

Getting Help

Objectives

- Browse a group of files
- Obtain help on AutoCAD commands and topics
- Use AutoCAD's context-sensitive help features

Vocabulary

context-sensitive help

AutoCAD provides a rich mix of online help and file-searching capabilities. Its digital versions of the *User's Guide, Command Reference, Driver and Peripheral Guide, Installation and Licensing Guides,* and *Customization Guide* are no more than a few clicks away no matter what you are doing in AutoCAD. Several of these guides are also available to you in PDF form. Adobe's® Portable Document Format (PDF) files open with the free Acrobat Reader® software. Several additional guides are also available as PDF files. Autodesk, the maker of AutoCAD, also provides online assistance on how you can obtain consulting, training, and support.

Browsing and Searching

When you open a drawing, AutoCAD provides visual help, making the file selection process faster.

1. Start AutoCAD and re-select the **2D Drafting & Annotation** workspace.

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I. Groundwork

8. Getting Help

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97

2. Pick the **Open** button on the **Quick Access** toolbar.

The **Select File** dialog box appears.

3. From the **Views** pull-down menu, located in the upper right area of the dialog box, select **Details**, unless it is grayed out, which means it is the current selection.

4. Rest the pointer over the name of each drawing file. If this does not produce a thumbnail in the **Preview** area, single-click the file.

As you have seen in previous chapters, the drawings appear in a preview box in the right area of the dialog box. This makes it easier to find drawings that you or others have created and saved.

Finding Drawing Files

Within the **Select File** dialog box, AutoCAD provides other options for reviewing and finding files.

1. From the **Tools** pull-down menu, located in the upper right area of the dialog box, select the **Find...** item.

This displays the **Find** dialog box, as shown in Fig. 8-1.

2. In the text box after **Named**, enter **db_samp** and pick the **Find Now** button.
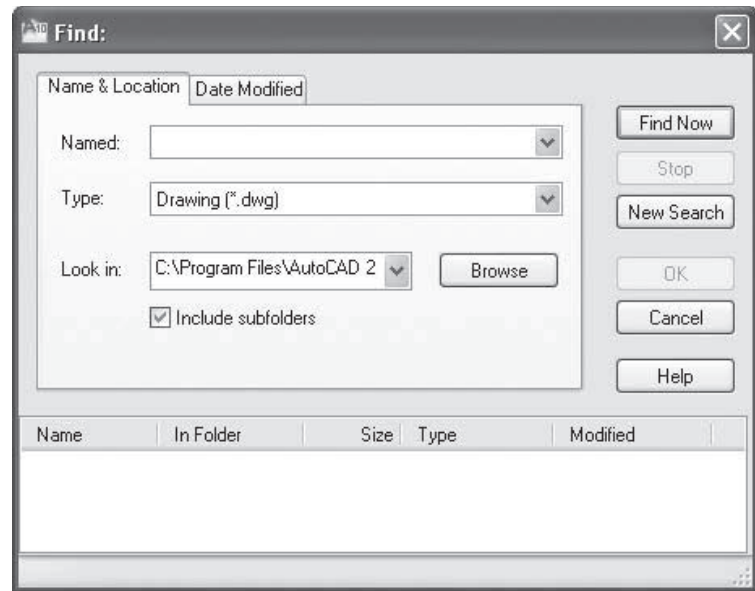
AutoCAD searches for the file named **db_samp.dwg**. If AutoCAD successfully finds it and displays the file details in the lower area of the dialog box, go to Step 4. If the search is unsuccessful, proceed to Step 3.

Fig. 8-1

3. Click the down arrow located at the right of **Look in,** select the disk drive on which AutoCAD was installed, and pick the **Find Now** button.

4. Pick the **OK** button to open the folder containing this file.

5. Select **Find...** again from the **Tools** pull-down menu.

6. Pick the down arrow located at the right of **Type**.

This permits you to specify other file types to search.

7. Click outside the pull-down menu to close it.

8. Pick the **Browse** button.

The **Browse** button enables you to select a specific folder or disk drive to search.

9. Pick the **Cancel** button.

10. Pick the **Date Modified** tab.

This gives you the option of specifying dates within which the file you are seeking was created or modified.

11. Pick the **Cancel** button.

12. Pick the **Cancel** button again to close the **Select File** dialog box.

Obtaining Help

The AutoCAD software offers many forms of help. Most of these can be obtained by entering the **HELP** command.

1. Find and open the **engine.dwg** file.

The cylinder head component that you created in Chapter 5 should now be visible on the screen.

RIBBON

2. Pick the **Help** button on the right side of the **Info Center**, or enter **HELP** or **?** at the keyboard.

> **HINT** You can also obtain help by pressing the **F1** function key.

The **AutoCAD 2010 Help** window appears.

Searching by Content Area

At this point, you have several options for obtaining help.

3. Pick the **Contents** tab, if it is not already selected.

4. Double-click **Command Reference** or click the small box next to it containing the plus (**+**) character.

5. Click the small box containing the plus character next to **Commands**.

6. Click the small box containing the plus character next to **C Commands**.

AutoCAD displays all of the commands and subtopics that begin with the letter C.

7. Click the **COPY** command.

This displays help on the **COPY** command.

8. In the upper left area of the **AutoCAD 2010 Help** window, pick the **Back** button.

This displays the previous screen.

Using the Help Index

Another way to search for help in AutoCAD is to use the help index. Using this method, you can find entries in all of the online help documents by entering just the first few letters of the command or topic for which you need help.

1. Pick the **Index** tab.

2. Use the scroll bar to scroll down through the list of topics.

Notice the blinking cursor inside the empty entry box.

3. Type **M** in upper- or lowercase, but do not press **ENTER**.

AutoCAD very quickly finds the first topic in the list box that begins with the letter M.

4. After the **M**, type **O** and then **V**, but do not press **ENTER**.

AutoCAD finds the first topic that begins with **MOV**.

5. Locate and double-click the item named **MOVE command**.

This displays the topics related to the **MOVE** command on the right side of the **AutoCAD 2010 Help** window.

6. Pick the **Concept, Procedure,** and **Quick Reference** tabs to display more information on these items.

Using the Search Tab

You can also use the **Search** tab to search for specific topics in AutoCAD.

1. Pick the **Search** tab.

2. Click in the edit box and type **move 3D objects**.

3. Pick the **Search** button or press **ENTER** to see a list of related topics.

AutoCAD lists 20 topics that contain any or all of the words *move, 3D,* or *objects*.

4. At the bottom of the list, pick **Show All Results**.

5. Use the scroll bar to move through the topics.

6. Locate and double-click on the item named **Move or Rotate Objects (Procedure)**.

7. On the right side of the **AutoCAD 2010 Help** dialog box, pick the **Concept** tab.

8. Pick **Rotate Objects** and scroll down to see the entire discussion.

The **Search** tab also permits you to phrase a specific question that you want answered.

9. Click in the edit box, type **How do I use the MOVE command?** Then press **ENTER**.

AutoCAD displays a list of topics that may answer your question.

10. Review the list of topics and double-click **MOVE (Quick Reference)**.

AutoCAD displays the **Quick Reference** tab for moving objects on the right side of the dialog box.

11. Close the **AutoCAD 2010 Help** window.

Other Sources of Help

The **Help** button on the **Info Center** provides several sources of help. Selecting the **Help** button is the same as entering the **HELP** command. If you select the arrow to the right of the button, AutoCAD offers other types of assistance, as described below.

New Features Workshop	Helps you learn what is new in AutoCAD 2010.
Additional Resources	Displays the following five options, each of which requires an Internet connection and Web browser.
Support Knowledge Base	Offers information on current issues and permits you to search the **Support Knowledge Base**.
Online Training Resources	Gives instructor-led and self-paced training options, how-to articles, and learning tips.

Online Developer Center	Provides software developers with tools and technologies to develop third-party software applications.
Developer Help	A **Help** window specifically for third-party AutoCAD developers.
Autodesk User Group International	An independent organization composed of local user groups that provides information and networking opportunities.
Send Feedback	Allows users to submit feedback about AutoCAD to Autodesk.
Customer Involvement Program...	Allows Autodesk to collect information about system configurations and how customers use AutoCAD and other Autodesk products.

1. Experiment with the **New Features Workshop**.

2. Close any help-related windows before proceeding to the next section.

Context-Sensitive Help

AutoCAD allows you to obtain help in the middle of a command, when you are most likely to need it. If another command is active when you enter the **HELP** command, AutoCAD automatically displays help for the active command. This is known as **context-sensitive help**. When you close the help window, the command resumes.

1. Enter the **GRID** command at the keyboard.

2. Press the **F1** function key to display help information on the **GRID** command.

HINT
To display the **AutoCAD 2010 Help** window, you may need to click the **AutoCAD 2010 Help** button on the Windows taskbar located at the bottom of the screen after you pick the **Help** button. This will bring the window to the foreground.

3. Close the help window, press the **ESC** key to cancel the **GRID** command, and exit AutoCAD without saving.

Chapter 8 Review & Activities

• REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Suppose you want to open a specific drawing file in the folder you named after yourself, but you have forgotten the name of it. Explain how to browse through the files in that folder to find the file you want to open.

2. When you pick the **HELP** button, what does AutoCAD display?

3. How do you obtain a listing of all AutoCAD topics?

4. Suppose you have entered the **MIRROR** command. At this point, what is the fastest way of obtaining help on this command?

5. Name and briefly describe each of the options available in the **Help** menu.

6. What is context-sensitive help? Explain how to use it in AutoCAD.

• CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. For some commands, the help information may be long and complicated. Discuss ways to make a permanent hard copy of commonly used help information to keep for reference.

2. Explain the benefit of typing questions in the **Search** tab of the **AutoCAD 2010 Help** window. When would you use this method instead of the **Index**?

Review & Activities Chapter 8

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. You know you have a drawing of a house somewhere on your hard drive, but you don't remember the full name of the drawing. You recall that it includes "**Textstyles,**" and because it's a drawing file, you know that it has a **DWG** file extension. Find the drawing. What is its name and what does it contain?

2. Obtain AutoCAD help on the following commands and state the basic purpose of each.

   a. **EXPORT**

   b. **SNAP**

   c. **TIME**

   d. **SCALE**

   e. **TRIM**

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Your company has purchased a new plotter for the AutoCAD network. You have been asked to configure it, but since you don't do many configurations, you need some help. Find the instructions in the AutoCAD help system for configuring a network nonsystem plotter. Write down the path for future reference.

2. You need to draw two concentric circles. You have already drawn the inner circle. The outer circle needs to be .5″ larger. Enter the **OFFSET** command from the **Modify** panel of the **Ribbon** and access context-sensitive help to offset the diameter of the first circle by .5″.

Continued

AutoCAD at Work

Mechanical Drafter

Mechanical devices are all around you. At the grocery store, conveyer belts help move shoppers' selections from their carts to the cashier. Mechanics use hydraulic lifts to raise cars in order to work on parts underneath. Military personnel use gauges, tracking devices, and other equipment. Have you ever taken a mechanical object apart just to see how it works? If so, you might be interested in learning more about mechanical drafting.

On the Job

Mechanical drafters make up the largest category of drafters in the United States. They design machinery and mechanical tools. Accurate, detailed drawings of mechanical equipment must be provided for manufacturers to use as guides during the building process. Today, most mechanical drafters use AutoCAD or a similar CAD program to assist with the overall design process.

A Mechanical Mindframe

Curiosity about how mechanical devices operate and the ability to visualize how

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an object might look from all angles are necessary skills for a mechanical drafter. The ability to communicate and work well with others is also important.

Mechanical drafters often consult with the individuals who initially thought up product ideas; they also confer regularly with other drafters and mechanical engineers. Perhaps most importantly, mechanical drafters must have a knack for understanding exactly how mechanical items work. Attention to detail is important both when studying existing devices to improve future designs and when creating the detail drawings for new products.

▶ Career Activities

1. Look around the room. Make a list of the mechanical devices you see.

2. Try to sketch a rough example of how you think a design drawing might look. Share your drawing with a classmate and compare ideas.

File Maintenance

Objectives

- Copy, rename, move, and delete drawing files
- Create and delete folders
- Audit and recover damaged drawing files

Vocabulary

audits
file attributes

When you work with AutoCAD, there will be times when you want to review the contents of a folder or delete, rename, copy, or move files. This chapter gives you practice in using these functions. The chapter also discusses commands that attempt to repair drawing files containing errors.

File and Folder Navigation

AutoCAD's **Select File** dialog box can help you organize and manage AutoCAD-related files.

1. Start AutoCAD and reselect the **2D Drafting & Annotation** workspace.
2. Pick the **Open** button from the **Quick Access** toolbar.

AutoCAD displays the **Select File** dialog box, as shown in Fig. 9-1 on page 104.

3. Find and open the folder with your name—the one you created in Chapter 4.
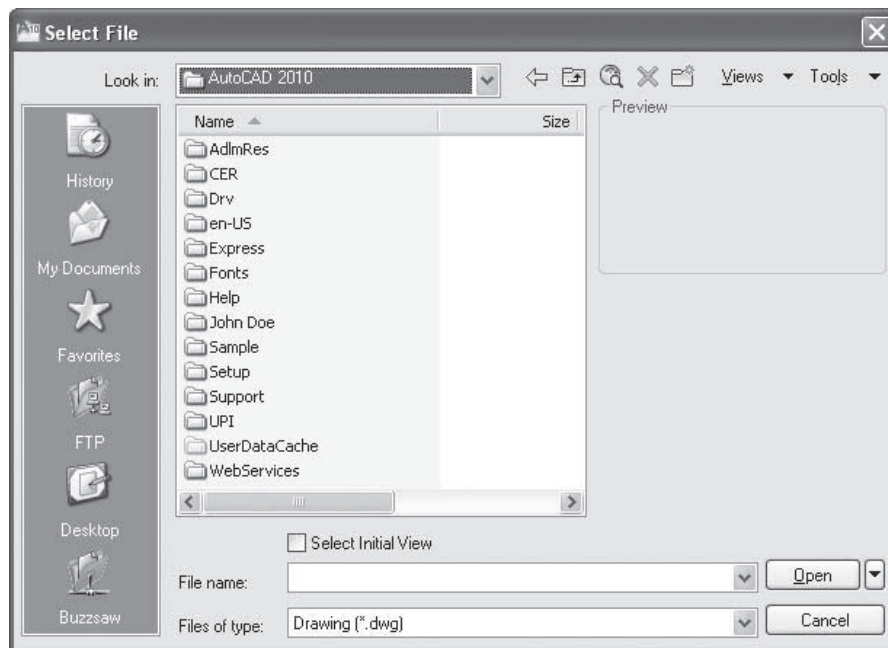
Fig. 9-1

Viewing File Details

4. Pick the **Views** drop-down menu and select **Details**, unless it is grayed out, which means it is the current selection.

This provides a listing of each file, along with its size, type, the author and the date and time it was created and last modified. These items are known as **file attributes**. In folders that contain many drawing files, it is sometimes easier to find files if they are listed in a certain order. By default, files are listed in alphabetical order by name. You can change the order in which the files and folders are listed by clicking the column names, which are actually buttons.

5. Click the gray bar labeled **Name**.

This reverses the order of the files. They are now in reverse alphabetical order (from Z to A).

6. Click the same gray **Name** bar again.

The files are once again listed in alphabetical order.

7. Click the gray bar labeled **Size**.

This places the files in order of size from smallest to largest. Clicking it again would reorder the files from largest to smallest. Clicking the gray bars labeled **Type** and **Date Modified** organizes the files according to type and date,

respectively. You may need to resize the columns or use the scroll bar to view
these options.

8. Click the gray bar labeled **Date Modified**; click it again.

9. From the **Views** drop-down menu, pick **List** to produce a basic listing of
the files.

Copying Files

The **Select File** dialog box permits you to copy files easily from within
AutoCAD.

1. Right-click and drag the **stuff.dwg** file to an empty area within the list box
area and release the right button on the pointing device.

2. Pick **Copy Here** from the shortcut menu that appears.

A new file named **Copy of stuff.dwg** appears.

Renaming Files

You can also rename files using the **Select File** dialog box.

1. Right-click the new file named **Copy of stuff.dwg**.

2. Select **Rename** from the shortcut menu.

You should now see a blinking cursor at the end of the file name with a box
around it.

3. Type the name **junk.dwg** and press **ENTER**.

The new file named **junk.dwg** has the same contents as **stuff.dwg**.

Creating New Folders

It's also fast and easy to create new folders.

1. Pick the **Create New Folder** button.

A new folder named **New Folder** appears.

2. To rename the folder, right-click the new folder, select **Rename** from the
menu, type **Junk Files**, and press **ENTER**.

Moving Files

Let's move a file to the new folder.

1. Right-click **junk.dwg** and drag and drop it into the **Junk Files** folder.

A shortcut menu appears.

2. Pick **Move Here**.

The **junk.dwg** file is now located in the **Junk Files** folder.

Deleting Files and Folders

AutoCAD also allows you to delete unwanted files and folders using the **Select File** dialog box.

1. Open the **Junk Files** folder.
2. Right-click **junk.dwg** and select **Delete** from the shortcut menu.
3. Pick the **Yes** button in the **Confirm File Delete** dialog box if you are sure you have selected **junk.dwg**.
4. Pick the **Up one level** button.
5. Right-click the **Junk Files** folder and select **Delete** from the shortcut menu.
6. If you are sure that you selected the **Junk Files** folder, pick the **Yes** button in the **Confirm Folder Delete** dialog box to send this folder to the Windows Recycle Bin.
7. Close the **Select File** dialog box.

> **NOTE**
>
> You can use methods similar to those discussed in this section even when AutoCAD is not open. The Microsoft Windows environment allows you to copy, rename, move, and delete files of all types, including AutoCAD drawing files.

Diagnosing, Repairing, and Recovering Files

Occasionally, AutoCAD drawing files may become corrupted. AutoCAD supplies two commands to help fix corrupted files.

Auditing a Drawing File

The **AUDIT** command is available as a diagnostic tool. It **audits**, or examines the validity of, the current drawing file. It can also correct some of the errors it finds. The **AUDIT** command automatically creates an audit report file containing an **ADT** file extension when the **AUDITCTL** system variable is set to **1** (**On**). The default setting is **0**.

Probable causes of damaged files include:

- An AutoCAD system crash
- A power surge or disk error while AutoCAD is writing the file to disk

When a file becomes damaged, AutoCAD may refuse to edit or plot the drawing. In some cases, a damaged file may cause an AutoCAD internal error or fatal error.

1. Open the file named **stuff.dwg**.
2. On the **Menu Browser**, select **Drawing Utilities** and **Audit**.

This enters the **AUDIT** command.

3. In reply to **Fix any errors detected?** press **ENTER** to accept the **No** default.
4. Press the **F2** function key to view the **AutoCAD Text Window** and read all of the message.

The **AutoCAD Text Window** displays information that is similar but not necessarily identical to that shown in Fig. 9-2.
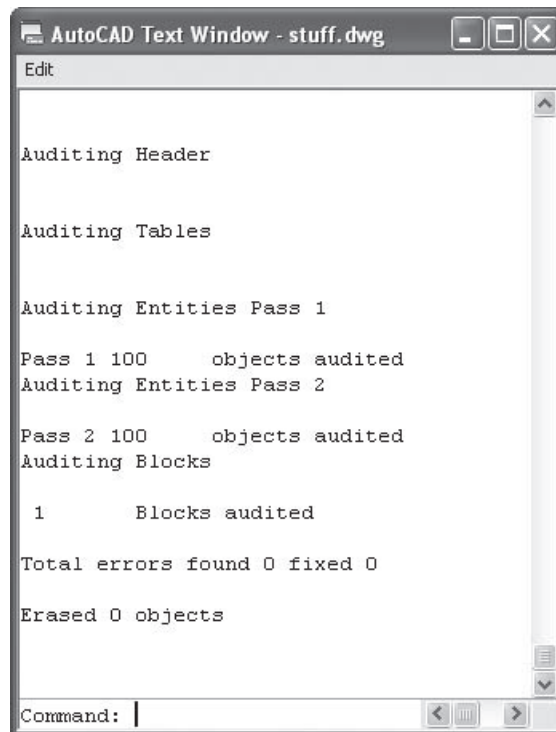
Fig. 9-2

If you had entered **Yes** in reply to **Fix any errors detected?** the report would have been the same, as long as the file contained no errors.

> AutoCAD offers a command similar to **AUDIT** named **RECOVER**. This command attempts to open and repair damaged drawing files. The recovery process has been embedded in the **OPEN** command, so when you try to open a damaged file, AutoCAD automatically executes the **RECOVER** command and attempts to repair it.
>
> If you know or suspect that a file has been damaged, you can enter the **RECOVER** command directly at the AutoCAD prompt. This produces the **Select File** dialog box, from which you can choose the file to be recovered.

Permanently Damaged Files

A drawing file may be damaged beyond repair. If so, drawing recovery with the **AUDIT** and **RECOVER** commands will not be successful.

Each time you save an AutoCAD file, AutoCAD saves the changes to the current **DWG** file. AutoCAD also creates a second file with a **BAK** (backup) file extension. This file contains the previous version of the file—the version prior to saving. If the **DWG** file becomes damaged beyond repair, you can rename the **BAK** file to a **DWG** file and use it instead. To prevent the loss of data, save often and produce a backup copy of your **DWG** files frequently.

The Drawing Recovery Manager

When a hardware problem, power failure, or software problem causes AutoCAD to terminate unexpectedly, the **Drawing Recovery Manager** opens the next time you start AutoCAD. It displays a list of the drawing files that were open at the time the program terminated.

If you pick a file, AutoCAD lists all the available versions of the drawing that may be recovered, including the **DWG** file, the **BAK** file, and the **SV$** (auto-save) file. These file types are listed in the order of their time stamps. You can double-click the most recent file, and AutoCAD will attempt to repair and open it. This allows you to recover the most recently saved version of the drawing that is available.

5. Exit AutoCAD without saving unless the file was damaged and recovered. In that case, save the changes to the file.

Chapter 9 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is the purpose of each of the five items located in the upper right area of the **Select File** dialog box?
2. How do you copy a file in the **Select File** dialog box?
3. In the **Select File** dialog box, what information can you display by picking **Details** from the **Views** drop-down menu?
4. How do you create a folder from within AutoCAD?
5. What is the primary purpose of the **AUDIT** command?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Is it possible to move and copy files from the **Select File** dialog box to folders on the Windows desktop? Is it possible to move and copy files from folders on the Windows desktop to the **Select File** dialog box? Explain why this might be useful.
2. AutoCAD gives you the option of creating a report file when you run the **AUDIT** command. Explain why such a report might be useful.

● APPLYING AUTOCAD SKILLS

Using the features described in this chapter, complete the following activities. Complete problems 1 through 5 from within the **Select File** dialog box.

1. Display a list of drawing files from one of AutoCAD's folders.
2. Display a list of template files found in one of AutoCAD's folders.
3. Rename one of your drawing files. Then change it back to its original name.

Continued

Chapter 9 **Review & Activities**

4. Make a copy of the file **Welding.dwg** in the AutoCAD **Sample/ DesignCenter** folder. Rename the copy **MyWelding.dwg** and move it to your named folder.

5. Delete **MyWelding.dwg** from your named folder.

6. Diagnose one of your drawing files and fix any errors detected. Display the text of the AutoCAD message. How many passes did the diagnostic tool run? How many objects were included?

• USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. It's time to build the bookcase you designed in Chapter 4. Make a copy of the bookcase you saved in your named folder as **prb4-1.dwg**. Modify the drawing as shown in Fig. 9-3 and save the modification as **bookcase.dwg**. Delete the copy of **prb4-1.dwg**.
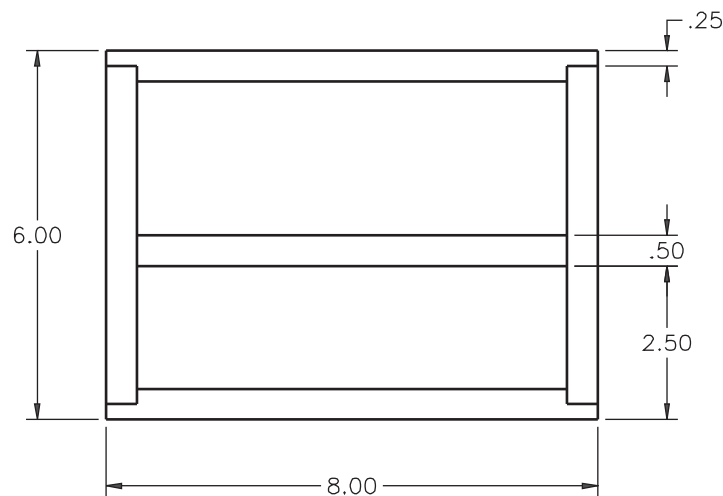
Fig. 9-3

2. To keep track of projects, you need to organize them into folders. Make a new folder called **Build** within your named folder. Move **bookcase.dwg** into the new folder and rename it **shelves.dwg**. Check the **shelves.dwg** file for corruption.

AutoCAD at Work

Ship Designer

If you have seen the movie "Titanic," you will surely remember the scene when the ship's designer shared his fears about the Titanic's fate. He knew there were not enough life boats to take all of the passengers to safety. He had been meticulous in every other aspect of ship design, but his superiors had not thought that adding more life boats was necessary. The designer was only allowed to plan room enough for a few of these small vessels. As a result, many people were left behind as the massive Titanic slowly sank into the ocean.

Giant ocean liners aren't the only types of ships that require careful planning during the initial design phase. Other marine vessels, such as tankers, barges, submarines, yachts, and aircraft carriers, also require the expertise of drafters to build safe and functional vessels.

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erly. Hull designers prepare drawings that provide exact measurements of ship dimensions. Ship detail drafters create plans for electrical and plumbing fixtures. Because the people who specialize in designing and building ships exhibit expertise in a variety of areas, the planning process is an excellent example of teamwork.

Special Skills

Drafters who want to specialize in one of the many specialties related to shipbuilding must acquire other skills in addition to basic drafting skills. They must develop an understanding of basic nautical safety. They must also understand how various building materials interact with fresh and salt water.

Who Designs Ships?

Currently, 22 major shipyards are operational in the United States. Drafters specializing in ship design work closely with naval architects and marine engineers to design the machinery ships need to operate prop-

▶ Career Activities

1. Visit the Shipbuilders Council of America (www.shipbuilders.org) and make a list of job descriptions pertaining to shipbuilding.

2. How might you become familiar with ships and ship design? Explain.

Part 1 Project | Applying Chapters 1–9

Rocker Arm

Engineering drawing refers to the creation of highly accurate drawings using drafting instruments or, more commonly today, CAD software. Engineering drawings are used, among other things, to describe a part so that the manufacturer can build it correctly.

▶ Description

The rocker arm shown in Fig. P1-1 was submitted to your manufacturing company as a hand sketch. The basic design has been approved, and now the marketing department needs an electronic version of the drawing for use in a new product brochure. The drawing must reflect the actual dimensions of the rocker arm, although for this use, it is not necessary to show the dimensions on the drawing.

Your task is to draw the rocker arm accurately using the AutoCAD commands and procedures you learned in Part 1. Be sure to read the "Hints and Suggestions" for this project before you begin. Save the drawing as **rockerarm.dwg** in your named folder.
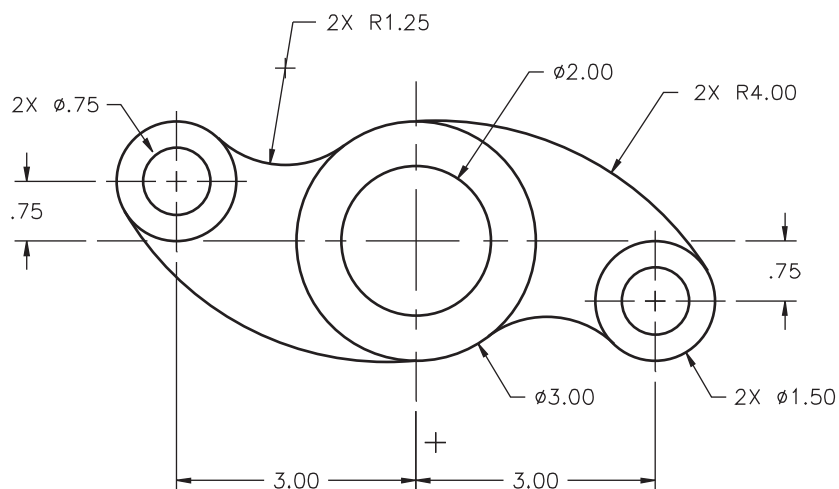
Fig. P1-1

▶ Hints and Suggestions

1. The lines made up of short and long dashes are center lines (lines that intersect at the exact center of a feature). Use these lines and your knowledge of coordinates to draw the rocker arm accurately.

2. Study the drawing to determine the best order in which to draw the parts of the rocker arm. Keep in mind that it is *not* necessary to construct the part linearly, so that each line connects to the next. For example, you may choose to create all of the holes and cylindrical features before you draw the other parts of the rocker arm.

3. If you have trouble visualizing how to draw the part in AutoCAD, try sketching it first on graph paper.

▶ Summary Questions

Your instructor may direct you to answer these questions orally or in writing. You may wish to compare and exchange ideas with other students to help increase efficiency and productivity.

1. Pinpoint any difficulties you had in drawing the rocker arm and explain how you solved them.

2. Describe the order in which you drew the parts to make up the rocker arm. Why did you choose this order?

3. Study the drawing in P1-1 once more. Does this drawing provide everything needed to manufacture the actual part? If not, what is missing?

4. In this project, you have created an electronic drawing file to be used by the marketing department because marketing needed the file in electronic format. However, there are other advantages to creating the file electronically. How else might this file be used? How would it need to be modified to be useful to other departments? Explain your answer.

Chapter 10

Object Snap

Objectives

- Set and use running object snap modes
- Apply object snaps that are not currently turned on
- Change object snap settings to increase productivity
- Use the **Quick Setup** wizard

Vocabulary

aperture box
object snaps
object snap tracking
perpendicular
quadrant
quadrant points
running object snap modes
running object snaps
system variable
wizard

One of the most useful features of AutoCAD is the **object snap** feature. Object snaps are like magnets that permit you to pick endpoints, midpoints, center points, points of tangency, and other specific points easily and accurately. **Object snap tracking**, which is used in conjunction with object snaps, provides alignment paths that help you produce objects at precise positions and angles. Combined with polar tracking, these features speed the precision drawing process.

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II. Drawing Aids and
Controls

10. Object Snap

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121

Running Object Snaps

You can use object snaps in different ways to make a drawing task easier. When you know that you will be needing one or more specific object snaps several times, you can preset them to "run" in the background as you are working. Object snaps that have been preset in this way are known as **running object snaps**.

Quick Setup Wizard

In preparation for using object snaps, we will use the **Quick Setup** wizard to establish basic settings quickly in AutoCAD. **Quick Setup** is based on the **acad.dwt** template file.

1. Start AutoCAD, and re-select the **2D Drafting & Annotation** workspace.

2. Enter **STARTUP** at the keyboard and enter **1**.

STARTUP is a system variable. A **system variable** is a setting that controls a specific function in AutoCAD. The default value of the **STARTUP** system variable is **0**. When it is set at **1**, AutoCAD displays the **Startup** dialog box when you start the software.

3. Enter the **NEW** command.

The **Create New Drawing** dialog box appears. The four buttons at the top permit you to open a drawing, start a drawing from scratch, use a template file, or use a wizard. A **wizard** is a prompted sequence of steps that streamlines a task or set of tasks. Using the **Start from Scratch** button is equivalent to beginning a new drawing with **STARTUP** set at **0**. In any case, all of the settings come from the **acad.dwt** template file.

4. Pick the **Use a Wizard** (fourth) button.

5. Select **Quick Setup** and pick **OK**.

AutoCAD displays a dialog box that focuses on the unit of measurement. **Decimal** is the default selection. Notice the sample in the right area of the box.

6. Accept the **Decimal** default selection and pick the **Next** button.

The dialog box concentrates on the drawing area. As you can see, the default drawing area is 12×9 units.

7. For **Width**, enter **95**, and for **Length**, enter **70**.

8. Pick the **Finish** button and save your work in a file named **bike.dwg**.

The drawing area now represents 95 × 70 units. These units can represent millimeters, kilometers, feet, miles, or whatever we want. We will use them as inches.

9. Enter **Z** for **ZOOM** and **A** for **All**.

This zooms the drawing area to the full 95 × 70 inches. You will learn more about the **ZOOM** command in Chapter 13.

Specifying Object Snap Modes

1. Use the pointing device to depress the **Polar Tracking, Object Snap, Object Snap Tracking**, and **Dynamic Input** toggle buttons on the status bar, unless they are already depressed.

When status bar toggle buttons are depressed, they have a blue background. When they are released, they have a grey background. The **Snap Mode, Grid Display,** and **Ortho Mode** buttons should not be depressed.

2. Enter the **OSNAP** command.

This displays the **Drafting Settings** dialog box, as shown in Fig. 10-1. Entering the **OSNAP** command is equivalent to right-clicking the **Object Snap** button on the status bar and selecting **Settings...** from the shortcut menu.
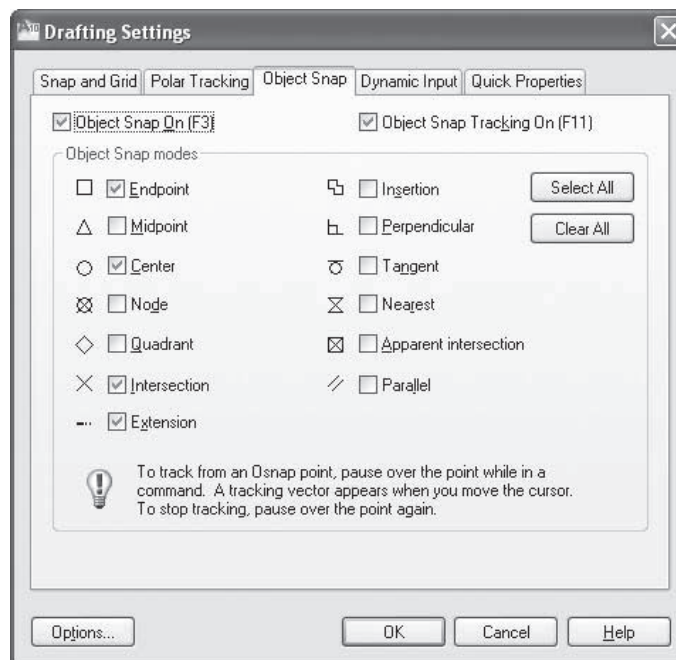
Fig. 10-1

3. Review the object snap modes.

The ones that are checked are called **running object snap modes**. These object snaps are "on" and function automatically as you work with drawing and editing commands in AutoCAD.

4. Check **Endpoint**, **Center**, **Intersection**, and **Extension**, and uncheck all the others. (Some or all of them may be checked already.)

Notice that **Object Snap On** and **Object Snap Tracking On** are checked. This is because the **OSNAP** and **OTRACK** buttons on the status bar are on (depressed). A third way to turn them on and off is to press the **F3** and **F11** function keys, as indicated in the dialog box.

5. Pick the **OK** button.

Using Preset Object Snaps

Let's use AutoCAD's object snaps to create a basic drawing of the new mountain bike frame design shown in Fig. 10-2. Be sure to save your work every few minutes as you work through this chapter.
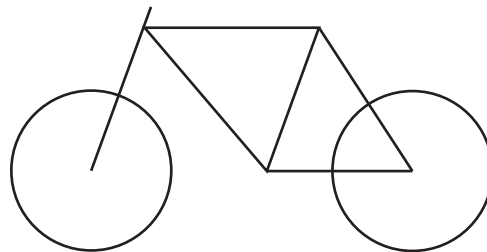
Fig. 10-2

1. In the lower left area of the screen, draw a 22″-diameter circle to represent the front wheel, as shown in Fig. 10-2.

The **Center** object snap selects the exact center of a circle or arc. Let's use it to start a 24″ line at the center of the bike's front wheel. This line will begin the outline of the bike frame.

2. Enter the **LINE** command.

3. At the **Specify first point** prompt, move the crosshairs so that it touches the circle or move the crosshairs to the circle's center.

A small yellow marker appears at the circle's center. The yellow marker indicates that AutoCAD has selected the point at the center of the circle.

DRAW
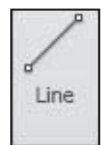
DRAW

4. While the yellow marker is present, pick a point.

AutoCAD snaps to the circle's center.

5. For the next endpoint, use polar coordinates to extend the line **24″** up and to the right at a **70°** angle.

HINT Type **24**, press the **TAB** key, type **70**, and press **ENTER**, or enter **@24<70** at the keyboard.

6. Press **ENTER** to terminate the **LINE** command.

Object Snap Tracking

Next, we are going to draw the horizontal line using AutoCAD's **object snap tracking** feature.

1. Reenter the **LINE** command and move the crosshairs to the second endpoint of the line until the **Endpoint** object snap marker appears.

The purpose of doing this is to "acquire" the **Endpoint** object snap. When you acquire an object snap, a small yellow **+** (plus sign) appears at the snap point. An acquired object snap also displays a small white **x** near the crosshairs. To remove an acquired object snap, pass over the snap point a second time.

2. Slowly move the crosshairs around the endpoint and notice the temporary alignment paths that appear.

As you may recall, alignment paths are temporary lines that display at specific angles from an acquired object snap. Object snap tracking is active when the alignment paths appear from one or more acquired object snaps. This feature is a part of AutoCAD's AutoTrack™. You can toggle AutoTrack on and off with the **Polar Tracking** and **Object Snap Tracking** buttons on the status bar. Because object snap tracking works in conjunction with object snaps, one or more object snaps must be set before you can track from an object's snap point.

3. Move the crosshairs down the line that you created earlier so that a small **x** appears on the line.

4. Enter **3**.

This starts a new line 3″ from the endpoint of the first line.

5. Draw a horizontal line segment 24″ to the right.

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II. Drawing Aids and
Controls

10. Object Snap

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125

HINT When creating the second point, use polar tracking to force the line horizontal and enter **24**.

6. In reply to **Specify next point**, click at the **Command** line and enter **@21<250** to create the next line segment.

NOTE The dynamic input prompt can also be used, but the result will depend on the position of the crosshairs at the time you type the angle value of 250. This is because AutoCAD measures the angle either clockwise or counterclockwise from the horizontal, depending on where you position the crosshairs.

7. In reply to **Specify next point**, move the crosshairs to the intersection of the first and second line segments.

NOTE The intersection of the two lines is also an endpoint of the second line. Therefore, either the **Intersection** or **Endpoint** object snap marker will appear. A yellow **x**-shaped marker depicts the **Intersection** object snap, and a square marker indicates the **Endpoint** object snap.

8. Snap to this point and press **ENTER** to terminate the **LINE** command.

9. Using the **Endpoint** object snap and polar tracking, create the second horizontal line. Make it 20″ long.

10. Create the final segment of the bike frame, using Fig. 10-2 on page 117 as a guide.

11. Create the second wheel for the bike. Use the **Endpoint** object snap to place the center of the wheel at the right endpoint of the lower horizontal line. Make it the same size as the front wheel.

The drawing of the new bike design is now complete.

12. Save your work.

Specifying Object Snaps Individually

Another way to use object snaps is to specify them individually as you need them. You can do this either at the keyboard or by displaying and using the **Object Snap** toolbar. Using this toolbar streamlines the selection of object snaps. Displaying a toolbar in the **2D Drafting & Annotation** workspace is a two-step process.

1. Click the down-facing arrow on the right side of the **Quick Assess** toolbar and select **Show Menu Bar** from the shortcut menu.

The **Menu Bar** appears between the title bar and the tabs of the **Ribbon**.

> **HINT** The **Menu Bar** provides easy access to many useful commands. It may be useful to keep the **Menu Bar** displayed at all times when working in AutoCAD.

2. Select **Tools** on the **Menu Bar**, then **Toolbars**, from the pull-down menu, then **AutoCAD**, and finally **Object Snap** from the cascading menus.

The **Object Snap** toolbar appears, as shown in Fig. 10-3.

Fig. 10-3

3. Rest the pointer on each of the buttons and read the tooltips.

Table 10-1 contains a list of object snap modes and their functions. To enter an object snap mode at the keyboard, enter only the capitalized letters.

Snapping to Quadrant Points

OBJECT SNAP

1. Enter the **LINE** command.
2. From the **Object Snap** toolbar, pick the **Snap to Quadrant** button.

Quadrant points on a circle are the points at 0°, 90°, 180°, and 270°. If you were to connect these points with horizontal and vertical lines, as shown in Fig. 10-4, you would divide the circle into four equal parts. Each part is a **quadrant** of the circle.
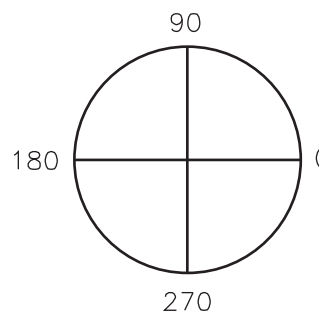
Fig. 10-4

Table 10-1. AutoCAD's Object Snaps

Object Snap	Button	Purpose
APParent intersection		Snaps to the apparent intersection of two intersection objects that may not actually intersect in 3D space
CENter		Center of an arc circle, ellipse, or elliptical arc
ENDpoint		Closest endpoint of a line or arc
EXTension		Objects along the extension paths of objects
INSert		Insertion point of text, block, or attribute
INTersection		Intersection of two objects
MIDpoint		Midpoint of an object
NEArest		Nearest point on an object
NODe		Nearest point entity or dimension definition point
NONe		Temporarily cancels all running object snaps (for one operation only)
PARallel		Snaps to a point on an alignment path that is parallel to the selected object
PERpendicular		Perpendicular to a line, arc, or circle
QUAdrant		Quadrant point of an arc, circle, ellipse, or elliptical arc
TANgent		Tangent to an arc, circle, ellipse, elliptical arc, or spline
FROm		Used in combination with other object snaps to establish a temporary reference point as a base point
Object Snap Settings		Displays the Object Snap tab of the Drafting Settings dialog box
Temporary Tracking Point		Allows you to set a temporary OTRACK point while a drawing command is active

3. Move the crosshairs over one of the two wheels until a small yellow diamond marker appears, along with the tooltip **Quadrant**.

4. Move to each of the four quadrant points and pick one of them.

5. Enter **ESC** to cancel the **LINE** command.

Snapping Perpendicularly

The **Perpendicular** object snap allows you to snap to a point perpendicular to an object such as a line, multiline, polyline, ray, spline, construction line, or even a point on a circle or arc. (A line is **perpendicular** to another line when the two form a right angle.)

1. Enter the **LINE** command and pick a point anywhere above the top horizontal segment of the bike.

2. Pick the **Snap to Perpendicular** button from the **Object Snap** toolbar.

3. Move the crosshairs along the top horizontal segment of the frame until a right angle marker appears, along with the tooltip **Perpendicular**, and pick a point.

OBJECT SNAP

AutoCAD snaps the line perfectly perpendicular to the horizontal segment of the frame.

4. Press **ENTER** to terminate the **LINE** command and pick the **Undo** button to remove the line.

QUICK ACCESS

Snapping Parallel Lines

You can use the parallel object snap to produce parallel line segments.

1. Reenter the **LINE** command, pick another point above the bike, and pick the **Snap to Parallel** object snap from the toolbar.

OBJECT SNAP

2. Touch the crosshairs on the fork of the frame (the first line you drew) and then move the crosshairs until a parallel marker appears, and then back and forth until a temporary line is parallel to the fork.

3. Pick a point anywhere to form the parallel line.

4. Press **ENTER** and pick the **Undo** button.

Snapping to Points of Tangency

The **Tangent** object snap permits you to snap to a point on an arc or circle that, when connected to the last point, forms a line tangent to that object.

1. Enter the **LINE** command and pick the **Snap to Tangent** button on the **Object Snap** toolbar.

OBJECT SNAP

2. Move the crosshairs along the bottom half of the front wheel until a **Snap to Tangent** marker appears, along with the tooltip **Deferred Tangent**.

AutoCAD defers the point of tangency until after you have picked both end-points of the line segment.

3. Pick a point anywhere along the bottom half of the front wheel.

4. Pick the **Snap to Tangent** button again.

5. Move the crosshairs along the bottom half of the rear wheel until the **Snap to Tangent** marker appears and pick a point.

OBJECT SNAP

AutoCAD draws a line tangent to both wheels.

6. Using polar tracking, extend the line 12″ to the right and press **ENTER** to end the **LINE** command.

Snapping to the Nearest Point

The **Nearest Point** object snap mode snaps to a location on the object that is closest to the crosshairs. This is particularly useful when you want to make certain that the point you pick lies precisely on the object and not a short distance away from it.

1. Pick the **Osnap Settings…** button from the **Object Snap** toolbar.

OBJECT SNAP

This also displays the **Object Snap** tab of the **Drafting Settings** dialog box. (You entered the **OSNAP** command to display it the last time.)

2. Pick the **Select All** button and pick **OK**.

All object snaps are now on.

3. Enter the **LINE** command and slowly move the crosshairs over the snap points of the objects you have drawn.

A yellow hourglass-shaped marker depicts the **Nearest** object snap.

4. Snap to a **Nearest** point on one of the objects.

Cycling Through Snap Modes

Pressing the **TAB** key cycles through all of the running snap modes associated with an object. This is useful when an area is densely covered with objects, making it difficult to snap to a particular point.

1. Rest the crosshairs on the rear wheel.

2. Press the **TAB** key.

3. Press **TAB** several more times, stopping for a moment between each.

As you can see, AutoCAD moves from one object snap to the next.

4. Press **ESC** to cancel the **LINE** command.

5. Pick the **Osnap Settings…** button on the **Object Snap** toolbar.

6. Pick the **Clear All** button, check **Endpoint**, **Center**, **Intersection**, and **Extension**, and pick **OK**.

OBJECT SNAP

Chapter 10: Object Snap   **123**

Object Snap Settings

Other settings allow you to tailor the behavior of object snaps to meet your individual needs.

OBJECT SNAP

Changing the Settings

1. Pick the **Osnap Settings...** button on the **Object Snap** toolbar.

2. Pick the **Options...** button in the dialog box.

Marker, **Magnet**, and **Display AutoSnap tooltip** should be checked. When **Marker** is checked, geometric markers appear at each of the snap points. (These are the yellow markers you have used throughout this chapter.) When **Magnet** is checked, AutoCAD locks the crosshairs onto the snap target. When **Display AutoSnap tooltip** is checked, AutoCAD displays a tooltip that describes the snap location.

3. Check the **Display AutoSnap aperture box** check box.

This causes an **aperture box** to appear at the center of the crosshairs when you snap to objects. The aperture box defines an area around the center of the crosshairs within which an object or point will be selected when you use the pointing device to pick it.

4. Pick the **Colors...** button below the **AutoSnap Settings** to display the **Drawing Window Colors** dialog box.

5. With **AutoSnap marker** highlighted under **Interface element**, pick the down arrow under **Colors** and select green.

6. Pick the **Apply & Close** button.

7. In the **AutoSnap Marker Size** area, move the slider bar slightly to the right, increasing the size of the marker.

In the **AutoTrack Settings** area of the dialog box, all three check boxes should be checked. When **Display polar tracking vector** is checked, AutoCAD displays an alignment path for polar tracking. When **Display full-screen tracking vector** is checked, AutoCAD displays alignment vectors as infinite lines. When **Display AutoTrack tooltip** is checked, AutoCAD displays AutoTrack tooltips.

The **Alignment Point Acquisition** area controls the method of displaying alignment vectors. When **Automatic** is picked, AutoCAD displays tracking vectors automatically when the aperture moves over an object snap. When **Shift to acquire** is picked, AutoCAD displays the tracking vectors only if you press the **SHIFT** key while you move the aperture over an object snap.

8. In the **Aperture Size** area, slightly increase the size of the aperture using the slider bar.

9. Pick the **Apply** and **OK** buttons.

10. Pick the **OK** button in the **Drafting Settings** dialog box.

Applying the New Settings

Now let's examine how the changes made in the dialog box affect the drawing process.

1. Enter the **ARC** command and move the crosshairs over the drawing.

DRAW

Notice that the object snap markers appear when the aperture box contacts the object.

2. Press **ESC** to cancel the **ARC** command.

The AutoSnap settings are stored with AutoCAD, not with the drawing. This means that AutoCAD remembers the changes. Likewise, running object snap modes are stored with AutoCAD.

3. Pick the **Osnap Settings…** button on the **Object Snap** toolbar and pick the **Options…** button.

OBJECT SNAP

4. Uncheck **Display AutoSnap aperture box**, change the marker color back to its original color by selecting the **Restore current element** in the **Drawing Window Colors** dialog box, and reduce the marker and aperture box to their original sizes.

5. Pick **Apply** and **OK**.

6. Pick **OK** again to close the **Drafting Settings** dialog box.

As you work with polar tracking, object snap, and object snap tracking, keep in mind that you can toggle them on and off at any time using the **Polar Tracking, Object Snap,** and **Object Snap Tracking** buttons on the status bar. Most of the time, you will want to use these features, but there will be times when you will want to disable them.

7. Practice using the remaining object snaps on your own.

8. Close the **Object Snap** toolbar.

9. Enter **STARTUP** at the AutoCAD prompt and enter **0**.

10. Save your work and exit AutoCAD.

Chapter 10   Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. How is the **Quick Setup** wizard useful?

2. Explain the purpose of the object snaps.

3. Describe two methods for using AutoCAD's object snap modes.

4. In order to snap a line to the center of a circle, what part of the circle must the crosshairs touch?

5. What is the benefit of using running object snap modes?

6. Describe a situation in which you would want to change the aperture box size.

7. Briefly describe the use of each of the following object snap modes.
   a. **Apparent Intersection**
   b. **Center**
   c. **Endpoint**
   d. **Extension**
   e. **Insertion**
   f. **Intersection**
   g. **Midpoint**
   h. **Nearest**
   i. **Node**
   j. **None**
   k. **Parallel**
   l. **Perpendicular**
   m. **Quadrant**
   n. **Tangent**

8. Explain the benefit of object snap tracking.

9. Explain the relationship between polar tracking, object snap, and object snap tracking. Can you use object snap tracking when object snap is turned off?

Review & Activities Chapter 10

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Devise a new object snap that does not currently exist in AutoCAD. The new snap mode must provide a useful service. Write a short paragraph describing the new object snap. Explain what it does and why it is useful.

2. Investigate the **Apparent Intersection** object snap. Of what use is an object snap that allows you to snap to a point where two objects only *seem* to intersect? Give an example.

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Draw the square on the left in Fig. 10-5. Then use object snaps to make the additions shown on the right to produce a top view of a jeweler's new design for a cut gemstone.
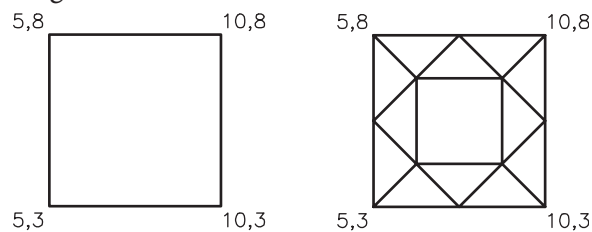
Fig. 10-5

2. Create a new drawing and set running snap modes for **Center** and **Endpoint**. Then create the shaft bearing in Fig. 10-6 using the dimensions shown. Start the lower left corner of the drawing at absolute coordinates **1,1** and work counterclockwise. Save the file as **shaftbearing.dwg**.
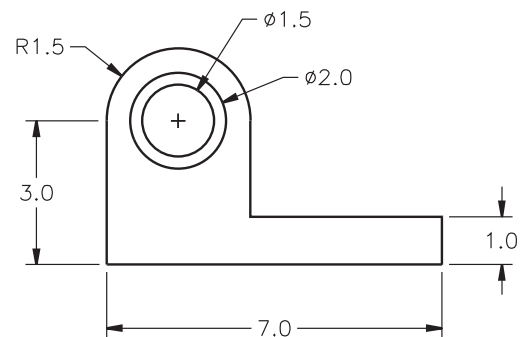
Fig. 10-6

Continued

Chapter 10 **Review & Activities**

3. Draw the top view of the video game part shown in Fig. 10-7. Start by drawing a circle of any size at any location. Then use the **Quadrant** and **Center** object snaps to draw the four lines to divide the circle into quadrants, as shown on the left in Fig. 10-8. (You must draw four lines, not two.) Use the **ARC** command with the **Start**, **Center**, **End** option and **Quadrant**, **Midpoint**, and **Center** object snap modes to draw the arcs. Erase the quadrant lines. Save the drawing as **gamepart.dwg**.
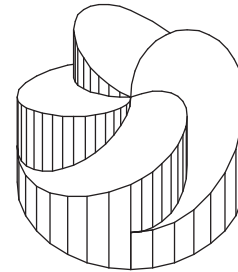
Fig. 10-7
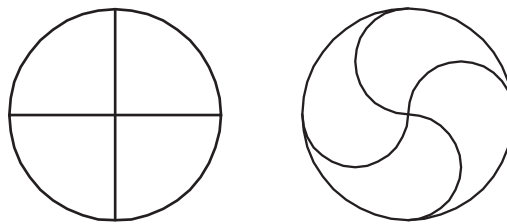
Fig. 10-8

Courtesy of Gary J. Hordemann, Gonzaga University

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. The drawing in Fig. 10-9 represents a variation of a magnet for a new motor design. Draw the magnet. Select the appropriate drawing unit of measurement, drawing area, and object snap modes. Do not include dimensions, but save the drawing as **magnet.dwg**.
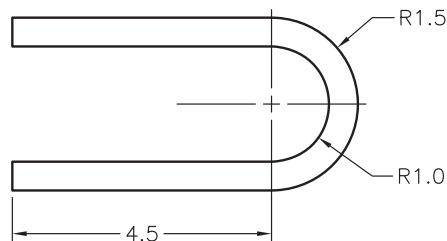
Fig. 10-9

Review & Activities Chapter 10

2. Suppose your company supplies special machine parts for a plastic injection molding company. Draw the new design for a pivot arm to swing the mold halves apart, as shown in Fig. 10-10. Use the **Quick Setup** wizard and select the appropriate drawing unit of measurement and drawing area. Set the appropriate object snap modes. Do not include dimensions. Save the drawing as **injection.dwg**.
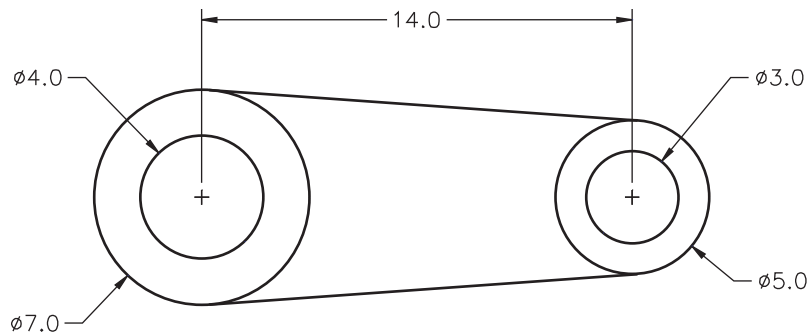
Fig. 10-10

3. The three figures in Fig. 10-11 are constructed of polygons according to the following rules:

a. There are as many polygons as there are sides to the polygons.

b. All polygons, except the two on the ends, share a single edge with each of two other polygons.

If we imagine these figures to be groups of blocks sitting on a horizontal surface, then all three groups are stable (assuming a fair amount of friction between the surfaces). Using the **Edge** option of the **POLYGON** command with either the **Intersection** or the **Endpoint** object snap, draw similar groupings of hexagons and heptagons. Draw them as though they were sitting on a horizontal surface, and be sure to follow the two rules stated above. Is either grouping unstable? Can you formulate a general rule about the stability of such groupings that would apply to polygons with any number of sides?
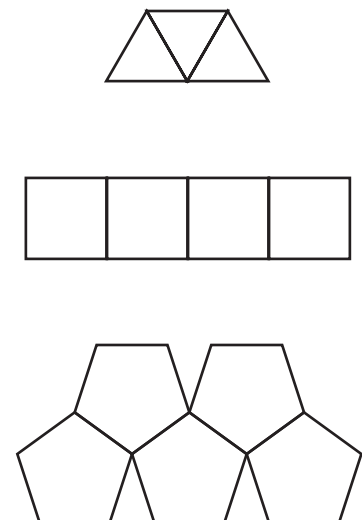
Fig. 10-11

Courtesy of Gary J. Hordemann, Gonzaga University

Chapter 11

Helpful Drawing Features

Objectives

- Use and change the display of coordinate information
- Restrict lines to vertical or horizontal orientations
- Track time spent on a drawing file
- Use the **AutoCAD Text Window** and the **Command Line** window
- Undo and redo multiple operations

Vocabulary

ortho
orthogonal

Several features in AutoCAD provide information to help users become more productive. The coordinate display, for example, gives size information on the drawing area. The ortho feature helps you draw horizontal and vertical lines quickly and accurately. The **TIME** command tracks the time you spend working with AutoCAD. The **AutoCAD Text** and **Command Line** windows provide historical information on your interactions with the software. With AutoCAD's undo and redo features, you don't have to worry about making a wrong move.

Coordinate Display

Coordinate information, in digital form, is located in the lower left corner of the AutoCAD window and is part of the status bar. The coordinate display tracks the current position (or coordinate position) of the crosshairs as you move the pointing device. It also gives the length and angle of line segments as you draw.

1. Start AutoCAD and re-select the **2D Drafting & Annotation** workspace.

2. Move the crosshairs in the drawing area by moving the pointing device and note how the coordinate display changes with the movement of the crosshairs.

3. Enter the **LINE** command and draw a line segment of a polygon. Note the coordinate display as you draw the line segment.

> **NOTE** AutoCAD provides shortcut keys for commands that are used frequently. Shortcut keys allow you to enter commands very quickly. AutoCAD's shortcut keys include the function keys (**F1**, **F2**, **F3**, and so on) and the combination of the **Control** (**CTRL**) key and another key pressed at the same time. The following steps use this shortcut method.

4. Press the **CTRL** key and the letter **I** at the same time, draw another line segment, and notice the coordinate display.

As you can see, polar coordinate information is now displayed.

5. Press the **CTRL** and **I** keys again, draw another line segment or two, and watch the coordinate display.

The coordinate display should now be off.

6. While in the **LINE** command, click the coordinate display and note the change in the coordinate display as you move the crosshairs.

7. Create additional line segments, clicking the coordinate display between each.

As you can see, clicking the coordinate display serves the same function as pressing the **CTRL** + **I** shortcut combination.

8. Press **ENTER** to terminate the **LINE** command.

The Ortho Mode

Now let's focus on an AutoCAD feature called ortho. **Ortho**, short for "orthogonal," allows you to draw horizontal and vertical lines quickly and easily. **Orthogonal**, in this context, means drawn at right angles.

Ortho is on when the **Ortho Mode** button on the status bar is depressed. You can toggle ortho on and off by clicking the **Ortho Mode** button.

STATUS BAR

1. Turn off object snap if it is currently on.

2. Click the **Ortho Mode** button in the status bar or press the **F8** function key. (Both actions perform the same function.)

3. Experiment by drawing lines with ortho turned on and then with ortho off. Note the difference.

> *HINT* Like the coordinate display feature, ortho can be toggled on and off at any time, even while you're in the middle of a command.

4. Attempt to draw an angular line with ortho on.

As you can see, it is not possible.

5. Clear the screen if necessary and draw the plug shown in Fig. 11-1, first with ortho off and then with ortho on. Don't worry about exact sizes and locations.
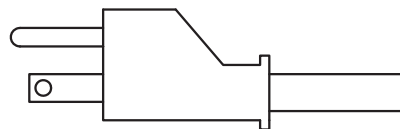
Fig. 11-1

Was it faster with ortho on?

> *NOTE* In many situations, polar tracking is more convenient and easier to use than ortho. However, ortho can be very useful when polar is set to angles other than 90° or when polar is turned off.

Tracking Time

AutoCAD keeps track of time while you work. With the **TIME** command, you can review this information.

1. Enter **TIME** at the keyboard or select the **Tools** menu in the **Menu Bar,** and pick **Inquiry** and **Time**.

The **AutoCAD Text Window** appears, providing information similar to that shown in Fig. 11-2. Of course, the dates and times will be different. Here's what this information means.

Current time:	Current date and time
Created:	Date and time the drawing was created
Last updated:	Date and time the drawing was last updated
Total editing time:	Total time spent editing the current drawing
Elapsed timer (on):	A timer you can reset or turn on or off
Next automatic save in:	Time before the next automatic save occurs

If the current date and time are not displayed on the screen, they were not set correctly in the computer, or the computer's battery may be weak or dead.

2. If the date and time are incorrect, right-click the time located in the lower right corner of the screen, pick **Adjust Date/Time**, make the necessary changes, and pick the **OK** button.

3. Press **ESC** to end the **TIME** command, but do not close the **AutoCAD Text Window**.

```
AutoCAD Text Window - Drawing1.dwg
Edit
Specify next point or [Undo]:
Specify next point or [Undo]:
Specify next point or [Close/Undo]:   <Ortho on>
Specify next point or [Close/Undo]:
Specify next point or [Close/Undo]:
Specify next point or [Close/Undo]:

Command:  LINE Specify first point:
Specify next point or [Undo]:
Specify next point or [Undo]:   <Ortho off>
Specify next point or [Close/Undo]:
Specify next point or [Close/Undo]:

Command: time

Current time:            Saturday, May 09, 2009  4:18:38:280 PM
Times for this drawing:
  Created:               Saturday, May 09, 2009  4:05:42:015 PM
  Last updated:          Saturday, May 09, 2009  4:05:42:015 PM
  Total editing time:    0 days 00:12:56:421
  Elapsed timer (on):    0 days 00:12:56:265
  Next automatic save in: 0 days 00:00:26:875

Enter option [Display/ON/OFF/Reset]: |
```

Fig. 11-2

4. Reenter the **TIME** command.

What information is different? Check the current time, total editing time, and elapsed time.

5. With AutoCAD's **TIME** command entered, type **D** for **Display** and notice what displays on the screen.

This provides updated time information.

6. Enter the **OFF** option and display the time information again.

The elapsed timer should now be off.

> **NOTE**
>
> An example of when you might specify **OFF** is when you want to leave the computer to take a break. When you return, you turn the time back **ON**. This keeps an accurate record of the actual time (elapsed time) you spend working on a project.

7. Reset the timer by entering **R** and display the time information once again.

The elapsed timer should show **0 days 00:00:00.000**.

8. Last, turn on the timer and display the time information.

Notice that the elapsed timer keeps track of the time only while the timer is turned on.

Why is all of this time information important? In a work environment, the **TIME** command can track the amount of time spent on each project or job, making it easier to charge time to clients.

9. Press **ESC** to end the **TIME** command, but do not close the **AutoCAD Text Window**.

AutoCAD Text Window

As you are aware, AutoCAD displays the time information in the **AutoCAD Text Window**. AutoCAD uses this window for various purposes, as you will see in future chapters. A basic understanding of how to use this window will help you use AutoCAD more efficiently.

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11. Helpful Drawing
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141

Displaying the Window

1. Pick a point in the drawing area but outside the **AutoCAD Text Window**.

This causes the graphics screen to appear in front of the **AutoCAD Text Window**, hiding it from view.

2. Press the **F2** function key to make the **AutoCAD Text Window** come to the front.

You can also make it reappear by picking the **AutoCAD Text Window** button on the Windows taskbar.

3. In the upper right corner of the **AutoCAD Text Window**, pick the **Minimize** button (the dash).

4. Press **F2** or pick the **AutoCAD Text Window** button on the Windows taskbar to make it reappear.

> You can press **F2** at any time to display the **AutoCAD Text Window**. **F2** serves as a toggle switch to toggle the display of the window.

5. Using the vertical scrollbar, scroll up the list of text in the **AutoCAD Text Window**.

Observe that AutoCAD maintains a complete history of your activity.

Copying and Pasting

Notice that the **AutoCAD Text Window** offers an **Edit** pull-down menu. This menu allows you to perform basic copying and pasting functions within the window.

1. Select the **Edit** pull-down menu from the **AutoCAD Text Window**.

Copy permits you to copy a selected portion of the text to the Windows Clipboard. The Clipboard is a memory space in the computer that temporarily stores information (text and graphics). After you have copied information to the Clipboard, it is easy to paste it into another software application. The **Copy** option is grayed out and not available because text has not been highlighted.

Copy History copies the entire contents of the **AutoCAD Text Window** to the Windows Clipboard. **Paste** enables you to paste the contents of the
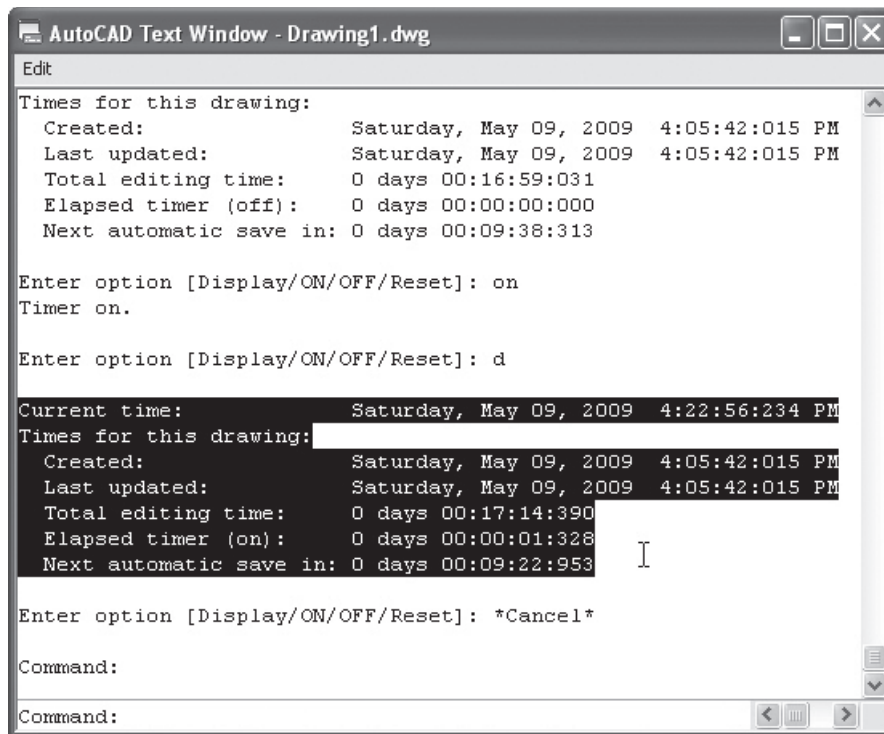
Fig. 11-3

Clipboard to the **Command** line. **Paste to CmdLine** pastes highlighted text to the **Command** line. The **Options...** menu item displays the **Options** dialog box, which allows you to change various settings to meet individual needs.

2. Using the pointer, highlight the most recent time information, as shown in Fig. 11-3.

3. Pick the **Copy** item from the **Edit** pull-down menu in the **AutoCAD Text Window** or press the **CTRL** and **C** keys to copy the text.

4. Close the **AutoCAD Text Window** by clicking the **Close** button (the **x**) in the upper right corner of the window.

5. Minimize AutoCAD by picking the **Minimize** button (the dash) located in the upper right corner.

6. Launch a text editor such as **Notepad**. (**Notepad** is a standard Windows program and is part of the **Accessories** group of Windows utilities.)

7. In the text editor, select **Paste** from the **Edit** menu or press the **CTRL** and **V** keys.

This pastes the time information from the Windows Clipboard. At this point, you could name and save this file and/or print this information.

8. Exit the text editor without saving the file.

9. Click the **AutoCAD** button on the Windows taskbar to make the AutoCAD window reappear.

Command Line Window

Now focus your attention on the **Command Line** window (the window that contains the **Command** prompt).

1. Position the pointer so that its tip is touching any part of the **Command Line** window's left border.
2. With the Windows pointer, click and drag the **Command Line** window upward into the drawing area until a floating window forms.

You can move the window anywhere on the screen, and you can resize its width and height.

3. Right-click in the title bar of the **Command Line** window and pick the **Transparency...** option.

This displays the **Transparency** dialog box that was discussed in Chapter 3.

4. Move the **General** slider to the far left and pick **OK**.

The **Command Line** window is now nearly transparent.

5. Right-click in the title bar of the **Command Line** window, pick the **Transparency...** option, move the **General** slider to the far right, and pick **OK**.
6. Drag the **Command Line** window downward and dock it into its original position.

Copying and Pasting

It is possible to copy and paste text into the **Command Line** window. You may find this feature useful as you work with AutoLISP, an AutoCAD programming language.

1. Use the up arrow at the right of the **Command Line** window to scroll up.
2. Identify a command (any command) and highlight it.
3. With the pointer inside the **Command Line** window, right-click and pick **Paste to CmdLine**.

AutoCAD pastes the highlighted text at the **Command** line.

4. With the pointer inside the **Command Line** window, right-click again and highlight **Recent Commands**.
5. Pick one of the commands from the list.

AutoCAD enters the command.

6. Press the **ESC** key.

Editing Command Entries

AutoCAD allows you to correct misspellings at the **Command** line.

1. Type **TME** (a misspelled version of the **TIME** command) at the **Command** line, but do not press **ENTER**.

2. Using the left arrow key, back up and stop between the **T** and **M**.

3. Type the letter **I**.

> **NOTE**
>
> The **Insert** key serves as a toggle to turn on and off the insert mode. If the **Insert** function is not active, press the **Insert** key to turn it on.

4. Assuming that **TIME** is now spelled correctly, press **ENTER**.

5. Press **ESC** to cancel.

Table 11-1 briefly describes the keys that you can use to edit text on the **Command** line.

6. Close the **AutoCAD Text Window**.

7. On your own, experiment with the **Command** line navigation options shown in Table 11-1.

Table 11-1. Editing Keys

Key	Action
Left arrow	Moves the cursor back (to the left)
Right arrow	Moves the cursor forward (to the right)
Up arrow	Displays the previous line in the command history
Down arrow	Displays the next line in the command history
HOME	Places the cursor at the beginning of the line
END	Places the cursor at the end of the line
INSERT	Turns on and off the insertion mode
DEL	Deletes the character to the right of the cursor
Backspace key	Deletes the character to the left of the cursor
CTRL + V	Pastes text from the clipboard

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145

Undoing Your Work

As you may recall from earlier chapters, you can undo or redo one or several actions by picking the **Undo** and **Redo** buttons in the **Quick Access** toolbar. AutoCAD also offers an **UNDO** command that provides further options.

Reversing Multiple Operations

The basic **UNDO** command is similar to the **Undo** button. It undoes, or reverses, one or more operations, beginning with the most recent.

1. Draw the objects shown in Fig. 11-4 at any size and location.
2. Enter the **UNDO** command at the keyboard.
3. Review the list of command options. Then enter **2** for the number of operations.
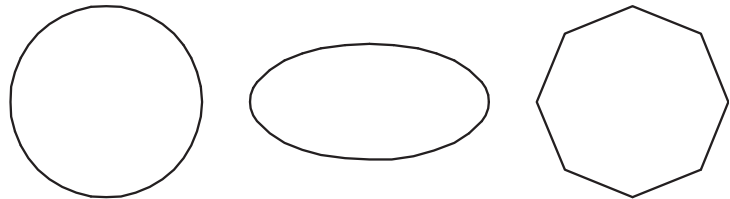
By entering **2**, you told AutoCAD to back up two steps.

Fig. 11-4

AutoCAD also includes the **U** command, which allows you to back up one step at a time. The action of the **U** command can be reversed by the **REDO** command.

Returning to a Previous Drawing Status

Suppose you want to proceed with drawing and editing, but you would like the option of returning to this point in the session at a later time. The **UNDO** command's **Mark** and **Back** options allow you to do this efficiently.

1. Draw a small rectangle.
2. Enter the **UNDO** command.
3. Enter the **Mark** option by typing **M** and pressing **ENTER**.

DRAW

AutoCAD has (internally) marked this point in the session.

4. Perform several operations, such as drawing and erasing objects and changing the buttons on the status bar.

Now suppose you decide to back up to the point where you drew the rectangle in Step 1.

5. Enter the **UNDO** command and then the **Back** option.

You have just practiced two of the most common uses of the **UNDO** command. Other **UNDO** options exist, such as **BEgin** and **End**. Refer to AutoCAD's online help for more information on these options.

Controlling the Undo Feature

The undo feature can use a large amount of disk space and can cause a "disk full" situation when only a small amount of disk space is available. You may want to disable the **UNDO** command partially or entirely by using the **UNDO Control** option.

Let's experiment with **UNDO Control**.

1. Enter the **UNDO** command and then the **Control** option.

2. Enter the **One** option.

3. Draw two arcs anywhere in the drawing area.

4. Enter the **UNDO** command.

UNDO now permits you to undo only your last operation.

5. Press **ENTER**.

6. Enter **UNDO**, **Control**, and **All**.

The undo feature is once again fully enabled.

7. Enter **UNDO** and note the complete list of options.

8. Enter **ESC** to cancel the command.

9. Exit AutoCAD without saving.

Wohlers: Applying
AutoCAD® 2010

II. Drawing Aids and
Controls

11. Helpful Drawing
Features

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147

Chapter 11 **Review & Activities**

• REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Of what value is the coordinate display?

2. What is the name of the feature that forces all lines to be drawn only vertically or horizontally? What function key controls this feature?

3. Of what value is the **TIME** command?

4. Briefly explain each of the following components of the **TIME** command.

    a. **Current time**

    b. **Created**

    c. **Last updated**

    d. **Total editing time**

    e. **Elapsed timer**

    f. **Next automatic save in**

5. Which function key toggles the display of the **AutoCAD Text Window** on and off?

6. Explain how you would copy a few lines from the **AutoCAD Text Window** to a text editor such as **Notepad**.

7. Describe two methods that let you undo your last five operations quickly.

8. Explain the use of the **UNDO Mark** and **Back** options.

• CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Experiment with ortho in combination with various object snap modes. What happens when you try to snap to a point that is not exactly horizontal or vertical to the previous point?

2. Why might you want to copy parts of the **AutoCAD Text Window** or **Command Line** window to the Windows Clipboard?

Continued

Chapter 11: Helpful Drawing Features   **141**

Chapter 11 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Set the elapsed timer to **ON**. Complete the design shown in Fig. 11-5 for an over-the-hood storage area for a garage. Use ortho to keep the lines perfectly horizontal and vertical. When you have completed the drawing, set the elapsed timer to **OFF** and write down the total time it took you to complete the drawing.
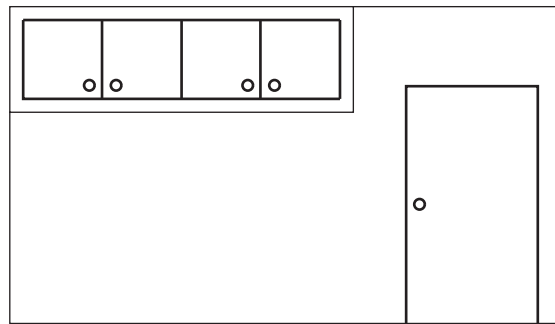
Fig. 11-5

2–4. Compare the use of ortho with object snap tracking and polar tracking by drawing the objects shown in Figs. 11-6 through 11-8. Create a new drawing to hold the objects. Turn on the coordinate display and note the display as you construct each object.
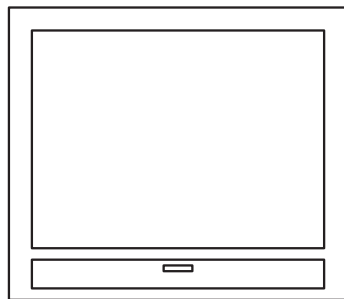
Fig. 11-6
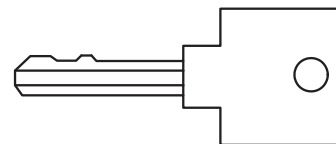
Fig. 11-7

Problems 3, 4, and 5 courtesy of Joseph K. Yabu, Ph.D., San Jose State University

Review & Activities Chapter 11
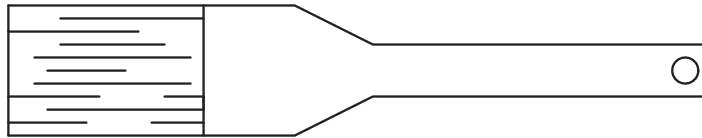
Fig. 11-8

5. Create a new drawing file and draw the lamp head shown in Fig. 11-9. Copy everything entered at the **Command Line** window to the Clipboard and paste it into a text editor such as **Notepad**. Then print it. (Read problem 6 before completing this problem.)

6. Use AutoCAD's timer to review the time you spent completing problem 5. Print this information or record it on a separate sheet of paper.

7. Draw the simple house elevation shown in Fig. 11-10 at any size with ortho or object snap tracking and polar tracking. Prior to drawing the roof, use **UNDO** to mark the current location in the drawing. Then draw the roof.

8. With **UNDO Back**, return to the point prior to drawing the roof. Draw the roof shown in Fig. 11-11 in place of the old roof. Use the **UNDO** command as necessary as you complete the drawing.
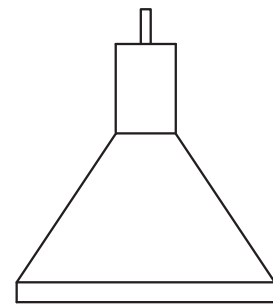
Fig. 11-9
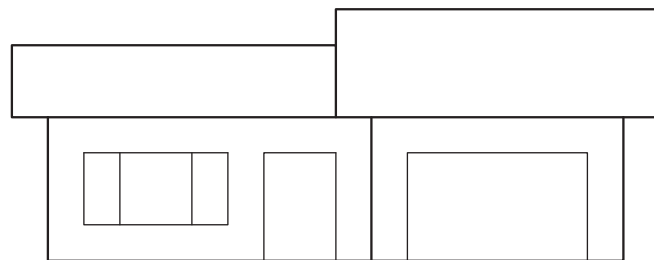
Fig. 11-10
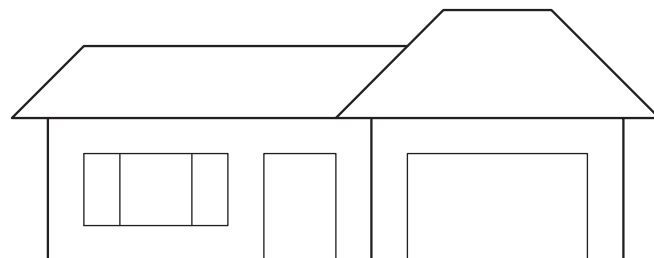
Fig. 11-11

Continued

Chapter 11: Helpful Drawing Features   **143**

Chapter 11 **Review & Activities**

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. As the packaging engineer for a major manufacturer, you have been asked to design a support for a new product to hold it in place within the container. The drawing (Fig. 11-12) is not to scale. Create a drawing to scale according to the dimensions given. Keep track of your hours, because the time will be billed to the customer. Do not dimension, but save the drawing.
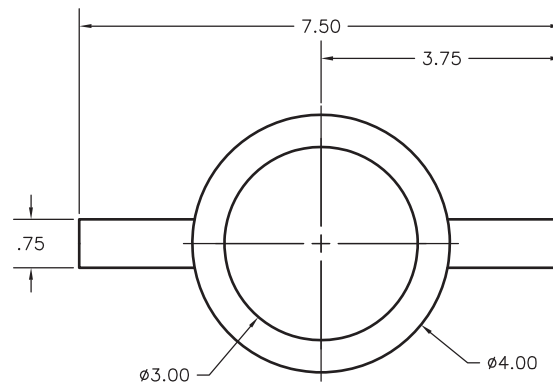
Fig. 11-12

2. The core of an electromagnet is made up of a series of thin "slices," or laminations. The laminations concentrate the electromagnetic field, adding strength. Draw the lamination shown in Fig. 11-13. Keep track of your time. Do not dimension, but save the drawing.
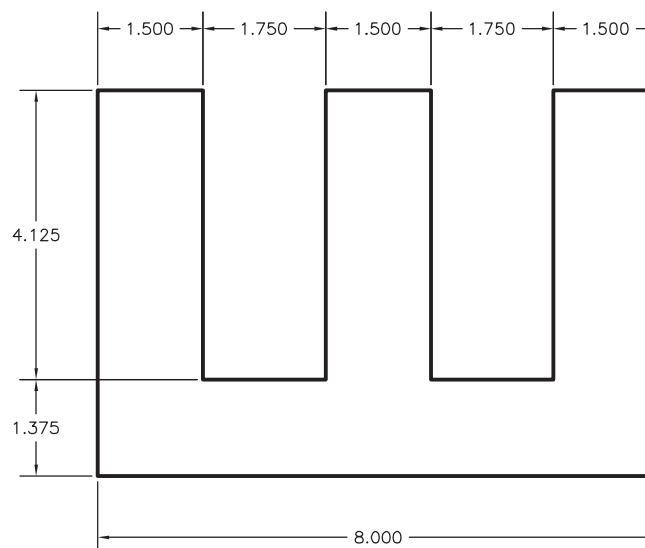
Fig. 11-13

Review & Activities Chapter 11

3. Draw the front view of the tube bulk-head shown in Fig. 11-14. Before drawing the holes, use the **UNDO** command to mark the current location in the drawing. After completing the bulkhead, use **UNDO** to replace the two holes with two new ones. Draw both of the new holes with a diameter of **1.00**. Change the center-to-center distance from **1.40** to **1.60**.

4. The two identical hubs shown in Fig. 11-15 are meant to be mounted back-to-back in an assembly. Draw the front view of one hub. Before drawing the four small holes, use the **UNDO** command to mark the current location in the drawing. After completing the hub, use **UNDO** to replace the four holes with eight new ones, equally spaced around the circumference. Use a new hole diameter of **.284** and a new center line diameter of **2.50**.
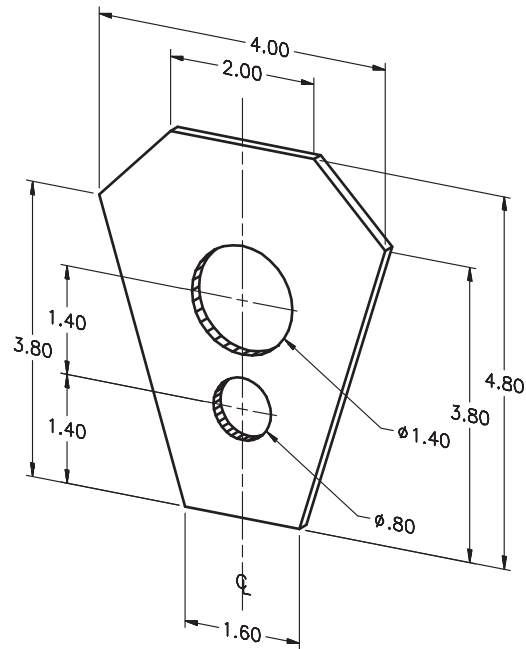
Fig. 11-14
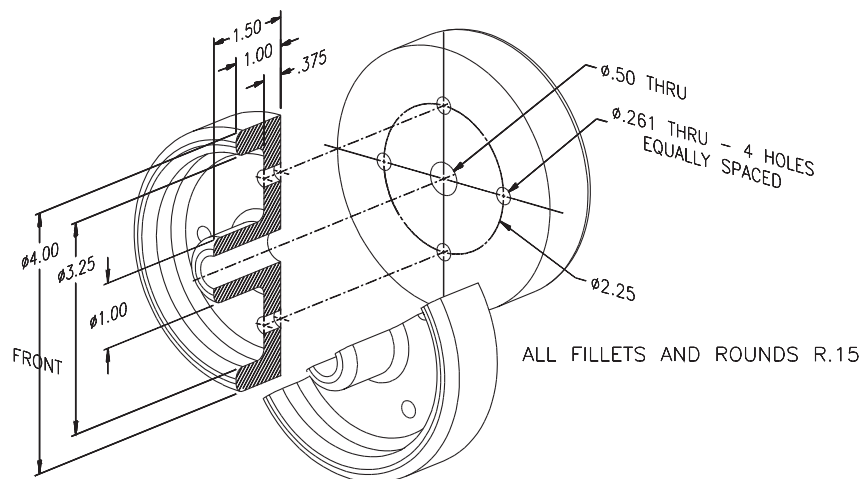
Fig. 11-15

Problems 3 and 4 courtesy of Gary J. Hordemann, Gonzaga University

Chapter **12**

Construction Aids

Objectives

- Set and use AutoCAD's visual grid system
- Establish a snap grid that is appropriate to the current drawing task
- Use construction lines to project views in a multiview drawing
- Become more familiar with the **AutoCAD Classic** workspace

Vocabulary

alignment grid
aspect ratio
construction lines
hidden lines
multiview drawing
offsetting
orthographic projection
plane
rays
snap grid
xlines

AutoCAD permits you to create a nonprinting grid system for visual layout of drawings. The snap grid complements the visual grid, assisting with the layout process. Construction lines provide a means for projecting lines from one view to another using orthographic projection. All of these construction aids make it easier to create accurate multiview drawings in the shortest amount of time possible.

Since you were introduced to workspaces in Chapter 3, you've worked exclusively in the **2D Drafting & Annotation** workspace. In this chapter, you will learn to use the **AutoCAD Classic** workspace.

Alignment Grid

The **GRID** command allows you to set an **alignment grid** of dots of any desired spacing, making it easier to visualize distances and drawing sizes. You can use the grid with both English (Imperial) and metric units and in both 2D and 3D modeling. In this chapter, we will create a drawing of a plastic extrusion using metric units.

1. Start AutoCAD and set the **STARTUP** system variable to **1**.

2. Enter the **NEW** command at the keyboard.

The **Create New Drawing** dialog box appears.

3. Pick **Start from Scratch**, pick the **Metric** radio button, and pick **OK**.

4. Pick the **Workspace Switching** button on the status bar and select the **AutoCAD Classic** workspace.

5. Close all floating toolbars and palettes that are open on the screen.

6. Move the crosshairs to the upper right corner of the drawing area and read the coordinate display.

Since you are now working in the metric system, the unit of measure is millimeters (mm). The drawing area of 12 × 9" is now approximately 500 × 350 mm, depending on the size of the window representing the drawing area on your computer.

STATUS BAR

7. Pick the **Grid Display** button located in the status bar.

A grid of dots appears on the screen.

8. Enter the **GRID** command at the keyboard.

Notice that the default value for the grid spacing is 10. This means that the dots are currently spaced 10 mm apart.

9. Change the spacing to 20 mm by entering **20**.

10. Reenter the **GRID** command and change the spacing back to 10 mm.

11. Turn the grid off by picking the **Grid Display** button.

12. Make the grid visible again.

STATUS BAR

You can also turn the grid on and off by pressing **CTRL G** or the **F7** function key.

Snap Grid

The **snap grid** is similar to the visual grid, but it is an invisible one. You cannot see it, but you can see the effects of it as you move the crosshairs. It is like a set of invisible magnetic points. The crosshairs jumps from point to point as you move the pointing device. This allows you to lay out drawings quickly, yet you have the freedom to toggle snap off at any time.

STATUS BAR

1. Pick the **Snap Mode** button in the status bar to turn on the snap grid.

> **NOTE**
>
> The snap grid is different from object snap, which permits you to snap to points on objects.
>
> You can also turn the snap grid on and off by pressing **CTRL B** or the **F9** function key.

When you turn the snap grid on, it appears at first as though nothing has happened.

2. Slowly move the pointing device and watch closely the movement of the crosshairs.

The crosshairs jump (snap) from point to point.

3. Enter the **SNAP** command at the keyboard.

4. Enter **5** to specify a snap spacing of 5 mm.

5. Move the pointing device and note the crosshairs movement.

> **HINT**
>
> If the grid is not on, turn it on to better see the movement of the crosshairs. Notice the spacing relationship between the snap resolution and the grid. They are independent of one another.
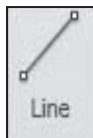
DRAW

6. Pick the **Line** button from the docked **Draw** toolbar and draw a line.

Notice that the endpoints of the line snap automatically to the invisible snap grid. As you can see, the snap grid can be useful for quickly creating fairly accurate drawings.

Changing the Aspect Ratio

The ratio of height to width of a rectangular region is called the **aspect ratio**. You can change this ratio so that the crosshairs snaps one distance vertically and a different distance horizontally. This can be useful when you are laying out a drawing.

1. Enter the **SNAP** command.

Several options appear at the **Command** line.

2. Enter **A** to select the **Aspect** option.

AutoCAD asks for the horizontal spacing.

3. Enter **5**.

Next AutoCAD asks for the vertical spacing.

4. For now, enter **10**.

5. Move the crosshairs up and down, then back and forth. Note the difference between the amount of vertical movement and horizontal movement.

Applying the Snap and Grid Features

The following steps demonstrate how snap and grid can be useful in creating drawings both quickly and accurately.

1. Set snap at **5** mm both horizontally and vertically.

2. Create the drawing of a plastic extrusion shown in Fig. 12-1. Use the pointing device (not the keyboard) to specify all points, and do not include the dimensions.
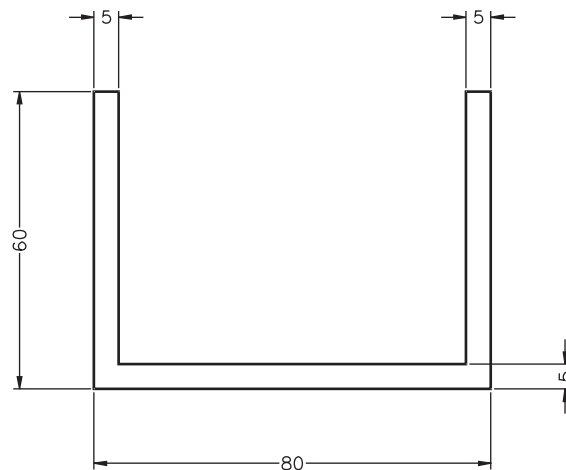
Fig. 12-1

Changing Snap and Grid Settings

The **Drafting Settings** dialog box permits you to review and make changes to the snap and grid settings.

1. Right-click on the **Snap Mode** or **Grid Display** button and select **Settings...** from the shortcut menu.

The **Snap and Grid** tab of the **Drafting Settings** dialog box appears.

As you can see, this dialog box enables you to change the *x* and *y* spacing of both the snap and grid. AutoCAD saves these settings with the drawing, so if you start a new drawing, the old settings return.

2. Close the dialog box.

3. Save your work in a file named **snap.dwg**, but do not close the drawing or exit AutoCAD.

Construction Lines

Construction lines, also called **xlines**, are lines of infinite length. They can be used to construct objects and lay out new drawings.

1. Select the **New** button on the **Quick Access** toolbar, or enter the **NEW** command to start a new drawing.

2. Select the **Start from Scratch** option and **Metric**.

3. Set the grid to **10** mm and the snap to **5** mm.

4. From the **Draw** toolbar, pick the **Construction Line** button.

QUICK ACCESS

This enters the **XLINE** command.

5. Pick a point anywhere on the screen.

6. Move the crosshairs and notice what happens.

DRAW

A line of infinite length passes through the first point and crosshairs.

7. Pick a second point anywhere on the screen.

A line freezes into place. As you move the crosshairs, notice that a second line forms.

8. Pick another point so that the intersecting lines approximately form a right angle.

9. Press **ENTER** to terminate the **XLINE** command.

Unlike the snap and grid construction aids, xlines are AutoCAD objects. You can select them for use with commands, and you can snap to them using object snaps.

10. Enter the **LINE** command and use the **Intersection** object snap to snap to the intersection of the two construction lines.

> **HINT** If **Intersection** is not currently set as a running object snap, you can open the **Object Snap** toolbar and pick the **Intersection** button. Another quick way to enter it is to type **INT** at the AutoCAD prompt.

11. Press **ESC** to terminate the **LINE** command.

12. Erase the xlines.

Constructing Horizontal and Vertical Xlines

The **XLINE** command provides several options that allow you to create specific types of construction lines.

1. Enter the **XLINE** command and enter **H** for **Horizontal**.

2. Pick a point anywhere on the screen; then move the crosshairs and pick a second point, and a third.

Because the **Horizontal** option is in effect, all of the construction lines are horizontal.

DRAW

3. Press **ENTER** to terminate the **XLINE** command and press **ENTER** or the spacebar to reenter it.

4. Enter **V** for **Vertical** and pick three points anywhere on the screen.

The **Vertical** option forces all the lines to be exactly vertical.

Constructing Xlines at Other Angles

To create construction lines at angles other than 0 or 90 degrees, you can use the **Angle** option.

1. Press **ENTER** to terminate the **XLINE** command and then reenter it.

2. Enter **A** for **Angle** and enter **45**.

3. Place three construction lines.

4. Terminate the **XLINE** command.

5. Erase all but one construction line.

Offsetting Construction Lines

You may find many occasions to use the **Offset** option of the **XLINE** command. **Offsetting** a line means creating the line at a specific distance from another line. For example, suppose you have created a line that represents the outer edge of a sidewalk on an architectural site drawing. If the sidewalk is to be 3 feet wide, you can create a construction line at an offset distance of 3 feet to show the width of the sidewalk.

1. Enter **XLINE** and enter **O** for **Offset**.
2. Enter **7** (for 7 mm), select the construction line, and pick a point on either side of it.

You will learn more about offsetting ordinary lines in Chapter 16.

Creating Perpendicular Construction Lines

You can also create a construction line that is perpendicular to other objects on the screen.

1. Terminate **XLINE**, reenter it, and select or enter the **Perpendicular** object snap.
2. Pick one of the construction lines and pick another point elsewhere on the screen.

The new construction line is perpendicular to the one you selected.

3. Terminate **XLINE** and erase all objects from the screen.

Rays

Rays are a special type of construction line. They extend from a single point into infinity in a radial fashion. Rays are useful when you are laying out objects radially.

1. From the **Draw** pull-down menu, pick **Ray**.

This enters the **RAY** command.

2. Pick a point near the center of the screen.
3. Move the pointing device in any direction and then pick several points around the first point.
4. Terminate the **RAY** command.
5. Close the current drawing file and do not save your work. (**Snap.dwg** should still be open.)

Orthographic Projection

Construction lines, rays, and the grid and snap features are often used to create multiview drawings. A **multiview drawing** is one that describes a three-dimensional object completely using two or more two-dimensional views. These views commonly include a front, top, and side view, although others are often necessary. The AutoCAD features discussed in this chapter, coupled with object snap tracking, help you to produce multiview drawings using a technique known as *orthographic projection*.

As mentioned in Chapter 11, *ortho* means "at right angles." **Orthographic projection**, therefore, is the projection of views at right angles to one another. Each adjacent view is projected at a right angle onto a plane, resulting in a two-dimensional view. A **plane** is an imaginary flat surface used to construct the two-dimensional view.

Figure 12-2 shows an example of a simple multiview drawing created using orthographic projection. The right-side view is located to the right of the front view. This positioning is standard. It is important to use the standard positions because doing so allows other people to understand the position of these views at a glance. The thin lines are temporary construction lines.

Top

Front   Right Side
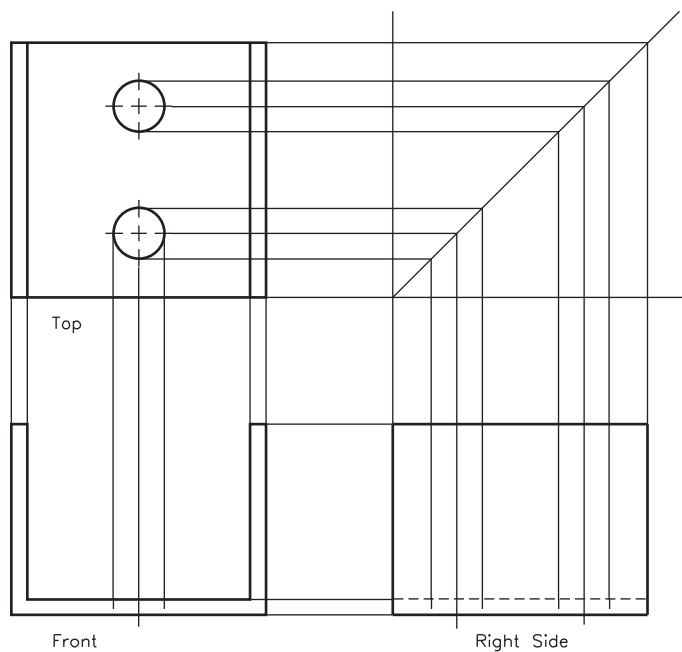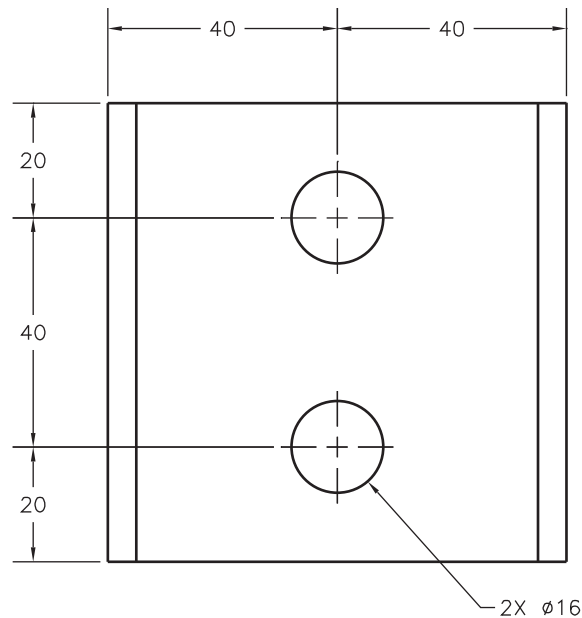
Fig. 12-2

Fig. 12-3

1. Using grips, move the drawing of the plastic extrusion to the lower left area of the screen.

2. Using object snap tracking, snap, and grid, create the top view as shown in Fig. 12-2 on page 153. Use the following information when constructing the top view:

   • From Fig. 12-3, obtain the length dimension of the extrusion and the location and size of the two holes.

   • The broken lines through the holes are center lines. For now, create these lines using a series of line segments. You will learn more about dashed lines, center lines, and other linetypes in Chapter 24.

   • You may need to turn off object snap when picking some of the points. However, object snap must be on for object snap tracking to work.

   • Consider using the **XLINE** command to draw a vertical construction line snapped to the midpoint of one of the horizontal lines to represent the centers of the holes. Then erase the xline.

3. Save your work.

4. After creating the top view, project the holes down to the front view. Consider the following suggestions:

   • Set **Quadrant** as a running object snap mode.

   • Use object snap tracking to acquire the diameter of the holes.

- Draw hidden lines to show the holes in the front view. (**Hidden lines** are the short dashed lines in the multiview drawing that are used to indicate features that are hidden from view.) For now, create them using two short line segments.

5. Save your work.

6. Create the right-side view by projecting lines from the top and front views. Consider the following suggestions:

- Create temporary lines as shown in Fig. 12-4. The vertical line is at an arbitrary distance from the top view.

- Draw the angled line at a 45° angle. (Use the **Angle** option of the **XLINE** command.) Use it to project the dimensions from the top view to the right-side view.

- Project lines from the top view to the 45° line. Then project vertical lines down from the intersections of the horizontally projected lines and the 45° lines to locate the right-side view.

Fig. 12-4

- Use the height dimensions from the front view to finish the right-side view.
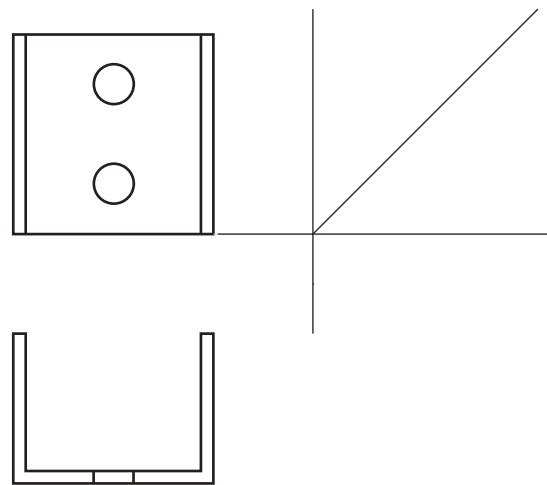
> **NOTE**
>
> In practice, the views in a multiview drawing are laid out carefully to ensure that the distance between the front and top views is approximately equal to the space between the front and side views. Keep this in mind as you create the construction lines in Step 6.

7. Erase all temporary construction lines.

Your multiview drawing should now look similar to the multiview drawing shown in Fig. 12-2 on page 153, except that your drawing will not include construction lines. You will learn more about multiview drawings in Chapter 26.

8. Close any open toolbars, save your work, and exit AutoCAD.

Chapter 12 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is the purpose of the grid? How can the grid be toggled on and off quickly?

2. When would you use the snap feature? When would you toggle off the snap feature?

3. How can you set the snap feature so that the crosshairs move a different distance horizontally than vertically?

4. What is the purpose of the **XLINE** and **RAY** commands?

5. Explain the purpose of construction lines and rays. Describe a drawing situation in which you could use construction lines or rays to simplify your work.

6. What is orthographic projection? How is it used in creating engineering drawings?

7. What is the purpose of a multiview drawing?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. AutoCAD provides several different grid and snap systems. Write a short paper describing their differences and similarities.

2. It is possible to accomplish the same tasks using either construction lines and rays or AutoCAD's AutoTrack™ feature. Describe a situation in which you would prefer to use one instead of the other.

Review & Activities Chapter 12

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Draw the triangle shown in Fig. 12-5. Turn the grid off and use a snap spacing of .25.
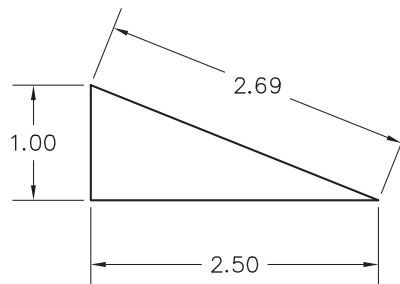
Fig. 12-5

2. Draw the rectangle shown in Fig. 12-6. Set the grid at 1 and snap at .5.
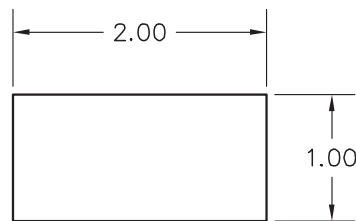
Fig. 12-6

3. Draw the front view of a pipe shown in Fig. 12-7. Use a grid of 2 and turn snap off.
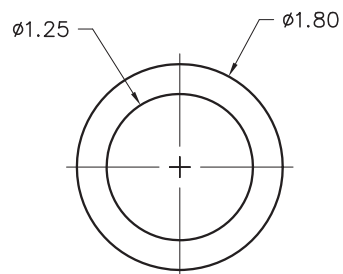
Fig. 12-7

Continued

Chapter 12: Construction Aids   157

Chapter 12 **Review & Activities**

4. Draw the two views of the rubberized hand grip used for closing the safety bar on a roller coaster car (Fig. 12-8).
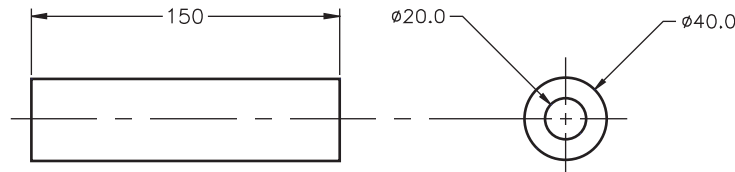
Fig. 12-8

Begin by drawing two construction lines, as shown in Fig. 12-9. Draw the front view as shown. Then draw the Ø40 circle using the tangent-tangent-radius method. Locate the Ø20 circle to be concentric with the first circle, and erase the construction lines. Do not dimension it, and do not draw the center lines.
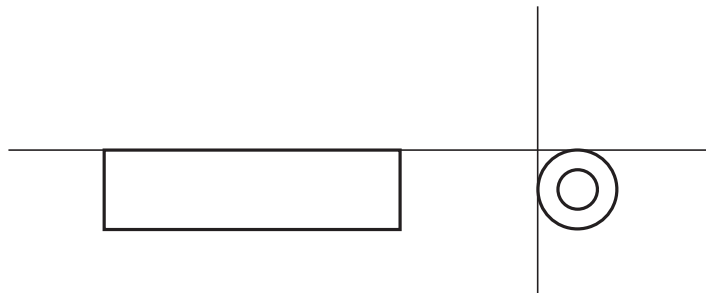
Fig. 12-9

5. Draw three views of the slotted block shown in Fig. 12-10. Start by drawing the front view using the **LINE** and **CIRCLE** commands. Do not worry about drawing the object exactly. Make the task easier by using ortho.

Slotted Block   Front View
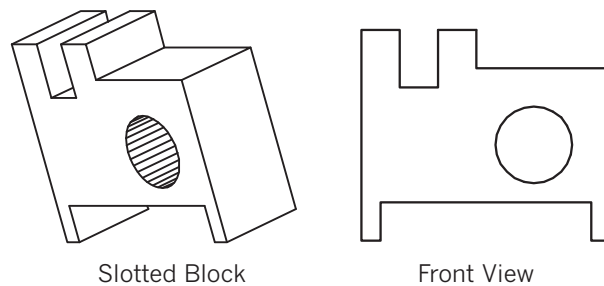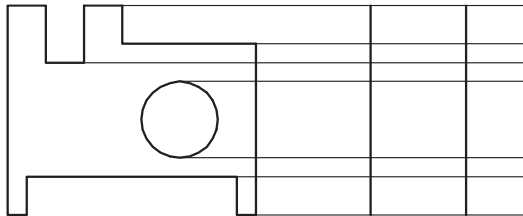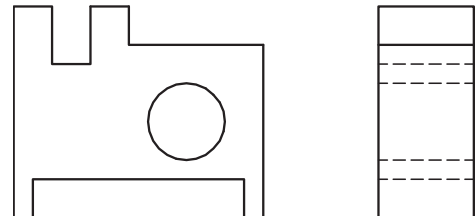
Fig. 12-10

Fig. 12-11

Fig. 12-12

Use the **RAY** command to extend the edges in the front view into what will become the right-side view. Use the appropriate object snaps to begin the rays at the edges of the front view, as shown in Fig. 12-11.

To begin drawing the right-side view, add two vertical lines (estimate the space between them), as shown in Fig. 12-11. Use the construction lines (rays) to draw the visible lines. Draw the hidden lines as a series of very short lines.

When you are finished with the right-side view, erase the construction lines to reduce confusion in the drawing. The drawing should now look like the one in Fig. 12-12.

Repeat the process to draw the top view. The finished drawing should look like the one in Fig. 12-13.
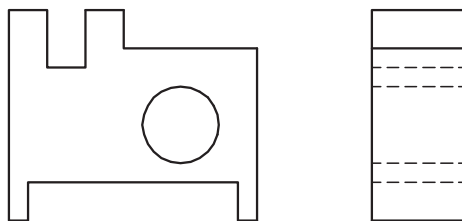
Fig. 12-13

Continued

Chapter 12 **Review & Activities**

6. Draw the top, front, and right-side views of the block support shown in Fig. 12-14. Use a grid spacing of .2 and a snap setting of .1. Consider using construction lines or rays to position the three views. For now, draw the hidden and center lines using very short lines. You may wish to change the snap spacing to .05.
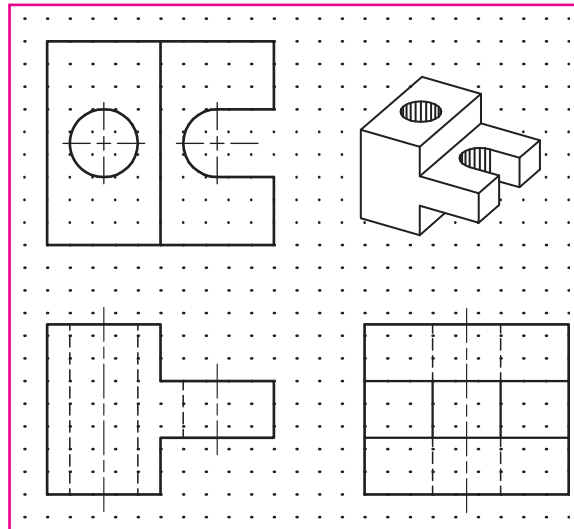
Fig. 12-14

7. Draw the three orthographic views of the bushing holder shown in Fig. 12-15. Use a grid spacing of .2 and a snap setting of .1. Consider using construction lines or rays to help you position the views. The front view is shown as a full section view; *i.e.*, it is shown as if the object has been cut in half. The diagonal lines indicate cut material. You may want to use the **Offset** option of the **XLINE** command to create them at even intervals.
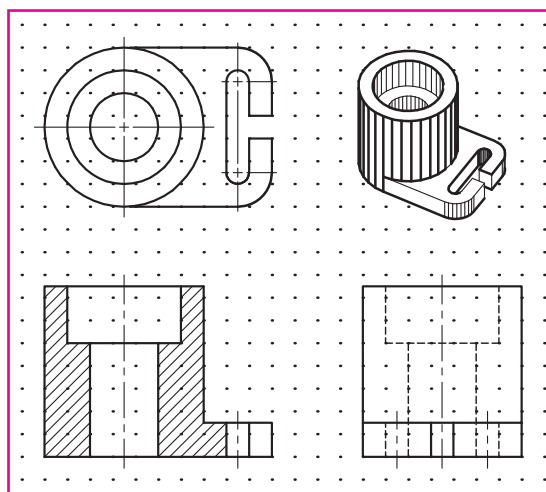
Fig. 12-15

Problems 5, 6, and 7 courtesy of Gary J. Hordemann, Gonzaga University

Review & Activities Chapter 12

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Create the front and right-side views of the block shown in Fig. 12-16. Then, using the horizontal line above the front view as a basis, complete the top view of the block.
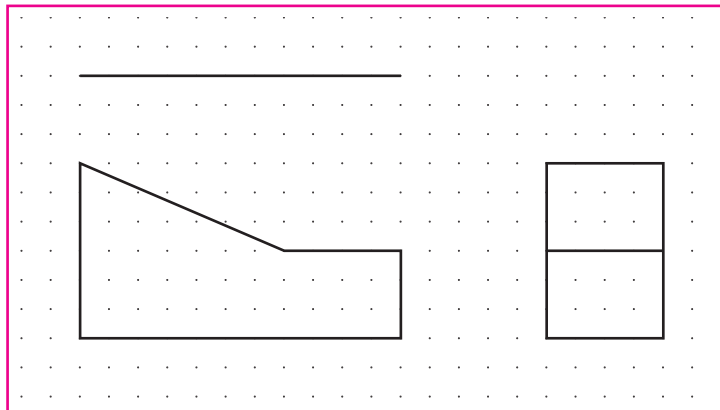
Fig. 12-16

2. Create the front and top views of the object shown in Fig. 12-17. Then, using the horizontal line to the right of the front view as a basis, complete the right-side view of the block.
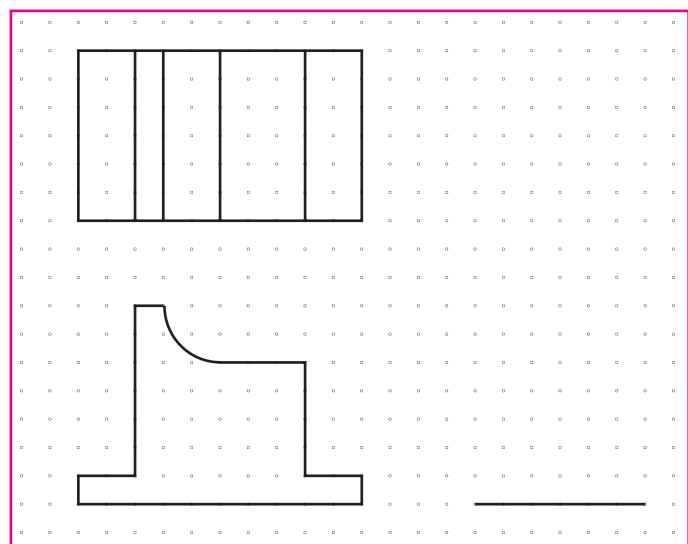
Fig. 12-17

AutoCAD at Work

Graphic Designer

Think about the last CD or DVD you purchased. Can you remember what the cover looks like? Does it include fancy artwork or wording? Almost every item available for purchase today is enclosed in some sort of packaging. To entice consumers to select their products, companies hire graphic designers to create an appealing image to represent their products or services.

While graphic designers do not typically create plans used for manufacturing or building products, as most drafters do, their duties are similar. Graphic designers create visual images for product packaging, company logos, posters, banners, and advertisements. They are responsible for producing the images and text that represent a particular product or service.

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Jill Braaten, photographer

Using Technology

Most graphic designers now use computer software to create their designs. Programs such as Adobe Illustrator® and Adobe Photoshop® are used to create and alter images. Desktop publishing programs such as Adobe PageMaker®, InDesign® CS2, and Quark XPress® help graphic designers position and align the text and images or lay out a design.

Conveying Concepts

Graphic designers work closely with clients to figure out a way to communicate an idea or concept to the targeted audience. An understanding of what the client wants is critical. Graphic designers must also understand who will be using the product (or reading the book, poster, or billboard). They target their designs for the intended customer group according to published demographic surveys.

► Career Activities

1. If you were considering becoming a graphic designer, what type of experience do you think would be beneficial to your goals?

2. Compare and contrast the skills required for a graphic designer and those required for a drafter.

Wohlers: Applying
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II. Drawing Aids and
Controls

13. Zooming

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169

Chapter 13

Zooming

Objectives

- Zoom in on portions of a drawing to view or add details
- Use the **ZOOM** command while another command is entered
- Control view resolution

Vocabulary

realtime zooming
transparent zooms
view resolution

Large drawings with detail can be difficult, and at times impossible, to view legibly on even the largest computer monitors. AutoCAD offers a zooming capability that magnifies areas of the drawing. Remarkably, this capability can magnify objects by as much as ten trillion times. While this may seem impractical for most work, scientists and astronomers who produce drawings and models of incredible scale find this capability very useful. Consider viewing a solar system, in which distances are measured in millions or billions of miles.

Taking a Closer Look

Let's apply the **ZOOM** command.

1. Start AutoCAD, select the **Use a Wizard** button from the **Startup** dialog box, pick **Quick Setup**, and pick **OK**.

> **HINT**
> If the **Startup** dialog box does not appear when you start AutoCAD, enter the **STARTUP** system variable and enter **1**. Then enter the **NEW** command.

2. Pick the **Architectural** radio button and **Next**.

3. Enter **20′** for the **Width** and **15′** for the **Length** of the drawing area and pick **Finish**.

> **HINT**
> Be sure to include an apostrophe (') for the foot mark. If you don't, AutoCAD will assume that you mean inches, which would be much too small for the following office drawing. Note that when you work in inches, adding a double quote (") after the number is optional.

4. Pick the **Workspace Switching** button on the status bar, select the **AutoCAD Classic** workspace, and close all floating tool palettes and toolbars.

5. Enter **Z** for **ZOOM** and **A** for **All**.

This zooms the drawing to the 20′ × 15′ area.

6. Enter **GRID** and enter **1′** or **12** (inches).

7. Set the snap grid to **2″**.

8. Draw the room shown in Fig. 13-1, including the table and chair. Estimate the distance from the wall to the table.
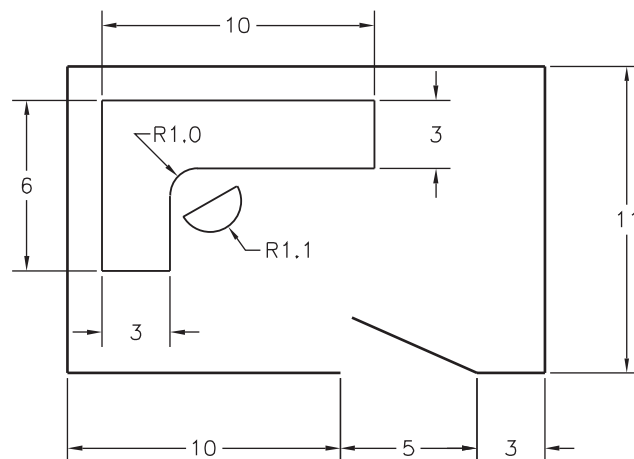
Fig. 13-1

9. Save your work in a file named **zoom.dwg**.

Methods of Zooming

AutoCAD provides several options for zooming in or out on a drawing. You can enter these options by typing **Z** (for **ZOOM**) at the keyboard and then entering the first letter of the option, or you can use the many zoom buttons. These buttons are located on the **Zoom** toolbar and on the flyout on the **Zoom Window** button on the **Standard** toolbar. (Depress and hold the pick button to view the flyout.)

STANDARD

1. Pick the **Zoom Window** button on the **Standard** toolbar.

This enters the **ZOOM** command.

2. In response to **Specify first corner**, place a window around the table and chair, as shown in Fig. 13-2. (Pick a point for the first corner and then select a point for the opposite corner.)

The table and chair should magnify to fill most of the screen.
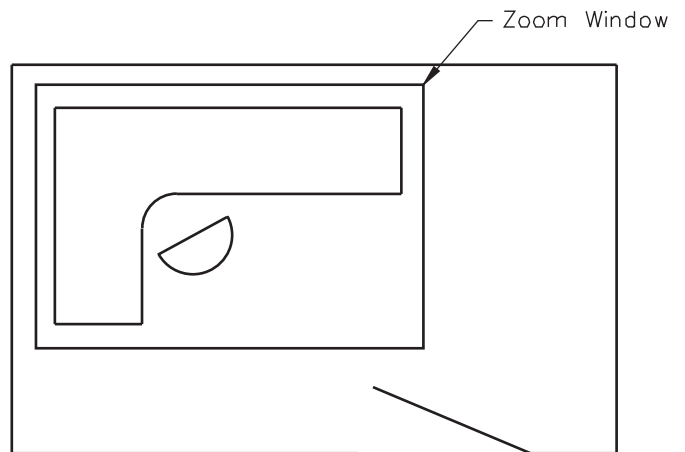
3. Next, draw the basic representations of the components that make up a CAD workstation, as shown in Fig. 13-3. Approximate their sizes, and omit the text. Save your work.

Zoom Window

Fig. 13-2

Printer

Mouse

Computer and Monitor
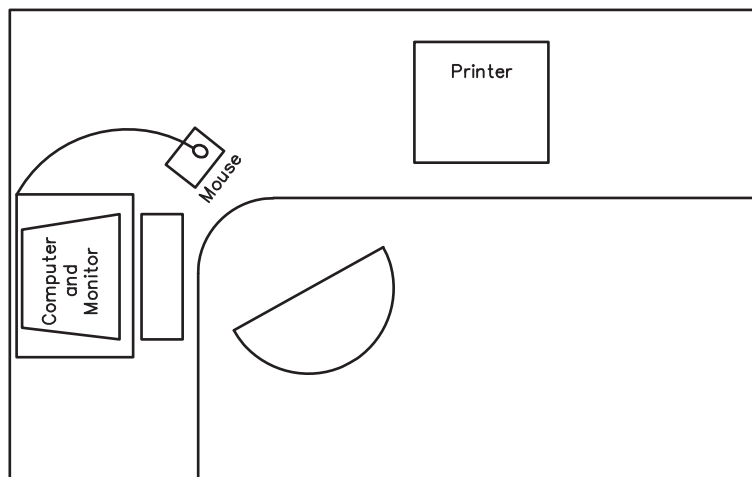
Fig. 13-3

Now let's zoom in on the keyboard.

STANDARD

4. Pick the **Zoom Window** button again, and this time place a window around the lower half of the keyboard.

Let's zoom in further on the keyboard, this time using the **Zoom In** option.

ZOOM

5. From the **Zoom** toolbar or the **Zoom** flyout on the **Standard** toolbar, pick the **Zoom In** button.

6. If you cannot see the bottom of the keyboard, zoom again (using the **Zoom In** or **Zoom Window** button) so that the bottom of the keyboard fills most of the screen.

7. In the lower left corner of the keyboard, draw a small square to represent a key, as shown in Fig. 13-4. (You may need to turn snap off first.)
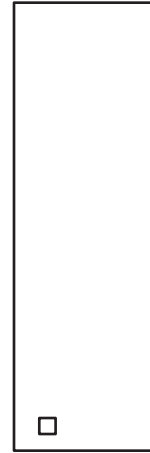
Fig. 13-4

8. Zoom in on the key.

9. Using the **POLYGON** command, draw a small trademark on the key, as shown in Fig. 13-5.

STANDARD

10. Next, pick the **Zoom Previous** button on the **Standard** toolbar and watch what happens.

You should now be at the previous zoom factor. The trademark should look smaller.
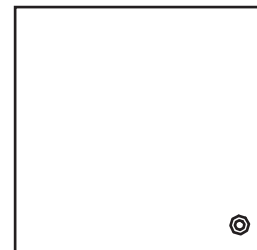
Fig. 13-5

STANDARD

11. Pick the **Zoom All** button, or enter **Z** and **A** at the keyboard.

ZOOM

AutoCAD zooms the drawing to its original size.

12. Pick the **Zoom Extents** button or enter **Z** and **E** at the keyboard.

AutoCAD zooms the drawing as large as possible while still showing the entire drawing on the screen.

ZOOM

You can also zoom to the extent of a select object or set of objects.

13. Pick the **Zoom Object** button or enter **Z** and **O** at the keyboard.

14. Use a window to select the computer and keyboard and press **ENTER**.

AutoCAD zooms in, making the selected objects the extent of the drawing screen.

ZOOM

15. Pick the **Zoom Extents** button or enter **Z** and **E** at the keyboard.

It is also possible to enter a specific magnification factor. AutoCAD considers the entire drawing area (the **ZOOM All** display) to have a magnification of 1.

By entering a different number with the **ZOOM** command entered, you can change the magnification relative to this. If you need to change the magnification relative to the current magnification (instead of 1), you can accomplish this by typing an **x** after the number.

16. Pick the **Zoom Scale** button and enter **2**.

This causes AutoCAD to magnify the drawing area by two times relative to the **ZOOM All** display (**All = 1**).

17. Enter **ZOOM** and **.5**.

This causes AutoCAD to shrink the drawing area by one half relative to the **ZOOM All** display.

18. Pick the **Zoom In** button; pick it again.

19. Pick the **Zoom Out** button.

20. Pick the **Zoom Extents** button again.

21. Enter **ZOOM** and **.9x**.

Because you placed an **x** after the number, the zoom magnification decreases to nine tenths (90%) of its current size, not nine tenths of the **ZOOM All** display area.

22. Continue to practice using the **ZOOM** command by zooming in on the different CAD components of the drawing and including detail on each.

23. Save your work.

Realtime Zooming

AutoCAD's **realtime zooming** allows you to change the level of magnification by moving the cursor. This is one of the fastest ways to zoom.

1. From the **Standard** toolbar, pick the **Zoom Realtime** button.

This places you in realtime zoom mode. Notice that the crosshairs has changed.

2. Using the pick button, click and drag upward to zoom in on the drawing.

3. Click and drag downward to zoom out on the drawing.

4. Press **ESC** or **ENTER** to cancel realtime zooming.

5. Roll the center wheel button on the mouse to zoom in or out on the drawing.

> The status bar contains a **Zoom** button too. Using this **Zoom** button is the equivalent of typing **ZOOM** (or **Z**) at the keyboard.

ZOOM

ZOOM

ZOOM

ZOOM

STANDARD

Zooming Transparently

With AutoCAD, you can perform **transparent zooms**. This means you can use **ZOOM** while another command is in progress. To do this, enter **'ZOOM** (notice the apostrophe) at any prompt that is not asking for a text string.

1. Enter the **LINE** command and pick a point on the screen.

2. At the prompt, enter **'Z**.

AutoCAD lists the options preceded by **>>** to remind you that you are in transparent mode.

3. Zoom in on any portion of the screen.

Notice the message **Resuming LINE command**.

4. Pick an endpoint for the line and terminate the command.

> NOTE
> Picking any of the zoom buttons also permits you to perform a transparent zoom.

5. Erase the line and enter **ZOOM All**.

Controlling View Resolution

The accuracy with which curved lines appear on the screen is known as **view resolution**. The **VIEWRES** command allows you to set the view resolution for objects in the current drawing.

1. Enter the **VIEWRES** command.

2. In reply to **Do you want fast zooms?** accept the **Yes** default value.

Notice that the circle zoom percent (screen resolution) is 1000 by default. AutoCAD allows you to enter any number from 1 to 20,000.

3. Enter **20** for the circle zoom percent and notice the coarse appearance of the arcs in the drawing.

4. Enter **VIEWRES** again; press **ENTER** and then enter **1000**.

Notice the smooth appearance of the arcs on the screen.

5. Save your work and exit AutoCAD.

Chapter 13 **Review & Activities**

• REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Explain why the **ZOOM** command is useful.
2. Cite one example of when it would be necessary to use the **ZOOM** command to complete an engineering drawing and explain why.
3. Describe each of the following **ZOOM** options.
   a. **All**
   b. **In**
   c. **Extents**
   d. **Previous**
   e. **Window**
4. What is a transparent zoom? When might a transparent zoom be useful?
5. Why might you often choose to use the realtime zoom option instead of other zoom methods?
6. Explain the purpose of the **VIEWRES** command.

• CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. AutoCAD drawings are stored in a vector format. Many other graphics programs, such as Microsoft Paint® and Adobe Photoshop®, store their drawings in a raster format. Find out the differences between the raster and vector formats. Can you convert a drawing in a vector format to a raster format? Can you convert a raster drawing into a vector format? Explain.
2. Investigate the **ZOOM Center** option. What is its purpose? When might you want to use it? Explain.

Continued

Chapter 13: Zooming   **169**

Chapter 13 **Review & Activities**

Work the following problems to practice the commands and skills you learned in this chapter.

1. As a general rule, the outside diameter of the washer face on a hex bolt head is the same as the distance across the hex flats. This dimension is approximately $1^1/2$ times the diameter of the bolt shank. Draw the head and washer face of the $1/4''$ hex bolt shown in Fig. 13-6.
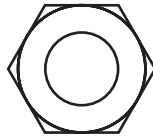
Fig. 13-6

Use the following guidelines:

- Create a new drawing using the **Quick Setup** wizard, fractional units, and a drawing area of 20×15.
- Create the outer edge of the washer face using a diameter of $7/16$. Place it near the center of the screen.
- Zoom a window so that the circle almost fills the drawing area.
- Draw a hexagon (6-sided polygon) with the same center as the circle and a distance across the flats of $7/16$. In other words, circumscribe the hexagon around a $7/16''$ circle.
- Draw a second circle with a diameter of $1/4''$ to represent the shank of the bolt.
- Enter **ZOOM All** and notice how small the bolt head is. Even if you had set the drawing area for the default of 12×9, it would be impossible to draw the bolt head accurately without using the **ZOOM** command.

2. Create a drawing such as an elevation plan of a building, a site plan of a land development, or a view of a mechanical part. Using the **ZOOM** command, zoom in on the drawing and include details. Zoom in and out on the drawing as necessary using the various **ZOOM** options and transparent zooms.

Review & Activities Chapter 13

3. Draw the kitchen floor plan shown in Fig. 13-7. Then zoom in on the kitchen sink. Edit the sink to include the details shown in Fig. 13-8. Save the file as **kitchen.dwg**.
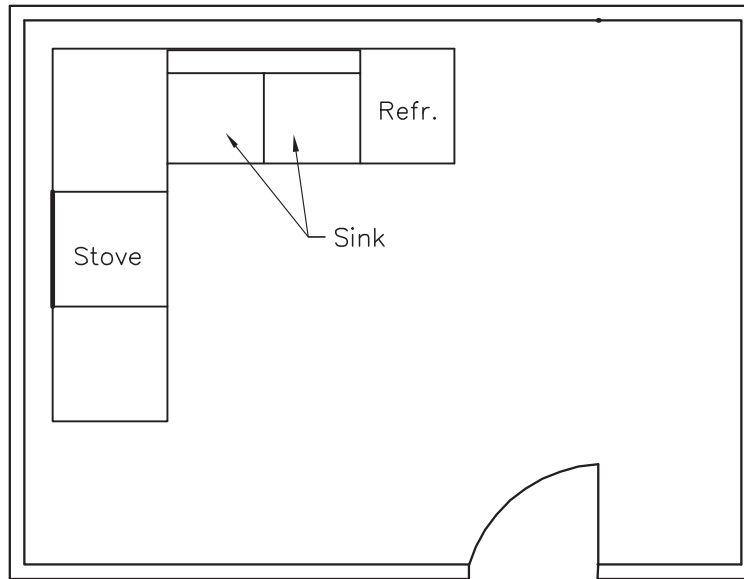
Fig. 13-7
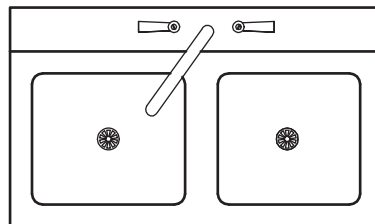
Fig. 13-8

Courtesy Kathleen P. King, Fordham University, Lincoln Center

4. Refer to **land.dwg** on the Online Learning Center (OLC) in the Drafting & Design Problems section. In the middle third of the drawing, zoom in on details such as picnic tables. Add and edit objects to improve the drawing. Zoom out. Apply **VIEWRES** and transparent zooms.

Continued

Chapter 13 **Review & Activities**

● **USING PROBLEM-SOLVING SKILLS**

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Use the **ZOOM** command to draw the door with a fan window shown in Fig. 13-9. Approximate all dimensions.

2. Draw the gasket shown in Fig. 13-10. Do not dimension it. Use construction lines for center lines and erase them when the drawing is complete. Are you sure the 1.5 radius is in fact a radius? Use the **ZOOM** command to find out. One way to create this drawing is to draw two R.8 circles and one R1.50 circle and locate the four lines with the **Tangent** object snap. You can then erase the circles and use the **ARC** command to connect the lines. When you have finished the drawing, use the **ZOOM** command to ensure that the intersections between lines and arcs are made correctly.
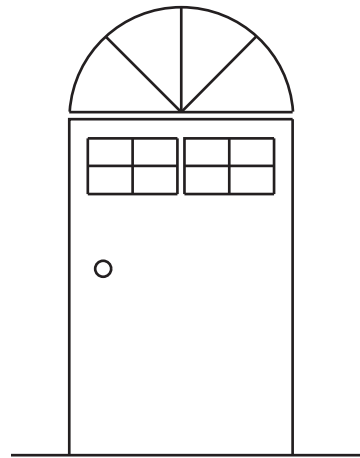
Fig. 13-9
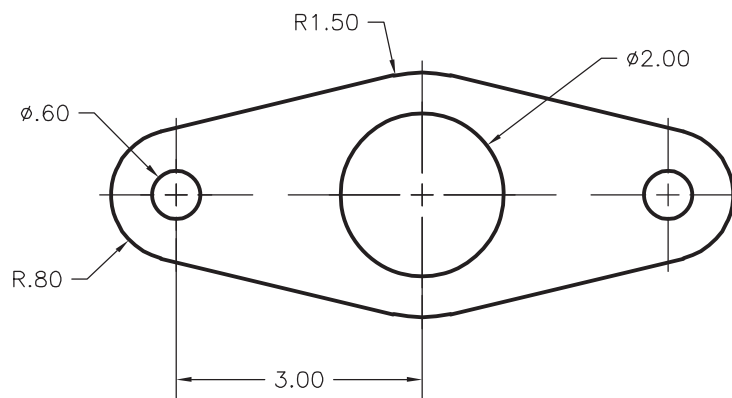
Fig. 13-10

AutoCAD at Work

Patent Drafter

Have you ever had a brilliant idea for an invention? Something so simple, perhaps, that you couldn't believe someone else didn't think of it first? Well, then, you have a lot of work to do to make sure that you really are the first one to come up with your idea. Conducting research on previously patented inventions is necessary to avoid "stealing" someone else's idea.

The process inventors go through to obtain a patent is complicated and time-consuming, but if their ideas are truly unique, the feeling of accomplishment is worth the effort. Inventors don't have to do all this work by themselves, however. Many people are available to help make the research and approval process easier. Attorneys and patent drafters are examples of people who play key roles in securing a patent for an invention.

© Stockbyte/Getty Images/RF

Drawing the Idea

Patent drafters produce drawings of both existing products and new product ideas. Drawings of existing products include carefully labeled components; this is necessary to identify parts that may have already been patented.

When designing drawings of new product ideas, drafters use information provided by the inventor (such as rough sketches and notes) to create a picture of what the new product will look like.

It Takes Teamwork

A patent drafter is just one of several individuals who help an idea become an actual product. The inventor's attorney is responsible for keeping the process of obtaining a patent moving. It is the inventor's attorney who typically selects and hires a patent drafter. The drafter must then work closely with the attorney and the inventor to ensure his or her work is consistent with the written description of the item.

▶ Career Activities

1. Research the patent process on the Internet. How would you go about applying for a patent?

2. Think of an invention you can't imagine living without. Find out who patented it, and when, and how it has changed since its first production.

Chapter 14

Panning and Viewing

Objectives

- Pan from one zoomed area to another
- Produce and use named views
- Use **Aerial View** to navigate in a complex drawing

Vocabulary

panning
view box

Zooming in on a drawing solves part of the problem of working with fine detail on a drawing. A method called **panning** solves the other part. Panning permits you to move the zoom window to another part of the drawing while keeping the same zoom magnification. This means you do not have to constantly zoom in and out to examine detail.

Panning

Note the degree of detail in the architectural floor plan shown in Fig. 14-1. The drafter who completed this CAD drawing zoomed in on portions of the floor plan in order to include detail. For example, the drafter zoomed in on the kitchen to place cabinets and appliances, as shown in Fig. 14-2. Suppose the drafter wants to include detail in an adjacent room but wants to simply "move over" to the adjacent room. This operation can be accomplished by panning.

There are several ways to pan in AutoCAD. You can use the **PAN** command, various panning buttons, the center button on the mouse, or the scroll bars to move around in a drawing.

LOWER LEVEL FLOOR PLAN
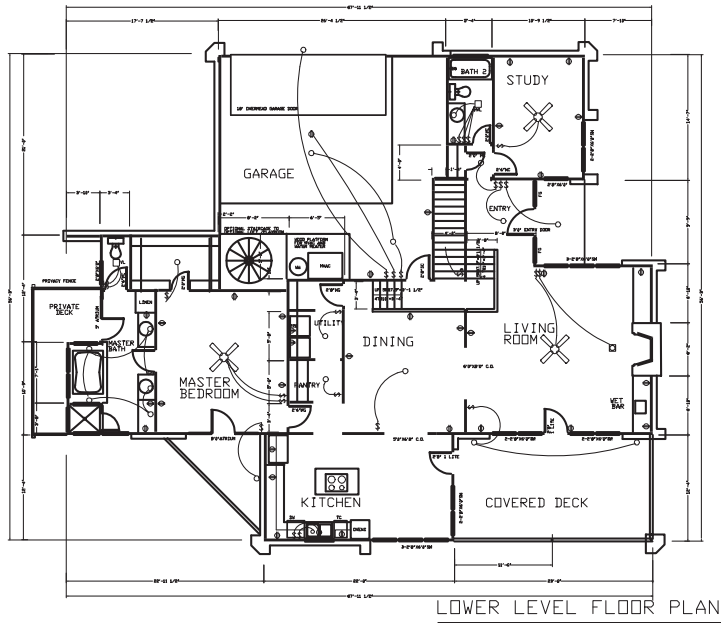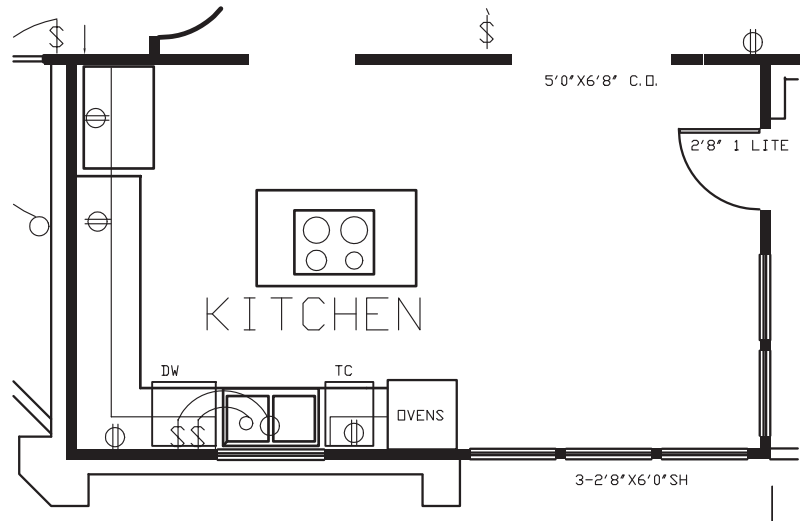
Fig. 14-1

Fig. 14-2

1. Start AutoCAD, select the **Open a Drawing** button from the **Startup** dialog box, and open the drawing named **db_samp.dwg** located in AutoCAD's **Sample/Database Connectivity** folder.

2. Pick **Save As...** from the **Menu Browser** or the **File** pull-down menu, open the folder with your name, and pick the **Save** button.

3. Pick the **Workspace Switching** button on the status bar and select the **2D Drafting & Annotation** workspace.

4. Zoom in on the lower right area of the drawing. (The zoom buttons are in the **Navigate** panel on the **View** tab of the **Ribbon.**)

Chapter 14: Panning and Viewing   175

Realtime Panning

Let's pan to the upper left side of the drawing.

1. Pick the **Pan** button in the **Navigate** panel of the **Ribbon** or enter **P** for **Pan**.

The crosshairs changes to a hand.

2. In the upper left portion of the screen, click and drag down and to the right.

The drawing window moves down and to the right.

3. Experiment further with the **PAN** command until you feel comfortable with it. Pan in different directions and at different zoom magnifications.

4. Press **ESC** or **ENTER** to exit the realtime pan mode.

5. Click the center wheel button on the mouse and drag to pan in the drawing area.

Transparent Panning

You can use the **PAN** command transparently. This is particularly useful for reaching points that are not currently visible. To do this, enter **'PAN** (notice the apostrophe) at any prompt that does not ask for a text string.

1. Enter the **LINE** command and pick a point anywhere on the screen.

2. Pick the **Pan** button on the **Ribbon** or enter **'P** at the keyboard.

3. Pan to a new location and press **ESC** or **ENTER** to exit the realtime panning mode.

4. Pick a second point to create a line segment and terminate the **LINE** command.

5. Erase the line.

Alternating Realtime Zooms and Pans

AutoCAD provides a fast method of alternating between realtime pans and zooms.

1. Pick the **Zoom Realtime** button on the **Utilities** panel of the **Ribbon.**

2. Zoom in on the drawing.

3. Right-click the pointing device and select **Pan** from the shortcut menu.

4. Pan to a new location.

5. Right-click and pick one of the **Zoom** options on the shortcut menu.

6. When you are finished, pick **Exit** from the shortcut menu or press **ESC** or **ENTER** to exit the realtime zoom and pan modes.

Capturing Views

Imagine that you are working on an architectural floor plan. You have zoomed in on the kitchen to include details such as appliances, and now you are ready to pan over to the master bedroom. Before leaving the kitchen, you foresee a need to return to the kitchen for final touches. However, by the time you're ready to do this final work on the kitchen, you may be working at a different zoom magnification or at the other end of the drawing. The **VIEW** command solves the problem.

1. Using realtime panning and zooming, find and zoom in on the four offices in the lower right area of the drawing.
2. Click the down-facing arrow on the right side of the **Quick Access** toolbar and select **Show Menu Bar** from the shortcut menu.
3. From the **View** menu on the **Menu Bar**, select **Named Views...**, or enter **VIEW** at the AutoCAD prompt.

This displays the **View Manager** dialog box, as shown in Fig. 14-3.

4. Pick the **New...** button to open the **New View/Shot Properties** dialog box.
5. For **View** name, enter **Executives** and pick **OK**.

AutoCAD adds the named view to the list under **Model Views**.

6. Pick the **OK** button to close the **View Manager** dialog box.
7. Pan to a new location on the drawing.
8. Enter the **VIEW** command, pick the **Executives** view from the list, pick the **Set Current** button in the dialog box, and pick **OK** to restore the named view.
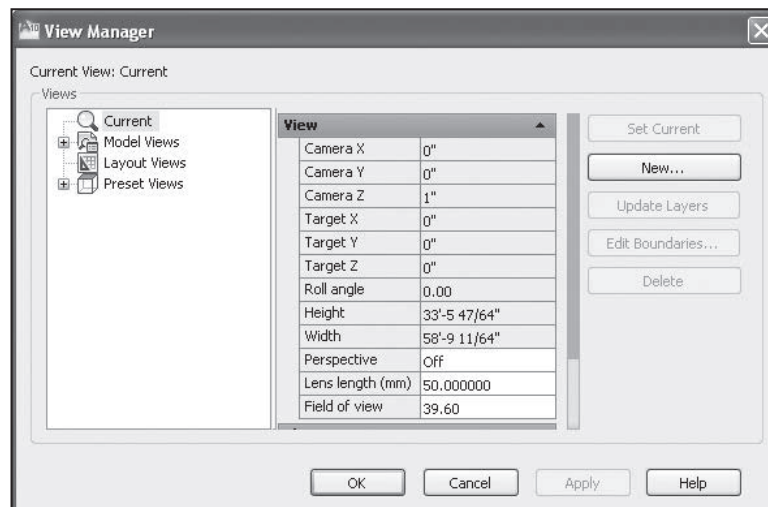
Fig. 14-3

Aerial View

Aerial View is a fast and easy-to-use navigational tool for zooming and panning.

1. From the **View** menu on the **Menu Bar**, pick **Aerial View**.

This displays the **Aerial View** window with the current drawing inside, as shown in Fig. 14-4. The buttons in the **Aerial View** window should look familiar. The black rectangle represents the zoomed area.
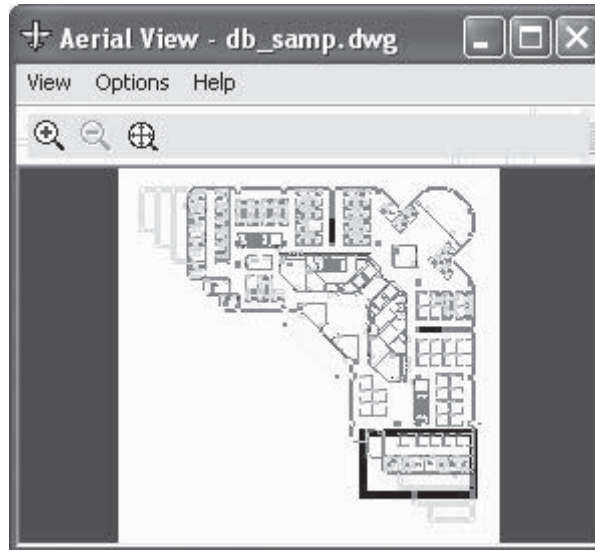
2. Move the crosshairs into the **Aerial View** window and click once.

A new crosshairs appears.

Fig. 14-4

3. Move the window (called a **view box**) around.

AutoCAD zooms in on the area defined by the view box. Meanwhile, **Aerial View** maintains a view of the entire drawing. The view box, depicted by a black rectangle, shows the size and location of the current view in the graphics screen.

4. Right-click to freeze the view box.

Aerial View shows the location of the current zoom window.

5. Press the pick button to activate the view box.

6. Press the pick button again to change the size of the view box and then pick another point to lock in a new size.

7. Move the view box to a new location and right-click.

8. On your own, experiment with the buttons and menu items in the **Aerial View** window.

9. Close the **Aerial View** window.

10. **ZOOM All** and exit AutoCAD without saving.

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II. Drawing Aids and
Controls

14. Panning and Viewing

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185

Chapter 14 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Explain why panning is useful.
2. How can you pan to another part of a drawing while another command is entered in AutoCAD?
3. Explain why named views are useful.
4. Explain the method of alternating between realtime pans and zooms.
5. What is the primary benefit of using **Aerial View**?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Many zoom and pan variations have been introduced in the last two chapters. Describe how and when you might use a combination of them, but explain also why it may be impractical to use all of them. Which methods of zoom and pan are the fastest, and which seem to be the most practical for most applications?
2. Is **Aerial View** an alternative to using the **PAN** command? Explain when you would use each most effectively.

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Open the **attrib.dwg** file located in the **Sample/ActiveX/ExtAttr** folder. Use AutoCAD's zoom feature to magnify the parts list located in the upper right area of the drawing. Use realtime zoom and pan to study all the items on the list. What is the part number of the Inner Plate? How many rivets are required for the assembly?

Continued

Chapter 14 **Review & Activities**
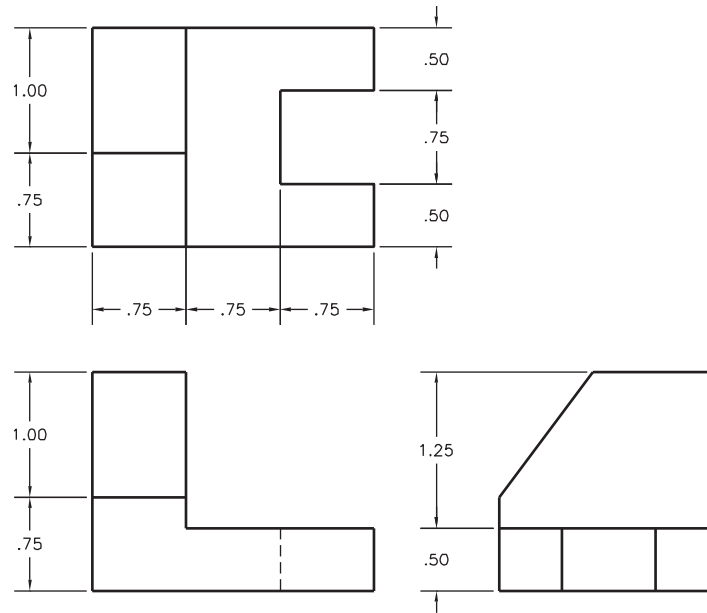
Fig. 14-5

2. Draw each of the views shown in Fig. 14-5 using the dimensions shown. However, do not include the dimensions on the drawing. Zoom in on one of the views and store it as a named view. Then pan to each of the other views and store each as a named view. Restore each named view and alter each shape. Be as creative as you wish, but be certain that all three of the views are consistent with one another.

3. Perform dynamic pans and zooms on the drawing in problem 2.

4. Open the **kitchen.dwg** file you created in problem 3 of Chapter 13. Add the details for the stove and refrigerator as shown in Fig. 14-6. Open **Aerial View** and use it to position the zoom window as you work. Save the file as **kitchen2.dwg**.
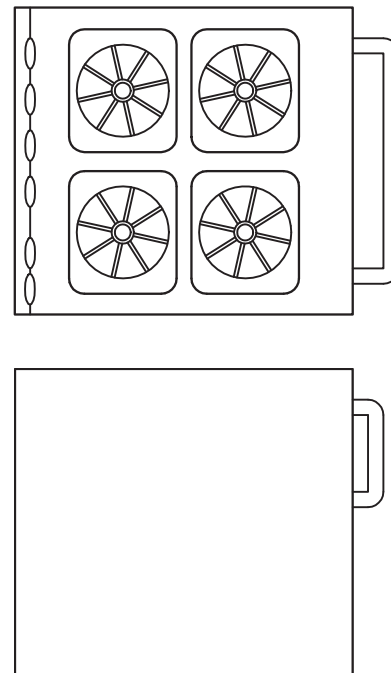
Fig. 14-6

Review & Activities Chapter 14

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. As the person responsible for computer allocation, you need to know how many offices are without computers. Open the **db_samp.dwg** file in AutoCAD's **Sample/Database Connectivity** folder (Fig. 14-7). Using dynamic zoom and pan as necessary, make a list by office number of offices that do not have computers. Do not save any changes you may have made to the drawing.
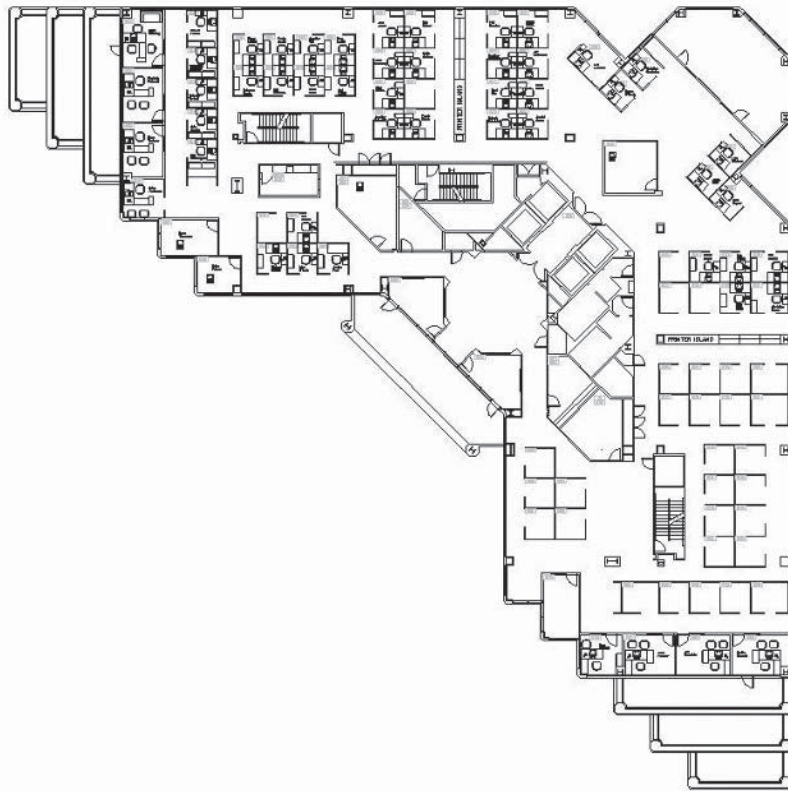
Fig. 14-7

2. Reopen **db_samp.dwg** and use **Aerial View** to make a list of rooms with designations of **COPY**, **I.S.**, **STOR.**, or **COFFEE**. Which of these rooms should have computers? Do you think office number 6001 should have a computer? Why or why not? Do not save the drawing.

Part 2 Project | Applying Chapters 10-14

Patterns and Developments

Many manufactured products begin as flat pieces of metal, plastic, or other material. Examples include cereal boxes, heating and cooling ducts, and even items of clothing. Manufacturers use flat *patterns* to cut material to the proper size and shape. The construction of these patterns is known as *development*. Fig. P2-1 shows a box constructed for a company that distributes breakfast cereals, as well as the pattern used to create the box.

▶ Description

Study the magazine holder shown in Fig. P2-2. It is to be manufactured from cardboard, and you are responsible for developing the pattern needed to cut the cardboard to size. Show the fold lines as well as the overall shape of the pattern, as demonstrated in Fig. P2-1.
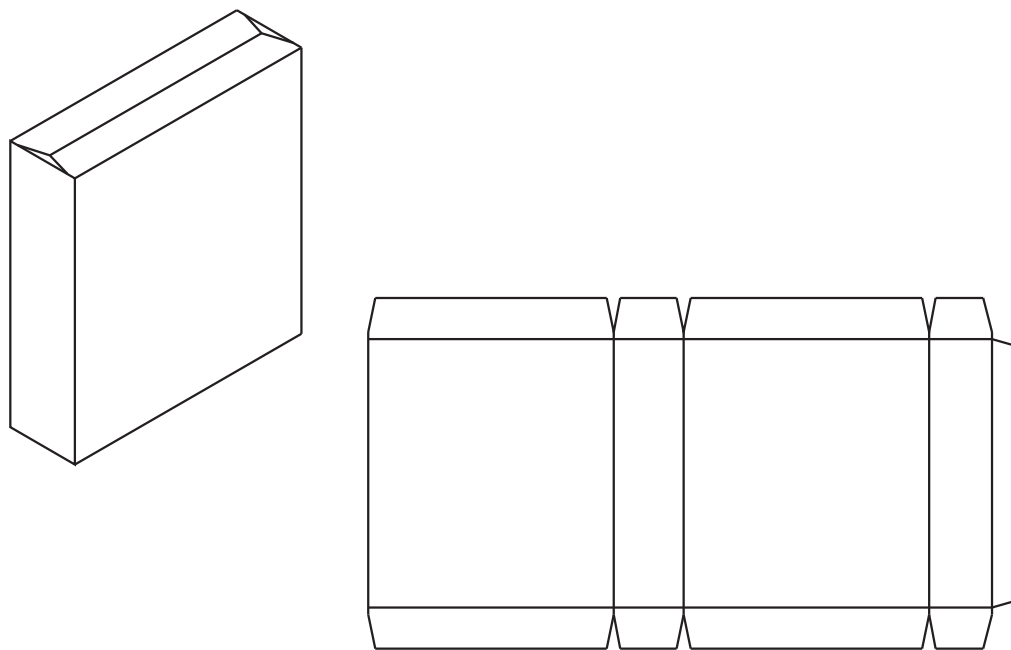
Fig. P2-1
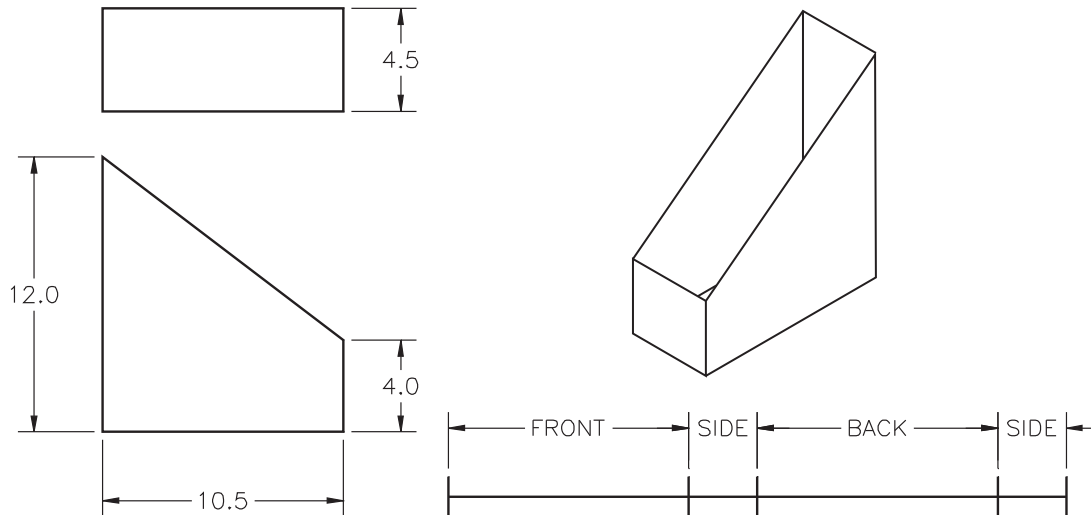
Fig. P2-2

Hints and Suggestions

1. Draw the orthographic views of the magazine holder.

2. Referring to the orthographic views as necessary, create a stretchout line for the magazine holder. A *stretchout line* is a line that shows the total length of the finished pattern and the relative lengths of each part. An example stretchout line for the magazine holder is shown in Fig. P2-2.

3. Using the orthographic views and the stretchout line, create the development. Use object snaps and other construction aids as necessary, and use **ZOOM** and **PAN** to simplify your drawing task.

▶ Summary Questions

Your instructor may direct you to answer these questions orally or in writing. You may wish to compare and exchange ideas with other students to help increase efficiency and productivity.

1. Pinpoint any difficulties you encountered with this project and explain how you solved them.

2. Which construction aids did you use to create the orthographic views of the magazine holder? Which aids were used for the development? Explain how these aids helped.

3. How would you create a pattern for a soup can? In what ways would the development be different from that for the magazine holder? Explain.

Chapter **15**

Solid and Curved Objects

Objectives

- Produce solid-filled objects
- Control the appearance of solid-filled objects
- Create and edit polylines
- Create and edit splines

Vocabulary

B-splines
control points
polyline
screen regeneration
splines
traces
vertex

Many technical drawings contain thick lines and solid objects. Drawings for printed circuit boards, for instance, include relatively thick lines called **traces**. The trace lines represent the metal conductor material that is etched onto the PC board. Drawings and models that use curved lines often take advantage of polylines and spline curves. Examples are consumer products and business machines ranging from phones, pagers, and cosmetic products to printers, copiers, and fax machines. Many of the plastic injection-molded parts on these products are designed using polylines and spline curves. You will learn more about polylines and spline curves later in this chapter.

Note the heavy lines in the drawing shown in Fig. 15-1. They were used to make the drawing descriptive and visually appealing.
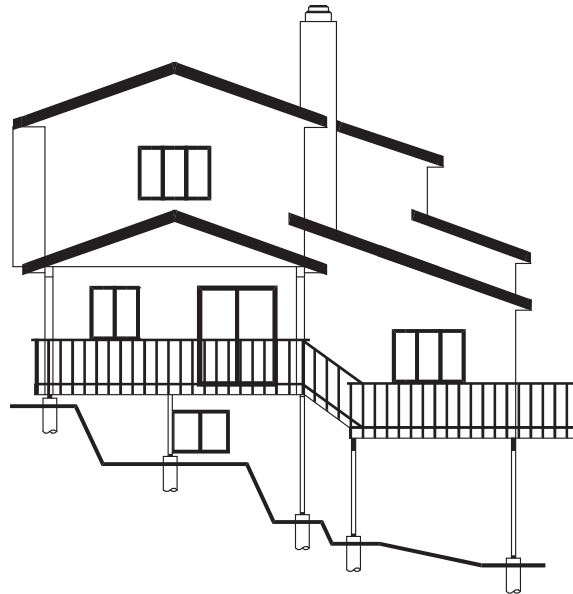
Fig. 15-1

Drawing Solid Shapes

The **SOLID** command produces solid-filled, two-dimensional objects.

1. Start AutoCAD, select the **Start from Scratch** button from the **Create New Drawing** dialog box, pick **Imperial** units, and pick **OK**.

> **HINT** If the **Startup** dialog box does not appear when you start AutoCAD, enter the **STARTUP** system variable and enter **1**. Then enter the **NEW** command.

2. Pick the **Workspace Switching** button on the status bar, select the **AutoCAD Classic** workspace, and close all floating toolbars and palettes.

3. Enter the **SOLID** command at the keyboard.

4. Produce a solid-filled object similar to the one in Fig. 15-2. Pick the points in the exact order shown.

Pt 1   Pt 3

Pt 2   Pt 4

Fig. 15-2

> **HINT** Consider using snap and grid.

5. Pick a fifth and sixth point.

AutoCAD adds a new piece to the solid object.

6. Experiment with the **SOLID** command. (If you pick the points in a different order, AutoCAD creates an hourglass-shaped object.)

7. Press **ENTER** to terminate the **SOLID** command.

Keep the objects on the screen because you will need them for the following section.

Controlling Object Fill

With the **FILL** command, you can control the appearance of solids, wide polylines, multilines, and hatches. Polylines are discussed in the next section of this chapter. You will learn more about multilines in Chapter 16 and hatches in Chapter 19.

FILL is either on or off. When **FILL** is off, only the outline of a 2D solid is represented on the screen. This saves time when the screen is regenerated.

1. Enter the **FILL** command at the keyboard.

2. Enter **OFF**.

To see the effect of turning **FILL** on or off, you must force AutoCAD to recalculate all of the vectors on the screen. This is known as **screen regeneration**. To force the screen to regenerate, AutoCAD provides the **REGEN** command.

3. From the **View** pull-down menu, pick **Regen**, or enter **REGEN** at the keyboard.

4. Reenter the **FILL** command and turn it **ON**.

5. Enter **REGEN** to force another screen regeneration.

6. Erase all objects on the screen.

Polylines

A **polyline** is a connected sequence of line and arc segments that is treated by AutoCAD as a single object. Polylines are often used in lieu of conventional lines and arcs because they are more versatile.

Drawing a Thick Polyline

Let's use the **PLINE** command to create the electronic symbol for a resistor, as shown in Fig. 15-3.

Fig. 15-3

1. Set snap at **.5**.
2. Pick the **Polyline** button on the **Draw** toolbar, or enter **PL** (the **PLINE** command alias).

DRAW

This enters the **PLINE** command.

3. Pick a point in the left portion of the screen.

The **PLINE** options appear.

4. Enter **W** (for **Width**) and enter a starting and ending width of **.15** unit. (Notice that the ending width value defaults to the starting width value.)
5. Draw the object by approximating the location of the endpoints. If you make a mistake, undo the segment. Press **ENTER** when you have finished drawing the resistor.
6. Click anywhere on the polyline to display its grips.

Notice that the entire polyline is treated as a single object.

7. Save your work in a file named **poly.dwg**.

Editing a Polyline

AutoCAD supplies a special editing command called **PEDIT** to edit polylines. **PEDIT** allows you to perform many advanced functions, such as joining polylines, fitting them to curve algorithms, and changing their widths. Let's edit the polyline using the **PEDIT** command.

MODIFY II

1. Display the **Modify II** toolbar.

2. Pick the **Edit Polyline** button on the **Modify II** toolbar.

This enters the **PEDIT** command.

3. Pick the polyline.

The **PEDIT** options appear.

4. Enter **W** and specify a new width of **.2** units.

The width of the polyline changes. As you can see, using polylines can be an advantage when there is a possibility that you will need to change the width of several connected line segments at a later time.

Let's close the polyline, as shown in Fig. 15-4.

5. Enter **C** for **Close**.

The **PEDIT Fit** option creates a smooth curve based on the locations of the vertices of the polyline. In this text, a **vertex** (plural *vertices*) is any end-point of an individual line or arc segment in a polyline. Let's do a simple curve-fitting operation.

Fig. 15-4

6. Enter **F** for **Fit**.

Notice how the drawing changed. The curve passes through all of the vertices of the polyline.

7. Enter **D** (for **Decurve**) to return it to its previous form.

Next, let's move one of the vertices, as shown in Fig. 15-5.

8. Enter **E** for **Edit vertex**.

Notice that a new set of choices becomes available. Also notice the **x** at one of the vertices of the polyline.

Fig. 15-5

9. Move the **x** to the vertex you want to change by pressing **ENTER** several times.

10. Enter **M** for **Move** and pick a new point for the vertex.

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III. Drawing and Editing

15. Solid and Curved
Objects

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195

11. To exit the **PEDIT** command, enter **X** (for **eXit**) twice.

12. Save your work.

PEDIT contains many more editing features. Experiment further with them on your own.

Exploding Polylines

The **EXPLODE** command gives you the ability to break up a polyline into individual line and arc segments.

1. Enter the **UNDO** command and select the **Mark** option. (We will return to this point at a later time.)

2. Pick the **EXPLODE** button on the **Modify** toolbar.

3. Select the polyline and press **ENTER**.

MODIFY

Notice the message: **Exploding this polyline has lost width information. The UNDO command will restore it.**

This is the result of applying **EXPLODE** to a polyline that contains a width other than the default. You now have an object that contains many line objects for easier editing, but you have lost the line width.

4. To illustrate that the drawing is now made up of several individual objects, pick one of them.

Drawing Curved Polylines

In some drafting applications, there is a need to draw a series of continuous arcs to represent, for example, a river on a map. If the line requires thickness, the **ARC Continue** option will not work, but the **PLINE Arc** option can handle this task.

1. Enter the **PLINE** command and pick a point anywhere on the screen.

2. Enter **A** for **Arc**.

DRAW

A list of arc-related options appears.

3. Move the crosshairs and notice that an arc begins to develop.

4. Enter the **Width** option and enter a starting and ending width of **.1** unit.

5. Pick a point a short distance from the first point.

6. Pick a second point, and a third.

7. Press **ENTER** when you are finished.

8. Enter **UNDO** and **Back**.

Spline Curves

Splines are curves that use sampling points to approximate mathematical functions that define a complex curve. AutoCAD allows you to create splines, also referred to as **B-splines**, using two different methods. You can use the **PEDIT** command to transform a polyline into a spline curve, or you can use the **SPLINE** command to generate the spline directly.

Transforming a Polyline into a Spline

The **PEDIT Spline** option uses the vertices of the selected polyline as the control points of the curve. **Control points** are points that exert a "pull" on a curve, influencing its shape. The curve passes through the first and last control points and is pulled toward the other points, but it does not necessarily pass through them. The more control points you specify in an area, the more pull they exert on the curve.

MODIFY II

1. Pick the **Edit Polyline** button on the **Modify II** toolbar and select the polyline.
2. Enter **S** for **Spline**.

Do you see the difference between the **Spline** and **Fit** options? The **Spline** option uses the vertices of the selected polyline as control points, whereas the **Fit** option produces a curve that passes through all vertices of the polyline.

3. Enter **D** to decurve the object.
4. Enter **X** to exit the **PEDIT** command.

Through the **PEDIT** command, AutoCAD offers two spline types: quadratic B-splines and cubic B-splines. An example of each is shown in Fig. 15-6. The type of spline created by the **Spline** option depends on the setting of the **SPLFRAME** system variable. (A system variable in AutoCAD is similar to a command, except that it holds temporary settings and values instead of performing a specific function.)

Original   Fit Curve   Quadratic B−spline   Cubic B−spline
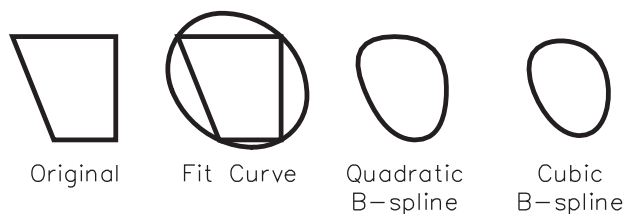
Fig. 15-6

The system variable **SPLINETYPE** controls the type of spline curve to be generated. Set the value of **SPLINETYPE** at **5** to generate quadratic B-splines. Set its value at **6** to generate cubic B-splines.

The **PEDIT Decurve** option enables you to turn a spline back into its frame, as illustrated by the previous Step 3. You can view the spline curve and its frame simultaneously by setting the **SPLFRAME** system variable to **1**.

Producing Splines Directly

With the **SPLINE** command, you can create spline curves using a single command. Spline curves are used extensively in industry to produce free-form shapes for automobile body panels and the exteriors of aircraft and consumer products. The spline curve itself is the building block for producing interesting and sometimes extremely complex designs.

1. From the **File** pull-down menu, pick **Save As...**, and enter **splines.dwg** for the file name.
2. Erase all the objects in the drawing.
3. Pick the **Spline** button on the **Draw** toolbar to enter the **SPLINE** command.
4. Pick a point and then a second point.
5. Slowly move the end of the spline back and forth and notice how the spline behaves.
6. Continue to pick a series of points.
7. Press **ENTER** three times to complete the spline and terminate the command.
8. Enter **SPLINE** again and pick a series of points.
9. Enter **C** for **Close**.
10. Press **ENTER** to terminate the command.

DRAW

DRAW

> **NOTE**
>
> The **Fit Tolerance** option changes the tolerance for fitting the spline curve to the control points. A setting of **0** causes the spline to pass through the control points. By changing the fit tolerance to **1**, you allow the spline to pass through points within 1 unit of the actual control points.
>
> You can easily convert a spline-fit polyline into a spline curve. Enter the **SPLINE** command, enter **O** for **Object**, and pick the polyline.

Editing Splines

Using grips, you can interactively edit a spline entity. The **SPLINEDIT** command provides many additional editing options.

1. Without any command entered, select one of the spline curves to display the grips.
2. Close the floating **Quick Properties** panel or move it out of the way.
3. Lock onto one of the grips and move it to a new location.

This is the fastest way to edit a spline.

MODIFY II

4. Press **ESC** to clear the grips.
5. From the **Modify II** toolbar, pick the **Edit Spline** button.

This enters the **SPLINEDIT** command.

6. Select your first spline.

The editing options appear. The **Close** option closes the spline curve. The **Move vertex** option enables you to edit a spline by moving its vertices. The **Refine** option permits you to enhance the spline data.

7. Enter **R** for **Refine** to display the refinement options.
8. Enter **A** for **Add control point**.
9. Pick several points on the spline to add new control points, and then press **ENTER**. (Turn off object snap before adding new control points.)
10. Press **ENTER** twice.

The **rEverse** option reverses the order of the control points in AutoCAD's database. The **Undo** option undoes the last edit operation.

11. Close the **Modify II** toolbar, save your work, and exit AutoCAD.

Chapter 15 Review & Activities

• REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. How would you draw a solid-filled triangle using the **SOLID** command?
2. What is the purpose of **FILL**, and how is it used?
3. What object types does **FILL** affect?
4. What is a polyline?
5. In what situations might you decide to use the **PLINE** command instead of the **LINE** and **ARC** commands?
6. Briefly describe each of the following **PLINE** options.
   a. **Arc**
   b. **Close**
   c. **Halfwidth**
   d. **Length**
   e. **Undo**
   f. **Width**
7. Briefly describe each of the following **PEDIT** command options.
   a. **Close**
   b. **Join**
   c. **Width**
   d. **Edit vertex**
   e. **Fit**
   f. **Spline**
   g. **Decurve**
   h. **Ltype gen**
   i. **Undo**
8. Explain a situation in which you might need to explode a polyline.
9. Describe the **SPLINETYPE** system variable.
10. What is the fastest method for editing a spline?

Continued

Chapter 15 **Review & Activities**

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Discuss possible uses for the **SOLID** and **FILL** commands. Under what circumstances might you use them? Give specific examples.
2. Discuss everyday applications of spline curves.

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Construct the building shown in Fig. 15-7 using the **PLINE** and **SOLID** commands. Specify a line width of .05 unit. Estimate the size and shape of the roof.

Fig. 15-7

Fig. 15-8

2. After you have completed problem 1, place the solid shapes as indicated in Fig. 15-8. Estimate their sizes and locations.
3. Use the **PLINE** command to draw a road sign indicating a sharp left corner, as shown in Fig. 15-9.

Fig. 15-9

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III. Drawing and Editing

15. Solid and Curved
Objects

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201
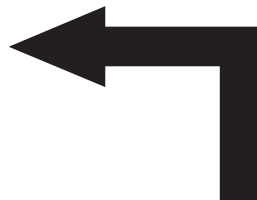
Review & Activities Chapter 15

4. The SI symbol shown in Fig. 15-10 is used on metric drawings that con-form to the SI system and use what is called *third-angle projection*. This projection system, which places the top view of a multiview drawing above the front view, is the normal way of showing views in the United States. Draw the symbol, including the outlines of the letters S and I; then use the **SOLID** command to fill in the letters.

Fig. 15-10

Courtesy of Gary J. Hordemann, Gonzaga University

5. Create the approximate shape of the racetrack shown in Fig. 15-11 using **PLINE**. Specify .5 unit for both the starting and ending widths. Select the **Arc** option to draw the figure.
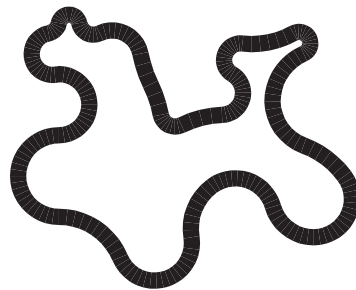
Fig. 15-11

6. Draw the symbols in Fig. 15-12 using the **PLINE** and **PEDIT** commands.

Fig. 15-12

Continued

Chapter 15 **Review & Activities**

7. Draw the car in Fig. 15-13 using the **PLINE** and **PEDIT** commands.
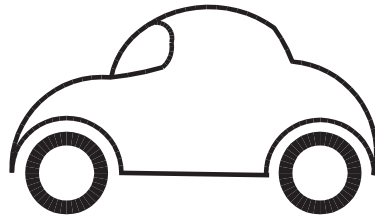
Fig. 15-13

8. The artwork for one side of a small printed circuit board is shown in Fig. 15-14. Reproduce the drawing using donuts and wide polylines. Use the grid to estimate the widths of the polylines and the sizes of the donuts.

GRID SPACING: .2"
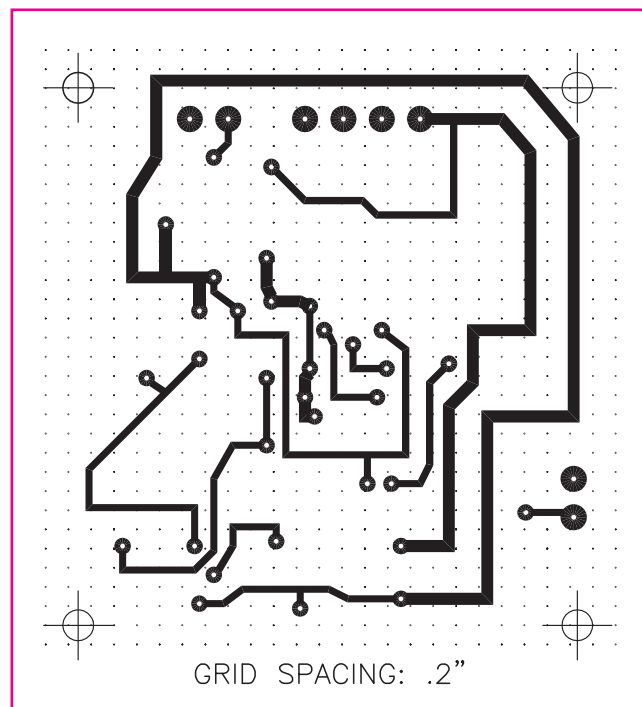
Fig. 15-14

Courtesy of Gary J. Hordemann, Gonzaga University

Review & Activities Chapter 15

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Use the **SPLINE** command to approximate the route of Interstate 89 as it passes through Chittenden County, Vermont, as shown in Fig. 15-15. Include the Lake Champlain shoreline, but do not include the text. Save the drawing as **89.dwg**.

To Montreal

Interstate 89 as it passes through Chittenden County, Vermont

Lake Champlain

Interstate 89

To Montpelier
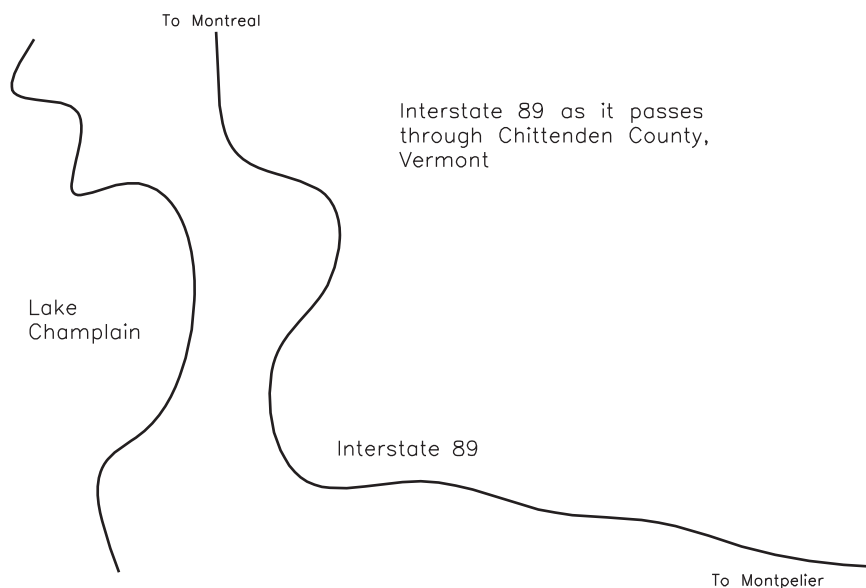
Fig. 15-15

2. Model railroads can be built to many different scales. One of the more popular scales is known as HO, in which $1/8'' = 1'$. With the **PLINE** command, design an HO scale railroad track layout in a figure-eight pattern. Make the center intersection at 90 degrees. At a convenient point in your layout, add a spur to terminate in a railroad yard with three parallel tracks for storing the rolling stock. Refer to books in your local library or search the Internet for any additional information you may need to complete this problem.

3. Construct the drawing shown in Fig. 15-1 on page 185.

Chapter 16

Adding and Altering Objects

Objectives

- Create chamfered corners
- Break pieces out of lines, circles, and arcs
- Produce fillets and rounds
- Offset lines and circles
- Create and edit multilines

Vocabulary

chamfers
fillets
image tiles
mline
multiline
rounds

Most consumer products contain rounded or beveled edges for improved aesthetics and functionality. AutoCAD offers quick ways to create rounded inside corners, called **fillets**, and outside corners, called **rounds**, as well as beveled edges, called **chamfers**. Other useful tools include breaking lines and offsetting them to create parallel lines. AutoCAD also gives you the flexibility to produce multiple lines at one time. This is especially useful for architectural drawing, as are the other capabilities mentioned here.

Creating Chamfers

The **CHAMFER** command enables you to place a chamfer at the corner formed by two lines.

1. Start AutoCAD and open the drawing named **gasket.dwg**. (This is a file you created in Chapter 7.)
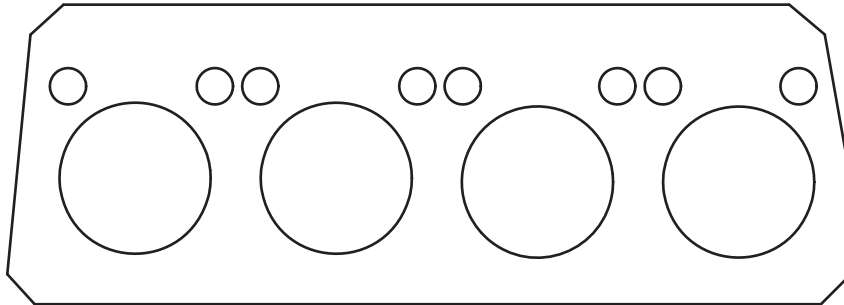
Fig. 16-1

2. Select the **2D Drafting & Annotation** workspace.

3. Pick the **Chamfer** button on the **Modify** panel of the **Ribbon.** (You may need to pick the down arrow next to the **Fillet** button and select the **Chamfer** button from the flyout.)

4. Enter **D** for **Distance**.
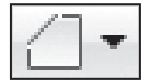
The distance you specify is the distance from the intersection of the two lines (the corner) to the start of the bevel, or chamfer. You can set the chamfer distance for the two lines independently.

5. Specify a chamfer distance of **.25** unit for both the first and second distances.

6. Place a chamfer at each of the corners of the gasket by picking the two lines that make up each corner.

When you are finished, the drawing should look similar to the one in Fig. 16-1, with the possible exception of the holes.

7. Obtain AutoCAD online help to learn about the other options offered by the **CHAMFER** command.

HINT Enter the **CHAMFER** command and press **F1**.

Breaking Objects

Let's remove (break out) sections of the gasket so that it looks like the drawing in Fig. 16-2 on page 200. As you know, the bottom edge of the gasket was drawn as a single, continuous line. Therefore, if you were to use the **ERASE** command, it would erase the entire line. The **BREAK** command, however, allows you to "break" certain objects such as lines, arcs, and circles.
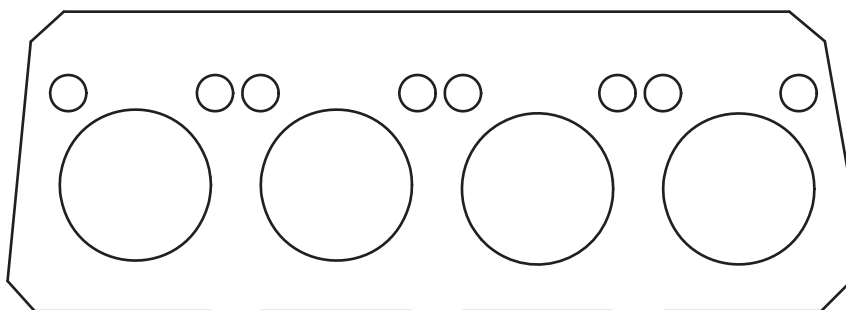
Fig. 16-2

MODIFY

1. From the expanded **Modify** panel on the **Ribbon,** pick the **Break** button.

This enters the **BREAK** command.

2. Turn off object snap if it is on.

3. Pick a point on the line where you would like the break to begin. Since the locations of the breaks in Fig. 16-2 are not dimensioned, you may approximate the location of each start point.

4. Pick the point where you would like the break to end.

The piece of the line between the first and second points disappears. Let's break out two more sections of approximately equal size, as shown in Fig. 16-2.

5. Repeat Steps 1 through 4 to create each break.

6. Experiment with the other **BREAK** options on your own. Refer to AutoCAD's online help for more information about these options.

7. Insert arcs along the broken edge of the gasket, as shown in Fig. 16-3.

8. Save your work.

Let's break out a section of one of the holes in the gasket, as shown in Fig. 16-4.

MODIFY

9. Pick the **Break** button or enter **BREAK** at the keyboard.

10. Pick a point anywhere on the circle. (You may need to turn off object snap.)

11. Instead of picking the second point, enter **F** for **first point**.

12. Pick the first (lowest) point on the circle.

13. Working counterclockwise, pick the second point.

A piece of the circle disappears.

14. Pick the **Undo** button on the **Quick Access** toolbar or enter **U**.

QUICK ACCESS
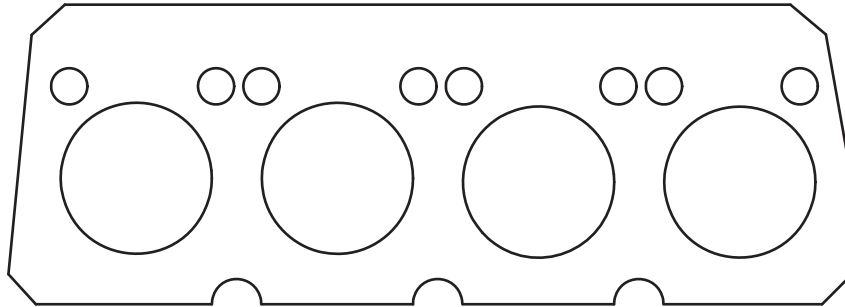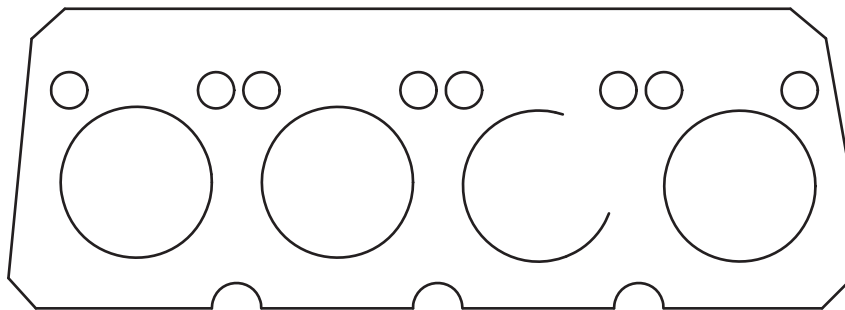
Fig. 16-3

Fig. 16-4

Creating Fillets and Rounds

In AutoCAD, the **FILLET** command creates both fillets and rounds on any combination of two lines, arcs, or circles. Let's change the chamfered corners on the gasket to rounded corners.

1. Erase each of the four chamfered corners.
2. Pick the arrow next to the **Chamfer** button on the **Modify** panel, pick the **Fillet** button, enter **R** for **Radius**, and enter a new radius of **.3**.
3. Produce fillets at each of the four corners of the gasket by picking each pair of lines.
4. Save your work.

MODIFY

The gasket drawing should now look similar to the one in Fig. 16-5 on page 202.
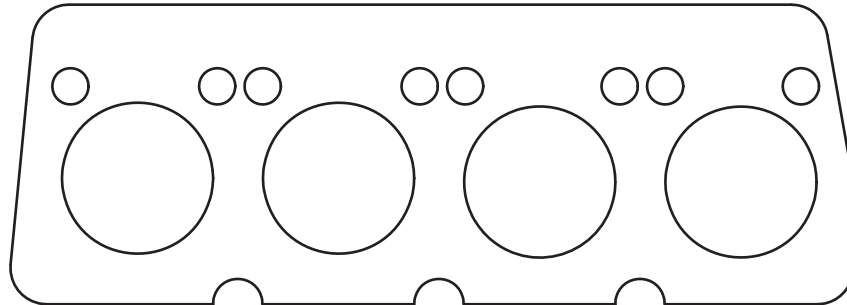
Fig. 16-5

Extending to Form a Corner

Let's move away from the gasket and try something new.

1. Above the gasket, draw lines similar to the ones shown in Fig. 16-6. Omit the numbers.

2. Enter the **FILLET** command and set the fillet radius at **0**.

MODIFY

3. Select lines 1 and 2.
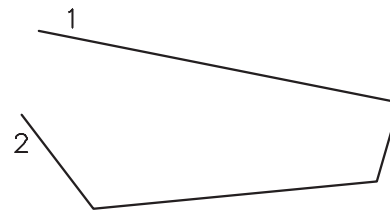
Fig. 16-6

The two lines extend to meet at a corner with a 0 radius, closing the figure. This technique works with the **CHAMFER** command, too.

Trim Option

The **Trim** option is useful for making slotted holes.

1. Above the gasket drawing, draw two relatively short lines that are parallel.

2. Enter the **FILLET** command, enter **T** for **Trim**, and press **ENTER** to accept the **Trim** default value.

MODIFY

3. In response to **Select first object**, pick the end of one of the two lines.

4. In response to **Select second object**, pick the same end of the other line.

AutoCAD fits an arc to the two lines.

5. Repeat Steps 2 through 4 for the other ends of the two lines.

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III. Drawing and Editing

16. Adding and Altering
Objects

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209

Offsetting Objects

Offsetting a line results in another line at a specific distance, or offset, from the original line. Offsetting a circle or arc results in another circle or arc that is concentric (shares the same center point) with the original one.

Offsetting Circles

1. From the **Modify** panel on the **Ribbon**, pick the **Offset** button.

This enters the **OFFSET** command.

MODIFY

2. For the offset distance, enter **.2**.

3. Select one of the large holes in the gasket drawing.

4. Pick a point inside the hole in reply to **Specify point on side to offset**.

5. Select another hole and pick a side to offset.

6. Press **ENTER** to terminate the command.

7. Delete the two offset circles.

Offsetting Through a Specified Point

1. Using the **LINE** command, draw a triangle of any size and shape.

2. Pick the **Offset** button and enter **T** (for **Through**).

MODIFY

3. Pick any one of the three lines that make up the triangle. (You may need to turn off object snap.)

4. Pick a point a short distance from the line and outside the triangle.

The offset line appears. Notice that the line runs through the point you picked.

5. Do the same with the remaining two lines in the triangle so that the object looks similar to the one in Fig. 16-7.

6. Enter **CHAMFER** and set the first and second chamfer distances to **0**.

7. Pick two of the new offset lines.

8. Do this again at the remaining two corners to complete the second triangle.

Fig. 16-7

9. Clean up the drawing so that only the gasket remains.

10. Save your work and close the drawing file.

210

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AutoCAD® 2010

III. Drawing and Editing

16. Adding and Altering
Objects

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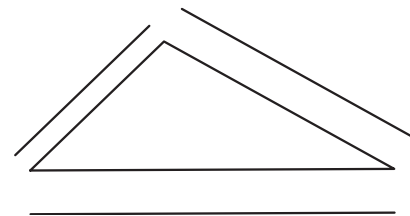
Drawing Multilines

The **OFFSET** command is useful for adding offset lines to existing lines, arcs, and circles. If you want to produce up to 16 parallel lines, you can do so easily—and simultaneously—with the **MLINE** command. This command produces parallel lines that exist as a single object in AutoCAD. The object is called a **multiline** or **mline** object.

QUICK ACCESS

1. Pick the down-facing arrow on the **Quick Access** toolbar and select **Show Menu Bar** from the pull-down menu.

2. On the **Quick Access** toolbar, pick the **QNew** button.

3. Select the **Start from Scratch** button, select **Imperial** units, and pick **OK**.

4. From the **Draw** menu on the **Menu Bar**, pick **Multiline**.

This enters the **MLINE** command.

5. Create a large triangle and enter **C** to close it.

Notice that the corners of the triangle meet perfectly. The triangle is a single object—an mline object.

6. Erase the triangle.

Creating Multiline Styles

The **Multiline Style** dialog box permits you to change the appearance of multilines and save custom multiline styles. Let's use this dialog box to produce the drawing of a room, as shown in Fig. 16-8.

1. Set snap at **.25** units and turn on either polar tracking or ortho.

2. From the **Format** menu on the **Menu Bar**, pick the **Multiline Style…** item.
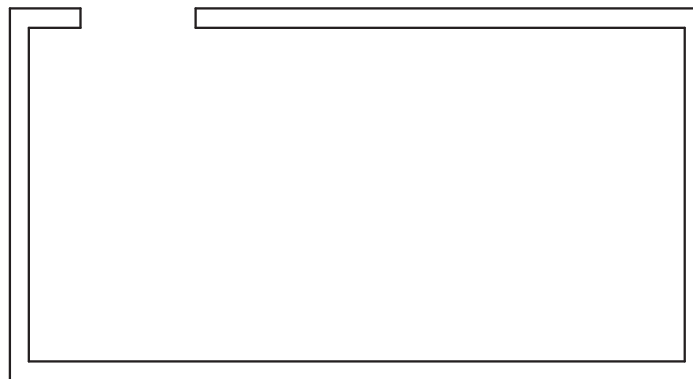
Fig. 16-8
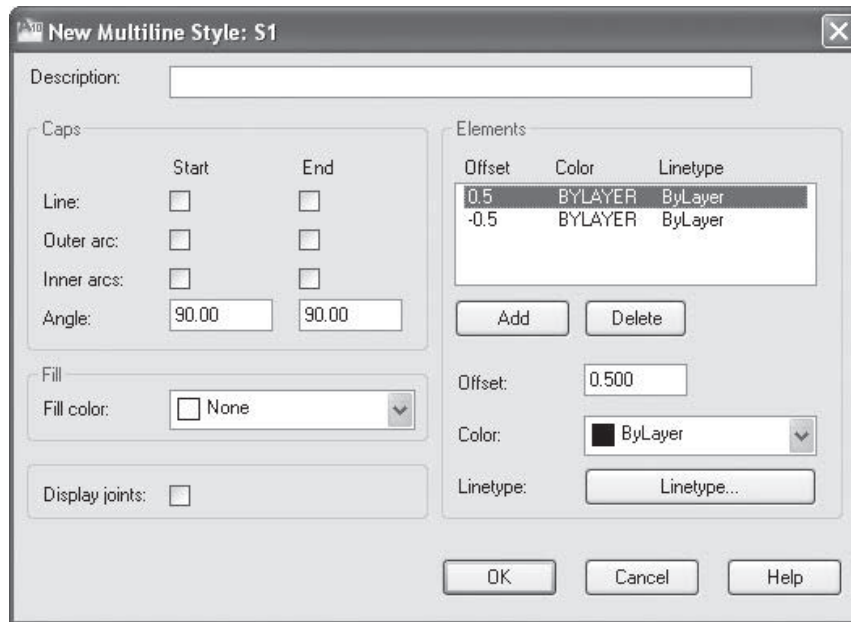
The **Multiline Style** dialog box appears.

3. Pick the **New...** button.

4. Type **S1** and pick the **Continue** button.

The **New Multiline Style** dialog box appears, as shown in Fig. 16-9.

5. In the box located to the right of **Description**, type **This style has end caps**.

> NOTE
>
> AutoCAD does not require you to enter a description for multiline styles you create. However, it is good practice to document your styles so that you can see at a glance which style is appropriate if you need to change or update the drawing later.

6. On the left side under **Caps**, check the **Line Start** and **Line End** boxes.

> NOTE
>
> Many other options are available in the **New Multiline Style** dialog box. You can control the number of line elements and the color, linetype, and lineweight of each element. You can also set the distance between lines by changing the **Offset** values.

7. Pick the **OK** button.

8. Pick the **Save...** button.

9. In the **Save Multiline Style** dialog box, pick the **Save** button to store the **S1** style in the file named **acad.min**.

Notice that the current style is still **STANDARD**. Before you can use a new style, you must make it the current style.

10. Select **S1** and pick the **Set Current** button.

Look in the **Preview** area in the **Multiline Style** dialog box. Notice that both ends of the **S1** multiline are "capped" with short connecting lines.

11. Pick the **OK** button.

When you create new multilines, the **S1** style will apply.

Scaling Multilines

You can set the distance between lines in a multiline using the **Scale** option of the **MLINE** command.

1. Enter **MLINE** at the keyboard.

2. Enter **S** for **Scale** and enter **.25**.

3. With **MLINE** entered, create the drawing shown in Fig. 16-8 on page 204, approximating its size.

> **HINT**
> Use the **Undo** button if you need to back up one line segment.

4. Press **ENTER** to terminate **MLINE**.

5. Save your work in a file named **multi.dwg**.

Let's add the interior walls shown in Fig. 16-10.

6. Display the **Multiline Style** dialog box.

7. Create a new multiline style named **S2** with the following description: **This style has a start end cap.**

8. Uncheck the **Line End** check box located under **Caps** and pick **OK**.

9. Save **S2** and make it the current style.

10. Pick **OK** to exit the **Multiline Style** dialog box.
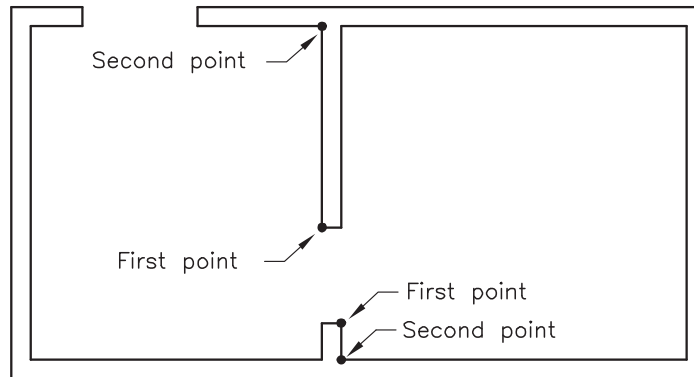
Fig. 16-10

11. Using **MLINE**, draw the interior walls to match those shown in Fig. 16-10. Press **ENTER** to complete each wall. (You may need to turn object snap off before performing this step.)

Editing Intersections

Using the **Multiline Edit Tools** dialog box, it is possible to edit the intersection of multilines.

1. Pick **Object** and then **Multiline…** from the **Modify** menu on the **Menu Bar**.

This enters the **MLEDIT** command and displays the **Multilines Edit Tools** dialog box, as shown in Fig. 16-11. The twelve items in the dialog box are called **image tiles**.

2. Pick the **Open Tee** image tile.

3. In response to **Select first mline**, pick one of the two interior walls.

4. In response to **Select second mline**, pick the adjacent exterior wall.

AutoCAD breaks the exterior wall.

5. Edit the second interior wall by repeating Steps 3 and 4.

6. Press **ENTER** to terminate the **MLEDIT** command.

7. On your own, explore the options in the **Multiline Style** and **Multilines Edit Tools** dialog boxes.

8. Save your work and exit AutoCAD.
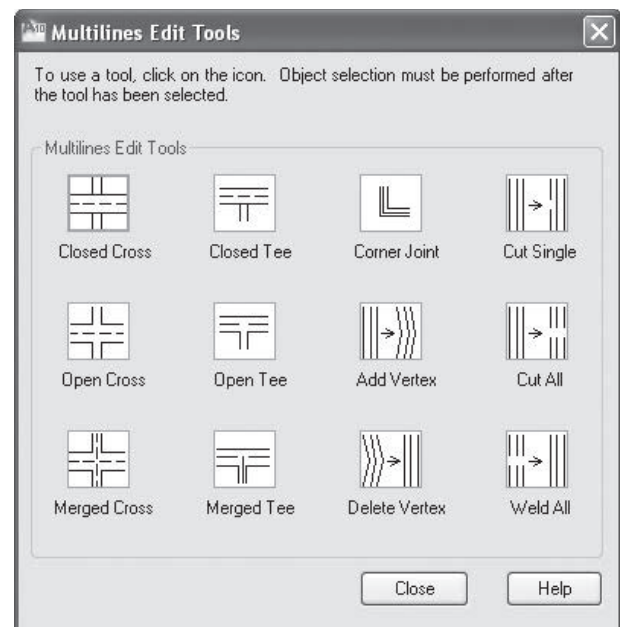
Fig. 16-11

Chapter 16 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is the function of the **CHAMFER** command?
2. How is using the **BREAK** command different from using the **ERASE** command?
3. If you want to break a circle or arc, in which direction do you move when specifying points: clockwise or counterclockwise?
4. On what panel on the **Ribbon** is the **Fillet** button found?
5. How do you set the fillet radius?
6. What can you accomplish by setting either **FILLET** or **CHAMFER** to 0?
7. Explain the purpose of the **OFFSET** command.
8. How might multilines be useful? Give at least one example.
9. Explain the purpose of the **Multiline Style** and **Multilines Edit Tools** dialog boxes.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Explore the **Angle** option of the **CHAMFER** command. How is it different from the **Distance** option? Discuss situations in which each option (**Angle** and **Distance**) may be useful.
2. To help potential clients understand floor plans, the architectural firm for which you work draws its floor plans showing the walls as solid gray lines the thickness of the wall. Exterior walls are 6″ thick, and interior walls are 5″ thick. Explain how you could achieve these walls using **MLINE** and the **Multiline Style** dialog box.
3. AutoCAD allows you to save multiline styles and then reload them later. Explore this feature and write a short paragraph explaining how to save and restore multilines.

Review & Activities Chapter 16

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create the first drawing on the left in Fig. 16-12. Don't worry about exact sizes and locations, but do use snap and ortho. Then use **FILLET** to change it to the second drawing. Set the fillet radius at **.2** units.
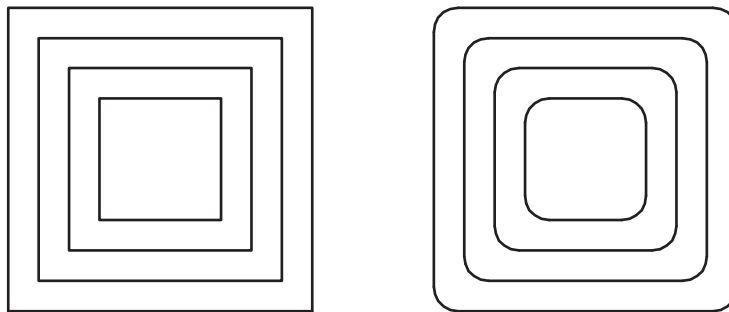
Fig. 16-12

2. Create the triangle shown in Fig. 16-13. Then use **CHAMFER** to change it into a hexagon. Set the chamfer distance at **.66** units.
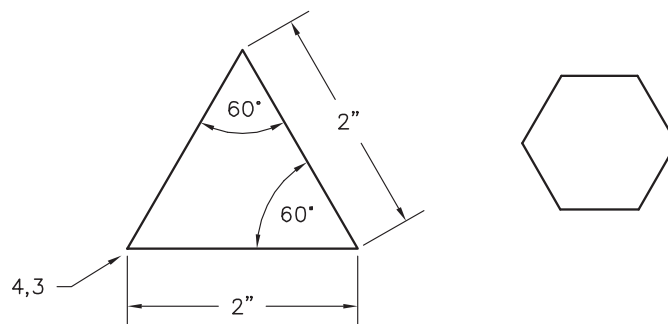
Fig. 16-13

Continued

Chapter 16 **Review & Activities**

3. Create the roller and cradle for a conveyor. Begin by drawing a circle and three lines, as shown in Fig. 16-14. Specify a fillet radius of **.25** and create the fillets on the left side, as shown in Fig. 16-15. Then fillet the right side. The finished roller and cradle should look like the one in Fig. 16-16.
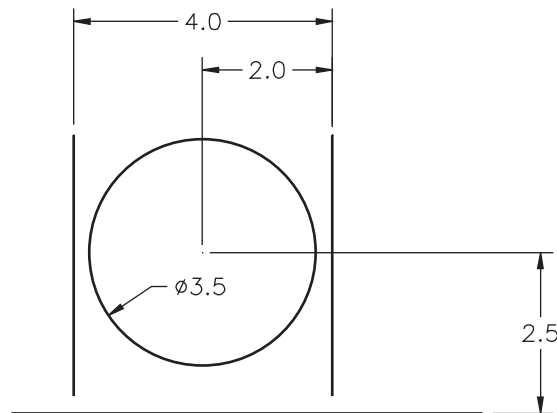
Fig. 16-14
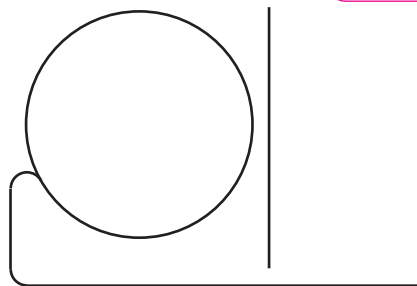
Fig. 16-15
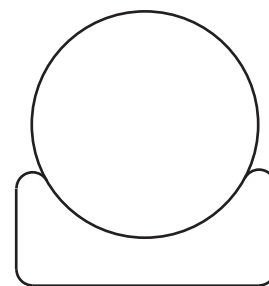
Fig. 16-16

4. Draw the steel base shown in Fig. 16-17. Use the **FILLET** command to place the fillets and rounds in the corners as indicated.

Fig. 16-17

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III. Drawing and Editing

16. Adding and Altering
Objects

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217
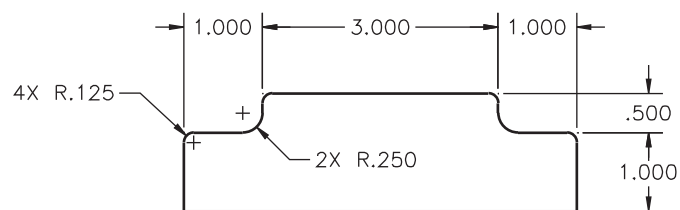
Review & Activities Chapter 16

5. Draw the shaft shown in Fig. 16-18. Make it **2** units long by **1** unit in diameter. Use the **CHAMFER** command to place a chamfer at each corner. Specify **.125** for both the first and second chamfer distances.

.125 X 45° BOTH ENDS
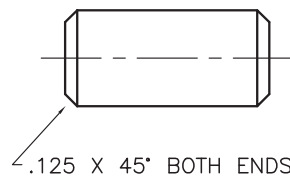
(**Fig. 16-18**)

6. Using the **ARC** and **CHAMFER** commands, construct the detail of the flat-blade screwdriver tip shown in Fig. 16-19.

.250

.500
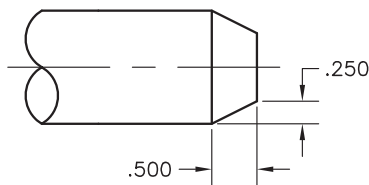
(**Fig. 16-19**)

7. Create the picture frame shown in Fig. 16-20. First create the drawing on the left using the **LINE** and **OFFSET** commands. Then modify it to look like the drawing on the right using the **CHAMFER** command.
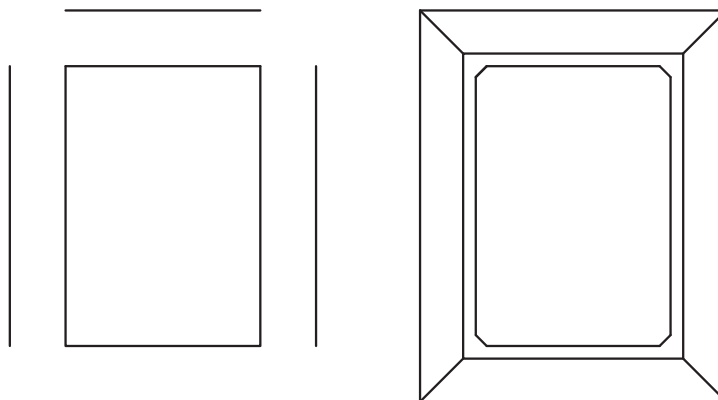
(**Fig. 16-20**)

Courtesy of Joseph K. Yabu, Ph.D., San Jose State University

Continued

Chapter 16 **Review & Activities**

8. Create the A.C. plug shown in Fig. 16-21 using the **MLINE**, **MLSTYLE**, **CHAMFER**, and **FILLET** commands.
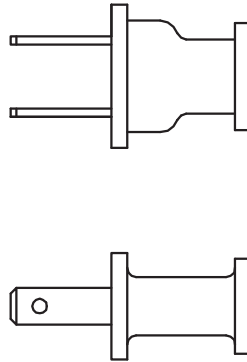
Fig. 16-21

Courtesy of Joseph K. Yabu, Ph.D., San Jose State University

9. Draw the front view of the rod guide end cap shown in Fig. 16-22. Use the **OFFSET** command to create the outer circular shapes and the inner horizontal lines. Use the **BREAK** and **FILLET** commands to create the inner shape with four fillets. Do not dimension the drawing.
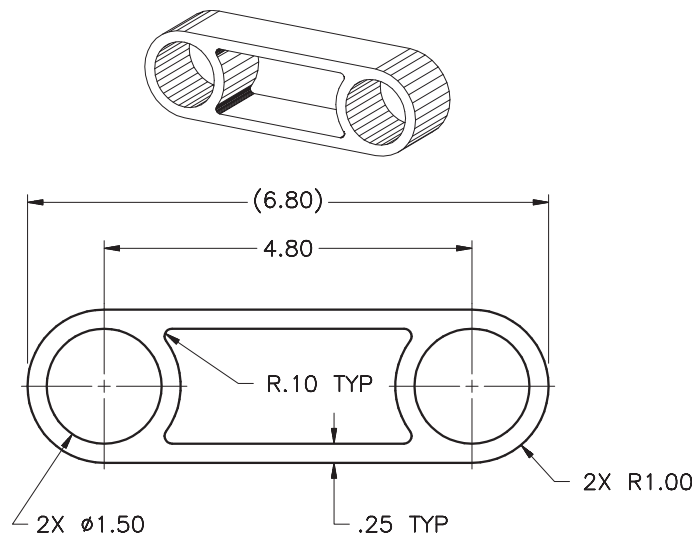
Fig. 16-22

Courtesy of Gary J. Hordemann, Gonzaga University

Review & Activities Chapter 16

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Draw the front view of the ring separator shown in Fig. 16-23. Start by using the **CIRCLE** and **BREAK** commands to create a pair of arcs. Then use **OFFSET** to create the rest of the arcs. Do not dimension the drawing.

2. Kitchens are arranged around their three most important elements: the sink, the range, and the refrigerator. The traffic pattern is arranged in a triangle, with a vertex at each of the three items. As a result, kitchens are usually arranged in one of three patterns: U-shaped, L-shaped, or parallel. Draw an example of each type of kitchen using the **MLINE** command with capped ends. Do not label the appliances, but draw the appliances as shown in Fig. 16-24. Show the triangular traffic pattern for each kitchen you design. Which seems most convenient to you? Why?
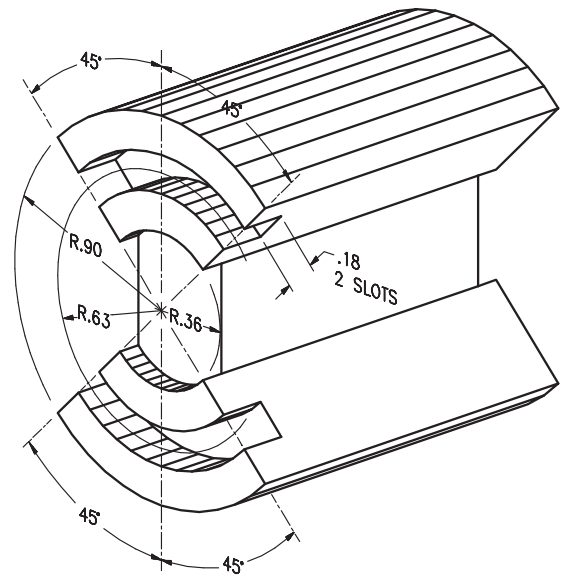
Fig. 16-23

Courtesy of Gary J. Hordemann, Gonzaga University
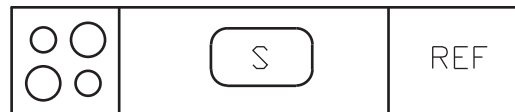
Symbols for Kitchen

Fig. 16-24

3. Select one of the kitchen arrangements you created in problem 2 and design a kitchen around it. Include the walls, door and window placements, and any other appliances or furniture that you think a kitchen should include.

AutoCAD at Work

Entrepreneur

One of the exciting things about our country is that we have the freedom to choose our own career paths. If you are interested in health care, you might decide to pursue a career in medicine or nutrition. If you wish to provide a specific type of product or service that few other companies offer, you may consider starting your own business.

Entrepreneurs—people who create and run their own businesses—are found in nearly every field. In the year 2000, about 10,000 of the noted 213,000 drafting jobs were held by self-employed individuals.

© Brand X Pictures/PunchStock

Working on Your Own

Experienced, independent individuals work in many areas of drafting, but few have entered the profession on their own without first obtaining specialized training. Most have worked initially for a drafting company to learn about drafting techniques. After accumulating years of experience, these individuals decided to obtain their own clients and work for themselves. Some drafting entrepreneurs use their business as their primary means of income, while others meet with clients in their spare time, in addition to working a full-time job.

Wanted: Motivated, Reliable Professionals

A successful entrepreneur must be self-motivated, patient, and persistent. Building a new business takes time, and when you're the boss, your company's fate is entirely in your hands. Good time-management and decision-making skills are helpful, too. Entrepreneurs must also be able to communicate with others; good relationships with both clients and employees are necessary in order to obtain business and to keep it. One of the most important characteristics, however, is reliability. Clients and employees must know that they can take you at your word.

▶ Career Activities

1. Have you ever tried to run your own business? What skills did you use?

2. Interview an entrepreneur from your area. What was the most difficult problem the person faced? How was the problem solved?

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III. Drawing and Editing

17. Moving and duplicating
Objects

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221

Chapter 17

Moving and Duplicating Objects

Objectives

- Change an object's properties
- Move and copy objects
- Mirror objects and parts around an axis
- Produce rectangular and polar arrays

Vocabulary

array
circular array
mirror
mirror line
polar array
rectangular array

Earlier in this book, you learned how to move and copy objects using the grips method. Now you will learn how to perform these and other operations using commands. Why would you want to? The commands give you additional options. Also, if you choose to write scripts or programs that automate certain AutoCAD operations, you could embed these commands into them, so knowing how they work is important.

You will also learn to create arrays of objects, both in rectangular and circular fashion. The arraying features are useful for many purposes, including architectural drafting, facilities planning, auditorium and stadium design, and countless machine design applications. The problems at the end of this chapter provide examples.

Changing Object Properties

The **Quick Properties** panel provides a method for changing several of an object's properties.

1. Start AutoCAD and open the drawing named **gasket.dwg**.
2. Select the **AutoCAD Classic** workspace and close all floating toolbars and palettes.
3. Select one of the large holes in the gasket.

The **Quick Properties** panel appears. You can turn **Quick Properties** on and off with the **Quick Properties** button on the status bar.
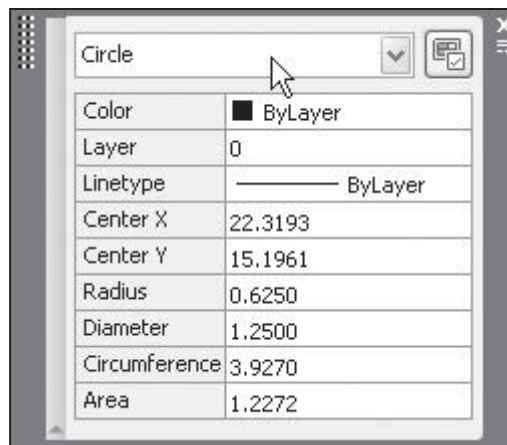
4. Rest the pointing device over one of the vertical bars on the left and right sides of the **Quick Properties** panel.

This displays the expanded **Quick Properties** panel**,** as shown in Fig. 17-1. Review the list of object properties. Note that the properties listed in the palette depend on the object selected. In this case, the options are specific to the circle you selected in Step 3.

5. Enter a new number in the **Diameter** box and press **ENTER**.

The diameter of the circle changes. You will use the **Quick Properties** panel and the larger, more thorough **Properties** palette many times throughout this book.

6. Change the diameter back to its original value and press **ENTER.**
7. Press the **ESC** key to remove the object selection and close the **Quick Properties panel.**

Moving Objects

Let's move the gasket to the top of the screen using the **MOVE** command.

MODIFY

1. From the **Modify** toolbar, pick the **Move** button, or enter **MOVE** at the keyboard.

AutoCAD displays the **Select objects** prompt.

2. Place a window around the entire gasket drawing and press **ENTER**.

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III. Drawing and Editing

17. Moving and duplicating
Objects

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223

> **HINT** **M** is the command alias for the **MOVE** command.

3. In reply to **Specify base point or displacement**, place a base point somewhere on or near the gasket drawing, as shown in Fig. 17-2.

4. Move the pointing device in the direction of the second point (destination).

5. Pick the second point.

The gasket should move accordingly.

6. Practice using the **MOVE** command by moving the drawing to the bottom of the screen.

7. Reenter the **MOVE** command.

8. In response to the **Select objects** prompt, pick two of the four large holes and press **ENTER** (or right-click).

9. Specify the first point (base point). Place the point anywhere on or near either of the two holes.

10. In reply to **Specify second point or displacement**, move the pointing device upward. (Polar tracking should be on.)

11. Enter **3** to move the holes vertically 3 units.

12. Undo the move.

MODIFY

Second Point (Destination)

First (Base) Point
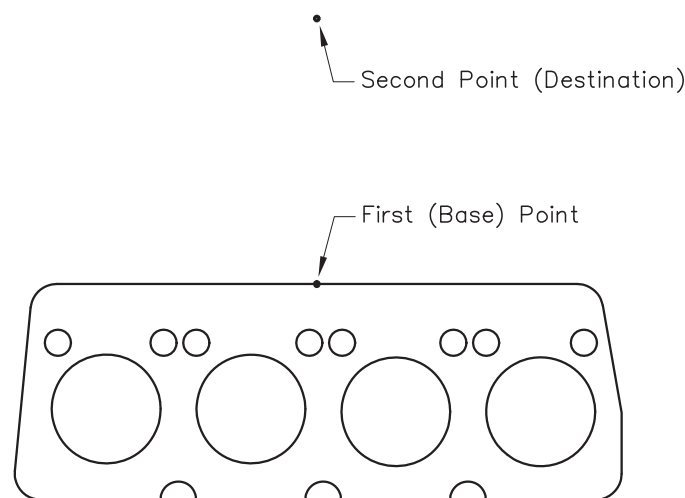
Fig. 17-2

Copying Objects

MODIFY

The **COPY** command is similar to the **MOVE** command. The only difference is that the **COPY** command keeps the selection in its original location and "moves" a copy of it to the location you specify.

1. Erase all of the large holes in the gasket except for one.
2. From the **Modify** toolbar, pick the **Copy** button.

This enters the **COPY** command.

3. Select the remaining large hole and press **ENTER** or right-click.
4. Select the center of the hole for the base point.

> **HINT** Use the **Center** object snap. Also, turn on ortho or polar tracking before completing the following step.

5. Move the crosshairs and place the hole in the proper location.
6. Repeat Step 5 until all four large holes are in place; then press **ENTER**.
7. Practice using the **COPY** command by erasing and copying the small holes.
8. Save your work.

Mirroring Objects

There are times when it is necessary to **mirror** (produce a mirror image of) a drawing, detail, or part. A simple copy of the object is not adequate because the object being copied must be reversed, as was done with the butterfly in Fig. 17-3. One side of the butterfly was drawn and then mirrored to produce the other side. The mirroring feature can save a large amount of time when you are drawing a symmetrical object or part.

Fig. 17-3

Place mirror line here
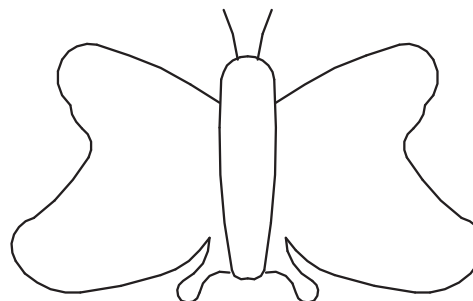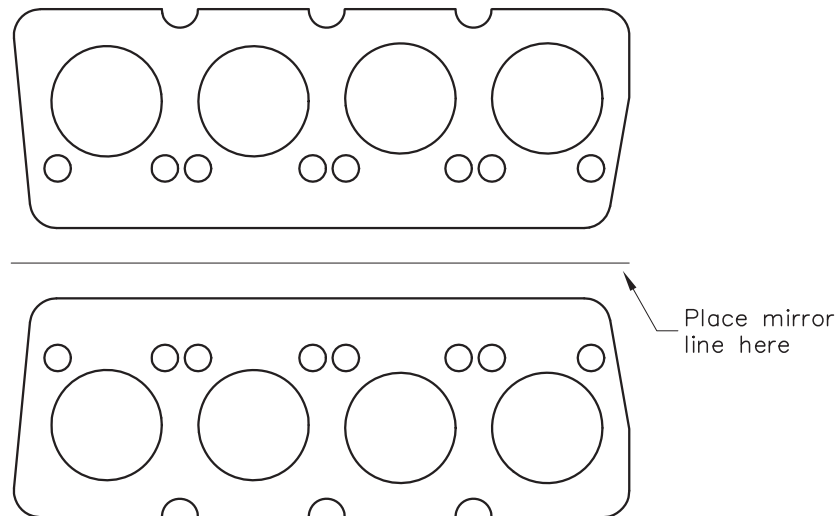
Fig. 17-4

Mirroring Vertically

The same is true if the engine head gasket we developed is to be reproduced to represent the opposite side of an eight-cylinder engine. In this case, we will mirror the gasket vertically (around a horizontal plane).

1. If necessary, move the gasket to the bottom of the screen to allow space for another gasket of the same size.

2. From the **Modify** toolbar, pick the **Mirror** button.

This enters the **MIRROR** command.

MODIFY

3. Select the gasket by placing a window around it and press **ENTER**.

4. Create a horizontal **mirror line** (axis) near the gasket by selecting two points on a horizontal plane, as shown in Fig. 17-4.

AutoCAD asks if you want to delete the source objects.

5. Enter **N** for **No**, or press **ENTER** since the default is **No**.

AutoCAD mirrors the gasket as shown in Fig. 17-4.

Mirroring at Other Angles

You can also create mirror images around a mirror line at angles other than horizontal.

1. Draw a small triangle and mirror it with an angular (*e.g.*, 45°) mirror line.

2. Erase the triangles.

3. Save your work and close the drawing.

Producing Arrays

An **array** is an orderly grouping or arrangement of objects. AutoCAD's arraying feature can save a large amount of time when you create a drawing that includes many copies of an object arranged in a rectangular or circular pattern.

QUICK ACCESS

1. Begin a new drawing by selecting the **QNew** button from the **Quick Access** toolbar.

2. Select **Use a Wizard** and **Quick Setup** and pick **OK**.

3. Pick **Architectural** for the units and pick the **Next** button.

4. Enter a width of **24′** and a length of **18′** and pick the **Finish** button.

5. Set the grid at **1′** and the snap at **1″**; **ZOOM All**.

6. Zoom in on the lower left quarter of the drawing area and create the chair shown in Fig. 17-5. Use the following information.

MODIFY

- Create the **1′**–**3″** radius arc first. Use the snap grid to snap to the start and end points of the arc.

- Produce the **9″** arc by offsetting by **6″** from the **1′**–**3″** arc.

- Create the first **3″** arc using the end-points of the larger arcs; then create the second **3″** arc using the **MIRROR** command.

- The **10″** arc passes through the centers of the **3″** arcs.
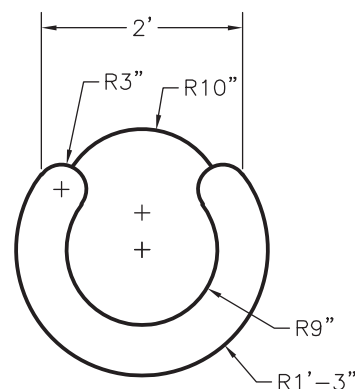
7. ZOOM All and save your work in a file named **chair.dwg**.

Fig. 17-5

Creating Rectangular Arrays

An array in which the objects are arranged in rows and columns is known as a **rectangular array**. An example is shown in Fig. 17-6.

MODIFY

1. Pick the **Array** button from the docked **Modify** toolbar.

This displays the **Array** dialog box. Notice that **Rectangular Array** is selected.

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III. Drawing and Editing

17. Moving and duplicating
Objects

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227

2. Enter **4** for **Rows** and **5** for **Columns**.

3. Enter **3′** for **Row offset** and **34″** for **Column offset**.

These offsets define the distance between the rows and columns. The distance is measured from the center of one chair to the center of the next.

4. Pick the **Select objects** button located in the upper right area of the dialog box.

5. Select the chair, press **ENTER**, and pick the **Preview** button.

AutoCAD displays an array of 20 chairs, as shown in Fig. 17-6, and a prompt to return to the **Array** dialog box or accept the array.

6. Right-click to accept the array.

7. Save your work and close the drawing.

NOTE
You can produce rectangular arrays at any angle by changing **Angle of array** in the **Array** dialog box.
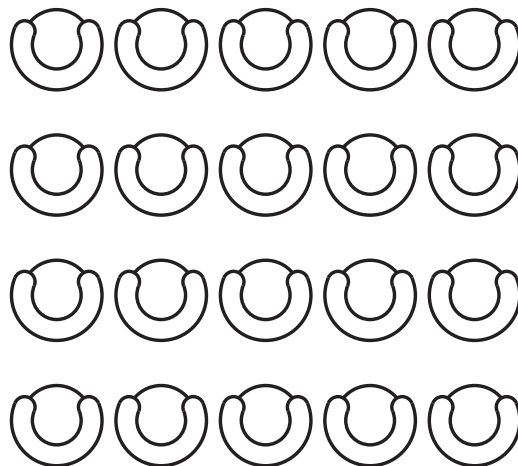
Fig. 17-6

Creating Circular Arrays

Next, we are going to produce the cycle wheel shown in Fig. 17-7. Notice the arrangement of the spokes. We can create them quickly by drawing only two lines and then using a circular array to create the remaining lines. A **circular array** (also called a **polar array**) is one in which the objects are arranged radially around a center point.

1. Begin a new drawing from scratch.

2. Draw a wheel similar to the one shown in Fig. 17-8. Make the wheel large enough to fill most of the screen. Use the **Center** object snap to make the circles concentric.

3. Draw two crossing lines similar to the ones in Fig. 17-9. Use the **Nearest** object snap to begin and end the lines precisely on the appropriate circles.

4. Save your work in a file named **wheel.dwg**.

5. Pick the **Array** button or enter **AR**, the command alias for **ARRAY**.

MODIFY

6. Use the following information to complete the options in the **Array** dialog box.

   • Select **Polar Array**.

   • Select the button to the right of **Center point** and pick the wheel's center.

   • Enter **18** for **Total number of items**.

   • Do not change **Angle to fill** because you want to array the two spokes 360 degrees.

   • Pick the **Select objects** button, select the two spokes, press **ENTER**, and pick **Preview**.
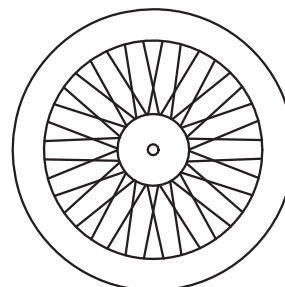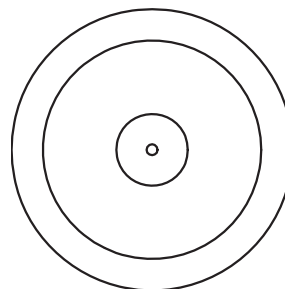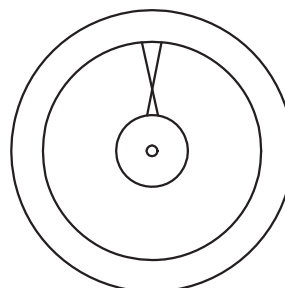
The wheel should look similar to the one in Fig. 17-7.

7. Right-click to accept the array and end the command.

8. Practice creating polar arrays using other objects. At least once, specify less than 360 degrees for **Angle to fill**.

9. Save your work and exit AutoCAD.

Fig. 17-7

Fig. 17-8

Fig. 17-9

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III. Drawing and Editing

17. Moving and duplicating
Objects

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229

Chapter 17 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is the purpose of the **Properties** palette?
2. In what toolbar is the **Move** button located?
3. Explain how the **MOVE** command is different from the **COPY** command.
4. Describe a situation in which the **MIRROR** command would be useful.
5. During a **MIRROR** operation, can the mirror line be specified at any angle? Explain.
6. Name two types of arrays.
7. State one practical application for each type of array.
8. What is the effect on a polar array if you specify an angle to fill of less than 360 degrees? Explain.
9. Explain how a rectangular array can be created at any angle.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Experienced AutoCAD users take the time necessary to analyze the object to be drawn before beginning the drawing. Discuss the advantages of doing this. Keep in mind what you know about the AutoCAD commands presented in this chapter.

2. Draw the dialpad for a pushbutton telephone. Create one of the pushbuttons and use the **ARRAY** command to create the rest. Each pushbutton is a 13 mm × 8 mm rectangle, and the buttons are arranged as shown in Fig. 17-10. What should you enter for the space between rows and spaces between columns?
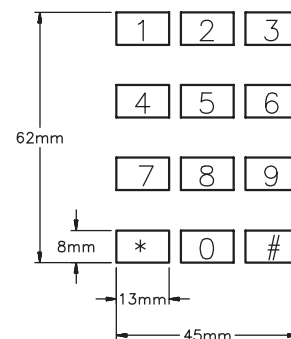
Fig. 17-10

Continued

Chapter 17 **Review & Activities**

• APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create the shim shown in Fig. 17-11. Begin by drawing the left half of the shim according to the dimensions shown, using construction lines for center lines. Then mirror the objects to create the right half of the shim. Erase any unneeded lines. Save the drawing as **shim.dwg**.
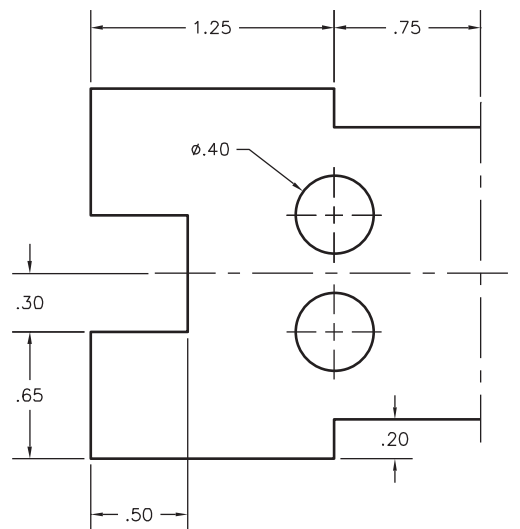
Fig. 17-11

Adapted from a drawing by Bob Weiland

2. Draw the chisel shown in Fig. 17-12 using the **ARC**, **FILLET**, **LINE**, **COPY**, and **MIRROR** commands.
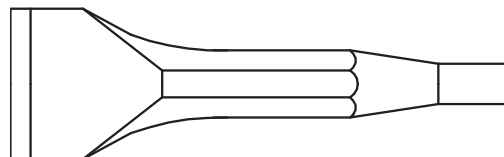
Fig. 17-12

Courtesy of Joseph K. Yabu, Ph.D., San Jose State University

Review & Activities Chapter 17

3. Draw the key shown in Fig. 17-13 using the **ARC**, **FILLET**, **LINE** or **PLINE**, **COPY**, and **MIRROR** commands.
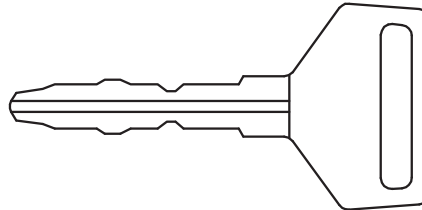
Fig. 17-13

Courtesy of Joseph K. Yabu, Ph.D., San Jose State University

4. Apply the commands introduced in this chapter to create the piping drawing shown in Fig. 17-14.
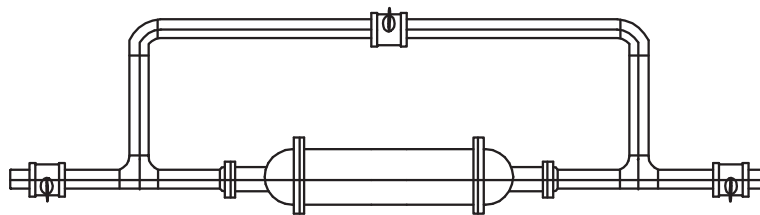
Fig. 17-14

Courtesy of Dr. Kathleen P. King, Fordham University, Lincoln Center

5. Develop a plan for an auditorium with rows and columns of seats. Design the room any way you'd like. Save your work as **auditorium.dwg**.

6. Create the cooling fan shown in Fig. 17-15.
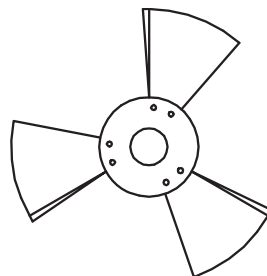
Fig. 17-15

Courtesy of Joseph K. Yabu, Ph.D., San Jose State University

Continued

Chapter 17 **Review & Activities**

7. Draw the front view of the bar separator shown in Fig. 17-16. Draw one quarter of the bar separator using the **LINE**, **ARC**, and **FILLET** commands with a grid setting of **.1** and a snap of **.05**. Use the **MIRROR** command to make half of the object; then use it again to create the other half.
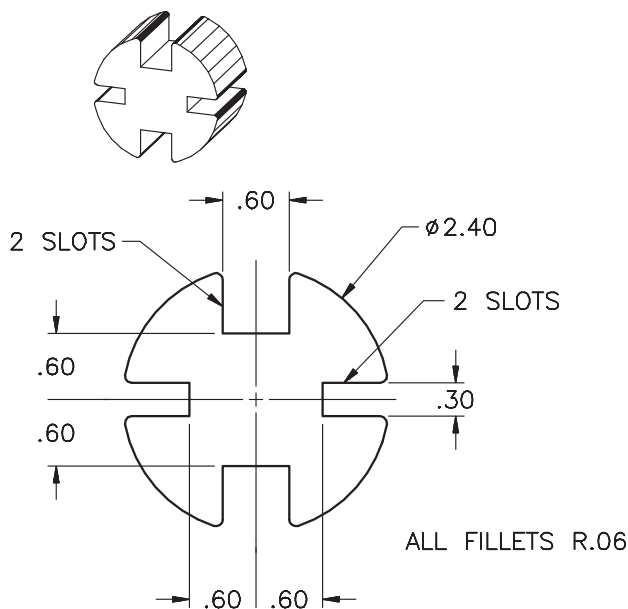
Fig. 17-16

Courtesy of Gary J. Hordemann, Gonzaga University

8. Draw the air vent shown in Fig. 17-17 using the **ARRAY** command. Other commands to consider using are **POLYGON** and **COPY**.
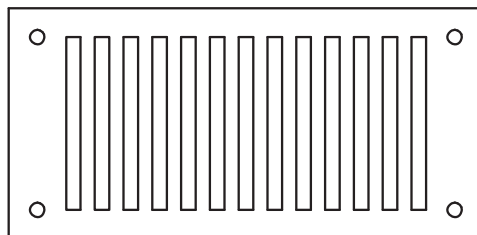
Fig. 17-17

Courtesy of Joseph K. Yabu, Ph.D., San Jose State University

9. Draw the front view of the wheel shown in Fig. 17-18. Begin by drawing one quarter of the object. Draw the three arcs and four lines, as shown in Fig. 17-19. Center the arcs on a known point. Next, use the **FILLET** command to create the cavity shown on the right. Erase the outside lines and the outside arc. Then use the **MIRROR** command to make three more copies of the cavity. Complete the drawing by adding arcs and circles. Try constructing the keyway by first drawing half of the shape, then using **MIRROR** to generate the other half. For more practice, try drawing one-eighth of a wheel.

.251
⌀8.00
⌀2.50
⌀1.50
.80 TYP
1.631
R1.50 TYP
⌀7.20
ALL FILLETS R.60
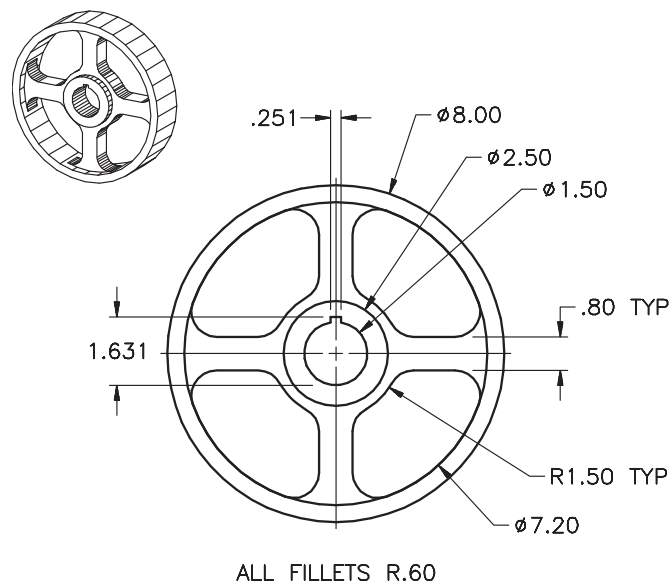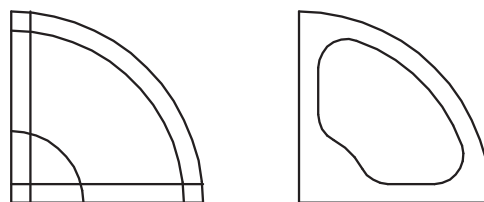
Fig. 17-18

Fig. 17-19

Courtesy of Gary J. Hordemann, Gonzaga University

Continued

Chapter 17 **Review & Activities**

10. Load the drawing named **gasket.dwg**. Erase each of the holes. Replace the large ones according to the locations shown in Fig. 17-20, this time using the **ARRAY** command. Reproduce the small holes using the **ARRAY** command, but don't worry about their exact locations. The diameter of the large holes is **1.25**; the small holes have a diameter of **.30**.
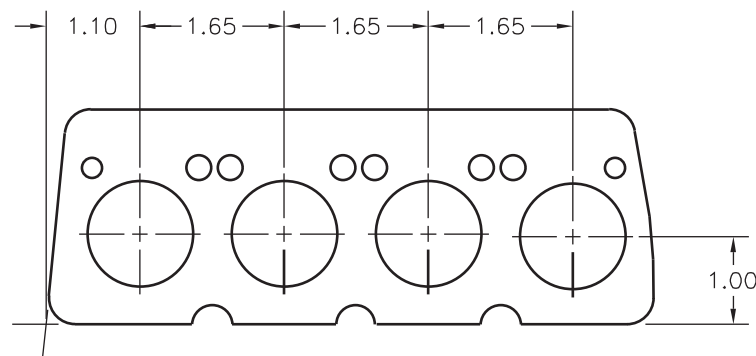
Fig. 17-20

11. Draw the snowflakes in Fig. 17-21 using the **ARRAY** command. Since snowflakes are six-sided, you may want to begin by drawing a set of concentric circles. Then use the **ARRAY** command to insert 6, 12, or more construction lines. The example in Fig. 17-22 shows the beginning of one of the snowflakes. Note that you only need to array three lines at 15° intervals for a total of 45° (two lines if you use the **MIRROR** command).
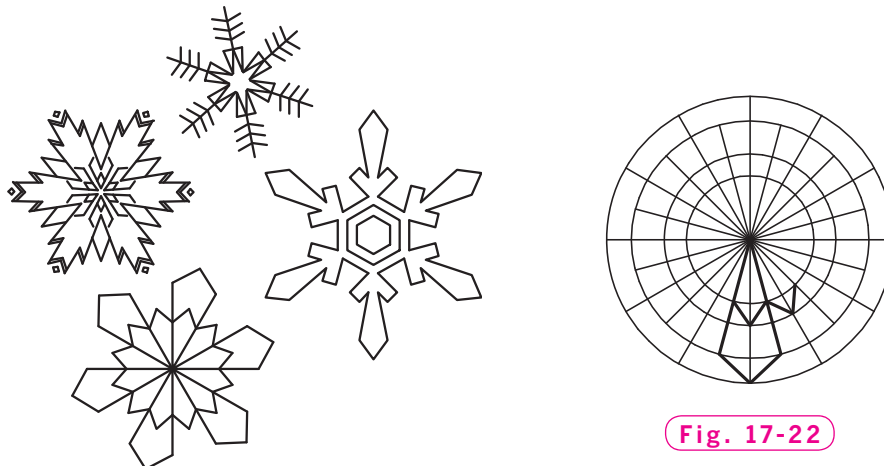
Fig. 17-22

Fig. 17-21

Courtesy of Gary J. Hordemann, Gonzaga University

12. Draw the top view of the power saw motor flywheel shown in Fig. 17-23. Use the **ARRAY** command to insert the 24 fins and arcs. You may find the **OFFSET** command useful in creating the first fin.
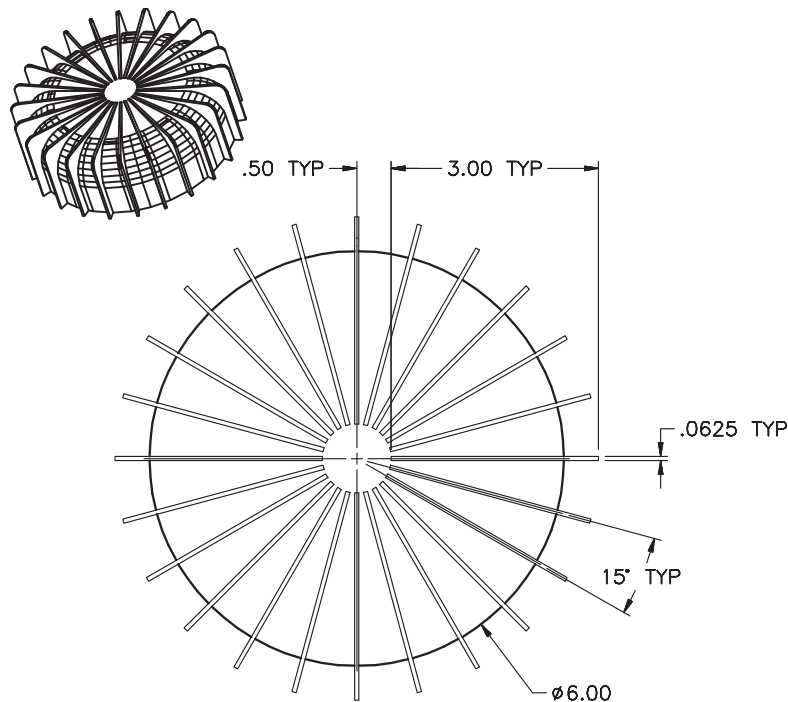
Fig. 17-23

Courtesy of Gary J. Hordemann, Gonzaga University

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. You are creating problems for an AutoCAD textbook and need to simulate the grid displayed with a **.5** spacing. With the **POINT** command (on the **Draw** toolbar), create a rectangular array within a drawing space of **12,9** so that the grid completely fills the space. If you find the point is too small to show on the screen, obtain help to find out how to make it larger. When you have completed the grid, turn AutoCAD's grid on and off to see how your grid corresponds to the real grid. Save the drawing as **grid.dwg**.

Continued

Chapter 17 **Review & Activities**

2. You have received a design change for the shim you created in problem 1 of this chapter's Applying AutoCAD Skills section. Open **shim.dwg** and use the **MOVE** and **COPY** commands to arrange the holes in a pattern approximately as shown in Fig. 17-24. Save the drawing as **shimrev.dwg**.
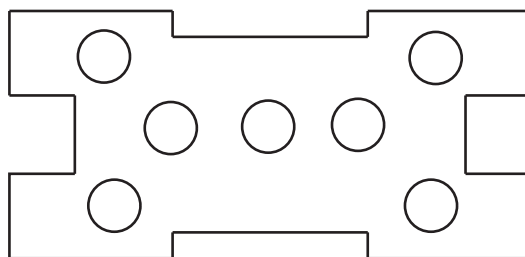
(**Fig. 17-24**)

3. Draw the wheel shown in Fig. 17-25 (you decide how).
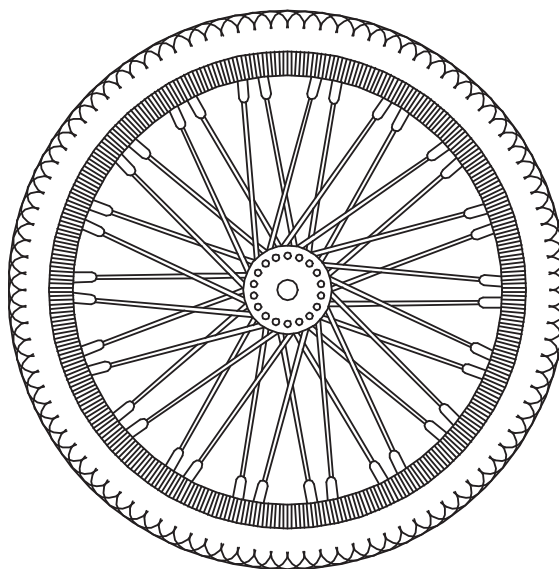
(**Fig. 17-25**)

Adapted from a drawing by Bob Weiland

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III. Drawing and Editing

18. Modifying and
Maneuvering

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237

Chapter 18

Modifying and Maneuvering

Objectives

- Stretch objects to change their overall shape
- Scale objects using a scale factor or reference length
- Rotate objects to exact angles
- Trim and extend lines to specific boundaries
- Trim and extend multilines
- Join individual lines to specific boundaries

Vocabulary

colinear
rotating
scale

The more you create and edit drawings with AutoCAD, the more you will discover a need to edit objects in many ways. You will find several uses for trimming, extending, joining, and lengthening lines as you experiment with new ideas. One of the benefits of using AutoCAD is that you are not penalized for drawing something at the wrong size or in the wrong place. You can easily correct it using one of the commands covered in this textbook.

Stretching Objects

Let's start by stretching objects using the **STRETCH** command.

1. Start AutoCAD and choose the **Quick Setup** wizard.
2. For the units, use **Architectural**, and for the area, enter **120′** and **90′**.

Be sure to include the foot symbols (′) so that AutoCAD knows that these are feet, not inches.
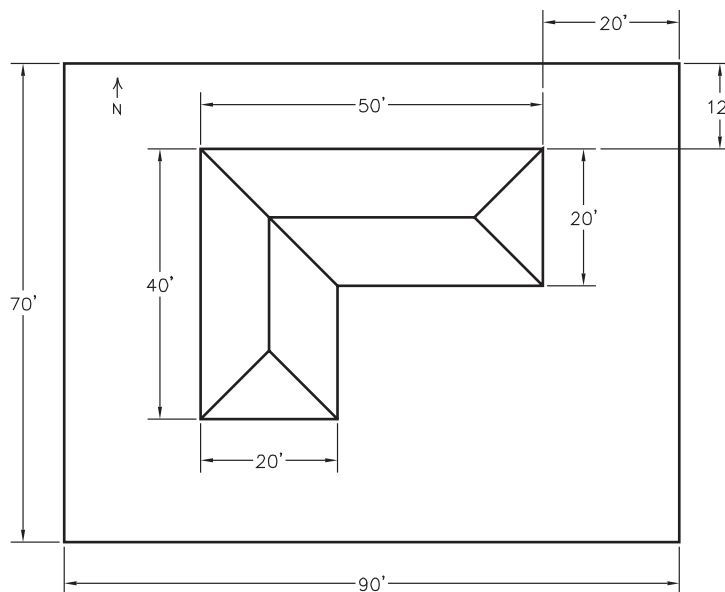
Fig. 18-1

3. Select the **2D Drafting & Annotation** workspace.

4. Enter **ZOOM All**.

5. Draw the site plan in Fig. 18-1 according to the dimensions shown. With snap set at **2′**, place the lower left corner of the property line at absolute point **10,10**. Omit dimensions. All points in the drawing should fall on the snap grid.

6. Save your work in a file named **site.dwg**.

MODIFY

7. Pick the **Stretch** button from the **Modify** panel on the **Ribbon** or enter **S**, the command alias for **STRETCH**.

This enters the **STRETCH** command.

8. Select the east end of the house, as shown in Fig. 18-2, and press **ENTER**. Use the crossing object selection procedure.

HINT

In reply to **Select objects**, pick a point to the right of the house and move the cursor to the left to create a green crossing window (Fig. 18-2).

9. Pick the lower right corner of the house for the base point, as shown in Fig. 18-2.

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AutoCAD® 2010

III. Drawing and Editing

18. Modifying and
Maneuvering

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239

10. In reply to **Second point of displacement**, stretch the house **10′** to the right and pick a point. (Snap should be on.) The house stretches dynamically.

The house should now be longer.

11. Stretch the south portion of the house **10′**.

12. Save your work.
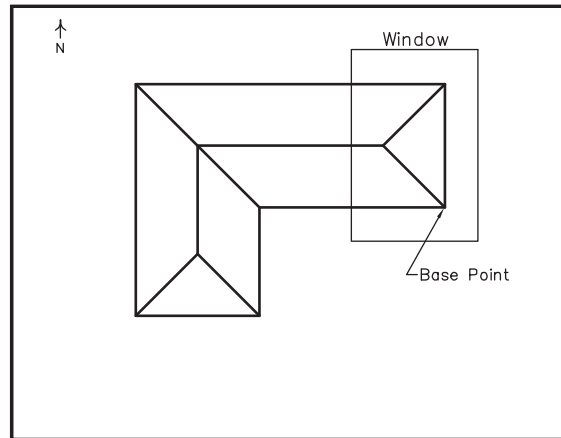
Fig. 18-2

Scaling Objects

Since the house is now slightly too large, let's scale it down to fit the lot. When you **scale** an object, you increase or decrease its overall size without changing the proportions of the parts to each other. You can use the **SCALE** command to scale by a specific scale factor, or you can use the existing size of the object as a reference length for the new size.

Using a Scale Factor

1. Pick the **Scale** button from the **Modify** panel on the **Ribbon** or enter **SC**, the command alias for **SCALE**.

2. Select the house by placing a window around it and press **ENTER**.

3. Pick the lower left corner of the house as the base point.

At this point, you can dynamically drag the house into place or enter a scale factor.

4. Enter **.5** in response to **Specify scale factor**.

Entering **.5** reduces the house to half (.5) of its previous size.

Scaling by Reference

Suppose the house is now too small. Let's scale it up using the **SCALE Reference** option.

1. Enter **SCALE**, select the house as before, and press **ENTER**.

2. Pick the lower left corner of the house again for the base point.

MODIFY

MODIFY

3. Enter **R** for **Reference**.

4. For the reference length, pick the lower left corner of the house, and then pick the corner just to the right of it as the second point.

This establishes the length of the south wall (10′) as the reference length.

5. In response to **Specify new length**, move the crosshairs **4** feet to the right of the second point (snap should be on) and pick a point.

The south wall becomes 14′ long, and the rest of the house is scaled by reference to match.

Rotating Objects

Let's rotate the house on the site. Unlike scaling, **rotating** doesn't change the actual size of the house. It merely places the house at a different angle. Using the **ROTATE** command, you can drag objects to a new angle, or you can rotate them by a specific number of degrees if the precise angle is important.

Rotating by Dragging

MODIFY

Let's rotate the house by dragging it into place.

1. Pick the **Rotate** button from the **Modify** panel on the **Ribbon**, select the entire house, and press **ENTER**.

2. Pick the lower left corner of the house for the base point.

3. Turn off ortho and snap. As you move the crosshairs, notice that the object rotates dynamically. Drag the house in a clockwise direction a few degrees and pick a point.

4. Pick the **Undo** button to undo the rotation.

Specifying an Angle of Rotation

AutoCAD also allows you to specify the angle of rotation. In practice, this is a more useful option than dragging because you can control the placement of objects more precisely.

MODIFY

1. Enter **ROTATE**, select the house, and press **ENTER** after you have made the selection.

2. Pick the lower left corner of the house for the base point.

3. For the rotation angle, enter **25** (degrees).

The house rotates 25° counterclockwise. Your drawing should now look similar to the one in Fig. 18-3 without the additional lines.
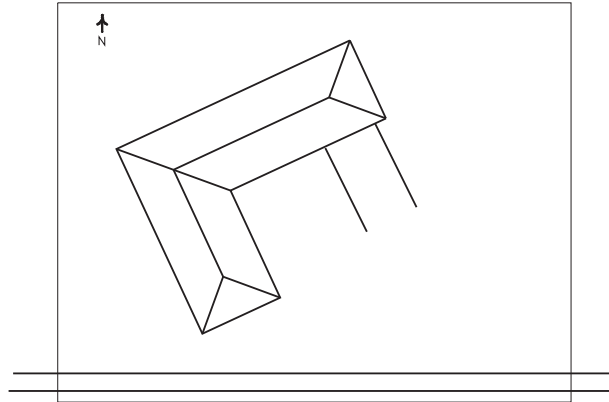
Fig. 18-3

Trimming Lines

The **TRIM** command allows you to trim lines that extend too far. You can use **TRIM** to clean up intersections and to convert construction lines and rays into useful parts of a drawing.

1. Draw two horizontal xlines **3** apart to represent a sidewalk, as shown in Fig. 18-3.
2. Draw a partial driveway, as shown in Fig. 18-3. (Ortho should be off.) Make the driveway **13**′ wide.

DRAW

> **HINT**
> Copy the line that makes up the east edge of the house for the first line of the driveway. Use the **OFFSET** command to create the second line of the driveway exactly **13**′ from the first line.

3. Pick the **Trim** button on the **Modify** panel of the **Ribbon** or enter **TR**, the command alias for the **TRIM** command.
4. Select the east property line as the cutting edge and press **ENTER**.
5. Select the sidewalk lines on the right side of the property line.

MODIFY

The sidewalk should now end at the property line.

6. Press **ENTER** to end the command.
7. Trim the west end of the sidewalk to meet the west property line.
8. Obtain online help to learn about the **Project** and **Edge** options.

Extending Lines

The most accurate way to extend the driveway lines so that they meet the south property line is to use the **EXTEND** command.

MODIFY

1. Pick the **Extend** button on the **Modify** panel of the **Ribbon.**
2. Select the south property line as the boundary edge and press **ENTER**.
3. In reply to **Select object to extend**, pick the ends of the two lines that make up the driveway and press **ENTER**.

The driveway extends to the south property line.

MODIFY

4. Using the **TRIM** command, remove the short intersecting lines so that the sidewalk and driveway look like those in Fig. 18-4.

> **HINT** Use a crossing window to select all four objects, press **ENTER**, and then select the parts you want to remove.

5. Save your work.

Fig. 18-4

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III. Drawing and Editing

18. Modifying and
Maneuvering

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243
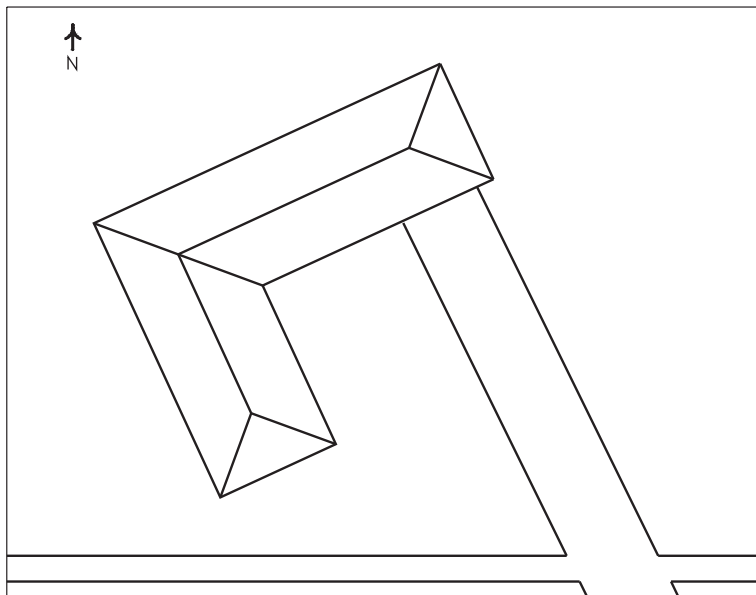
Working with Multilines

The **TRIM** and **EXTEND** commands work with multilines as well.

1. Open the drawing named **multi.dwg**.
2. Open the **Multiline Style** dialog box, set the style **S1** as the current style, and pick **OK**.
3. Create a new multiline, as shown in Fig. 18-5.
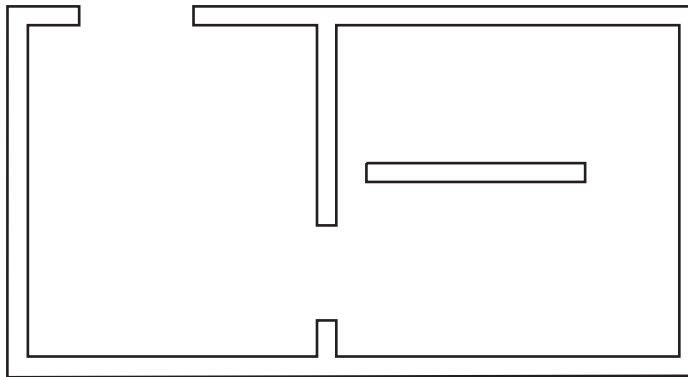
Fig. 18-5

4. Use the **EXTEND** command to extend the new wall to the interior wall. In reply to **Enter mline junction option**, type **O** for **Open**.

MODIFY

Your drawing should look like the one shown in Fig. 18-6.
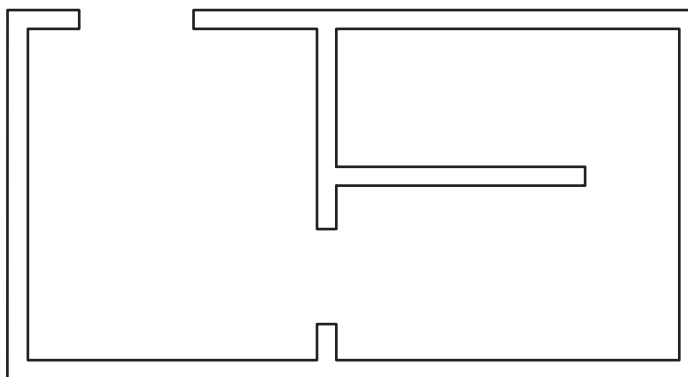
Fig. 18-6

5. Reenter the **MLINE** command and create another wall, as shown in Fig. 18-7.
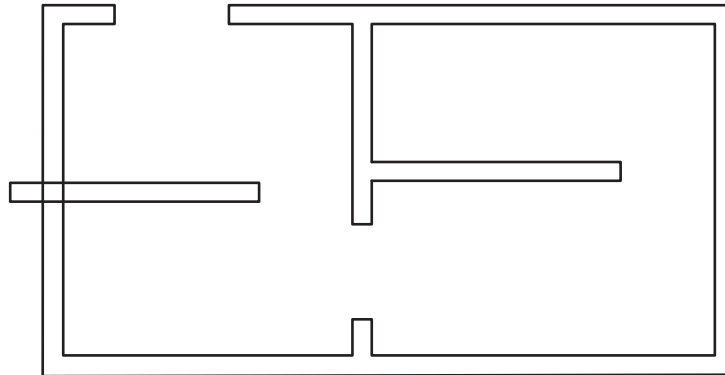
Fig. 18-7

MODIFY

6. Use the **TRIM** command to trim away the part of the wall that extends beyond the exterior wall of the building. This time, select the **Merged** option at the **Enter mline junction option** prompt.

The drawing should now look like the one in Fig. 18-8.
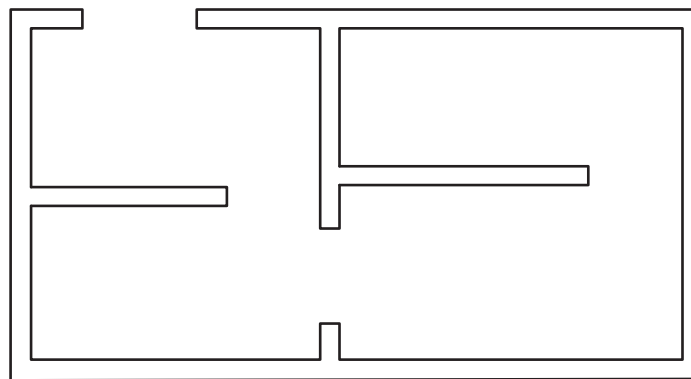
Fig. 18-8

7. Save your work.

8. Close **multi.dwg**.

Joining Lines

The **JOIN** command allows you to join similar individual objects into a single object. Let's use the **JOIN** command to change the roof on the house you created earlier in this chapter.

1. If you closed **site.dwg**, reopen it now.

2. Change the roof line to match the one shown in Fig. 18-9. On both ends of the house, erase the diagonal lines; then draw the short line segments to connect the roof ridge to each end of the house.
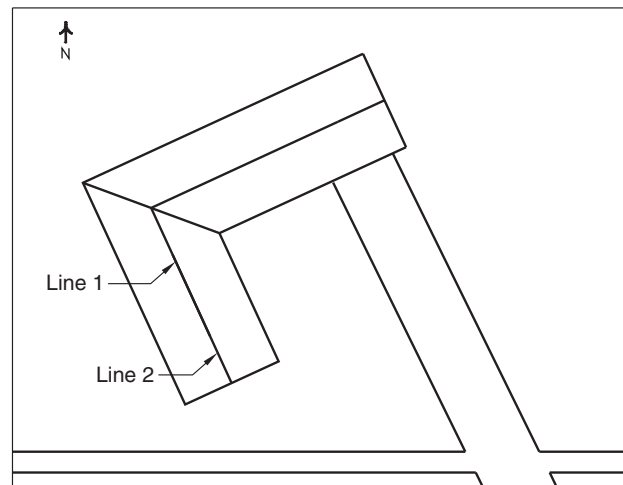
Fig. 18-9

> **HINT** Use the **Midpoint** object snap.

3. Pick the **Join** button on the expanded **Modify** panel of the **Ribbon** or enter **J**, the command alias for **JOIN**.

MODIFY

4. Select the first line segment in reply to **Select source object**, select the second line segment in response to **Select lines to join to source**, and press **ENTER**.

5. Repeat Steps 3 and 4 for the other end of the house.

The roof ridge line on each side of the house is now a single line.

> **NOTE** The **JOIN** command only works for two lines that are **colinear** (lying on the same infinite line).

6. Save your work and exit AutoCAD.

Chapter 18 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Explain the purpose of the **STRETCH** command.
2. Using the **SCALE** command, what number would you enter to enlarge an object by 50%? ...to enlarge it to 3 times its present size? ...to reduce it to half of its present size?
3. Explain how you would dynamically scale an object up or down.
4. Can you rotate an object dynamically? Explain.
5. How would you specify a 90° clockwise rotation of an object accurately?
6. Explain the purpose of the **TRIM** command.
7. Describe a situation in which the **EXTEND** command would be useful.
8. Explain the similarities between the **EXTEND** and **JOIN** commands.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Refer to **site.dwg**, which you completed in this chapter. If you were doing actual architectural work, you would need to place the features on the drawing much more precisely than you did in this drawing. Describe a way to place a **14**′-wide driveway exactly **2** feet from the corner of the house. The driveway should be perpendicular to the house.
2. Both the **ZOOM** command and the **SCALE** command make objects appear larger and smaller on the screen. Write a paragraph explaining the differences between the two commands. Explain the circumstances under which each command should be used.

Review & Activities   Chapter 18

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create a new drawing and draw the kitchen range symbol shown in Fig. 18-10. The square is **5** units on each side, the large circles are **R.8**, and the small circles are **R.6**. Locate the circles approximately as shown: equidistant from the adjacent sides, with the large circles in the upper left and lower right corners.
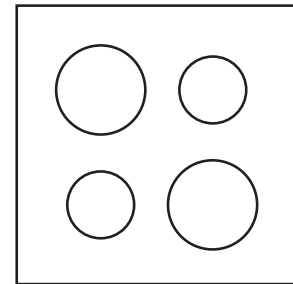
   Notice that this is an incorrect symbol for a kitchen range. The large and small circles should be reversed. You can fix this by rotating the circles 90°. To make sure the circles stay in their same relative locations, rotate them around the center of the square. To do this, first draw two diagonal lines, as shown in Fig. 18-11. Then enter the **ROTATE** command and select the four circles and two diagonal lines. After you have completed the rotation, erase both of the diagonal lines. Then scale the range to 25% of its current size. Save the drawing as **range.dwg**.
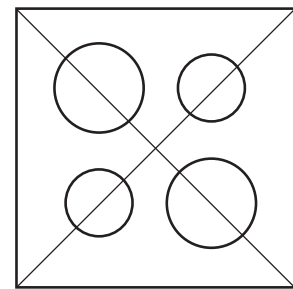
Fig. 18-10

Fig. 18-11

2. Open **site.dwg**, the drawing you created in this chapter. Perform each of the following operations on the drawing.

   - Change the roof line by adding angled (hipped) ridges on both ends of the house.

   - Stretch the driveway by placing a (crossing) window around the house and across the driveway. Stretch the driveway to the north so that the house is sitting farther to the rear of the lot.

   - Add a sidewalk parallel to the east property line. Use the **TRIM** and **BREAK** commands to clean up the sidewalk corner and the north end of the new sidewalk.

   - Reduce the entire site plan by 20% using the **SCALE** command.

   - Stretch the right side of the site plan to the east **10′**.

   - Rotate the entire site plan 10° in a counterclockwise direction.

Continued

Chapter 18 **Review & Activities**

- Place trees and shrubs to complete the site plan drawing. Use the **ARRAY** command to create a tree or shrub, as shown in Fig. 18-12. Duplicate and scale the tree or shrub using the **COPY** and **SCALE** commands.

When you finish, your drawing should look similar to the one in Fig. 18-12.
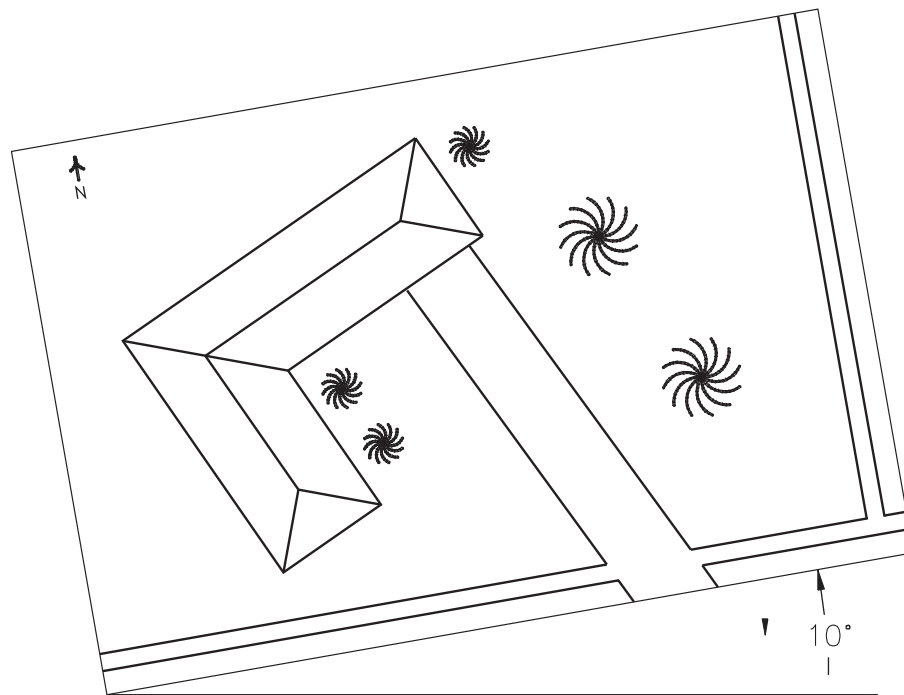
Fig. 18-12

3. Marketing wants to use the original design of your TV remote control device, but you must change it to meet competitive standards. Create the first drawing of the device (shown on the left in Fig. 18-13). Then copy the drawing. On the copy, make the following changes using the commands you learned in this chapter.

- Scale the device to 90% of its original size.
- Change the two buttons at the bottom into one large button.
- Reduce the overall length.

The final device should look similar to the drawing on the right in Fig. 18-13. Save the drawing as **remote.dwg**.

Review & Activities Chapter 18
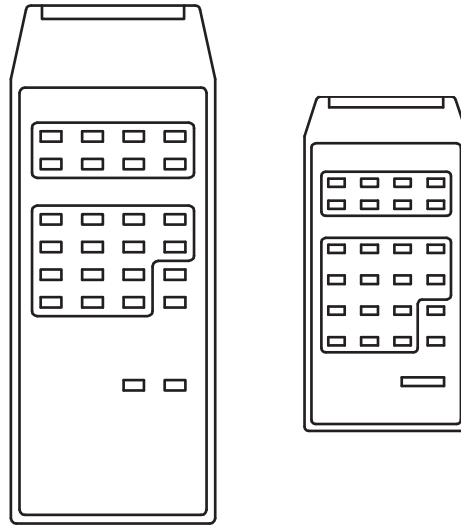
Fig. 18-13

4. Create the two armchairs shown in Fig. 18-14. Approximate their sizes, but align them as shown in the illustration so that the backs of the chairs are colinear. Use the chairs to make a matching sofa. To do this, make a copy of both chairs. Then remove the inside arms on the copies of both chairs and extend the horizontal lines to meet, as shown. Finally, use the **JOIN** command to join the lines that represent the back of the sofa so that the back of the sofa consists of a single line. Join the other horizontal lines as well. Save the file as **furniture.dwg**.
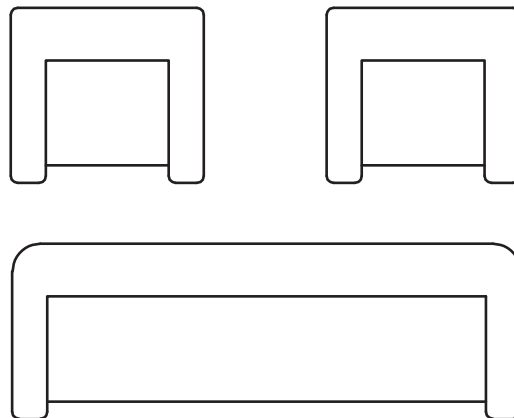
Fig. 18-14

Continued

Chapter 18: Modifying and Maneuvering   243

Chapter 18 **Review & Activities**

5. Draw the front view of the tube bundle support shown in Fig. 18-15. Proceed as follows:

- Draw the outside circle and one of the holes.
- Offset both circles to create the width of the material.
- Draw a line connecting the center of the outside circle and the center of the hole.
- Offset the line to create the internal support structure that connects the hole with the center of the support.
- Draw another line from the center of the large circle at an angle of 30° to the first line.
- Use **TRIM** to create half of one cavity.
- Use **MIRROR** to create the other half of the cavity.
- Fillet the three corners.
- Use **ARRAY** to insert five more copies of the hole and cavity.

When you have completed the drawing, use the **SCALE** command to shrink it by a factor of **.25** using the obvious base point. Save the drawing as **tubebundle.dwg**.
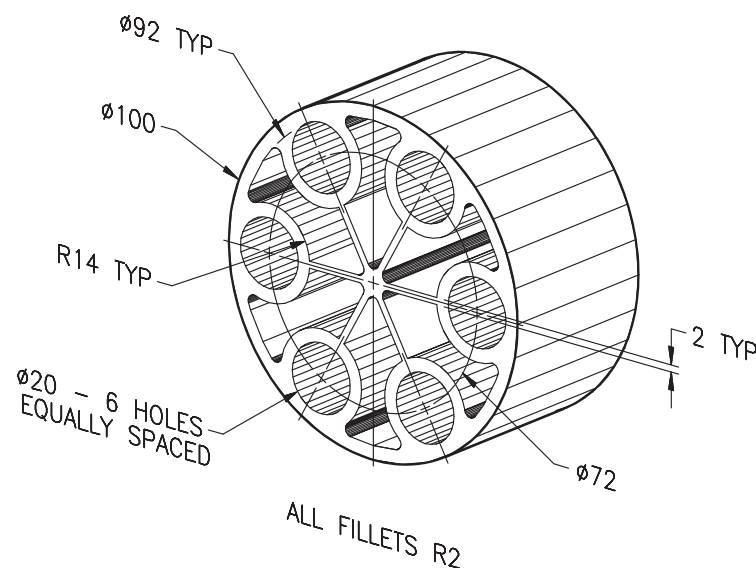
Fig. 18-15

Courtesy of Gary J. Hordemann, Gonzaga University

Review & Activities Chapter 18

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. You have a full single garage on your lot. The garage measures **16′ × 25′** and has a **9′** door opening. Draw a floor plan of the garage with **6″**-thick walls. Show the opening, but do not include the door. Your spouse's car necessitates a larger garage. Use the **EXTEND** command to convert the full single garage to a full double garage measuring **25′ × 25′**. Include two **9′** door openings separated by a **1′** center support. Trim where necessary. Save the drawing as **garage.dwg**.

2. Local zoning laws will not permit you to have a full double garage on your narrow lot. Use the **TRIM** command to convert the proposed full double garage to a small double garage measuring **20′ × 20′** with one large **16′** door opening. Save the drawing as **garage2.dwg**.

3. Create the house shown in Fig. 18-16 on a **100′ × 35′** lot.
   - Make the drawing limits the size of the lot.
   - Create a new multiline style named **FRAME** with two elements a total of **4″** apart to represent the walls.
   - Estimate the size of the house and of each room.
   - Use **MLEDIT**, **TRIM**, and **EXTEND** to create the wall intersections.
   - Save the file as **frame.dwg**.
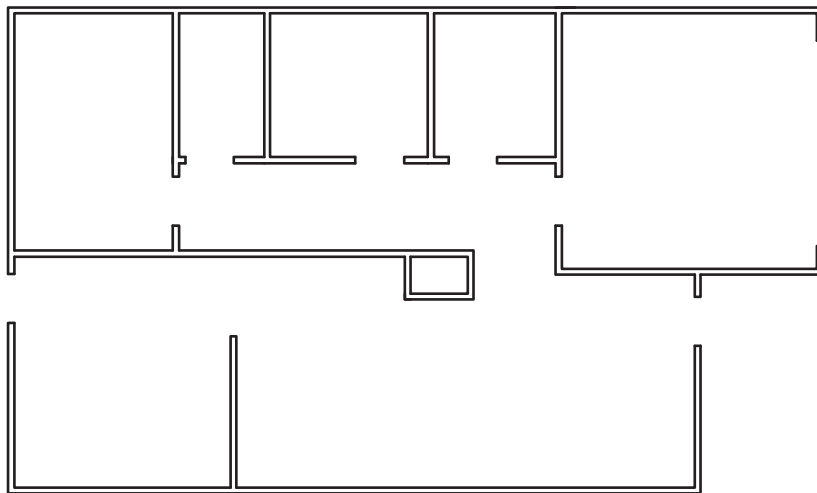
Fig. 18-16

AutoCAD at Work

Interior Designer

A variety of related careers fall under the umbrella of "drafting." While many drafters and designers create plans to manufacture or build products, others work to create attractive surroundings for daily living. Try to think of an old, run-down building in your area that has recently been restored. Although architectural designers certainly played a role in ensuring that the building was structurally secure, other design professionals were responsible for making the inside usable and aesthetically appealing. These individuals, known as *interior designers*, are responsible for designing and supervising interior construction and restoration efforts.

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Details of Design

Interior designers cover a lot of territory when planning a comfortable indoor atmosphere. They create drawings and plans specifying the types and quantities of materials necessary for construction, the types of lighting systems that will best complement a room, and how to furnish an area creatively. Interior designers may be employed by design firms or may work as independent consultants. They become involved with building and restoration projects for homes, offices, and even sets for films and television shows.

Being Versatile

Interior designers wear many different hats; that is, they are responsible for various aspects of the design process. They should be able to estimate the costs involved with construction and restoration, such as specific costs of materials and labor. Leadership skills are also important, as interior designers need to be able to convey their ideas to those involved with creating or restoring an area. Interior designers must also be able to think creatively in order to design an appropriate look for various types of indoor areas.

▶ **Career Activities**

1. Compare the interior designer's job with that of an architectural drafter.

2. Find out about courses of study available for interior designers. What are the prerequisites for these courses?

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III. Drawing and Editing

19. Hatching and Sketching

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253

Chapter 19

Hatching and Sketching

Objectives

- Hatch objects and parts according to industry standards to improve the readability of drawings
- Edit a hatch by changing, adding, and removing boundaries
- Change the characteristics of an existing hatch
- Use the **Tool Palettes** window to apply hatching
- Use sketching options to create objects with irregular lines

Vocabulary

associative hatch
crosshatch
full section
hatch
island
section drawings

AutoCAD's hatching and sketching features can help enhance the visual appearance and readability of a drawing. For example, on the map shown in Fig. 19-1 on page 248, hatching makes it easy to tell which parts of the area are national forest and which are urban areas. The irregular lines representing the roads were created using the **SKETCH** command.

In some cases, hatching and sketching are used to help make a drawing attractive. This may be true when drawings are created for use by a manufacturer's marketing or sales team. In other cases, hatching and sketching are used according to industry standards to make a technical drawing easier to read at a glance.
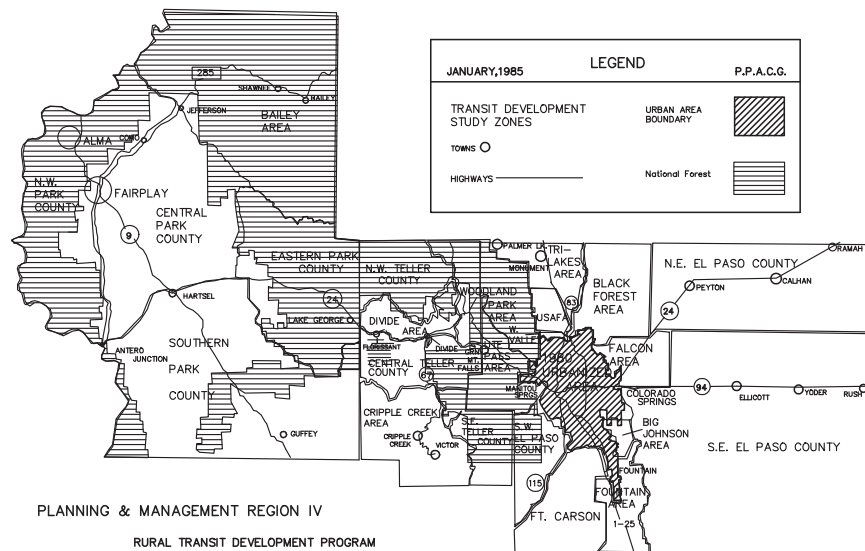
Fig. 19-1

Courtesy of David Salamon, Pikes Peak Area Council of Governments

Hatching

A **hatch** or **crosshatch** is a repetitive pattern of lines or symbols that shows a related area of a drawing. Hatching is used extensively in engineering drawing and production drafting to show cut surfaces. For example, when you draw a section of a mechanical part, you use hatching to reflect the cut, or internal, surface. **Section drawings** are used to show the interior detail of a part. Figure 19-2 shows an example of a **full section** (one that extends all the way through the part). The drawing on the left is the front view, and the drawing on the right is the section view.

Creating a Boundary Hatch

The **BHATCH** command creates associative hatches. An **associative hatch** is one that updates automatically when its boundaries are changed.

1. Start AutoCAD and begin a new drawing from scratch.
2. Select the **AutoCAD Classic** workspace and close all floating toolbars and palettes.
3. Create a sectional view of the center support shown in Fig. 19-3. Use the dimensions shown in the figure to create it accurately. Do not include the text (**Center Support**), the center lines, or the dimensions.
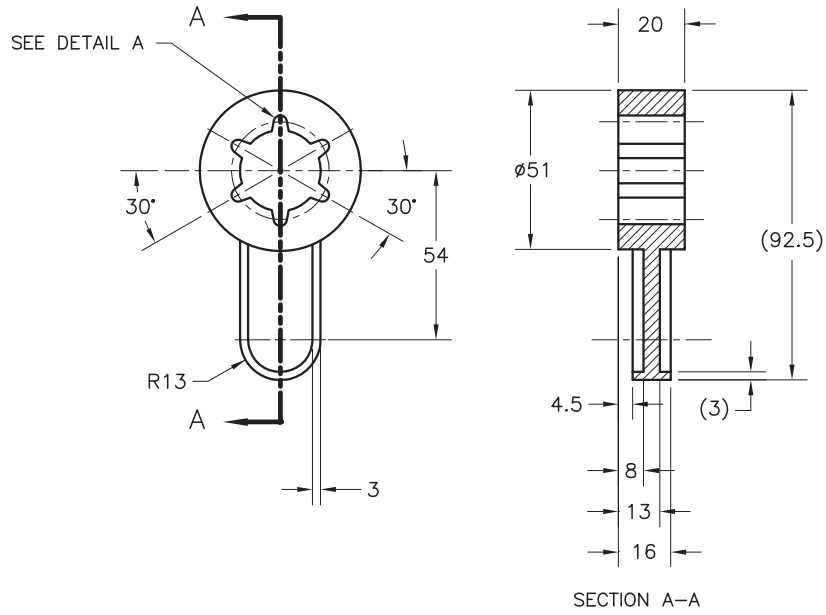
Fig. 19-2

Courtesy of Julie H. Wickert, Austin Community College
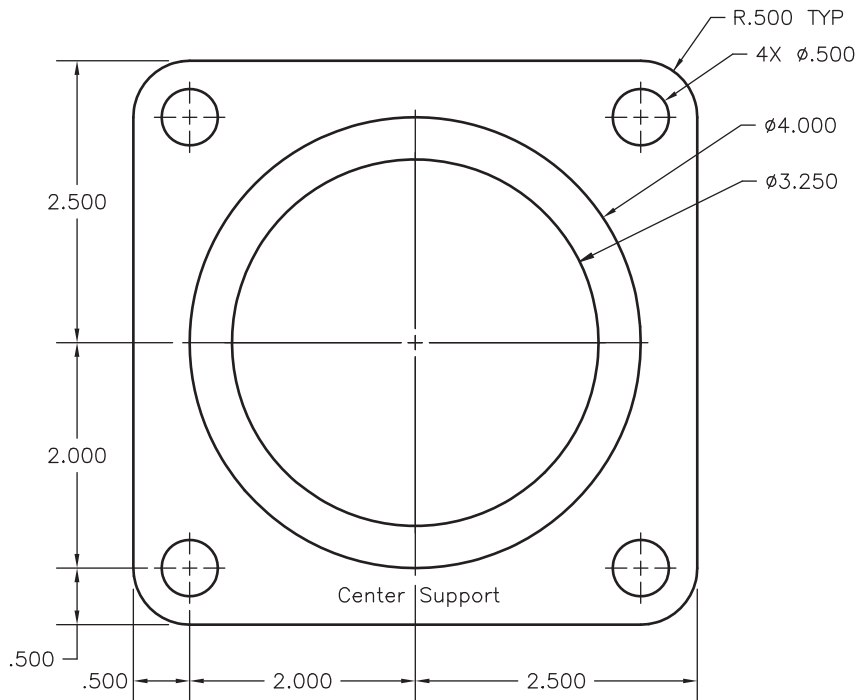
Fig. 19-3
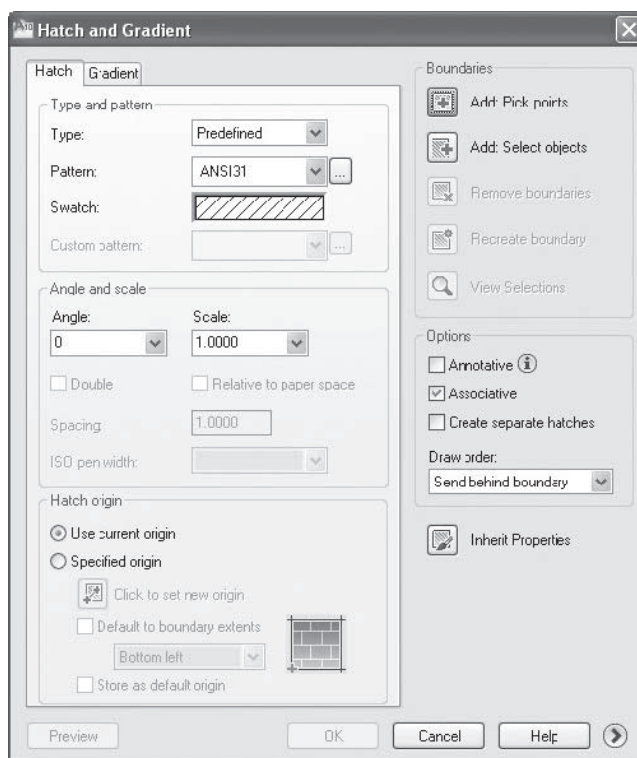
Fig. 19-4

DRAW

4. Save your work in a file named **hatch.dwg**.

5. Pick the **Hatch** button on the **Draw** toolbar.

This enters the **BHATCH** command and displays the **Hatch and Gradient** dialog box, as shown in Fig. 19-4.

6. Pick the **More Options** arrow (**>**) located in the lower right corner of the dialog box.

7. In the **Island display style** area of the dialog box, pick **Normal**, unless it is already selected.

8. Under the **Hatch** tab, pick the **Pattern** down arrow.

A list of hatch patterns appears. ANSI31 is the standard crosshatch pattern.

9. Pick the **...** button located to the right of the down arrow you just picked.

The **Hatch Pattern Palette** appears with ANSI hatch patterns. ANSI stands for the American National Standards Institute, an industry standards organization in the United States.

10. Pick the **ISO** tab.

ISO hatch patterns appear. ISO stands for the International Standards Organization, which also produces industry standards.

11. Pick the **Other Predefined** tab.

AutoCAD displays other predefined hatch patterns that are available to you.

12. Pick the **ANSI** tab, pick **ANSI31**, and pick **OK**.

Notice the **Angle and scale** area in the dialog box. **Angle** permits you to rotate the hatch pattern, while **Scale** enables you to scale it. The scale value should be set at the reciprocal of the plot scale so that the hatch pattern size corresponds to the drawing scale. This information will be useful when you begin to print or plot your drawings.

13. Under **Boundaries**, pick the **Add: Select objects** button, select the entire drawing, and press **ENTER**.

The **Hatch and Gradient** dialog box reappears.

14. Pick the **Preview** button located in the lower left area of the dialog box to preview the hatch.

15. As instructed by AutoCAD, right-click to accept the hatch.

Your drawing should now look similar to the one in Fig. 19-5.

16. Save your work.
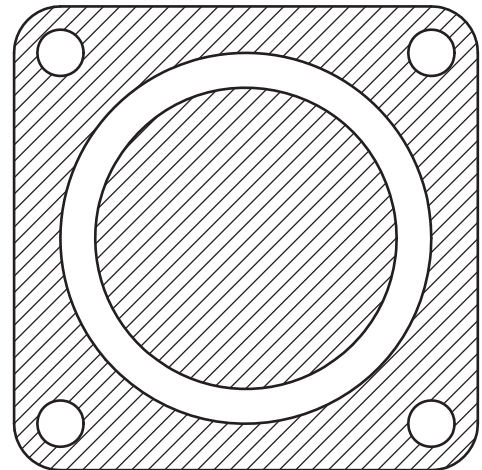
Fig. 19-5

Editing a Hatch

By default, hatches created with the **BHATCH** command are associative. Remember, an associative hatch is one that updates automatically when you change the size or shape of the hatched object. Associative hatches also update when you add or remove islands inside the hatched area.

Changing the Boundary

1. Select the inner cylindrical part of the center support. Be sure to select the circle and not the hatch.

2. Move the pointing device over the sidebars of the **Quick Properties** panel so that it expands. (Make sure the **Quick Properties** button on the status bar is toggled on.)

3. In the **Quick Properties** panel, change **Diameter** to **2.75** and press **ENTER**.

The diameter changes, along with the associative hatch.

4. Press **ESC** to close the **Quick Properties** panel and de-select the circle.

5. Change the diameter of the lower right hole to **.375** and press **ESC** to close the **Quick Properties** panel.

Once again, AutoCAD updates the hatch pattern.

6. Undo the two changes you just made.

Adding and Removing Islands

The boundaries of a hatch can also be edited by adding and removing islands from the hatch. An **island** is an area within an object, such as a hole, that is not hatched.

The **HATCHEDIT** command permits you to change the boundaries of an associative hatch. To enter this command, you can enter **HATCHEDIT** at the keyboard, right-click and use the shortcut menu, or pick the **Edit Hatch** button on the **Modify** toolbar**.**

MODIFY II

1. Display the **Modify II** toolbar and pick the **Edit Hatch** button.

This enters the **HATCHEDIT** command.

2. Pick the hatch in the drawing.

This causes the **Hatch Edit** dialog box to display. As you can see, it is very similar to the **Hatch and Gradient** dialog box.

3. Under **Boundaries**, pick the **Remove boundaries** button.

4. Select the inner cylindrical part of the center support and press **ENTER**.

5. Pick the **Preview** button. Then right-click to accept the edit.

AutoCAD computes the area of a hatch and updates it automatically when you edit the hatch.

6. Select the hatch.

7. Pick the **Properties** button on the **Standard** toolbar.

STANDARD

The **Properties** palette appears, which provides much more information about objects than the **Quick Properties** panel. Notice that the hatch area is 11.4336 square inches. (You may need to enlarge the palette and expand the **Geometry** table to see the **Area** value.)

MODIFY II

8. Reopen the **Hatch Edit** dialog box.

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III. Drawing and Editing

19. Hatching and Sketching

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259

9. Pick the **Add: Select objects** button and add the center cylinder back into the hatched area. Pick **OK** when you are finished.

10. Select the hatch and check the area displayed in the **Properties** palette.

The hatch area is automatically updated to 19.7294 square inches.

11. Close the **Properties** palette and save your work.

Changing Other Hatch Characteristics

The **Hatch Edit** dialog box also allows you to change other characteristics of a hatch. For example, you can choose a different hatch pattern or change the angle or scale of the current hatch pattern.

1. Pick the **Edit Hatch** button and select the hatch.

2. Make the following changes:
   Pattern: **ESCHER**
   Angle: **30**
   Scale: **1.3**

3. Pick the **OK** button.

MODIFY II

AutoCAD applies the changes to the drawing.

4. Experiment further on your own with creating and changing associative hatch objects.

5. Using the **Edit Hatch** button, change the hatch pattern to **ANSI31**, the angle to **0**, and the scale to **1**. Pick **OK** when you are finished.

6. Save your work.

MODIFY II

Tool Palettes Window

The **Tool Palettes** window provides another method for inserting hatch patterns.

1. Move the center support to the right side of the drawing area, produce a copy of it on the left side of the drawing area, and remove the hatch pattern from the copy.

2. Select the **Tool Palettes Window** from the **Standard** toolbar.

This displays the **Tool Palettes** window.

STANDARD

3. Review the contents of the **Tool Palettes** window by clicking each of its tabs.

4. Pick the **Hatches and Fills** tab and pick the **Imperial Steel** hatch pattern.

To see the name of a pattern, rest the pointer on it.

The **Imperial Steel** hatch pattern locks onto the crosshairs.

5. Click inside the center support's cylindrical feature.

AutoCAD hatches only the cylindrical feature.

6. Undo the hatch.
7. Select the **Steel** hatch pattern again from the **Tool Palettes** window and pick a point just inside the outer boundary of the center support. Do not select the cylindrical area.

AutoCAD hatches the center support correctly.

8. Erase the second center support and center the first one in the drawing area.
9. Close the **Tool Palettes** window and save your work.

You will learn more about the **Tool Palettes** window in Chapter 33.

Sketching

The **SKETCH** command is rarely used to create "sketches" because AutoCAD offers many other commands that make sketching (in its traditional sense) a faster, more accurate process. However, the **SKETCH** command is used for other purposes, such as to show irregular lines on maps like the one shown in Fig. 19-1 on page 248.

Let's try some freehand sketching.

1. Using the **Save As...** option in the **Menu Browser**, create a new drawing named **sketch.dwg**.
2. Erase the objects that are currently in the new drawing.
3. Enter **SKPOLY** at the keyboard and specify a value of **1**.

SKPOLY is a system variable that controls whether the **SKETCH** command creates lines (**0**) or polylines (**1**). Setting **SKPOLY** to **1** allows you to smooth sketched curves using the **PEDIT** command.

4. Turn off snap and enter the **SKETCH** command.

Table 19-1. SKETCH Options

Option	Function
Pen	Raises and lowers the pen
eXit	Records all temporary lines and exits
Quit	Discards all temporary lines and exits
Record	Records all temporary lines
Erase	Selectively erases temporary lines
Connect	Connects to an existing line endpoint
. (period)	Line to point

5. Specify **.1** unit for the **Record increment**. (This should be the default value.)

The sketching options appear at the **Command** line. Table 19-1 provides a brief description of each of the **SKETCH** options.

6. To begin sketching, pick a point where you would like the sketch to begin. The pick button toggles the pen down.

7. Move the pointing device to sketch a short line.

8. Pick a second point (to toggle the pen up), and type **X** to exit.

9. Move to an open area on the screen and sketch the golf course sand trap shown in Fig. 19-6. Set the **Record increment** at **.2**.

It's okay if your sketch doesn't look exactly like the one in Fig. 19-6.

10. When you are finished sketching the sand trap, type **R** for **Record** or **X** for **eXit**.

11. Practice sketching by using the remaining **SKETCH** options. Draw anything you'd like.

12. When you have finished, save your work and exit AutoCAD.
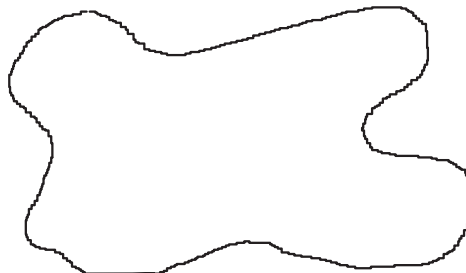
Fig. 19-6

Chapter 19 **Review & Activities**

• REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Explain why hatch patterns are useful.
2. With which command can you change a hatch pattern after the hatch has been applied to an object?
3. The **BHATCH** command permits you to hatch drawings. Describe a second method.
4. Briefly describe the purpose of each of the following options of the **SKETCH** command.
   a. **Pen**
   b. **eXit**
   c. **Quit**
   d. **Record**
   e. **Erase**
   f. **Connect**
   g. (period)
5. **SKETCH** requires a **Record increment**. What does this increment determine? (If you are not sure, specify a relatively large increment such as **.5** or **1** and notice the appearance of the sketch lines.)
6. What is the purpose of the **SKPOLY** system variable?

• CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Find out what the **Inherit Properties** button on the **Hatch and Gradient** dialog box does. Explain how it can save you time when you create a complex drawing that has several hatched areas.
2. Investigate the **Gradient** tab of the **Hatch and Gradient** dialog box. What does this tab allow you to do? When might you want to use this feature?

Review & Activities   Chapter 19

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Draw the wall shown in Fig. 19-7. Replace the angled break line on the right with an irregular line drawn with the **SKETCH** command to indicate a continuing edge. Hatch the wall in the **Brick** pattern. Save the drawing as **brickwall.dwg**.

2. Create the simplified house elevation shown in Fig. 19-8. Use the **SKETCH** command to define temporary boundaries for the hatch patterns on the roof.

3. Use the **SKETCH** command to create a map of the United States. Use Fig. 19-9 as a guide.
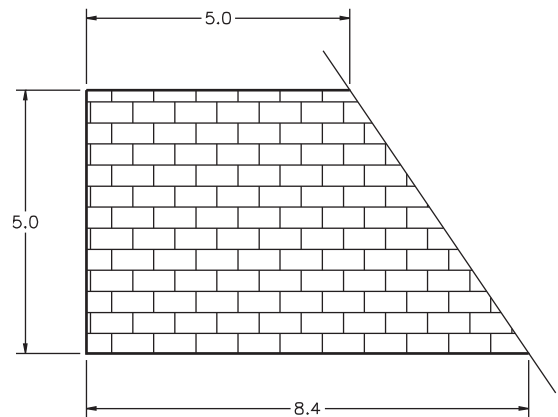
Fig. 19-7
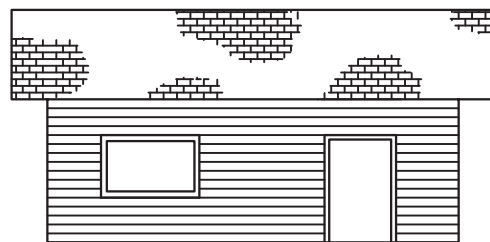
BRICK AND LINE

Fig. 19-8

Fig. 19-9

Continued

Chapter 19 **Review & Activities**

4. Use the **SKETCH** command to draw a logo similar to the one shown in Fig. 19-10. Before sketching the lines, be sure to set **SKPOLY** to **1**. Use the **BHATCH Solid** pattern to create the filled areas.

Fig. 19-10

Courtesy of Gary J. Hordemann, Gonzaga University

5. Draw the right-side view of the slider shown in Fig. 19-11 as a full section view. Assume the material to be aluminum. See Appendix D for a table of dimensioning symbols. For additional practice, draw the top view as a full section view.
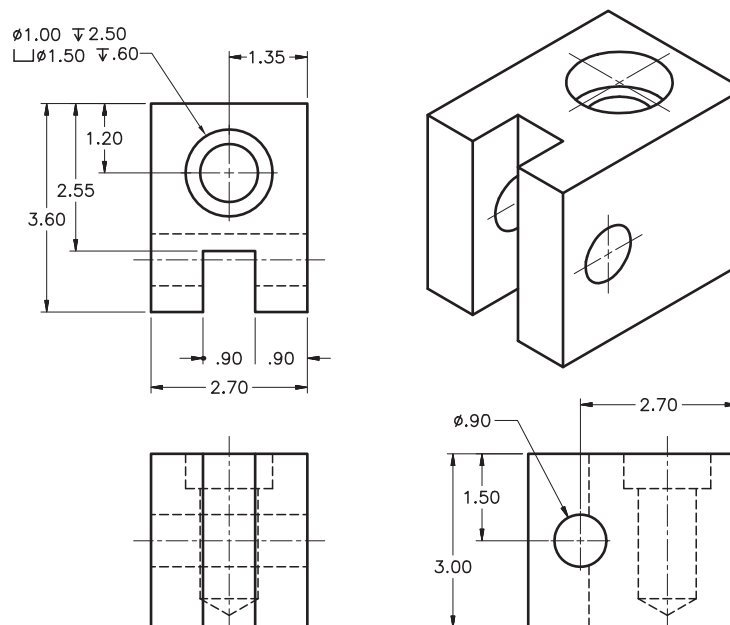
Fig. 19-11

Courtesy of Gary J. Hordemann, Gonzaga University

Review & Activities Chapter 19

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Your company is trying some new bushings for the swings in the child's playground set it manufactures. Draw the front view and full section of the bushing shown in Fig. 19-12. Hatch the section as appropriate. Save the drawing as **bushing.dwg**.
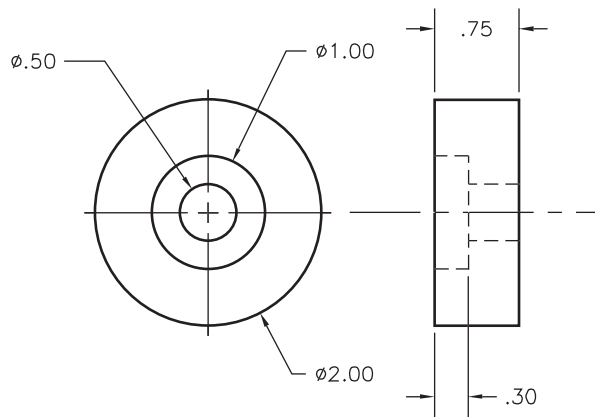
Fig. 19-12

2. An alternate design of the bushing described in problem 1 provides a hard insert to act as a wear surface within the bushing (Fig. 19-13). Make the change in the sectional view and hatch accordingly. Save the drawing as **altbushing.dwg**.
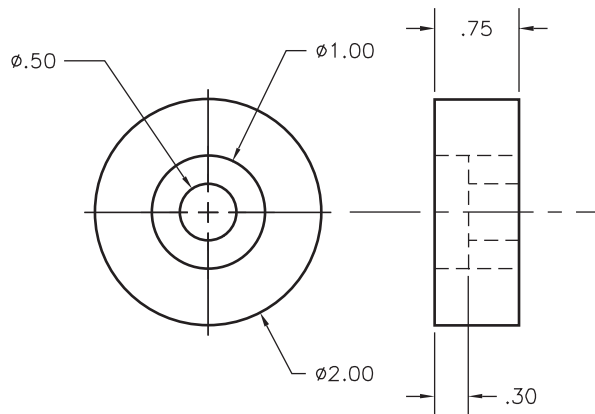
Fig. 19-13

Continued

Chapter 19 **Review & Activities**
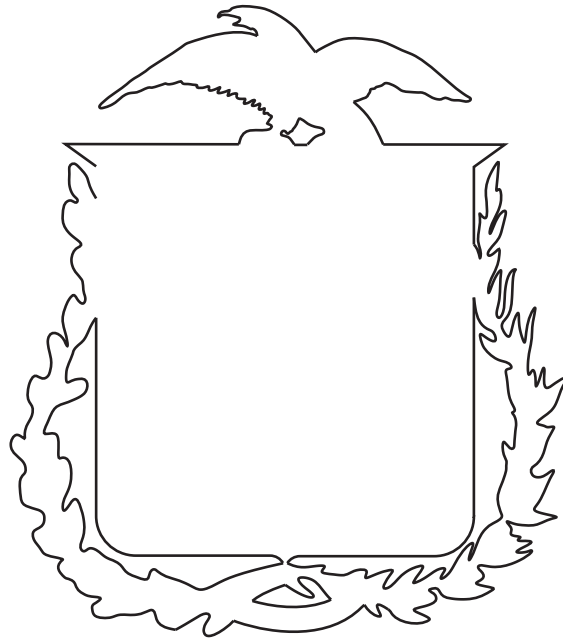
Fig. 19-14

3. The drawing shown in Fig. 19-14 was used with a computer-driven, high-pressure water cutter to cut a shield out of a piece of $1/4''$ brass. Use the **SKETCH** command with **SKPOLY** set to **1** to draw a similar shield. Experiment with the **PEDIT** command to make the sketched drawing as smooth as possible.

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III. Drawing and Editing

Careers Using AutoCAD:
Checker

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267

CAREERS Using AutoCAD

Checker

Like everyone else, drafters who create detailed technical drawings can make mistakes. Before drawings are submitted for use in the actual building or manufacturing process, they therefore must obtain clearance from a checker.

Checkers ensure that mathematical equations are correct, diagrams are drawn to scale, and notes and abbreviations are clear and accurate. They also ensure that the drawings contain all of the information necessary to create the part or component. Drawings that do not meet the checker's standards are returned to the drafter for revision. After the checker reviews the corrections, a supervisor, architect, or engineer reviews the corrected drawings also. Mistakes, if left unnoticed, can cost a company a great deal of money.

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Communication Is the Key

Good communication skills and teamwork are required, because checkers often work directly with both drafters and supervisors. They must be able to convey information on both verbal and visual levels.

The ability to get along with other people is also very important. A checker must be able to point out errors and suggest corrections without belittling the drafter or making the drafter feel uncomfortable.

On the Job

Checkers work in a variety of drafting areas, including architecture, manufacturing, construction, transportation, and communication. Checkers are not out to make drafters' lives miserable by returning drawings—which may require days, weeks, or even months of work—marked with corrections. Checkers simply try to ensure that all of a company's drawings are as complete and accurate as possible— for practical, financial, and safety reasons.

► Career Activities

1. Explain how the job of a checker is similar to that of a proofreader or editor.

2. What skills do you think a checker needs to do the job well?

Part 3 Project | Applying Chapters 15-19

Signage

Throughout history, people have used visual cues in the form of signs to communicate information. Signs are typically easy to interpret, quick to read (if indeed they contain any actual text at all), and unobjectionable. Signs without text, by the way, are often considered universal in nature: they are capable of being understood in different cultures regardless of the language spoken. The purpose of a sign can be varied—advertising, public service, directions, warnings, identification—almost any message that needs to be transmitted. A very common use of signs is to regulate vehicular traffic. Traffic signs identify routes taken, geographic direction of the vehicle, speed limits, expressway exit/entrance numbers, nearby hospitals, and sources of food and fuel for the traveler, among other uses.

▶ Description

In AutoCAD, create the signs in Fig. P3-1 as assigned by your instructor, or improve upon some of the signs as they currently exist. For example, you might create a more effective graphic. Actual sizes and legal descriptions of many signs are available on the Internet.

Fig. P3-1

Wohlers: Applying
AutoCAD® 2010

III. Drawing and Editing

Part 3 Project: Applying
Chapters 15–19

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269

Alternatively, your instructor may ask you to design your own original signs. Suggestions include:

elephant crossing	**no junk food**
cash only	**idea factory**
no fish here	**no fishing**
no tire-changing	**barbecues available**
wet concrete	**drive safely or don't drive**

Use your imagination or seek advice from the instructor for further ideas.

▶ Hints and Suggestions

1. Consider using **DRAW>SURFACE>2D SOLID** to create the sign as a solid color; alternatively, you may want to use **POLYGON** with the **Solid** hatch pattern to color in the background.

2. Consider using gradients from the **Gradient** tab of the **Hatch and Gradient** dialog box. Would a gradient be effective on a highway sign?

3. Polylines allow a choice of thickness as well as color for lines, arrows, and other images that might be appropriate for your sign's communication.

4. If you use polylines, filleting closed polygons can be accomplished in a single command.

5. Utilize the various modifying commands to reuse pieces of one sign that might lend themselves to another section or sign.

▶ Summary Questions

Your instructor may direct you to answer these questions orally or in writing. You may wish to compare and exchange ideas with other students to help increase efficiency and productivity.

1. Why is technical drawing sometimes referred to as a "universal language"? Explain.

2. What might be a reason to keep all signs in a particular category, such as "informational" or "regulatory," based on a single color scheme or shape?

3. What characteristics make traffic signs easy to read as drivers go by?

4. How is it that so many countries can use similar traffic signs, yet not have a shared language or culture? Explain your reasoning.

5. Do you think there is a relationship between the size of a sign and its importance to the driver? Explain.

<div style="text-align:right">C h a p t e r   **20**</div>

Notes and Specifications

Objectives

- Create and edit text dynamically
- Specify the position and orientation of text
- Create and use new text styles
- Review options for fonts and adjust the quality of TrueType fonts
- Produce multiple-line text using AutoCAD's text editor
- Import text from an external word processor
- Format text to create lists

Vocabulary

font
justify
mtext
notes
specifications
title block

Notes and specifications are a critical part of most engineering drawings. Although the two terms are often used interchangeably, notes and specifications are technically two different types of text. **Notes** generally refer to the entire drawing, rather than to any one specific feature. For example, a note might describe the thickness of a part. **Specifications**, on the other hand, provide information about size, shape, and surface finishes that apply to specific portions of an object or part.

Without text, drawings would be incomplete. The drawing in Fig. 20-1 shows the amount of text that is typical in many drawings. Some drawings contain even more text. As you can see, the text is an important component in describing the drawing. With traditional drafting, the text is carefully placed by hand, which is not only time consuming, but tedious. With CAD, you can place the words on the screen almost as fast as you can type them.
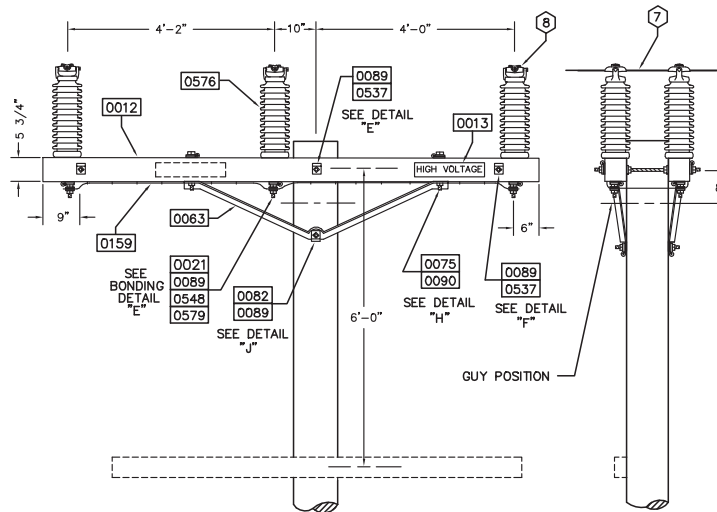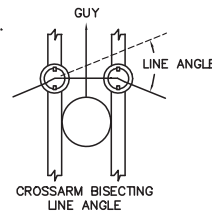
Fig. 20-1

Types of Text

AutoCAD supplies two different ways of inserting text objects. The one you use should depend on the type and amount of text you are inserting. The **DTEXT** command inserts one or more lines of text as single-line objects. **MTEXT**, on the other hand, treats one or more paragraphs of text as a single object. This chapter presents both ways of inserting text.

Dynamic Text

The **DTEXT** command enables you to display AutoCAD text dynamically in the drawing as you type it. It also allows you to **justify**, or align, the text in several ways, including left-aligned, right-aligned, and centered.

1. Start AutoCAD and start a new drawing from scratch.
2. Select the **2D Drafting & Annotation** workspace.
3. Set snap to **.25**.

TEXT

4. Select the **Annotate** tab on the **Ribbon** and pick the **Single Line Text** button on the **Text** panel, or enter **DT** (the command alias for **DTEXT**).

This enters the **DTEXT** command.

5. In response to **Specify start point of text**, place a point on the left side of the screen. The text will be left-justified beginning at this point.

6. Reply to the **Specify height** prompt by moving the crosshairs up **.25** units from the starting point and picking a point.

7. Enter **0** (degrees) in reply to **Specify rotation angle of text**.

A blinking cursor appears inside a narrow text box.

8. Type your name using both upper- and lowercase letters and press **ENTER**.

You should again see the blinking cursor, now inside a text box of two lines.

9. Type your street address or P.O. box and press **ENTER**.

Notice where the text appears in relation to the previous line.

10. Type your city, state, and zip code and press **ENTER**.

11. Press **ENTER** again to terminate the **DTEXT** command.

Let's enter the same information again, but this time in a different format.

12. Reenter the **DTEXT** command (press the space bar), and **J** for the **Justify** option.

The justification options appear.

13. Enter **C** for the **Center** option.

14. Place the center point near the top of the screen and set the text height by entering **.2** at the keyboard. Do not insert the text at an angle.

15. Repeat Steps 8 through 11. Be sure to press **ENTER** twice at the end.

When you are finished, your text should be centered like the example in Fig. 20-2. If it isn't, try again.

16. Save your work in a file named **text.dwg**.

Ms. Jane Doe
507 East 104th Avenue
Cadsville, CA 30160

Fig. 20-2

Multiple-Line Text

The **MTEXT** command creates a multiple-line text object called **mtext**. AutoCAD uses a text editor to create mtext objects.

1. Erase the text from the upper half of the screen.

2. From the **Annotate** tab of the **Ribbon,** pick the **Multiline Text** button on the **Text** panel or enter **T** at the keyboard.

TEXT

This enters the **MTEXT** command and displays the text style and height at the **Command** line.

3. Pick a point in the upper left area of the screen.

Notice the new list of options.

> **HINT** Recall that you can access the list of options at the dynamic input cursor by pressing the down arrow key on the keyboard.

4. Enter **W** for **Width**.

5. Enter **4.5** in response to **Specify width** or, with ortho and snap on, pick a point **4.5** units to the right of the first point.

AutoCAD displays a new **Ribbon** tab, **Text Editor,** with text formatting panels, and a ruler above the selection point. Notice that the font is **Txt** and the default text height is **.2000** units.

> **NOTE** A **font** is a set of characters, including letters, numbers, punctuation marks, and symbols, in a particular style. **Arial** and **Roman Simplex** are examples of fonts used in AutoCAD.

6. From the font drop-down list on the **Formatting** panel, change the font from **Txt** to **Arial**.

7. Type the following sentence:

   I'm using AutoCAD's text editor to write these words.

8. Highlight the word **using** and pick the **B** (bold) button.

9. Highlight the word **words** and pick the **U** (underline) button.

The sentence should now look like the one in Fig. 20-3 on page 268.

10. Pick the **Close Text Editor** button on the **Ribbon** and save your work.

CLOSE

Fig. 20-3

Importing Text

Instead of typing text within AutoCAD, you can create it using a standard word processing program and then import it. Another option is to copy text from a word processing program and paste it into AutoCAD. Many people find it easier to create lengthy text passages using word processors with which they are familiar.

1. Minimize AutoCAD.
2. From the Windows **Start** menu, pick **Programs** (or **All Programs**), **Accessories**, and **Notepad**.
3. Enter the following text:

   AutoCAD's text editor permits you to import text from a file. This text will appear in AutoCAD shortly.

4. From Notepad's **File** pull-down menu, pick **Save As...**, select the folder with your name, and name the file **import.txt**.
5. Exit Notepad and maximize AutoCAD.
6. From the **Annotate** tab of the **Ribbon**, pick the **Multiline Text** button on the **Text** panel.
7. Near the top center of the screen, pick the first corner.
8. Produce a rectangle that measures **1** unit tall by **3** units wide.

The multiline text editor appears.

9. Right-click inside the text box to display the shortcut menu.
10. Select **Import Text...** from the menu.
11. Locate and select the new file named **import.txt**.

This imports the text into the multiline text editor.

12. Highlight the text and pick the **Arial** font from the font drop-down list on the **Ribbon**.

The text changes to the **Arial** font.

13. Pick the **Close Text Editor** button on the **Ribbon** and save your work.

TEXT

CLOSE

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AutoCAD® 2010

IV. Text and Tables

20. Notes and
Specifications

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275

Formatting Text

The multiline text editor in AutoCAD offers many of the same formatting capabilities as standard word processing programs. For example, you can format text into bulleted, numbered, or lettered lists. You can also create paragraph indents and tabs.

TEXT

1. From the **Text** panel on the **Ribbon**, pick the **Multiline Text** button, or enter **T** at the keyboard.

2. Pick a point in an open area of the screen, enter **W** for **Width**, and **6** for the width.

3. Type the following lines. Be sure to press **ENTER** at the end of each line.

   Notes:
   All dimensions in inches.
   Material: Aluminum 6061-T6 bar stock.
   All fillets and rounds R.125 unless otherwise specified.
   Break all sharp corners.

4. Highlight all of the typed text, change the font to **Arial**, and make the text height **.2000** unit.

5. Highlight the first line and underline it.

6. Highlight every line except the first line, then pick the **Bullets and Numbering** button on the **Paragraph** panel, and select **Numbered** from the menu.

7. If there is too much space between the numbers and text, pick the small rectangle and triangle in the ruler bar and slide them to the left to align the text closer to the numbers.

The list should now look like the one in Fig. 20-4.

8. Pick the **Close Text Editor** button.

9. Save your work.

Notes:
1. All dimensions in inches.
2. Material: Aluminum 6061-T6 bar stock.
3. All fillets and rounds R.125 unless otherwise specified.
4. Break all sharp corners.

F i g .   2 0 - 4

Text Styles and Fonts

It is possible to create new text styles using the **STYLE** command. During their creation, you can expand, condense, slant, and even draw the characters upside-down and backwards.

Creating a New Text Style

1. Select the **Text Style** pull-down menu on the top of the **Text** panel on the **Ribbon** and select the **Manage Text Styles...** option, or enter **ST** at the keyboard for the **STYLE** command.

The **Text Style** dialog box appears, as shown in Fig. 20-5.

> **HINT**
>
> You can also open the **Text Style** dialog box from the **Format** menu in the **Menu Bar.** In the **AutoCAD Classic** workspace, the **Text Style...** button is on the docked **Styles** toolbar. It is located at the right of the **Standard** toolbar.

2. Pick the **New...** button, enter **comp1** for the new text style name, and pick the **OK** button.
3. Under **Font Name**, display the list of fonts by picking the down arrow.
4. Using the scroll bar, find the font file named **complex.shx** and select it.

Notice that the text sample in the Preview box changes to show the appearance of the **complex.shx** font.

Fig. 20-5

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AutoCAD® 2010

IV. Text and Tables

20. Notes and
Specifications

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277
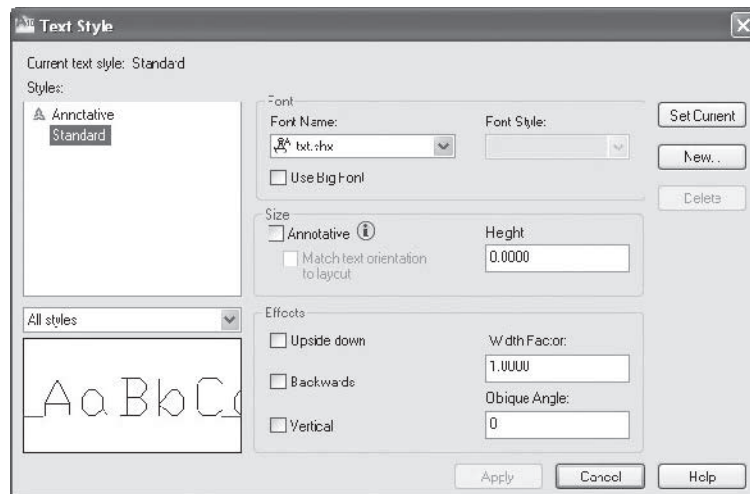
5. Study the other parts of the dialog box and then pick the **Apply** and **Close** buttons.

You are ready to use the new **comp1** text style. Notice that the **comp1** name appears in the **Text Style** box on the **Text** panel.

6. Enter the **DTEXT** command and the **Justify** option.

7. Right-justify the text by entering **R** (for **Right**).

8. Place the endpoint near the right side of the screen.

9. Set the height at **.3** unit.

10. Set the rotation angle at **0**.

11. For the text, type the three lines shown in Fig. 20-6. Be sure to press **ENTER** twice after typing the third line.

Assemble and
ream for No. 0
taper pin.

Fig. 20-6

With the **STYLE** command, you can develop an infinite number of text styles. Try creating other styles of your own design. The **romans.shx** font is recommended for most engineering drawings.

As you create more text styles within a drawing file, you may occasionally want to check their names.

12. Select the down arrow to the right of the **Text Style** text box on the **Text** panel to display the defined text styles.

If you want to learn more about the text styles, you will need to display the **Text Style** dialog box.

13. Select the **Manage Text Styles...** option from the pull-down menu.

14. In the **Text Style** dialog box, under **Styles**, select **Standard**.

The font name (**txt.shx**) and other characteristics of the style appear. This is the default text style.

15. Pick the **Cancel** button.

Text Fonts

AutoCAD supports TrueType fonts and AutoCAD compiled shape (**SHX**) fonts.

1. Reopen the **Text Style** dialog box and pick the **New…** button.
2. Type **tt** for the new text style name and pick the **OK** button or press **ENTER**.
3. Under **Font Name**, find and select **Swis721 BT**.

The **Swis721 BT** font displays in the preview area. Notice the overlapping T's located at the left of the font name. This indicates that it is a TrueType font. Notice also that **Roman** displays under **Font Style**.

4. Under **Font Style**, display the list of options, pick **Italic**, and notice how the font changes in the Preview area.

Note the **0.0000** value in the text box under **Height**. This **0** value indicates that the text is not fixed at a specific height, giving you the option of setting the text height when you enter the **DTEXT** command.

5. Pick the **Apply** and **Close** buttons.
6. Enter the **DTEXT** command and pick a point anywhere in the drawing area.
7. Enter **.2** for the height and **0** for the rotation angle.
8. Type **This is TrueType.** and press **ENTER** twice.

This is TrueType.

Fig. 20-7

The TrueType text should look like the text in Fig. 20-7.

Text Quality

The **TEXTQLTY** system variable sets the resolution of text created with TrueType fonts. A value of **0** represents no effort to refine the smoothness of the text. A value of **100** represents a maximum effort to smooth the text. Lower values decrease resolution and increase plotting speed. Higher values increase resolution and decrease plotting speed. You will learn more about plotting in Chapter 25.

1. Enter the **TEXTQLTY** system variable at the keyboard.
2. Press the **ESC** key.
3. Experiment with other TrueType fonts on your own.
4. Save your work.

Setting the Current Style

Let's set a new current text style.

1. Select the down arrow to the right of the **Text Style** text box on the **Text** panel and pick the **comp1** text style.

The **comp1** style is now the current text style.

2. Place some new text on the screen and then terminate the **DTEXT** command.

3. Change back to the **tt** text style.

4. Save your work.

Applying Text in Drawings

Now that you have learned many ways of creating text, let's apply that knowledge to the creation of a title block. A **title block** is a portion of a drawing that is set aside to give important information about the drawing, the drafter's name, the company, and so on.

QUICK ACCESS

1. Pick the **QNew** button on the **Quick Access** toolbar, or type **NEW** at the keyboard and begin a new drawing from scratch. Save it as **title.dwg**.

2. Set the grid at **.25** and snap at **.0625**.

3. Create the title block shown in Fig. 20-8 using the following information:

Fig. 20-8

- For **Dynamic Design, Inc.**, create a new style named **Swiss** using the **Swis721 Ex BT** font. Make the text **.25** tall.

- Create a new text style named **Roms** using the **romans.shx** font and use it to produce the small text, which is **.06** tall.

- Use the **Justify** options of the **DTEXT** command to position the text accurately.

4. Save your work and exit AutoCAD.

NOTE
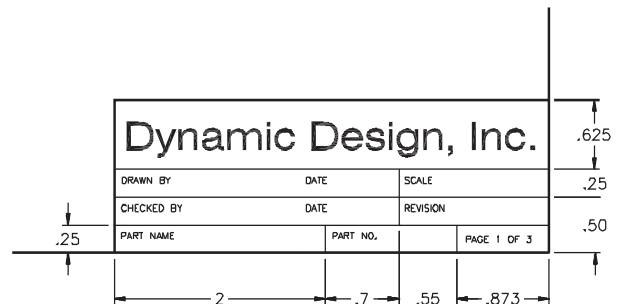
Many companies use their own customized title blocks. AutoCAD also supplies several ready-to-use title blocks in the **Templates** folder.

Chapter 20 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is the most efficient way to enter a single line of text? What command should you use?
2. What might be a benefit of using the **MTEXT** command?
3. What is the purpose of the **DTEXT Style** option?
4. Name at least six fonts provided by AutoCAD.
5. What command do you enter to create a new text style?
6. Briefly describe how you would create a tall, thin text style.
7. Explain how to import text from a word processor into AutoCAD. Why might you choose to do this?
8. What is the purpose of a title block?
9. Explain the differences between the **DTEXT** command and the **MTEXT** command. Describe a situation in which each command would clearly be a better choice.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Obtain on-screen help and then experiment with the **COMPILE** command. What are the advantages and disadvantages of compiling fonts? When might you want to use this option?
2. In addition to standard AutoCAD and TrueType fonts, AutoCAD can also work with PostScript fonts. Write a paragraph comparing TrueType and PostScript fonts. Are there any situations in which one type might be preferable to the other? Explain.
3. Which word processor do you use on your computer? Make a list of all the word processing programs you have used. Most likely, any of these can be used to import text into AutoCAD. Try importing text from several word processing programs.

Review & Activities Chapter 20

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.
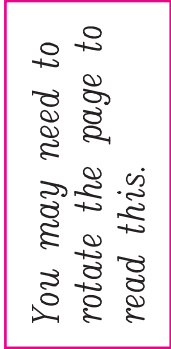
1. Type your name and today's date in Windows Notepad. Import this text into the drawings you created in Chapter 18. Save your work.

2. Create a new text style using the following information:

Style name: **cityblueprint**
Font file: **CityBlueprint**
Height: **.25** (fixed)
Width factor: **1**
Oblique angle: **15**

3. Create a new text style using the following information:

Style name: **ital**
Font file: **italic.shx**
Height: **0** (not fixed)
Width factor: **.75**
Oblique angle: **0**

4. Use the **DTEXT** command to place the text shown in Fig. 20-9. Use the **cityblueprint** text style you created in problem 2. Right-justify the text. Do not rotate the text.

Apply a light coat of primer after sand-blasting the surface.

(**Fig. 20-9**)

5. Use the **MTEXT** command and the **ital** text style you created in problem 3 to create the text shown in Fig. 20-10. Set the text height at **.3** unit. Rotate the text **90** degrees. Set the width of the text line to **4** units.

You may need to rotate the page to read this.

(**Fig. 20-10**)

Continued

Chapter 20: Notes and Specifications   **275**

Chapter 20 **Review & Activities**

6. Open the **remote.dwg** file you created in Chapter 18, "Applying AutoCAD Skills" problem 3. If you have not yet created the drawing, do so now. Then add the text shown in Fig. 20-11.

7. Shown in Fig. 20-12 is a block drawing algorithm for a program that sorts numbers into ascending order by a method known as *sorting by pointers*. Use the **LINE** and **OFFSET** commands to draw the boxes. Then insert the text using the **romans.shx** font. For the words **MAIN**, **SORT**, and **SWAPINT**, use the **romanc.shx** font and make the text larger. For the symbol **>**, use the **symath.shx** font, character **N**.
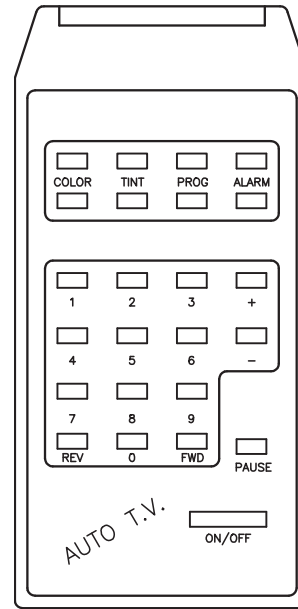
Fig. 20-11
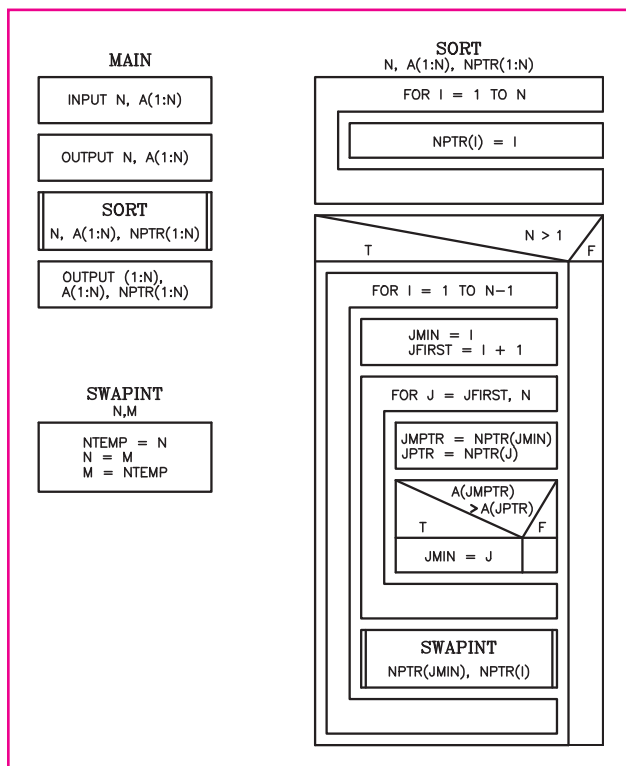
Fig. 20-12

Courtesy of Gary J. Hordemann, Gonzaga University

Review & Activities Chapter 20

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Most companies have standard operating procedures for creating engineering drawings. Your company is new, and the procedures have not yet been formalized. However, it has become standard practice to place a rectangular border around drawings and enclose the drawing identification in a title block. Open **shaftbearing.dwg**, which you created in Chapter 10 ("Applying AutoCAD Skills" problem 2). Draw a border around the periphery of the drawing. Across the bottom, draw a title block **.75** unit high and divide it into three equal parts. In the first part, place your name; in the second part, place the name of the drawing; and in the third part, place today's date. Save the drawing.

2. Open the **shim.dwg** or **shimrev.dwg** drawing from Chapter 17. Make a border and title block and complete the information as in problem 1. Add the following information to the drawing in the form of drawing notes. Use the text style and font of your choice, or as specified by your company's procedures. Place the notes in the lower left corner of the drawing, just inside the border. Save the drawing.

Notes:
1. Material: SAE 1010 Carbon Steel
2. Break sharp edges
3. All dimensions +/−.01 unless otherwise specified

Text Editing and Spell Checking

Objectives

- Edit text using AutoCAD's text editor
- Create special text characters
- Find and replace text
- Check the spelling of text in a drawing using AutoCAD's spell checker

Vocabulary

in-place text editor
special characters

AutoCAD offers several commands and features for editing and enhancing both types of text objects. Its text editor and spell checker help you produce error-free notes and specifications. With AutoCAD, it is possible to find and replace text using a feature similar to that in word processing programs.

Editing Text

AutoCAD provides two commands to change the contents of text. **DDEDIT** edits text created with the **DTEXT** command. **MTEDIT** edits multiple-line text created with the **MTEXT** command. In each case, AutoCAD displays an edit box, or **in-place text editor**, that allows you to edit the selected text.

1. Start AutoCAD and open the file named **text.dwg**.
2. Select the **AutoCAD Classic** workspace and close all floating toolbars and palettes.
3. Create a new text style named **roms** using the **romans.shx** font. Apply all of the default values and close the dialog box.

4. Using the **DTEXT** command, enter the following text at a height of **.15**.

NOTE: ALL ROUNDS ARE .125

Editing Dynamic Text

You can edit text that was created with the **DTEXT** command right on the screen.

1. Double-click the text you just created.

The text becomes highlighted against a blue background.

2. Move the pointer to the area containing the text and pick a point between the words **ALL** and **ROUNDS**.

3. Enter **FILLETS AND** so that the line now reads:

NOTE: ALL FILLETS AND ROUNDS ARE .125

4. Press **ENTER** twice to terminate the command.

Editing Mtext Objects

You can use a similar method to edit text created with the **MTEXT** command.

1. Double-click the mtext object located in the upper left area of the screen.

This displays an in-place text editor and the **Text Formatting** toolbar.

2. With the pointing device, position the cursor in the text and change a couple of words in the same way as you would with a word processor or text editor.

3. Change the style to **roms** and the height to **.125** and pick **OK**.

HINT To change the text height, you must first highlight the text.

The mtext changes according to the instructions you placed in the dialog box.

Justifying Mtext

1. Double-click the mtext object located in the upper left area of the screen.

2. Change the text style to **tt**.

3. Highlight the text.

4. In the **Text Formatting** toolbar, select the **MText Justification** button (the second button on the bottom row).

AutoCAD offers nine different ways to justify mtext objects. **Top Left** is the default setting. Text can be center-, left-, or right-justified with respect to the left and right text boundaries. Also, text can be middle-, top-, or bottom-aligned with respect to the top and bottom text boundaries.

5. Select **Top Right** and pick **OK** on the **Text Formatting** toolbar.

AutoCAD right-justifies the text along the top and right boundaries.

Creating Special Characters

The in-place text editor permits you to insert special characters such as the degree (°) and plus/minus (±) symbols. **Special characters** are those that are not normally available on the keyboard or in a basic text editor.

A

DRAW

1. Enter the **MTEXT** command.

2. In an empty area of the screen, create a rectangle measuring about **1** unit tall by **2** units wide.

3. Enter the following specification, but omit the special characters.

   Drill a Ø.5 hole at a 5° angle using a tolerance of ±.010.

4. Pick a point prior to **.5** and right-click.

5. From the shortcut menu, select **Symbol** and **Diameter**.

AutoCAD inserts the diameter symbol.

6. Pick a point after **5**, right-click, and select **Symbol** and **Degrees**.

The degree symbol appears.

7. Pick a point before **.010**, right-click, and select **Symbol** and **Plus/Minus**.

The plus/minus symbol appears.

8. Pick the **OK** button on the **Text Formatting** toolbar.

9. Double-click the text to redisplay the in-place text editor.

10. Move the pointer inside the text box, right-click, and rest the pointer on **Symbol**.

The menu includes several commonly used symbols such as **Angle**, **Center Line**, **Delta**, **Ohm**, and **Not Equal**.

11. Select **Other...** from the menu.

The character map displayed provides access to many additional characters.

12. Close the character map and pick the **OK** button to close the text editor.

Finding and Replacing Text

Suppose you have lengthy paragraphs of text and you want to find and replace certain words.

1. Rest the crosshairs on top of the mtext object located in the upper left area of the screen.

2. Right-click and pick the **Find...** item.

3. Pick the **More Options** arrow from the lower left corner of the dialog box.

This displays the expanded **Find and Replace** dialog box, as shown in Fig. 21-1.

4. In the **Find what:** text box, enter **write**.

5. In the **Replace with:** text box, enter **draft**.

6. Pick the **Find** button.

AutoCAD displays the text in the dialog box and finds and highlights the word **write**.

Fig. 21-1

7. Pick the **Replace** button.

AutoCAD replaces the word **write** with the word **draft**.

8. Pick the **OK** button to close the information dialog box that reports the results of the find and replace.

The **Find and Replace** dialog box offers additional options. If you check the **Match case** check box, AutoCAD finds the text only if the case of all the characters in the text matches the text in the **Find what:** text box. If you check the **Find whole words only** check box, AutoCAD matches the text in the **Find what:** text box only if it is a single word. If the text is part of another word, AutoCAD ignores it. The **Text Types** section allows you to select the types of text you want to search. By default, all options are selected.

9. In the upper right area of the dialog box, pick the down arrow under the **Find where:** heading.

As you can see, you can search the entire drawing, the current space/layout, or the selected objects.

10. Pick the **Close** button.
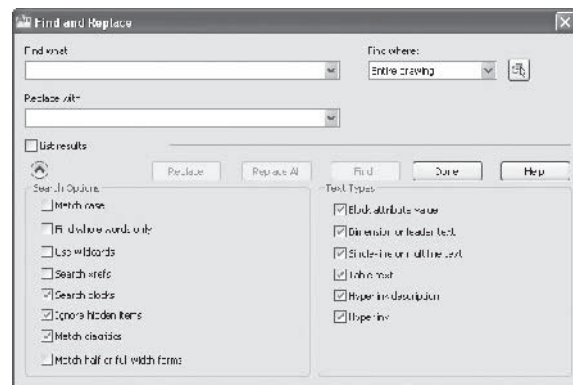
Using AutoCAD's Spell Checker

AutoCAD's spell checker examines text for misspelled words.

1. Enter **SP** at the keyboard to enter the **SPELL** command.

The **Check Spelling** dialog box appears.

2. Pick the down arrow under **Where to check,** and pick the **Selected objects** option.

3. Pick the **Select text objects** button, select the text **NOTE: ALL FILLETS AND ROUNDS ARE .125**, and press **ENTER**.

4. Select the **Start** button.

If you spelled the words correctly, AutoCAD displays the message box shown in Fig. 21-2.
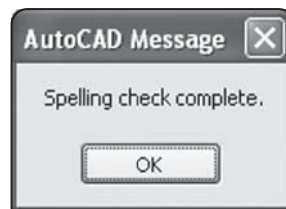
Fig. 21-2

5. Pick **OK** and close the **Check Spelling** dialog box.

6. Using the **DTEXT** command, add the term **TrueType** to the screen.

7. Enter the **SPELL** command. With **Entire drawing** selected in the **Where to check** text box, pick the **Start** button.

AutoCAD displays the words in the document that are not in the spell checker's dictionary. These include words you might have misspelled and words like **TrueType** (and perhaps your last name). For each word that AutoCAD displays, you can choose to **Ignore** the spelling or select one of AutoCAD's suggestions and **Change** the word's spelling. You can also choose to **Ignore All** instances of the word, **Change All** instances of the word, or add the word to AutoCAD's dictionary by picking the **Add to Dictionary** button.

8. Pick the **Change** button to correct words that are misspelled and the **Ignore** button to skip words that are not misspelled.

9. When the spelling check is complete, pick **OK** and close the **Check Spelling** dialog box.

Other Options

AutoCAD provides other options for changing the appearance of text created with MTEXT.

1. Double-click the mtext in the top right corner of the drawing area.

2. Move the pointer inside the text box and right-click.

3. Select the **Paragraph...** item.

This displays the **Paragraph** dialog box, which permits you to adjust the indentation and tabs for mtext objects.

> **NOTE** You can also display the **Paragraph** dialog box by right-clicking the ruler above the mtext and selecting the **Paragraph**... item.

4. Pick the **Cancel** button.

5. Highlight all of the mtext.

6. Right-click to display the shortcut menu and rest the pointer on **Change Case**.

This allows you to change the mtext to all upper- or lowercase letters. Below **Change Case** is **AutoCAPS**. This converts all newly typed and imported text to uppercase, but it does not affect existing text.

7. Select **Cancel** to close the shortcut menu.

8. Save your work and exit AutoCAD.

Chapter 21 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. How does selecting and right-clicking text permit you to edit dynamic text? How does it permit editing of mtext?

2. Why might the character map be useful?

3. How would you add a diameter symbol to mtext?

4. Explain a situation in which AutoCAD's spell checker may find a word that is spelled correctly. What might cause this?

5. Suppose you have paragraphs of text that contain several words or phrases that need to be replaced with new text. What is the fastest way to replace the text?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Look again at the mtext object you edited in this chapter. If you changed the word **text** to **test** and the word **write** to **rite**, what misspellings do you think the spell checker would find? Try it and see. Then write a short paragraph explaining the proper use of a spell checker.

2. As you read in this chapter, AutoCAD allows you to change the dictionary it uses to check the spelling of words. AutoCAD also allows you to create one or more custom dictionaries. Under what circumstances might you want to use a custom dictionary? Might you ever need more than one? Explain.

Review & Activities Chapter 21

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Open **shaftbearing.dwg**, which you updated in the previous chapter to include text. Change the date to be the date you actually created the original drawing. Add your middle initial to your name. If you already have your middle initial, remove it. Spell-check the title block. Did your name appear for correction? If it did, add it to the dictionary. Save the drawing as **shaft-rev1.dwg**.

2. Edit the drawing notes in **shimrev.dwg**. Remove note number 2 (**Break sharp edges**). Renumber note 3, and add periods at the end of the notes. Spell-check the notes and the title block. Your name should not appear for correction. Save the drawing.

3. Refer again to **shimrev.dwg**. Your method of entering **+/−** for a plus/minus value in the original note 3 is not acceptable to your supervisor. You have been asked to change it to the standard format of **±**. Erase your notes, reenter the text using the **MTEXT** command, and insert the proper symbol. Save the drawing.

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Open **Mechanical - Text and Tables.dwg** located in AutoCAD's **Sample/ Mechanical Sample** folder and save it in your named folder as **Text and Tables 2.dwg**. Zoom in so that you can read the individual lines. Search and replace each occurrence of **PIPE SUPPORTS** with the term **PIPE BRACKETS**. Create a new text style called **Notes** using the **Myriad Web Pro** font. Select all the text in the **General Notes** and change the text to Notes, make the text **4** units in height, and pick **OK**. Save your work.

Continued
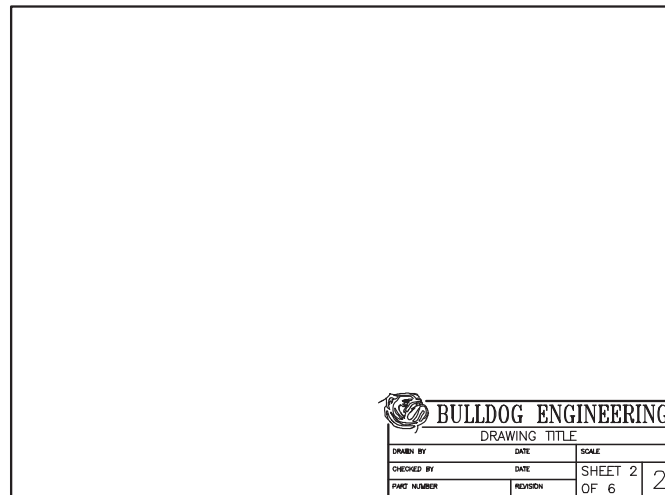
Fig. 21-3

2. The border and title block shown in Fig. 21-3 are suitable for a paper size of **8.5″ × 11″**. The border allows **.75″** white space on all four sides. Draw the border and title block using the dimensions in the detail drawing shown in Fig. 21-4. The three sizes of lettering in the title block are **.0625″**, **.125″**, and **.25″**. Use the **romans.shx** font. Replace "Bulldog Engineering" with your own name and logo. Position the text precisely and neatly. Use the snap and grid as necessary. Omit words, such as **BY** in **DRAWN BY**, and misspell a few words. Then, using AutoCAD's editing and spell-checking capabilities, fix the text.
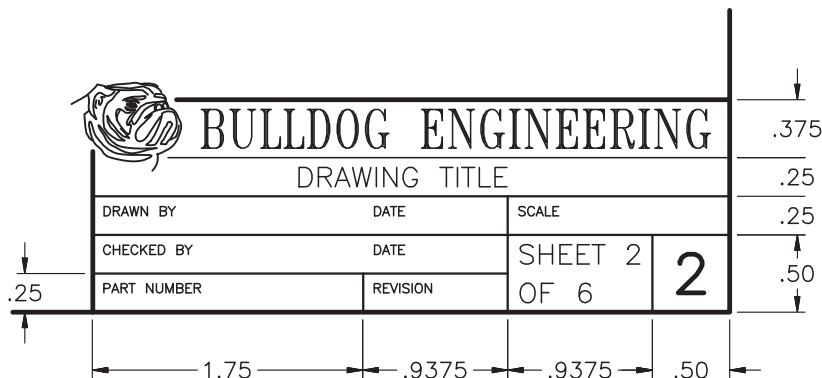
Fig. 21-4

Chapter 22

Tables

Objectives

- Create a table and populate it with text
- Apply styles to tables
- Edit tables
- Use formulas in tables to perform calculations

Vocabulary

cells
formula
table

Engineering drawings often contain a parts list or bill of materials in tabular form. These tables list the requirements for the raw materials used to manufacture parts, all the individual detail parts that make up an assembly, and other critical information. AutoCAD allows you to create tables in your drawings and provides easy editing capabilities. You can even use tables to perform calculations as you would in a spreadsheet program.

Creating Tables

A **table** is a set of rows and columns that includes text. The basic units of a table—the individual boxes within the table's grid—are called **cells**. Creating a table in AutoCAD is similar to creating one in Microsoft Word®. After creating a table, you are permitted to populate it with text.

1. Start AutoCAD and open **text.dwg**.
2. Select the **2D Drafting & Annotation** workspace.
3. Pick the **Table** button on the **Tables** panel under the **Annotate** tab of the **Ribbon,** or enter the **TABLE** command.

TABLES
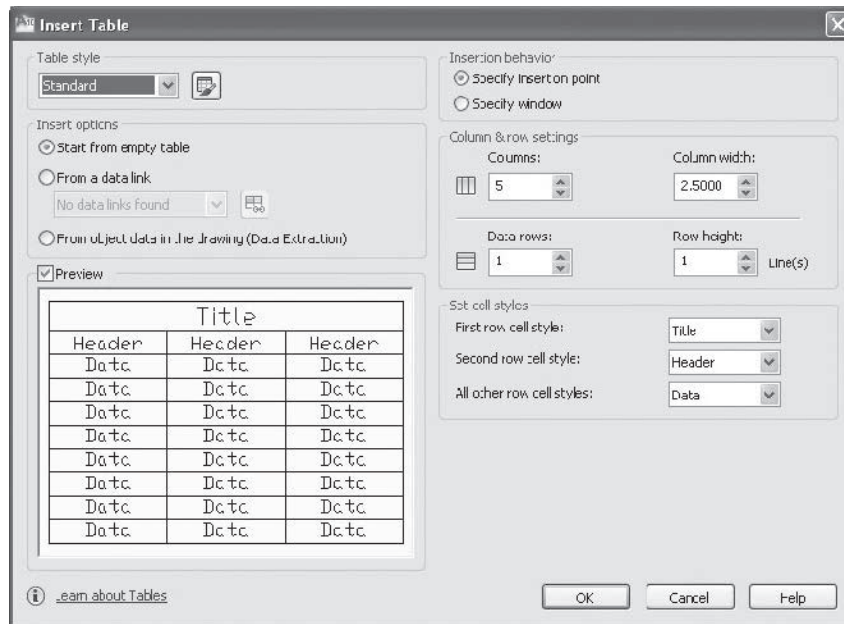
Fig. 22-1

This displays the **Insert Table** dialog box, as shown in Fig. 22-1.

4. In the **Insertion behavior** area, pick **Specify insertion point**.

5. In the **Columns & rows settings** area, enter **3** for the number of columns and **5** for the number of rows, and pick the **OK** button.

A table appears that you can drag on the screen.

6. Insert the table in an open area in the lower part of the screen. Pan and zoom as needed.

7. On the **Style** panel of the **Text Editor** tab of the **Ribbon,** select **tt** for the text style and enter **.18** for the text height.

8. Type **Fasteners** and then press the tab key.

9. Select **tt** for the text style, enter **BOLT**, and press the tab key.

10. Select **tt** for the text style, enter **.3125 x 16 x .75**, and press the tab key.

11. Select **tt** for the text style, enter **10**, and press the tab key.

12. Pick the **Close Text Editor** button to terminate the process.

CLOSE

Table Styles

In most cases, the same or similar text styles are used in all of a table's cells. AutoCAD allows you to create a table style to streamline the process of entering text.

1. Undo the insertion of the table.

2. Pick the **Table** button on the **Ribbon.**

3. In the **Table style** area of the **Insert Table** dialog box, pick the Table Style button located to the right of the down arrow.

This displays the **Table Style** dialog box.

TABLES

4. Pick the **New...** button, enter **Fasteners** for the new style name, and pick the **Continue** button.

The **New Table Style** dialog box appears, as shown in Fig. 22-2.

5. Select **Data** in the **Cell styles** text box and pick the **Text** tab.

6. Select **tt** for the text style and enter **.125** for the text height.

The **Borders** tab controls the appearance of the border lines. The **General** tab allows you to format the cells in the table, specify the alignment of the data in the cells, and control the spacing of the horizontal and vertical margins between the border of the cell and the cell contents. Under **General**, the **Table direction** text box allows you to change the table so that it reads from bottom to top instead of from top to bottom.

7. Select **Header** in the **Cell styles** text box, pick the **Text** tab, enter **tt** for the text style, and enter **.15** for the text height.

8. Select **Title** in the **Cell styles** text box, enter **tt** for the text style, and enter **.18** for the text height.
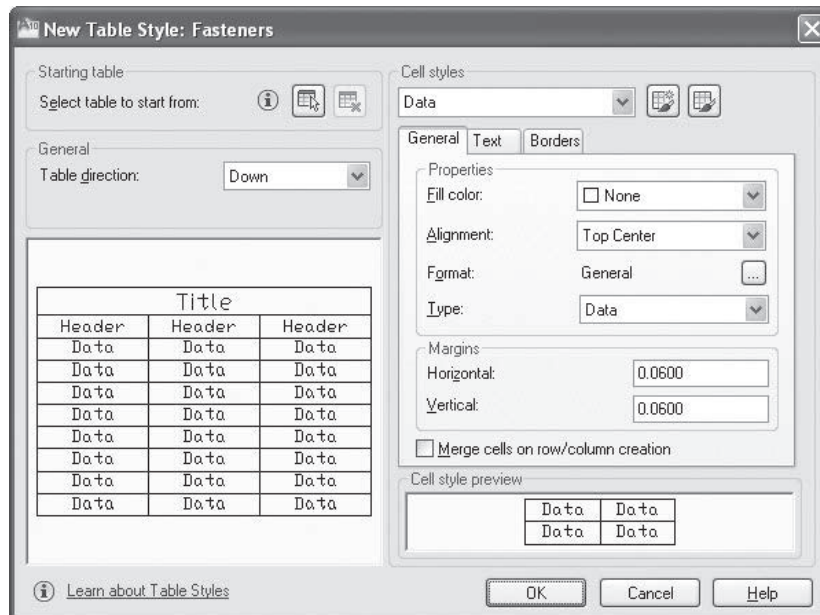
Fig. 22-2

FASTENERS		
TYPE	SIZE	QUANTITY
HEX BOLT	.375 x 16 x 1	10
HEX NUT	.375 - 16	10
PLAIN WASHER	.5 x 1 x .0625	20
CAP SCREW	.5 x 13 x 2	5

Fig. 22-3

9. Pick the **OK** button; pick the **Close** button in the **Table Style** dialog box.

10. In the **Insert Table** dialog box, select **Fasteners** for the table style and pick the **OK** button.

11. Insert the table in an open area in the lower part of the screen.

12. Enter **FASTENERS** and press the tab key.

13. Enter the text shown in Fig. 22-3 into the cells. Be sure to press the tab key to advance from one cell to the next. Click outside the table when you are finished.

14. Save your work.

Editing Tables

It is possible to edit the contents of a table. In fact, you can also change the properties of the table itself.

Editing Text in Tables

Making changes to the text within a table is very similar to editing mtext.

1. In the table, double-click **TYPE** in the first cell.

This selects the cell and displays the **Text Editor** tab on the **Ribbon.**

2. Double-click **TYPE**, this time to highlight it.

3. Type **Type** with an uppercase **T** and lowercase **y, p,** and **e.**

4. Repeat Steps 1 through 3 with **SIZE** and **QUANTITY**.

5. Double-click to activate (place a text cursor in) the left-most cell in the bottom (empty) row of cells.

6. Type **CAP SCREW** and press the tab key.

7. For the size, type **.5 x 13 x 1.5**; for the quantity, type **15**.

8. Pick the **Close Text Editor** button on the **Ribbon** or click outside the table to finish the editing session.

9. Pick the arrow in the lower right corner of the **Tables** panel on the **Ribbon.**

This displays the **Table Style** dialog box.

10. Select the **Fasteners** table style and pick the **Modify...** button.

11. Under **Cell styles**, pick **Data**.

12. In the **General** tab, pick **Middle Left** from the drop-down menu located to the right of **Alignment**.

13. In the **Text** tab, select red from the drop-down menu to the right of **Text color**.

14. Pick **OK** and close the **Table Style** dialog box.

The data text is now red and left-justified.

15. Undo your changes.

Changing the Table Border Lines

AutoCAD allows you to change the table itself, which is represented by border lines.

1. Select any one of the border lines.

This selects the entire table.

2. On the table's left-most vertical line, click the middle grip box and move it to the right to reduce the width of the column.

3. With the table still highlighted, right-click to display the shortcut menu and select **Size Columns Equally**.

AutoCAD adjusts the columns so they are of equal width.

4. Pick the arrow in the lower right corner of the **Tables** panel to display the **Table Style** dialog box and pick the **Modify...** button.

5. Under **Cell styles**, pick **Title**.

6. In the **Borders** tab, select **0.30 mm** for **Lineweight** and **Yellow** for **Color**.

7. Pick the first button at the bottom (to make the title border lines yellow), pick the **OK** button, and pick the **Close** button.

NOTE If the line weight did not change, pick the **Show/Hide Lineweight** button on the status bar.

8. Reopen the Modify Table Style dialog box and change the title border lineweight and color to **ByBlock**.

9. Save your work.

Inserting Formulas in Tables

A **formula** performs an automatic mathematical equation. You can insert simple formulas into cells in tables to calculate sums, counts, and averages.

1. In a blank area of the screen, create a table with two columns and five rows. Use the **Fasteners** table style. (It should be the current table style.)

2. Populate the cells with the text shown in Fig. 22-4.

Part Count	
Item	*Quantity*
Bolt	*8*
Washer	*16*
Nut	*8*
Spacer	*4*
TOTAL	

(**Fig. 22-4**)

3. Make the word **TOTAL** in the bottom left cell bold.

4. Select the empty cell in the lower right corner of the table.

5. Right-click and pick **Insert > Formula > Sum** from the shortcut menu.

6. In response to **Select first corner of table cell range**, click in the first cell below **Quantity**.

7. In response to **Select second corner of table cell range**, click in the cell in the number column next to **Spacer**.

The formula **=Sum(B3:B6)** appears in the cell. This formula will add the contents of cells B3 through B6.

8. Pick the **Close Text Editor** button.

The cell now contains the number 36, which is the sum of the part quantities listed in the table. Notice that the number in the cell is shaded, indicating that the cell contains a formula.

> **NOTE** You can also type formulas directly into cells. In this example, typing **=Sum(B3:B6)** directly into the cell would have yielded the same result.

9. Save your work and exit AutoCAD.

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IV. Text and Tables

22. Tables

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299

Chapter 22 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is a table in AutoCAD?
2. Describe the quickest method of editing text in a cell of a table.
3. Suppose that you created a table, but later decided to change the weight of the table's border lines. How would you go about it?
4. How many cells are in a table that contains five rows and seven columns?
5. Describe how to create a table style. Explain why this method is faster than formatting cells individually.
6. What is the purpose of a formula in a table?
7. How can you tell whether or not a number in a cell is the result of a formula?
8. Explain how you could make sure that all of the columns in a table are exactly the same width.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Other than for parts lists and bills of materials, when might you use a table in AutoCAD?
2. AutoCAD allows you to build a table with the title and headers on the bottom and the list on top. Why do you think this option exists? When would it be useful?

Continued

Chapter 22 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create the door schedule shown in Fig. 22-5 using a table style that includes cell data that is **.125** in height, column heads that are **.18** in height, and a title that is **.22** in height. All text should use the **Romans** font. Name the file **schedule.dwg**.

DOOR SCHEDULE			
QUAN.	TYPE	SIZE	REMARKS
2	FLUSH	3−0 x 6−8	HOLLOW CORE
3	FLUSH	2−8 x 6−8	HOLLOW CORE
1	FLUSH	2−6 x 6−8	SOLID CORE, OAK
3	SLIDING	4−0 x 6−8	HOLLOW CORE

Fig. 22-5

2. Using **Save As…**, create a new file from **schedule.dwg** and name the file **schedule2.dwg**. In this new file, change the text in the door schedule so that it matches the schedule in Fig. 22-6. Use the **tt** text style for the column heads and title and make all of the border lines green.

Door Schedule			
Quantity	Type	Size	Comments
2	FLUSH	2−10 x 6−8	HOLLOW CORE
4	FLUSH	2−8 x 6−8	HOLLOW CORE
1	FLUSH	3−0 x 6−8	SOLID CORE, OAK
2	BI−FOLD	5−0 x 6−8	HOLLOW CORE

Fig. 22-6

Review & Activities Chapter 22

● USING PROBLEM-SOLVING SKILLS

Complete the following activity using problem-solving skills and your knowledge of AutoCAD.

1. The **frame.dwg** file, which you created in "Using Problem-Solving Skills" problem 3 in Chapter 18, is shown in Fig. 22-7. (If you have not yet created the drawing, create it now.) Use the drawing and editing commands to insert windows and doors into the floor plan. Do research if necessary to find standard window and door widths so that you can create them accurately. Then create a table to show the sizes, kinds, and prices for the windows and doors. In the last column, show the number of each type of window or door. In the last row, use one or more formulas to find the total number of windows and doors.
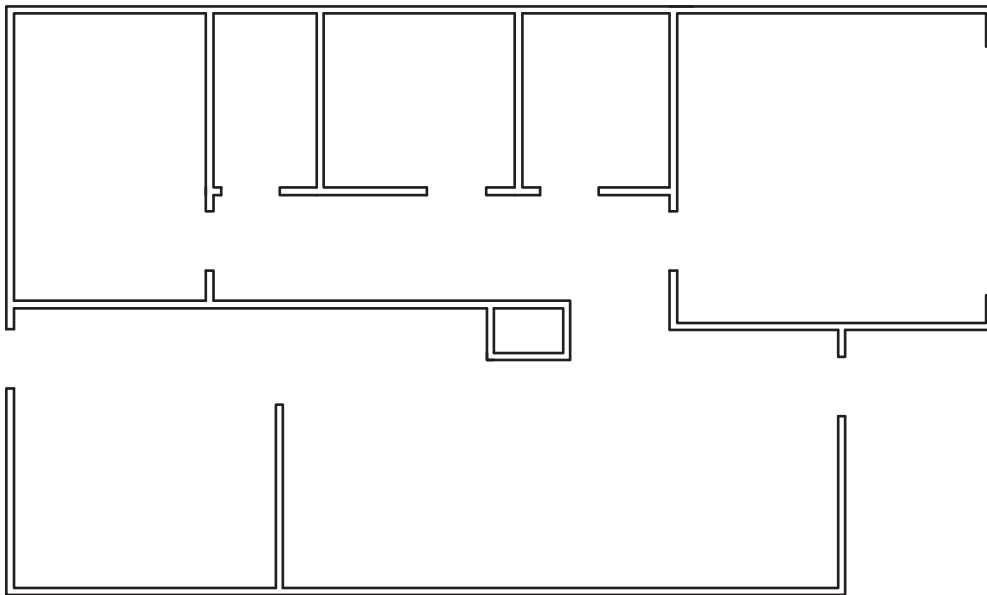
Fig. 22-7

Part 4 Project   Applying Chapters 20–22

Manufacturer's Product Table

Even in the modern electronic age, written communication remains a significant form of exchanging information. Even in a technical drawing, dimensions alone often do not convey sufficient information to enable the drawing to be used as intended. A few well-chosen words can provide enough information to minimize the number of views or necessary dimensions, making the drawing cleaner in appearance.

▼ Description

In this project, you will use written language to convey the information needed for the following scenario: A manufacturer of automotive wheels wishes to organize its product line in various ways to make it easier for consumers seeking to find a product that suits them. A decision has been made to organize the wheels in four different ways: by type of finish/color, by rim size (in diameter only—it is assumed each size is available in the most commonly needed widths), by material, and by price range.

Organization

Organize the following data into tables using AutoCAD's **TABLE** command.

- Finish/colors available include brushed aluminum and shiny aluminum; shiny and matte black; chromed steel and gunmetal gray steel (6 categories total)
- Rim sizes include diameters from 16″ to 24″ in 1″ increments (9 categories)
- Materials include aluminum, magnesium-aluminum alloy, and steel alloy (3 categories)

Price Structure

The price structure, per wheel, is as follows: aluminum 16″ wheels start at $299, magnesium-aluminum alloy wheels at $349, and steel alloy wheels at $229. For each line of wheel, the price increases by 5% (of the base price) for each incremental increase in diameter. Thus an aluminum wheel costs $299 plus 5% of $299 for a 17″ rim; an 18″ aluminum wheel goes for $299 plus 10% of $299, and so on. There is no distinction in price for color or finish.

Fig. P4-1

Accompanying Drawings

Create drawings of one or more styles of wheel that might be available. If you would like to see some of the many wheel designs being sold, simply do an Internet search for "automobile wheels." See Fig. P4-1 for other ideas.

▼ Hints and Suggestions

1. Consider including instructions for consumers on how to use the information in the table to find and calculate the cost of their order.

2. AutoCAD can insert formulas directly into tables. Your instructor may therefore ask you to include, for example, the potential to calculate taxes for the purchase of one or more wheels.

3. Make sure each table and column heading is clearly labeled.

4. Take advantage of text editing features such as bulleted and numbered lists.

5. When illustrating the wheels, consider using polar arrays.

▼ Summary Questions

Your instructor may direct you to answer these questions orally or in writing. You may wish to compare and exchange ideas with other students to help increase efficiency and productivity.

1. Why might it be preferable to use a table instead of a list or graph to convey this information?

2. How would a company benefit by facilitating the ability of consumers to initiate and price out their own orders?

Chapter 23

Drawing Setup

Objectives

- Explain the purpose of a template file and list settings that are commonly included
- Choose the appropriate unit of measurement for a drawing
- Determine appropriate sheet size and drawing scale for a drawing
- Check the current status of a drawing file

Vocabulary

drawing area
limits
template file

The first 22 chapters of this book have provided a foundation for using AutoCAD. Now it is time to apply many of the pieces. As part of this effort, you will identify the scale and sheet size for a particular drawing. In doing this, you will determine the drawing units and drawing area—key elements of the drawing setup process. You will also consider settings of features such as the grid and snap that you can store in a template file and use over and over again.

Template Files

In AutoCAD, a **template file** is a file that contains drawing settings that can be imported into new drawing files. This feature is helpful to people who need to create the same types of drawings (drawings with similar settings) frequently. The purpose of a drawing template file is to minimize the need to change settings each time you begin a new drawing. Some users and

Table 23-1. Template Development

Step	Description
1.	Determine what you are going to draw (mechanical detail, house elevation, etc.).
2.	Determine the drawing scale.
3.	Determine the sheet size. (Steps 2 and 3 are normally done simultaneously.)
4.	Set the drawing limits.
5.	Set the drawing area.
6.	Set the grid.
7.	ZOOM All. (This will zoom to the new drawing area.)
8.	Set the snap resolution.
9.	Enter STATUS to review the settings.
10.	Determine how many layers you will need and what information will be placed on each layer; establish the layers with appropriate colors, linetypes, lineweights, etc.
11.	Set the linetype scale.
12.	Create new text styles.
13.	Set the dimension scale, dimension text size, arrow size, etc.
14.	Store as a drawing template file.

companies include a standard border and title block in their template files. This further shortens the time they spend preparing for a new drawing.

Once you have set up a template file, subsequent drawing setup for similar drawings is quick and easy. When you use a template, its contents are automatically loaded into the new drawing. The template's settings thus become the settings for the new drawing. Template development includes the steps shown in Table 23-1. This and the next several chapters will provide an opportunity to practice these steps in detail.

Using a Predefined Template File

AutoCAD provides many standard templates that can be used as is or modified to fit individual needs.

1. Start AutoCAD and pick the **Use a Template** button on the **Startup** dialog box.

AutoCAD displays a list of template files available to you. Notice that all template files have a **DWT** file extension.

2. Scroll down the list and single-click several of the files, noting the files with borders and blocks.

Notice that the names of a few of the templates contain "**3d**." These templates are for 3D modeling. You will learn more about 3D modeling and these templates in Parts 8 and 9 of this book.

3. Find and double-click **Tutorial-imfg.dwt**.

The contents of the template file display on the screen in the default **2D Drafting & Annotation** workspace and the **D-Size Layout** tab. This template is set up to fit a D-size drawing sheet according to standards established by the American National Standards Institute (ANSI).

4. Pick the **Model** tab at the button of the drawing area.

As you can see, the **Model** space does not display the border and title block. You will learn more about **Model** (model space) and **Layout** (paper space) in Chapter 26.

Initial Template Setup

Many AutoCAD users choose to create custom template files. We will create one also. The first step is to identify the type of object (mechanical part, building, etc.) for which the new template will be used. For this exercise, let's create a template that can be used for stair details in drawings for homes and commercial buildings.

Next, we will determine the drawing scale for the stair detail. In this sense, a scale is a means of reducing or enlarging a representation of an actual object or part that is too small or too large to be shown on a drawing sheet. This information will give us a basis for establishing the drawing area, the linetype scale, and the scale for the dimensions. Let's use a $1/2'' = 1'$ scale, and let's base the template on a sheet size of $8.5'' \times 11''$.

1. Close the current drawing (do not save changes), and pick the **New** button from the **Quick Access** toolbar.

2. Pick the **Use a Wizard** button.

QUICK ACCESS

This begins our setup of the template file. We will create and save it as a regular drawing file. Then, when we have finished defining the settings, we will save it as a template file.

3. Read the description of the **Advanced Setup** wizard.

Advanced Setup uses the template file **acad.dwt**. So does **Quick Setup**.

4. Pick the **OK** button to proceed with the **Advanced Setup**.

Advanced Setup consists of five steps: **Units**, **Angle**, **Angle Measure**, **Angle Direction**, and **Area**. Completing these steps as part of the process of creating a new file can simplify part of the template setup process.

Specifying the Unit of Measurement

The unit of measurement you choose for the template file depends on the types of drawings for which you will use the template. AutoCAD provides several different types of units from which to choose.

1. Pick the **Architectural** radio button and review the sample drawing located at the right.

2. Under **Precision**, pick the down arrow and review the list.

3. Choose the **0′-0 1/16″** default setting.

This means that $1/16″$ is the smallest fraction that AutoCAD will display.

4. Pick the **Next** button to go to the next step.

NOTE

If you choose not to use a wizard to set up your drawing files and templates, you can set up the drawing units using AutoCAD's **UNITS** command. This command displays the **Drawing Units** dialog box, from which you can set drawing units and precisions for lengths and angular units.

Setting Angle Measurements

Decimal Degrees is the default setting for angle measurements. This is what we want to use for our template, so no selection is required. When **Decimal Degrees** is selected, AutoCAD displays angle measurements using decimals.

1. Under **Precision**, pick the down arrow to adjust the angle precision, and pick **0.0**.

A setting of **0.0** means that AutoCAD will display angle measurements to one decimal place, as shown in the sample.

2. Pick the **Next** button to proceed to the next step.

The next dialog box permits you to control the direction for angle measurements. As discussed earlier in the book, AutoCAD assumes by default that 0 degrees is to the right (east). Let's not change it.

3. Pick the **Next** button.

We also established that angles increase in the counterclockwise direction by default. Do not change this, either.

4. Pick the **Next** button to proceed.

Setting the Drawing Area

The next step is to set the **drawing area**, or **limits**, of the drawing. The drawing area defines the boundaries for constructing the drawing, and it should correspond to both the drawing scale and the sheet size. The default drawing area is 12×9 units. Using architectural units, the drawing area is $1' \times 9''$. (See Appendix C for a chart showing the relationships among sheet size, drawing scale, and drawing area.)

> **NOTE**
>
> Actual scaling does not occur until you plot the drawing, but you should set the drawing area to correspond to the scale and sheet size. The drawing area and sheet size can be increased or decreased at any time using the **LIMITS** command. The plot scale can also be adjusted prior to plotting. For example, if your drawing will not fit on the sheet at $1/4'' = 1'$, you can enter a new drawing area to reflect a scale of $1/8'' = 1'$. Likewise, you can enter $1/8'' = 1'$ instead of $1/4'' = 1'$ when you plot.

As mentioned before, the drawing area should reflect the drawing scale and sheet size. Let's look at an example.

If the sheet size is $11'' \times 8.5''$ and the drawing scale is $^{1}/4'' = 1'$, what is the drawing area? Since each plotted inch on the sheet would occupy 4 scaled feet, it's a simple multiplication problem: $11 \times 4 = 44$ and $8.5 \times 4 = 34$. The drawing area, therefore, would be $44' \times 34'$ because each plotted inch represents $4'$.

Since our scale is $^{1}/2'' = 1'$, what should the drawing area be?

HINT   How many $^{1}/2''$ units would $11''$ occupy, and how many would $8.5''$ occupy?

1. Enter **22**′ for the width and **17**′ for the length. Be sure to include the apostrophe for the foot mark.
2. Pick the **Finish** button.

You have completed AutoCAD's **Advanced Setup** process.

3. Save your work in a file named **tmp1.dwg**.

Establishing Other Settings

Let's continue setting up the template file by setting the grid and snap and zooming out to display the new drawing area.

1. Enter the **GRID** command and set the grid at **1**′. (Be sure to enter the apostrophe.)

The purpose of setting the grid is to give you a visual sense of the size of the objects and the drawing area. After you enter **ZOOM All** in the next step, the grid will fill the drawing area, with a distance of $1'$ between grid dots.

2. Enter **ZOOM All**.

The grid reflects the size of the sheet.

3. Enter **SNAP** and set it at **6**″.
4. Position the crosshairs in the upper right corner of the grid and review the coordinate display.

It should read approximately **22′-0″, 17′-0″, 0′-0″**. The **0′-0″** is the z coordinate.

Status of the Template File

Let's take a moment to review the settings we have created up to this point.

1. Select **Show Menu Bar** from the **Quick Access Toolbar's** pull-down menu to display the **Menu Bar.**

2. Select **Tools** on the **Menu Bar** and pick **Inquiry** and **Status**.

This enters the **STATUS** command. Note each of the components found in **STATUS**.

3. Close the **AutoCAD Text Window.**

Tmp1 now contains several settings specific to creating architectural drawings at a scale of $1/2'' = 1'$. It will work well for the stairway detail mentioned earlier.

We have developed the basis for a template file. Technically, we have not yet created the template, because we have not stored the contents of our work in a template (**DWT**) file. We will do this later when we continue its development.

The next steps in creating a template file deal with establishing layers. You will create layers in the next chapter and complete the final steps for creating a template file in a subsequent chapter.

4. Save your work and exit AutoCAD.

5. Produce a backup copy of the file named **tmp1.dwg**.

Backup copies are important because if you accidentally lose the original (and you may, sooner or later), you will have a backup. You can produce a backup copy in seconds, and it can save you hours of lost work. If you don't remember how to make a backup copy, see Chapter 9 for details on producing copies of files.

Chapter 23 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Explain the purpose and value of template files.
2. What settings are commonly included in a template file?
3. If you select architectural units, what does a precision of **0′-0 1/4″** mean?
4. What information is used to determine the size of the drawing area?
5. If the sheet measures 22″ × 16″ and the scale is 1″ = 10′, what should you enter for the drawing area?
6. Describe the information displayed as a result of entering the **STATUS** command.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Discuss unit precision in your drawings. If necessary, review the precision options listed in the **Drawing Units** dialog box for units and angles. (*Hint:* You can display this dialog box by entering the **UNITS** command.) Then describe applications for which you might need at least three of the different settings.
2. For a drawing to be scaled at $1/8″ = 1′$ and plotted on a 17″ × 11″ sheet, what drawing area should you establish?

Continued

Chapter 23 **Review & Activities**

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create a new drawing using the **Acad -named plot styles.dwt** template file. Use the **STATUS** command to review its settings. Create another new drawing using the **Acadiso -named plot styles.dwt** template file and review its settings also. What differences do you see between AutoCAD's templates for ANSI and ISO standards?

2. Establish the settings for a new drawing based on the information below, setting each of the values as indicated. Save the file as **prb23-2.dwg**.

   Scale: $1'' = 2''$
   Sheet size: $17'' \times 11''$
   Units: Decimal
   Drawing area: (You determine it.)
   Grid value: .5″
   Snap resolution: .25″

   Be sure to **ZOOM All**.

   Review settings with the **STATUS** command.

3. Establish the settings for a new drawing based on the information below, setting each of the values as indicated. Save the file as **prb23-3.dwg**.

   Scale: $^1/_8'' = 1''$
   Sheet size: $24'' \times 18''$
   Units: Architectural (You choose the appropriate options.)
   Drawing area: (You determine it.)
   Grid value: 4″
   Snap resolution: 2′

   Reminder: Be sure to **ZOOM All**.

   Review settings with the **STATUS** command.

4. Consider the following drawing requirements:

   Drawing type: Architectural drawing of a detached garage
   Dimensions of garage: $32' \times 20'$

   Other considerations: Space around the garage for dimensions, notes, specifications, border, and title block

   Based on this information, write the missing data for drawing setup on a separate sheet of paper. Suggest values for scale, paper size, units, drawing area, grid, and snap resolution.

Review & Activities Chapter 23

5. Consider the following drawing requirements:

Drawing type: Mechanical drawing of the wheel and bearings for an in-line skate

Approximate diameter of wheel: 2.5″

Other considerations: Space for dimensions, notes, specifications, border, and title block

Based on this information, write the missing data for drawing setup on a separate sheet of paper. Suggest values for scale, paper size, units, drawing area, grid, and snap resolution.

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Your architectural office needs a drawing template setup created that will accommodate the floor plans of houses measuring 16 m × 8.4 m. Create the template. In addition to the units, angles, and area, specify reasonable settings for both snap and grid. Save the template as **arch1.dwt**.

2. Electronic engineering requires large drawings of small parts. Set up a drawing for electronic circuits measuring $1/4″ \times 3/32″$. Use decimal settings, and set the appropriate snap and grid. Save the template as **elec.dwt**.

AutoCAD at Work

Landscape Designer

Parks and outdoor recreation centers are perfect locations for picnics, Frisbee games, and other social activities. The next time you visit your local park, try to take notice of the surroundings: healthy green grass, immaculate flower beds, a ball diamond or two, a playground, and perhaps even a water fountain for kids to splash in. The design and placement of such items, all of which make up a visually appealing outdoor atmosphere, require strategic planning.

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Working Together

Landscape designers typically work closely with a landscape architect. While architects supply practical and artistic views of how they want to arrange outdoor items, landscape designers possess the knowledge to create the actual plans for the design. Before construction begins, the landscape designer must prepare detailed drawings for plans that result in landscapes that are both attractive and economical.

Employment Opportunities

Large development corporations, private sectors, and government agencies need people with landscape design skills. Landscape designers create plans for golf courses and college campuses. They also design outdoor lighting systems and irrigation systems.

All landscape designers must be able to produce accurate drawings based on conceptualizations and rough sketches. It is also important for landscape designers to understand mathematical and measurement concepts, since the plans must be drawn to scale.

▶ Career Activities

1. Interview someone who can tell you about landscape designers in your area. Are apprenticeships available?

2. Do you think landscape designers should have a background in ecology and science? Why?

Layers and Linetypes

Objectives

- Create layers with appropriate characteristics for the current drawing or template
- Use layers to control the visibility and appearance of objects in a drawing
- Change an object's properties
- Use object properties to filter object selections
- Apply a custom template file

Vocabulary

center lines
color palette
filtering
freeze
layers
phantom lines

AutoCAD gives you the option of separating classes of objects into layers. It may be helpful to think of AutoCAD's **layers** as transparency film that helps you organize your drawing.

One of the benefits of using layers is the ability to make them visible and invisible. For example, you can place construction lines and reference notes on a layer and then turn the layer off when you're not using it. A house floor plan could be drawn on a layer called **Floor** and displayed in red. The dimensions of the floor plan could be drawn on a layer called **Dimension** and displayed in yellow. A layer called **Center** could contain blue center lines.

Table 24-1 on page 310 shows an example set of layers. Note the layer names, colors, linetypes, and lineweights. We will create these layers in this chapter.

Table 24-1. Typical Layers

Layer Name	Color	Linetype	Lineweight
0	white	Continuous	Default
Border	cyan	Continuous	0.50 mm
Center	magenta	Center	0.20 mm
Dimensions	blue	Continuous	0.20 mm
Hidden	green	Hidden	0.30 mm
Notes	magenta	Continuous	0.30 mm
Objects	red	Continuous	0.40 mm
Phantom	yellow	Phantom	0.50 mm

NOTE Be sure to make a backup copy of **tmp1.dwg** before you begin working with it in this chapter if you have not already done so.

Creating New Layers

LAYERS

1. Start AutoCAD and open the drawing named **tmp1.dwg**.
2. Select the **2D Drafting & Annotation** workspace.
3. Select the **Home** tab of the **Ribbon** and pick the **Layer Properties** button on the **Layers** panel, or enter **LA**, the command alias for the **LAYER** command.

This enters the **LAYER** command and displays the **Layer Properties Manager** palette. First, let's create layer **Objects**.

4. Pick the **New Layer** button near the top center of the palette.

This creates a new layer with a default name of **Layer1**. Notice that the name **Layer1** is highlighted and a cursor appears in the edit box.

5. Using upper- or lowercase letters, type **Objects** and press **ENTER**.

The name **Objects** replaces **Layer1**. If you make a mistake, you can single-click the layer name and edit it.

6. Create the layers **Border**, **Center**, **Dimensions**, **Hidden**, **Notes**, and **Phantom** on your own, as listed in Table 24-1. (You will set the color, linetype, and lineweight of each layer later in this chapter.)

Changing the Current Layer

Let's change the current layer to **Objects**.

1. Pick **Objects** and pick the **Set Current** button.

> **NOTE**
>
> The **Set Current** button is the green check mark at the top of the palette. You can also make a layer current by double-clicking the icon in the **Status** column next to the layer name.

Objects is now the current layer.

2. Close the **Layer Properties Manager** palette.

Notice that **Objects** now appears in the **Layer** text box on the **Layers** panel.

3. Save your work.

Assigning Colors

The **Layer Properties Manager** permits you to assign screen colors to the layers.

LAYERS

1. Pick the **Layer Properties** button or enter **LA**.

2. Find the **Color** heading, which is located at the left of the **Linetype** column heading. If the **Color** column heading and the color names are visible, skip to Step 5. If not, proceed to Steps 3 and 4.

3. Position the pointer between the two headings until the pointer changes to a double arrow.

4. Click and drag to the right until the names of the colors appear.

5. At the right of **Objects**, pick the black box under the **Color** heading.

The **Select Color** dialog box appears. It contains a color palette. A **color palette** is a selection of colors, similar to an artist's palette of colors.

6. Select the **True Color** tab.

This offers a choice of more than 16 million colors.

7. Select the **Color Books** tab.

This allows you to specify colors using third-party color books, such as PANTONE®, or user-defined color books.

8. Move the vertical slider bar up and down and select the down arrow in the **Color book** area to review options.

9. Select the **Index Color** tab.

10. In the bottom half of the dialog box, pick the color red (next to yellow) and pick the **OK** button.

AutoCAD assigns the color red to **Objects**.

11. Assign colors to the other layers as indicated in the layer listing in Table 24-1 on page 310. Cyan is light blue and magenta is purple.

12. When you have finished assigning the colors, auto-hide the **Layer Properties Manager** palette and save your work.

Notice that the color red appears beside **Objects** in the **Layers** panel.

AutoCAD's **COLOR** command allows you to set the color for subsequently drawn objects, regardless of the current layer. This allows you to set the color of each object individually, giving you a great deal of flexibility, but it can become confusing. It is recommended that you avoid use of the **COLOR** command and that its setting remain at **ByLayer**. The **ByLayer** setting means that the color is specified by the layer on which you draw the object.

The **Color Control** drop-down box on the **Properties** panel also allows you to set the current color. The recommended setting is **ByLayer**.

13. Draw a circle of any size on the current **Objects** layer.

It should appear in the color red.

14. Select **Hidden** from the **Layer** drop-down box in the **Layers** panel to make **Hidden** the current layer.

15. Draw a concentric circle inside the first circle.

It should appear in the color green.

Assigning Linetypes

Various types of lines are used in drafting to show different elements of a drawing. By convention, for example, hidden lines (those that would not be visible if you were looking at the actual object) are shown as dashed or broken lines. **Center lines** (imaginary lines that mark the exact center of

an object or feature) are shown by a series of long and short line segments. Let's look at and load some of the different linetypes AutoCAD makes available to you.

1. Display the **Layer Properties Manager** palette.
2. At the right of layer **Hidden**, and under the **Linetype** heading, pick **Continuous**.

This displays the **Select Linetype** dialog box.

3. Pick the **Load...** button.

This displays the **Load or Reload Linetypes** dialog box, as shown in Fig. 24-1.

AutoCAD stores the list of linetypes in a file named **acad.lin**, as listed in the box at the right of the **File...** button. These linetypes conform to International Standards Organization (ISO) standards. This is important because ISO is the leading organization for the establishment of international drafting standards.
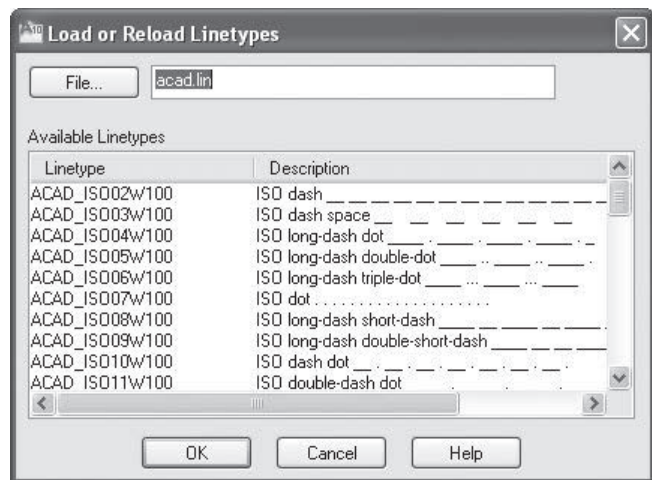
Fig. 24-1

At this point, you can select the individual linetypes that you want to load and use in the current drawing.

4. Using the scroll bar, review the list of linetypes.
5. Find and select the **CENTER**, **HIDDEN**, and **PHANTOM** linetypes.

HINT

Press the **CTRL** key when selecting them. This allows you to make multiple selections.

Hidden lines are used to show invisible edges on drawings. This is why they are called "hidden." Center lines are used to show the centers of holes, cylinders, rounded corners, and fillets. In AutoCAD, lineweights are applied to **phantom lines** based on their use. When used for cutting planes in sectional views, a thick lineweight is applied.

6. Pick the **OK** button in the **Load and Reload Linetypes** dialog box and notice the new list of linetypes in the **Select Linetype** dialog box.

7. Pick **HIDDEN** for the linetype and pick **OK**.

8. Auto-hide the **Layer Properties Manager** palette.

The inner circle on layer **Hidden** changes from a continuous line to a hidden line. However, it's unlikely that you will be able to see the hidden line until after we scale the linetypes.

9. Save your work.

> **NOTE**
>
> You can also load linetypes using the **LINETYPE** command. Pick the **Linetype…** item from the **Format** menu in the **Menu Browser** or enter **LT** (for **LINETYPE**) and pick the **Load** button.

Scaling Linetypes

The **LTSCALE** command permits you to scale the linetypes so that they are correct for the drawing scale.

1. Enter **LTS** for the **LTSCALE** command.

Let's scale the linetypes to correspond to the scale of the template drawing. This is done by setting the linetype scale at $1/2$ the reciprocal of the plot scale. When you do this, broken lines, such as hidden and center lines, are plotted to ISO standards.

> **NOTE**
>
> As you may recall, we are creating a template file based on a scale of $1/2'' = 1'$. Another way to express this is $1'' = 2'$ or $1'' = 24''$. This can be written as $1/24$. The reciprocal of $1/24$ is 24, and half of 24 is 12. Therefore, in this particular case, you should set **LTSCALE** at **12**.

2. In response to **Enter new linetype scale factor**, enter **12**.

The inner circle should now appear in the hidden linetype.

3. If you are not sure, zoom in on it.

4. Display the **Layer Properties Manager** palette.

5. Assign the **CENTER** linetype to layer **Center** and the **PHANTOM** linetype to layer **Phantom**.

6. Save your work.

> **NOTE**
>
> The **Linetype** drop-down box, located on the **Properties** panel, permits you to change the current linetype regardless of the current layer. This gives you a great deal of flexibility, but it can become confusing. It is recommended that you leave the linetype at its default setting of **ByLayer**.

Setting Lineweights

Lineweights are important in technical drawing. For example, object lines should be thicker so they stand out more than dimension lines. Normally, object lines are thick, hidden lines and text are of medium thickness, and center lines, dimensions, and hatch lines are thin. The exact lineweights vary from industry to industry and from company to company. In AutoCAD, the default lineweight is .25 mm.

1. Display the **Layer Properties Manager** palette.

2. At the right of layer **Objects**, and under the **Lineweight** heading, pick the word **Default**.

This displays the **Lineweight** dialog box, as shown in Fig. 24-2.

3. Scroll down the list to review the lineweight options.

4. Pick **0.40 mm** and pick **OK**.

This assigns a lineweight of 0.40 mm to layer **Objects**. This lineweight replaces the word **Default** in the **Layer Properties Manager** palette.

5. Pick the **Show/Hide Lineweight** button on the status bar.

This turns on the display of lineweights in the drawing area.

6. Pick the **Show/Hide Lineweight** button to toggle it off; toggle it back on.
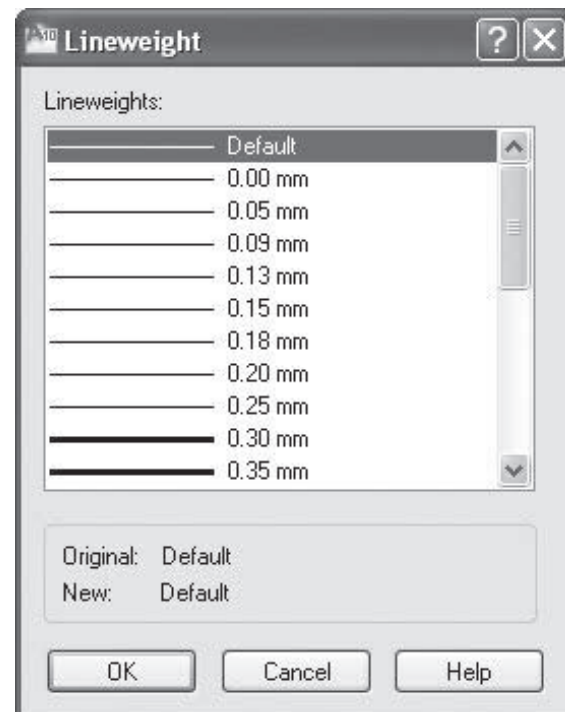
Fig. 24-2

7. Assign lineweights to the other layers, as shown in Table 24-1 on page 310.

8. Auto-hide the **Layer Properties Manager** palette, and save your work.

> **NOTE**
> The **Lineweight** drop-down box on the **Properties** panel allows you to set the lineweight for objects regardless of layer. However, it is recommended that you leave the lineweights at their default setting of **ByLayer** to avoid confusion.

Working with Layers

Setting up the appropriate layers for a drawing gives you much more control over the drawing. It also gives you more flexibility. For example, suppose you are working on the plans for a new residence. You can set up separate layers for plumbing, electrical, and other work that is often subcontracted. By placing the plans for these elements on separate layers, you can use the same drawing to generate plans for each of the subcontractors. For the electrical plans, you can turn off or **freeze** the plumbing layer so that the plumbing specifications don't clutter up the electrical drawing, and so on.

Turning Layers On and Off

1. Display the **Layer Properties Manager** palette.

2. At the right of **Objects**, click the light bulb.

This toggles off layer **Objects**.

3. Auto-hide the palette and notice that the red circle disappears.

4. Display the **Layer Properties Manager** palette.

5. Toggle on layer **Objects** by clicking the darkened light bulb.

6. Notice that the red circle reappears.

Freezing and Thawing Layers

1. Display the **Layer Properties Manager** palette and make **Hidden** the current layer.

2. In the **Freeze** column, and at the right of **Objects**, pick the symbol representing the sun.

This changes the symbol to a snowflake and freezes the layer.

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V. Preparing and Printing a
Drawing

24. Layers and Linetypes

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323

3. Move the pointing device to auto-hide the palette and notice what has happened to the circle.

It has disappeared.

4. Display the **Layer Properties Manager** palette again, and change the snow-flake into a sun by picking it.

The circle reappears. As you can see, freezing and thawing layers is similar to turning them off and on. The difference is that AutoCAD regenerates a drawing faster if the unneeded layers are frozen rather than turned off. Therefore, in most cases, **Freeze** is recommended over the **Off** option. Note that you cannot freeze the current layer.

Locking Layers

AutoCAD permits you to lock layers as a safety mechanism. This prevents you from editing objects accidentally in complex drawings.

1. Display the **Layer Properties Manager** palette.

2. At the right of **Objects**, click the symbol that looks like a padlock.

The lock closes, and the red circle appears less bright, indicating that the layer is now locked.

3. Try to edit the circle.

A locked padlock appears next to the crosshairs. AutoCAD will not permit you to edit the circle because it now resides on a locked layer.

> **NOTE** AutoCAD does allow you to create new objects on a locked layer. However, once you have created them, you cannot edit them.

4. Redisplay the **Layer Properties Manager** palette, and unlock layer **Objects** by clicking the locked padlock.

Managing Layers

There are several ways to control layers, layer properties, and the current layer in AutoCAD. The **Make Object's Layer Current** button allows you to change the current layer based on a selected object. The **Layer Properties Manager** allows you to change layer properties and delete unused layers.

1. Pick the **Make Object's Layer Current** button on the **Layers** panel of the **Ribbon.**

LAYERS

Notice that the AutoCAD prompt reads **Select object whose layer will become current**.

2. Pick the red circle.

Objects is now the current layer.

3. Display the **Layer Properties Manager** palette and pick the **New Property Filter** button located in the upper left area.

This displays the **Layer Filter Properties** dialog box. This dialog box is available to filter layers based on name, state, color, and linetype. **Filtering** means selectively including or excluding items using specific criteria. Filtering layers is especially useful in complex drawings that contain a large number of layers.

4. Pick the **Show example** button located in the upper right area, review the layer filter examples, and then close the box.

5. Pick **Cancel** to close the **Layer Filter Properties** dialog box.

The **New Group Filter** button creates a layer filter containing layers that you select and add to the filter. The **Layer States Manager** button displays the **Layer States Manager**. This dialog box enables you to save the current property settings for layers in a named layer state that you can later restore.

6. Pick the **Objects** layer and then right-click to display the shortcut menu.

7. Select **Change Description** from the shortcut menu.

This produces a text box under the **Description** column.

8. Enter **Object lines** for the description and press **ENTER**.

The description can be a helpful reference when you use these layers in the future.

Focus your attention on the **Plot Style** and **Plot** headings. Colors appear under **Plot Style**. These are the colors you see in the drawing area. Color **1** is red, color **2** is yellow, and so on, as shown. They are grayed out and unavailable because this drawing does not use plot styles.

Under the **Plot** heading, you can pick the printer icon to turn off plotting for a layer. When plotting is turned off, AutoCAD still displays the objects on that layer, but they do not plot.

9. Pick the printer icon located at the right of layer **Center**.

A red symbol appears on top of the printer, indicating that plotting has been turned off for this layer. You will learn more about plotting in the following chapter.

10. Turn on plotting for layer **Center** and auto-hide the palette.

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V. Preparing and Printing a
Drawing

24. Layers and Linetypes

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325

Changing and Controlling Layers

AutoCAD offers a convenient way of changing and controlling certain aspects of layers.

1. In the **Layers** panel of the **Ribbon,** pick the down arrow to display the **Layer** drop-down box.

This displays the list of layers in the current drawing. By picking the symbols, you can quickly change the current status of the layers.

2. Pick **Center**.

Layer **Center** becomes the current layer.

3. Display the list again.

4. Beside **Objects**, click the padlock.

The padlock changes to its locked position, and layer **Objects** becomes locked.

5. Pick it again to unlock layer **Objects**.

6. Beside **Hidden**, pick the sun.

The sun changes to a snowflake. Layer **Hidden** is now frozen.

7. Display the list and pick the snowflake.

This thaws layer **Hidden**. Notice that you can toggle layers on and off and freeze and thaw layers using buttons on the **Layers** panel.

8. Make **Objects** the current layer and save your work.

Working with Objects

Now that you have created a number of layers and assigned properties to them, you have much more flexibility in working with objects in the drawing. You can control their appearance and style by placing them on appropriate layers. You can also change an object's properties and even filter objects according to their characteristics.

Changing an Object's Properties

The **Properties** palette provides a quick method for reviewing and changing the properties of an object.

326

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AutoCAD® 2010

V. Preparing and Printing a
Drawing

24. Layers and Linetypes

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1. Double-click the green circle.

This displays the **Properties** palette. The **Properties** palette provides a lot of information about the circle, including its color, linetype, lineweight, and layer name. Suppose you created the object on the wrong layer—something you will certainly do in the future—and want to move it to a different layer.

2. At the right of **Layer**, pick **Hidden**.

This produces a down arrow.

3. Pick the down arrow to display the list of layers and pick **Center**.
4. With the cursor anywhere in the drawing area, press **ESC** to remove the selection.

The circle now resides on layer **Center**.

5. Using the same method, change the same circle back to layer **Hidden**.
6. Close the **Properties** palette.

The **Properties** palette is a handy way of quickly making changes to the objects in the drawing.

Matching Layer Properties

CLIPBOARD

The **Match Properties** button provides a quick way to copy the properties of one object to another.

1. Pick the **Match Properties** button on the **Clipboard** panel of the **Ribbon.**

This enters the **MATCHPROP** command.

2. In reply to **Select source object**, pick the red circle.

Notice the change to the pointer.

3. In reply to **Select destination object(s)**, pick the green circle.

This copies the properties of the first object to the second object.

4. Press **ENTER** to terminate the command.
5. Undo the last operation.

PROPERTIES

6. Pick the **Match Properties** button again and pick the green circle.
7. Enter **S** for **Settings**.

This displays the **Property Settings** dialog box. The checked items are copied from the source object to the destination object(s), allowing you to control the properties that are copied.

Table 24-2. Layers for Stair Detail

Layer Name	Color	Linetype	Lineweight
Risers	yellow	Continuous	0.30 mm
Treads	green	Continuous	0.60 mm

8. Pick the **Cancel** button in the dialog box and press **ENTER** to terminate the command.

9. Close the **Layer Properties Manager** palette.

10. Erase both circles and save your work, but do not close the drawing.

Filtering by Object Property

The **Quick Select** dialog box allows you to filter a selection set based on an object's properties. You can use the **Quick Select** dialog box with the **Properties** palette to make changes quickly to a large number of objects.

1. Start a new drawing by picking the **New** button on the **Quick Access** toolbar or by typing **NEW** at the keyboard. Use the **Quick Setup** wizard to specify **Architectural** units and a drawing area of **17′ × 11′**.

2. Create two new layers as shown in Table 24-2 and **ZOOM All**.

3. Create the stair detail shown in Fig. 24-3. Place the risers on layer **Risers** and the treads on layer **Treads**. (The *risers* are the vertical lines, and the *treads* are the horizontal lines.)

4. Zoom in so that the stairs fill most of the screen.

5. Save your work in a file named **stairs.dwg**.
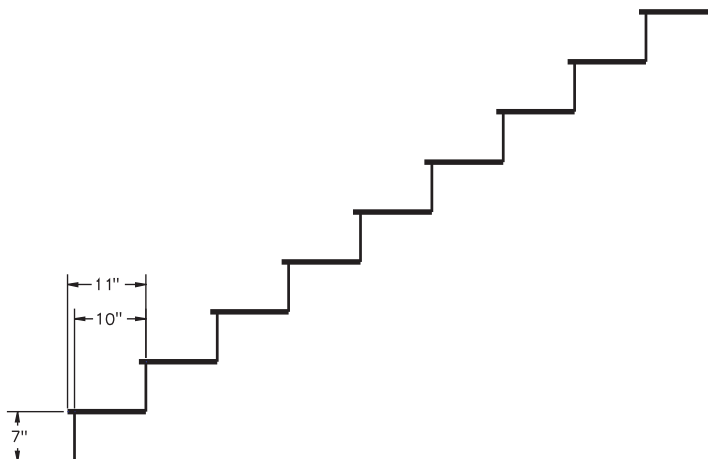
Fig. 24-3

UTILITIES

Suppose your supervisor reviews the drawing and asks you to place both the treads and the risers on the **Risers** layer. However, you must keep the differences in lineweight and color for display purposes.

6. Pick the **Quick Select** button on the **Utilities** panel of the **Ribbon.**

This displays the **Quick Select** dialog box. The **Properties** box lists several properties by which you can select objects.

7. Pick **Layer** in the **Properties** selection box.

8. Pick a point in the **Value** box to see a list of layers, and pick **Treads**.

We will leave the rest of the values at their default values. The operator is an equal sign, and the radio button for **Include in new selection set** is selected. This means that all the lines in the drawing that are on layer **Treads** will be included in the selection set.

Properties

PALETTES

9. Pick **OK**.

All of the stair treads are selected in a single operation.

10. In the **View** tab of the **Ribbon,** pick the **Properties** button on the **Palettes** panel.

11. Pick **Layer** in the **Properties** panel, pick the down arrow to see a list of layers, and choose **Risers**.

12. Auto-hide the **Properties** palette and press **ESC** to remove the grips.

All of the stair treads are now located on layer **Risers,** but they are now all the same lineweight and color. Because they are all on the same layer, you can't select according to layer this time. However, you know that all of the treads are exactly 11 inches long. The **Quick Select** dialog box allows you to select them based on their length.

UTILITIES

13. Pick the **Quick Select** button on the **Utilities** panel of the **Home** tab again.

14. Scroll down the **Properties** selection box and pick **Length**.

15. In the **Value** box, enter **11** and pick **OK**.

All of the 11″ lines in the drawing—all of the treads—are selected.

16. Pick **Lineweight** in the **Properties** palette.

17. Pick the down arrow and select **0.60 mm** from the drop-down list.

18. Pick **Color**, pick the down arrow, and choose green.

The treads are now on the layer **Risers,** but they are still 0.60 mm thick and green.

19. Close all open palettes, save your work, and close the file.

Wohlers: Applying
AutoCAD® 2010

V. Preparing and Printing a
Drawing

24. Layers and Linetypes

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329

Applying the Custom Template

The **tmp1.dwg** file that you created in the last chapter should still be open. This template file is nearly complete. The last few steps (12, 13, and 14 in Table 23-1 on page 299) typically involve creating new text styles and setting the dimensioning variables and **DIMSCALE**. All of this is covered in Chapters 27 and 28.

The template file concept may lack meaning to you until you have actually applied it. Therefore, let's convert **tmp1.dwg** to a template file and use it to begin a new drawing. First we will create the template file.

1. Be sure to save your work if you haven't already.
2. Select **Save As…** from the **Menu Browser.**
3. To the right of **Files of type**, pick the down arrow and select **AutoCAD Drawing Template (*.dwt)**.
4. Find and select the folder with your name, enter **tmp1** for the file name, and pick the **Save** button.

The **Template Options** box appears.

5. Enter **Created for stair details** and pick **OK**.
6. Close **tmp1.dwt**.
7. Select the **QNew** button from the **Quick Access** toolbar and pick the **Use a Template** button.
8. Pick the **Browse…** button.
9. Find and open the **tmp1.dwt** template file. Be sure to look in your named folder.

AutoCAD loads the contents of **tmp1.dwt** into the new drawing file.

10. Review the list of layers.

Look familiar? All the layers from the template file are already loaded into your new drawing file. You can see why template files are valuable time-saving tools.

11. Save the new drawing file as **staird.dwg** and exit AutoCAD.

Chapter 24 **Review & Activities**

• REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Name at least two purposes of layers.
2. How do you change the current layer using the **Layer Properties Manager**?
3. Describe the purpose of the **LTSCALE** command and explain how to set it.
4. Describe a situation in which you would want to freeze a layer.
5. Name five of the linetypes AutoCAD makes available.
6. What is the purpose of locking layers?
7. If you accidentally draw on the wrong layer, how can you correct your mistake without erasing and redrawing?
8. Explain how to freeze a layer using the **Layer** drop-down box.
9. Describe a way to select several similar objects using a single operation.

• CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. When might you use the **Layer** drop-down box in the **Layers** panel instead of using the **Layer Properties Manager** palette? Explain.
2. AutoCAD allows you to use linetypes that correspond to ISO standards. Find out more about ISO standards. When and where are they used?

• APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1–2. Create two new drawings using the **tmp1.dwt** template file. Change the layers to match those shown in Tables 24-3 and 24-4. Name the drawings **prb24-1.dwg** and **prb24-2.dwg**. Make **Objects** the current layer in **prb24-1.dwg**, and make **0** the current layer in **prb24-2.dwg**.

Review & Activities   Chapter 24

Table 24-3. Layers for Problem 1

Layer Name	State	Color	Linetype	Lineweight
0	Frozen	white	Continuous	Default
Border	On	cyan	Continuous	0.50 mm
Center	On	yellow	CENTER	0.20 mm
Dimensions	On	green	Continuous	0.20 mm
Hidden	On	yellow	HIDDEN	0.30 mm
Objects	On	red	Continuous	0.40 mm
Phantom	On	blue	PHANTOM	0.50 mm
Text	Frozen	magenta	Continuous	0.30 mm

Table 24-4. Layers for Problem 2

Layer Name	State	Color	Linetype	Lineweight
0	On	white	Continuous	Default
Center	Frozen	blue	CENTER	0.20 mm
Dimensions	On	yellow	Continuous	0.20 mm
Electrical	On	cyan	Continuous	0.30 mm
Found	On	magenta	DASHED	0.40 mm
Hidden	On	blue	HIDDEN	0.30 mm
Notes	On	yellow	Continuous	0.30 mm
Plumbing	Frozen	white	Continuous	0.30 mm
Title	Frozen	magenta	Continuous	0.30 mm
Walls	On	red	Continuous	0.40 mm

3. Create a drawing template for use with an A-size drawing sheet. Use a scale of $1/4'' = 1'$. Use architectural units with a precision of $1/16''$. Set up the following layers: **Floor**, **Dimensions**, **Electrical**, **Plumbing**, and **Furniture**. Name the template **ch24tmp.dwt**.

Continued

332 Wohlers: Applying
AutoCAD® 2010

V. Preparing and Printing a
Drawing

24. Layers and Linetypes

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Chapter 24 **Review & Activities**

4. Create a new drawing from scratch using **Quick Setup**. Set up the layers shown in Table 24-5. Then create the slide shown in Fig. 24-4 on layer **Object**. Place the center lines on layer **Center**, positioning them as shown in the illustration. Change back to the **Object** layer and create the hole (circle) for the slide so that its center point is at the intersection of the two center lines. To finish the drawing, offset the hole by **.2** units to the outside and trim the center lines to the outside circle. Erase the temporary trim circle, and save the drawing as **slide.dwg**.

Table 24-5. Layers for Problem 4

Layer Name	Color	Linetype	Lineweight
Object	white	Continuous	0.40 mm
Center	blue	CENTER2	0.20 mm
Dims	red	Continuous	Default
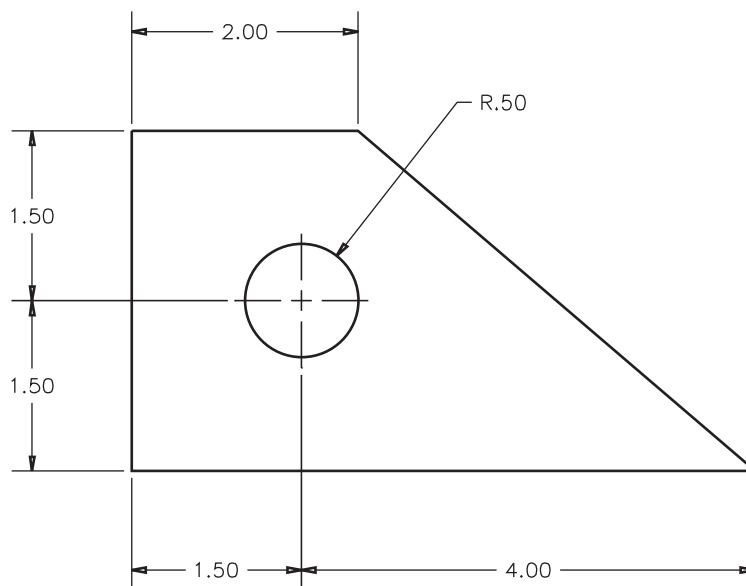
Fig. 24-4

5. Using the **ch24tmp.dwt** template file you created in problem 3 on page 325, create a simple floor plan for a house. Make the floor plan as creative and as detailed as you wish. Place all the objects on the correct layers.

6. The graph in Fig. 24-5 shows the indicated and brake efficiencies as functions of horsepower for a small engine. Reproduce the graph as follows: Using a suitable scale, draw the grid and plot the given points as shown; then draw a spline through each set of points. Place the border, title block, and curves on a layer named **Visible**; the grid and point symbols on layer **Grid**; and the text on layer **Text**. Trim the grid around the text and arrows, and trim the curves and grid out of the symbols. Use the appropriate text justification to align the axis numbers and titles properly.

HP	INDICATED	BRAKE
1.53	3.60	2.55
2.83	5.30	3.86
4.11	5.90	5.00
5.30	7.25	4.52
6.60	8.82	5.62
9.10	8.30	5.83
10.45	6.27	5.40
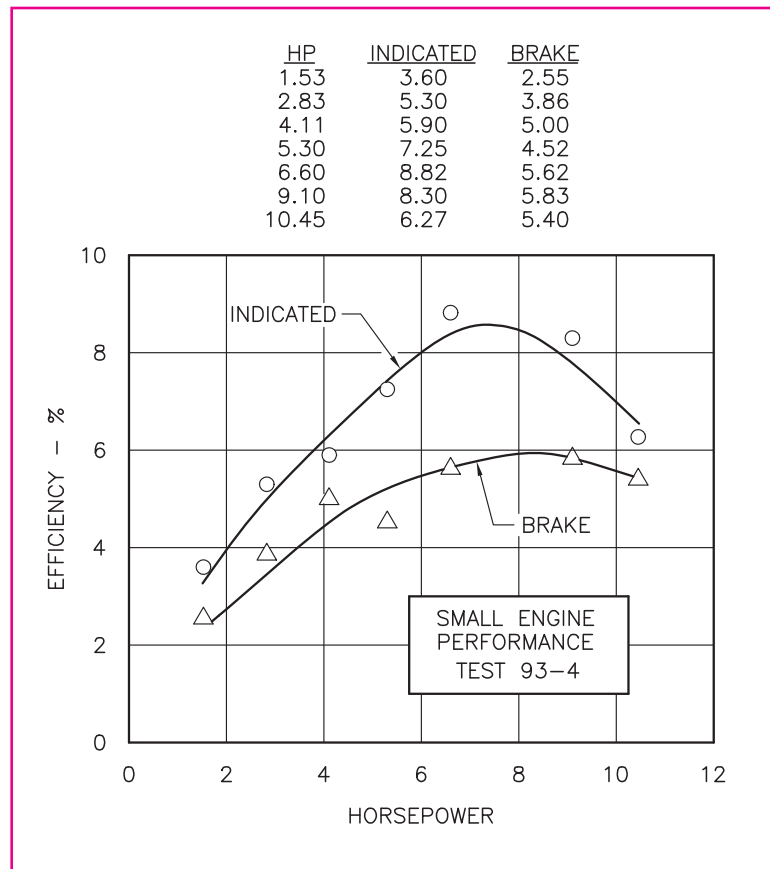
Fig. 24-5

Courtesy of Gary J. Hordemann, Gonzaga University

Continued

Chapter 24 **Review & Activities**

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Create the drawing of the washing machine spacer from the graph paper sketch in Fig. 24-6. Each square represents .25 in. Add a title and include a note saying that the material is SAE 1010 carbon steel. Make the appropriate drawing setup and create the necessary layers. Add center lines for all holes. Save the drawing as **ch24spacer.dwg**.
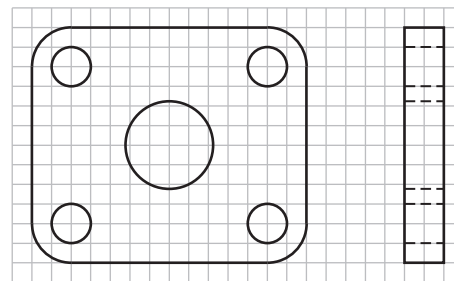
Fig. 24-6

2. Figure 24-7 shows a sketch of the step block for a bowling alley pinsetter. Create the drawing using appropriate layers. Each square represents 5 cm. Add a title and these notes: **1) Break all sharp edges**, **2) fillets and rounds 5 cm**, and **3) material is UNS S30451 stainless steel**. Save the drawing as **ch24pinsetter.dwg**.
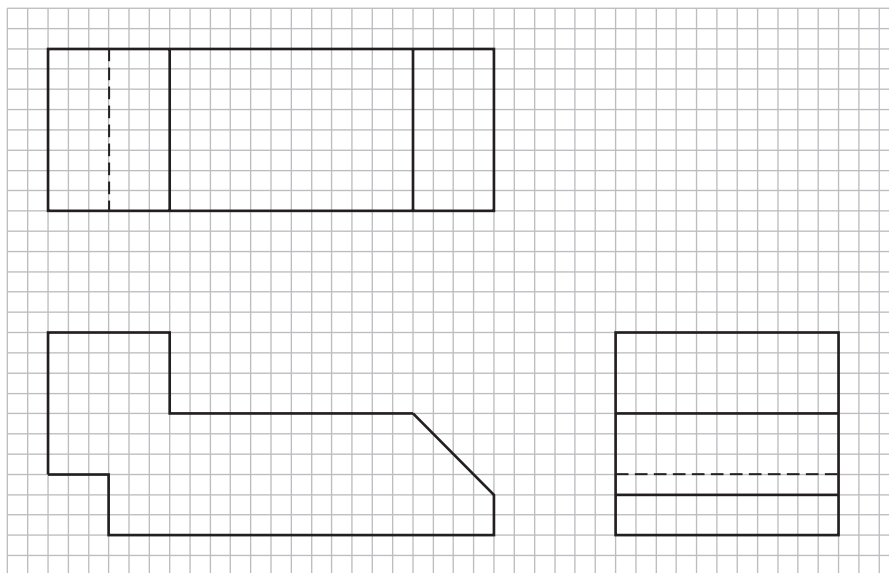
Fig. 24-7

Chapter 25

Plotting and Printing

Objectives

- Preview a plot
- Adjust plotter settings
- Plot an AutoCAD drawing to scale
- Create DWF and PDF files

Vocabulary

landscape
plot scale
plot style
plotters
portrait
printers
rendered
viewports
visual style
wireframe

CAD users plot and print drawings using plotters and printers. Many years ago, there was a distinct difference between a plotter and a printer, but today, the distinction has blurred to the extent that there is now little difference between the two. Both use the same basic technology—usually inkjet or laser—and both print in black, grayscale, or color onto paper and other sheet materials. Some people still refer to devices that handle large sheets, such as $36'' \times 48''$ and even larger, as **plotters**. They think of **printers** as smaller, tabletop devices handling sheets up to $11'' \times 17''$. Note that this is not a universally accepted difference, however. The terms *plotter* and *printer* and *plotting* and *printing* are used interchangeably in this book.

Let's create and plot a set of stairs.

1. Start AutoCAD and open the file named **staird.dwg**.

This file, which is based on the **tmp1.dwt** template file, was created in the previous chapter.

2. Select the **2D Drafting & Annotation** workspace.

3. On layer **Objects**, draw the stair step shown in Fig. 25-1. Use polar tracking and enter the lengths at the keyboard. (The **Show/Hide Lineweight** button on the status bar should *not* be depressed.)
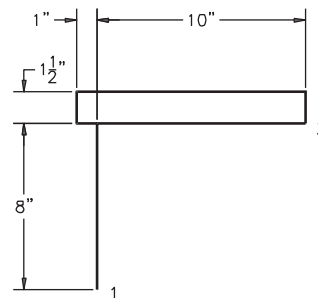
4. Copy the objects five times to produce the stairs as shown in Fig. 25-2. Use points 1 and 2 in Fig. 25-1 as the base point and second point, respectively. Be sure to add the lines that make it complete.

5. **ZOOM All** and save your work.
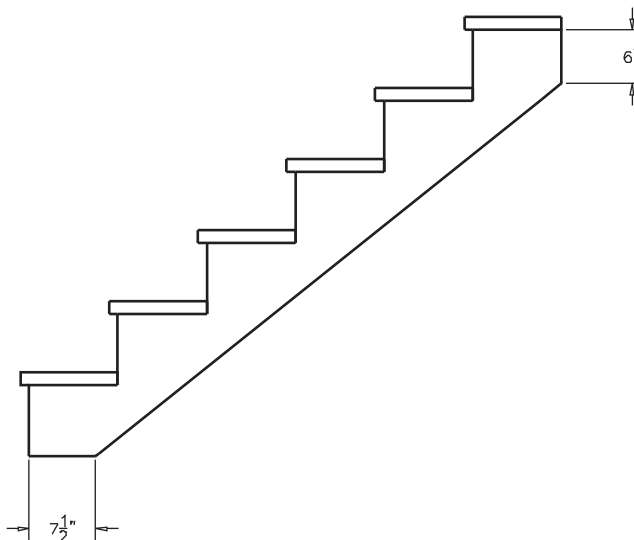
Fig. 25-1

Fig. 25-2

Previewing a Plot

AutoCAD allows you to preview a plot before you send it to the plotter. This feature allows you to catch mistakes before you spend the time and supplies to create the actual plot.

1. From the **Quick Access** toolbar, pick the **Plot** button or enter the **PLOT** or **PRINT** command at the keyboard.

QUICK ACCESS
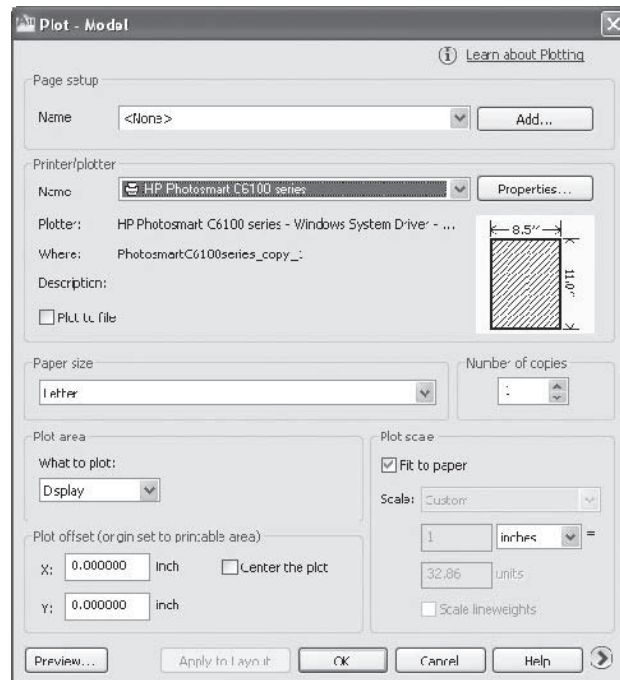
Fig. 25-3

The **Plot** dialog box appears, as shown in Fig. 25-3. In the illustration, notice that **HP Photosmart C6100 series** is the current printer/plotter. The device you see in this area will probably be different. AutoCAD stores plotter settings for each configured plotter, so other settings may also differ.

> **NOTE**
> If no plotter names appear in the upper left area named **Printer/plotter**, you must configure one before you proceed with the following steps. Consult your instructor or the head of your company's IT department for details.

Note the preview button located in the lower left area of the dialog box.

2. Pick the **Preview...** button.

This feature gives you a preview of how the drawing will appear on the sheet, as shown in Fig. 25-4 on page 332.

3. Use the pick button to zoom in on the drawing.

4. Right-click to display a shortcut menu, and pick **Pan** to move around in the drawing.
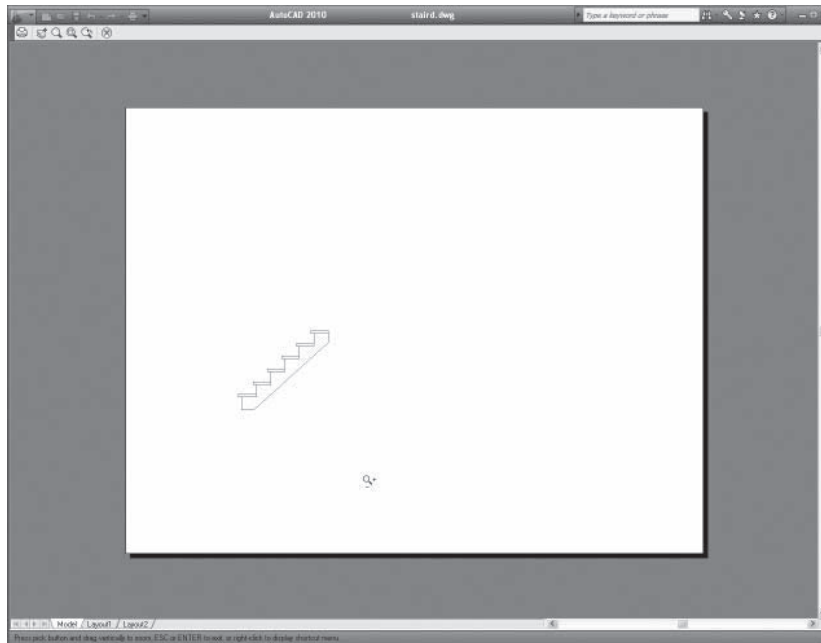
Fig. 25-4

Zooming and panning enable you to examine the drawing closely before you plot it. Plotting large drawings can take considerable time and supplies, so spotting errors before you plot can save time and money.

5. Press **ESC** or **ENTER**.

6. If your computer is connected to the configured plotter or printer and it is turned on and ready, continue with Step 7. If it is not, prepare the device. If you do not have access to an output device, pick the **Cancel** button and skip to the next section, titled "Tailoring the Plot."

7. Pick **OK**.

Since we did not make changes to the plot settings before plotting, the plot may be incorrect.

Tailoring the Plot

QUICK ACCESS

The following steps consider sheet size, drawing scale, and other plotter settings and parameters.

1. Pick the **Plot** button.

2. In the bottom right corner, pick the arrow (**>**) to expand the dialog box to include other options.

Table 25-1. Plotting Options

Option	Function
Display	Plots the current view
Extents	Similar to ZOOM Extents; plots the portion of the drawing that contains objects
Limits	Plots the entire drawing area
Window	Plots a window whose corners you specify

Specifying Sheet Size

1. Under **Paper size**, display the list of options.

2. Review the list of sizes available for the configured printer.

3. Pick **Letter** if it is available. If not, pick the option that is closest to $11'' \times 8.5''$.

Defining the Plot Area

1. In the **Plot** area, select the **Limits** option.

We selected **Limits** because we want to plot the entire drawing area as defined by the $22'' \times 17''$ drawing limits. In the future, you may choose to select one of the other plotting options. Table 25-1 explains what each option means.

Setting the Drawing Scale

1. Focus your attention on the **Plot scale** area of the dialog box.

The **plot scale** is the scale at which the drawing is plotted to fit on a drawing sheet.

2. Uncheck the **Fit to paper** check box.

The scale options become available.

3. Pick the down arrow next to **Scale:** and review the list of options.

As you may recall, the scale for the stairs drawing is $1/2'' = 1'$.

4. Select **1/2″ = 1′-0″**.

The numbers **1** and **24** (or **0.5** and **12**) are entered automatically in the **inches** and **units** boxes because **1 inch = 24 units** is another way to express our plot scale.

> **NOTE**
>
> You can manage the list of scales you use for viewports, page layouts, and plotting in the **Edit Scale List** dialog box. To open the list, pick **Scale List…** from the **Format** pull-down menu or type the command **SCALELISTEDIT**. Scales can be added, deleted, and modified, and you can rearrange the scale list to display the most commonly used scales at the top.

Setting the Drawing Orientation

Notice the illustration to the right of the **Portrait** and **Landscape** radio buttons in the **Drawing orientation** area. In **portrait** orientation, the drawing is positioned so that "north" or "up" falls on the narrower edge of the paper. In **landscape**, the wide edge of the paper is north.

1. Pick each radio button while watching the illustration to see the difference between portrait and landscape.
2. In the **Drawing orientation** area, pick **Landscape**.

Reviewing Other Settings

Let's look briefly at the other settings available from the **Plot** dialog box.

1. Focus on the **Page setup** area and pick the **Add** button.

This button allows you to save the current settings in the **Plot** dialog box to a named page setup. You can modify the page setup using the **Page Setup Manager**.

2. Pick **Cancel** in the **Add Page Setup** dialog box.
3. Notice the **Plot to file** check box in the **Printer/plotter** area.

When the **Plot to file** check box is checked, AutoCAD sends plot output to a file rather than to a device and creates a **PLT** file type.

4. Pick the **Properties…** button located in the **Printer/plotter** area.

This displays the **Plotter Configuration Editor**, which presents information relevant to the device.

5. Pick the **Cancel** button to close the **Plotter Configuration Editor**.
6. Now focus your attention on the **Plot offset** area of the dialog box.

It specifies an offset of the plotting area from the left corner of the sheet.

7. Focus your attention on the **Plot style table (pen assignments)** area.

The **Plot style table (pen assignments)** area allows you to create and edit plot style tables. A **plot style** in AutoCAD is a collection of property settings that are saved in a plot style table.

8. Focus your attention on the **Shaded viewport options** area.

This area specifies how shaded and rendered **viewports** are plotted and determines their resolution. A viewport is a bounded area that displays some portion of a drawing. Viewports are discussed further in the next chapter.

9. Select the down arrow located to the right of **Shade plot**.

The **Shade plot** area allows you to select a visual style for the plot. The **As displayed** option plots objects the way you see them on the screen. The other options—**Wireframe**, **Hidden**, **3D Wireframe**, **3D Hidden**, **Realistic**, **Conceptual**, and **Rendered**—are different visual styles for plotting your drawing. A **visual style** is the visual representation of objects in the drawing area or on a plot. It includes the way the edges are displayed and the way the object is shaded. A **wireframe** is the representation of a 3D object using lines and curves to show its boundaries. **Hidden** plots objects with the hidden lines removed. In this context, hidden lines are those lines and curves of an object that are hidden from view. A **rendered** 3D object is one that has shading applied. The **Realistic**, **Conceptual**, and **Rendered** options all plot rendered 3D objects. You will learn more about visual styles and rendering in Parts 8 and 9.

10. Select the down arrow located to the right of **Quality**.

The **Quality** area allows you to select the resolution of the plot, from **Preview** quality—150 dots per inch (dpi)—up to **Presentation** quality (600 dpi). You can also specify the maximum quality or specify the dpi resolution yourself by selecting **Custom** in the **Quality** box.

11. Focus on the **Plot options** area.

This area specifies options for lineweights, plot styles, and the order in which objects are plotted.

Plotting the Drawing

Let's preview and plot the drawing.

1. Pick the **Preview...** button.

2. Review the preview and then press **ESC**.

3. Prepare the device for plotting.

4. Pick the **OK** button to initiate plotting.

After plotting is complete, examine the output carefully. The dimensions on the drawing should measure correctly using a $1/2'' = 1'$ scale.

Creating DWF and PDF Files

AutoCAD users often need to share a design or drawing with business associates who do not use AutoCAD. These associates may be managers, sales representatives, customers, or suppliers. AutoCAD offers other output options in addition to printing that allow users to share drawings with non-AutoCAD users. Note that these output options allow non-AutoCAD users to view a file, but do not offer them any of the commands or features of AutoCAD. We will discuss two such output options: the **DWF** format and the **PDF** format.

Autodesk (the maker of AutoCAD) offers a free program called DWF Viewer. An AutoCAD user can save a drawing in the **DWF** format, then send the **DWF** file to or share it with a non-AutoCAD user who has installed the DWF Viewer software.

QUICK ACCESS

1. Pick the **Plot** button.

2. In the **Printer/plotter** area, pick the down arrow to the right of **Name**.

The drop-down menu displays a list of output options, including printing options and file output options.

3. Select **DWF6 ePlot.pc3** from the drop-down menu and pick **OK**.

4. Save the new **DWF** file as **staird-Model.dwf** in the folder with your name.

Another common file sharing format is the Adobe® Portable Document format (**PDF**). **PDF** files can be created in most popular programs, and can be read by Abode Reader, another free file-viewing program.

QUICK ACCESS

5. Pick the **Plot** button.

6. In the **Printer/plotter** area, pick **DWG To PDF.pc3** from the drop-down menu, then pick **OK**.

7. Save the new **PDF** file as **staird-Model.pdf** in the folder with your name and exit AutoCAD.

After saving a **DWF** or **PDF** file, you would typically share it with others by sending the file as an e-mail attachment or copying it to a secure server. Keep in mind that the recipient of the file must have the viewing software installed to be able to view the file.

Chapter 25 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Explain why a plot preview is useful.
2. In the **Plot** dialog box, what is the purpose of the **Plot to file** check box?
3. Briefly describe each of the following plot options.
   a. **Limits**
   b. **Extents**
   c. **Display**
   d. **View**
4. The drawing plot scale for a particular drawing is 1 = 4″. What does the 1 represent and what does the 4″ represent?
5. Describe how you would create a version of an AutoCAD file that could be viewed by someone who does not have the AutoCAD software.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Explore ways of sending **PLT** files to a printer or plotter.
2. Suppose you wanted to plot all of your drawings to **PLT** files and store them automatically in the folder of your choice. How can you set the default folder for **PLT** files?

Continued

Chapter 25 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter. In problems 1 and 2, prepare to plot a drawing using the information provided. Choose any drawing to plot.

1. Paper size: Letter

Units: Inches
Drawing orientation: Landscape
Plot scale: 1 inch × 2 units
Plot with object lineweights
Do not plot with plot styles
Shade Plot: As Displayed
Quality: Normal
Number of copies: 1
Do not plot to file
Perform a preview

2. When starting a drawing from scratch, select **Metric**.

Paper size: A4
Units: Millimeters
Drawing orientation: Landscape
Plot area: Limits
Plot scale: 1 mm × 10 units
Plot with lineweights
Number of copies: 1
Perform a preview
Plot to file

3. Open **slide.dwg** (from problem 4 in Chapter 24). Zoom in so that the slide fills most of the screen. Enter the **PLOT** command and choose the **Limits** option in the **Plot area** portion of the dialog box. Plot the drawing. Then create two more plots, one with the **Extents** radio button selected, and one with the **Display** radio button selected. Compare the drawings.

4. Choose and plot a drawing that you created in an earlier chapter. Consider the scale and drawing area so that dimensions measure correctly on the plotted sheet. Text and linetypes should also measure correctly on the sheet. For example, $1/8''$ text should measure $1/8''$ in height.

Review & Activities Chapter 25

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. The R&D department has requested a full-size plot of the step block for the bowling alley pin-setter you created in Chapter 24. Adjust the plot area, scale accordingly, and plot the drawing.

2. An outside sales representative in your company has requested a **PDF** of the washing machine spacer you created in Chapter 24. Open the file and plot the spacer as a **PDF** file. Open the **PDF** file of the spacer in Adobe Reader® to verify that it is accurate. (Adobe Reader is free software that you can download from the Adobe Web site if it is not already installed on your computer.)

AutoCAD at Work

Technical Illustrator

You have just purchased a new stereo and you are eager to hear how your favorite CD sounds through the system's powerful speakers. To figure out how to connect each component properly, you consult the instruction sheet:
Connect wire A from component 1 to component 2. Connect wire B from component 1 to component 3. Connect wires C and D from component 1 to external speakers. How do you know which is wire A? How is it different from wire D? Fortunately, the instructions also include an illustrated example of the setup procedure.

Assisting the User

Technical illustrators produce drawings for instruction manuals and appliance or equipment operating manuals. These illustrations allow users to identify specific parts and verify that they have followed the directions properly. Technical illustrators consult with the manufacturer of the product to ensure that their understanding of the item, its parts, and the intended procedure is accurate.

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Attention to Detail

As with many other types of drafting careers, technical illustrators must pay close attention to detail. If an unusual piece of equipment is mentioned in writing but a visual example is not provided, the person following the directions may become confused. A combination of illustrations and text offers multiple opportunities to understand exactly how to build or operate an item.

Many drafters are drawn to the field of technical illustration. Even though technical illustrators rarely create dimensioned technical drawings, their work must be accurate and complete. In fact, many technical illustrators have a strong drafting background.

▶ Career Activities

1. Find and compare instructions that are completely text-based and instructions that consist only of illustrations. Which do you find easier to follow? Why?

2. Investigate technical illustrator careers using the Internet. What are forecasts for future employment?

Wohlers: Applying
AutoCAD® 2010

V. Preparing and Printing a
Drawing

26. Multiple Viewports

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347

Chapter

26

Multiple Viewports

Objectives

- Create and use multiple viewports in model space
- Create and use viewports in paper space
- Position objects in paper space viewports
- Edit, position, and plot paper space layouts

Vocabulary

layout
model space
paper space

In AutoCAD, viewports are portions of the drawing area that you define to show a specific view of a drawing. By default, AutoCAD begins a new drawing using a single viewport. By defining and using multiple viewports, you can see more than one view of a drawing at once.

AutoCAD includes two different environments—**model space** and **paper space**—for working with objects. Paper space is used to arrange views of an object or 3D model and embellish them for plotting. Model space is used to construct and modify the objects that make up the drawing. Because model space is where you wll perform most of your drafting and design work, AutoCAD opens a single viewport in model space when you create a new drawing without using a template.

Each new drawing also has two default layouts. A **layout** is an arrangement of one or more views of an object on a single sheet. When you pick a layout, AutoCAD automatically switches to paper space. After you have set up the paper space parameters, AutoCAD positions model space objects against a white "paper" background to show you how they will appear on the printed sheet.

Viewports in Model Space

You can apply viewports to both model space and paper space. In model space, you can use viewports to draw and edit in more than one view at a time. The magnification of each viewport can be set individually, so viewports provide capabilities that would otherwise be impossible.

After creating a drawing in model space, you can create floating viewports in paper space to display different views of the drawing. Because they float, you can easily position them for plotting. Depending on your needs, you can set options that determine what is plotted and how the viewports fit on the sheet.

Figure 26-1 gives an example of applying multiple viewports in model space to **zoom.dwg**. Notice that each viewport is different both in its window extents and in magnification.
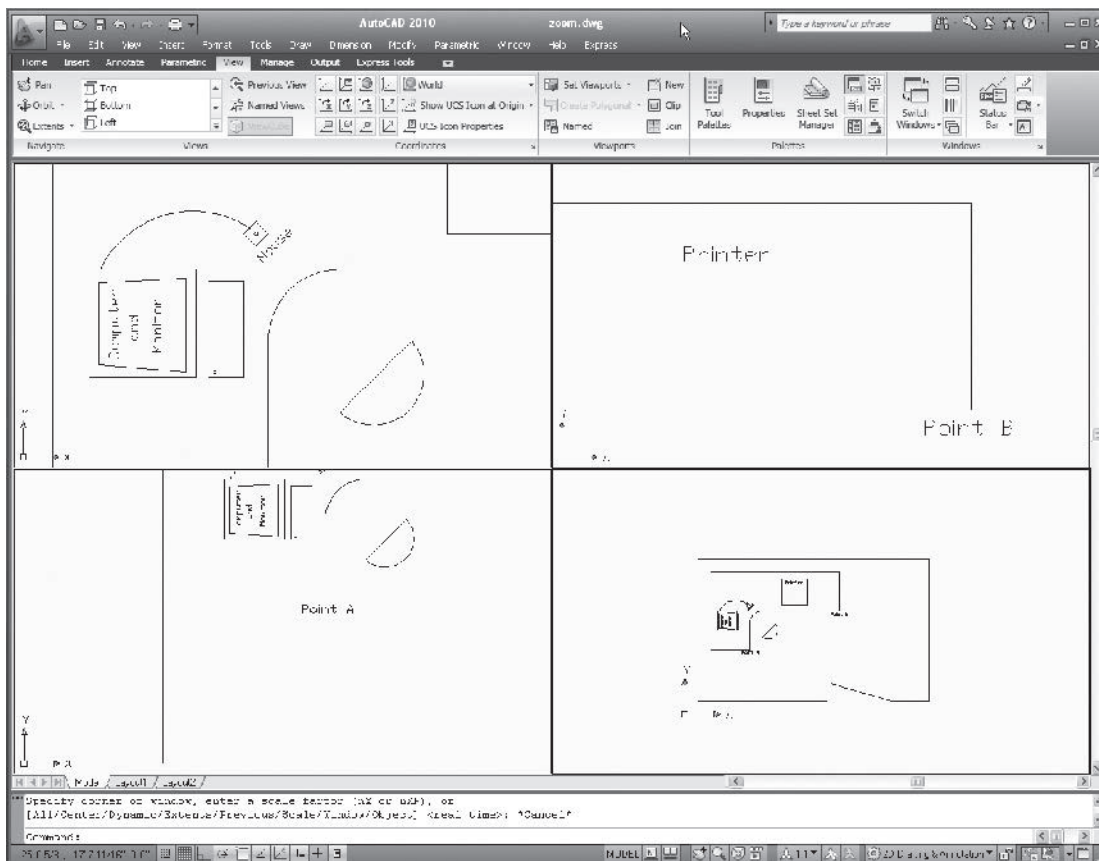
Fig. 26-1

Creating Viewports in Model Space

Viewports are controlled with the **VPORTS** (or **VIEWPORTS**) command.

1. Open the drawing file named **zoom.dwg**.
2. Select the **2D Drafting & Annotation** workspace.
3. Enter **ZOOM All** to make the drawing fill the screen.
4. Select the **View** tab on the **Ribbon** and pick the **New** button on the **Viewports** panel, or enter the **VPORTS** command.
5. Under **Standard viewports**, click on each of the options.

New
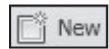
VIEWPORTS

Each viewport option appears in the **Preview** area.

6. Pick **Four: Equal** and pick the **OK** button.

AutoCAD produces four viewports of equal size, each with an identical view of the drawing.

7. Move the pointer to each of the four viewports.

The crosshairs appears only in the viewport with the bold border. This is the current (active) viewport.

8. Move to one of the three nonactive viewports.

An arrow appears in place of the crosshairs.

9. Press the pick button on the pointing device.

This viewport becomes the current one.

Using Viewports in Model Space

Let's modify **zoom.dwg** using the viewports.

1. Refer to Fig. 26-1 and create four similar viewports using AutoCAD's zoom and pan features.

> **HINT**
>
> Make one of the viewports current and then zoom and pan. Repeat this process for the other three viewports.

2. Make the lower left viewport current.
3. Enter the **LINE** command and pick point **A**. (Refer to Fig. 26-1 for point **A**.)
4. Move to the upper right viewport and make it current.

Notice that the **LINE** command is now active in this viewport.

5. Pick point **B** and press **ENTER**.

View all four viewports. The line represents the edge of a hard surface for the chair.

So you see, you can easily begin an operation in one viewport and continue it in another. Any change you make is reflected in all viewports. This is especially useful when you are working on large drawings with lots of detail.

Let's move the printer from one viewport to another.

6. Make the upper right viewport current and select the printer.

7. Right-click and pick **Move** from the shortcut menu.

8. Pick a corner of the printer as the base point.

9. Move to the lower left viewport and make it current. (Make sure ortho is off.)

10. Place the printer in the open area on the table by picking a point at the appropriate location.

Notice that the printer location changed in the other viewports.

Viewport Options

Let's combine two viewports into one.

VIEWPORTS

1. Pick the **Join Viewports** button on the **Viewports** panel.

2. Choose the upper left viewport in reply to **Select dominant viewport**.

3. Now choose the upper right viewport in reply to **Select viewport to join**.

As you can see, the **Join** option enables you to expand—in this case, double—the size of a viewport.

4. Pick the arrow in the **Viewport Configurations** text box in the **Viewports** panel and select the **Single** option.

The screen changes to single viewport viewing. This single viewport is inherited from the current viewport at the time you selected **Single**.

AutoCAD also allows you to subdivide current viewports into two or more additional viewports.

5. Pick the arrow in the **Viewport Configurations** text box in the **Viewports** panel and select the **Three: Right** option.

6. Make the upper left viewport the current one.
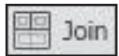
7. Pick the **New Viewport** button to redisplay the **Viewports** dialog box and pick **Two: Vertical**.

8. Under **Apply to**, located in the lower left corner, pick the down arrow, select **Current Viewport**, and pick **OK**.

New

VIEWPORTS

As you can see, AutoCAD applies **Two: Vertical** to the current viewport.

9. Try the remaining viewport options on your own. Practice drawing and editing using the different viewport configurations.

10. Save your work and close the drawing file, but do not exit AutoCAD.

> NOTE
>
> The **REGEN** command affects only the current viewport. If you are using multiple viewports and you want to regenerate all of them, you can use the **REGENALL** command.

Viewports in Paper Space

When you are creating a layout in AutoCAD, you can consider viewports as objects with a view into model space that you can move and resize. By default, AutoCAD presents a single viewport in the paper space layout.

1. Open the file named **staird.dwg**.

2. Pick the **Quick View Layouts** button on the status bar.

STATUS BAR

This produces three thumbnails of the views in the drawing: **Model, Layout1** and **Layout2**. Notice the four buttons below the thumbnails. You can pin the quick views open, create a new layout, publish the sheet set, or close the **Quick View Layouts.**

3. With the pick button, pick the **Layout1** thumbnail.

When you pick a layout view, a sheet with margins displays, reflecting the paper size of the currently configured plotter and printable area of the sheet.

In paper space, you can view the exact size of the sheet and see how the drawing will appear on the sheet when you plot. The dashed line represents the plotting boundary, and the solid line represents the single viewport. (AutoCAD's paper space defaults to a single viewport if you do not specify more than one.)

The paper space icon, located in the lower left corner of the drawing area, replaces the standard coordinate system icon. The paper space icon is

present whenever paper space is the current space. The coordinate system icon is present whenever model space is the current space.

4. Right-click the **Layout1** thumbnail, select **Rename** from the menu, enter **My Layout,** and press **ENTER.**

AutoCAD renames the layout tab.

5. Pick the **Close** button to close **Quick View Layouts.**

In the status bar, notice that **LAYOUT** is displayed on the **Model or Paper space** toggle button indicating that you are now in paper space.

6. Pick the viewport (solid red border).

As you can see, a viewport in paper space is an object.

7. Move the viewport a short distance.

The stair detail moves too, because it belongs to the viewport.

8. Undo the last step.

Plotting a Single Viewport in Paper Space

Plotting a viewport containing an object in paper space is different than plotting the same object in model space. Before plotting to scale, you must fit the objects in the viewport using the **ZOOM** command.

1. Double-click inside the viewport.

The outline of the viewport becomes bold and the coordinate system icon returns, indicating that you are now in model space within the layout.

2. Enter the **ZOOM** command and enter the **Scale** option.

3. For the scale factor, enter **1/24xp**.

The "1 to 24" reflects the scale of the drawing. As you may recall, we established a scale of $1/2'' = 1'$ for **tmp1.dwt**, which we used for the stair detail. You can also express the scale as $1'' = 24''$. When you enter a value such as **1/24** followed by **xp**, AutoCAD specifies the scale relative to the paper space scale. (The term **xp** means "times paper space scale.") If you enter **.5xp**, AutoCAD displays model space at half the scale of the paper space scale.

QUICK ACCESS

4. Double-click outside the viewport to return to paper space.

5. Pick the **Plot** button from the **Quick Access** toolbar.

Notice under **Plot area** that **Limits** is not available; **Layout** appears in its place.

6. Produce the following settings:

Paper size: Letter or 11″ × 8.5″
Drawing orientation: Landscape
Plot area: Layout
Scale: 1:1
1 inch = 1 unit

The plot scale is 1=1 because we do not want to scale the layout up or down. We did that in Step 3. This is one distinct difference between plotting from model space and plotting from paper space.

7. Perform a full preview and then press **ESC**.

8. Make sure that your plotter is ready, and pick the **OK** button to initiate plotting.

AutoCAD plots the stairs to scale.

9. Save your work and close the drawing file.

Adding Viewports in Paper Space

You can add viewports in paper space using a method similar to the one you used in model space, but you must be in paper space. Let's set up a drawing with multiple paper space viewports.

1. Create a new drawing and pick the **Quick Setup** wizard.

2. Pick **Next** to accept **Decimal** units.

3. Enter **11** for the width and **8.5** for the length, pick the **Finish** button, and **ZOOM All**.

4. Create the layers shown in Table 26-1, making **Vports** the current layer.

5. Set snap at **.5**″ and create a new text style named **romans** using the **romans.shx** font. Use the default settings.

6. Save your work in a file named **pspace.dwg**.

7. Pick the **Layout1** tab at the bottom of the drawing area to enter paper space.

8. Erase the viewport that appears on layer **Vports**.

Table 26-1. Layers

Layer Name	Color	Linetype	Lineweight
Objects	red	Continuous	0.40 mm
Border	blue	Continuous	0.60 mm
Vports	magenta	Continuous	0.20 mm

9. Display the **Menu Bar** by choosing **Show Menu Bar** from the pull-down menu on the **Quick Access** toolbar.

10. From the **View** menu on the **Menu Bar** pick **Viewports** and **4 Viewports**, and press **ENTER** to accept the **Fit** default value.

AutoCAD inserts four equally sized viewports on layer **Vports**.

11. Double-click inside the viewports.

Coordinate system icons appear in each of the four viewports, showing that you are now in model space.

Objects in Viewports

You will better understand the benefits of using viewports in paper space when you create an object.

1. Make **Objects** the current layer.

2. Enter the **THICKNESS** system variable and set it to **1**.

The **THICKNESS** system variable enables you to specify the thickness of an object in the *z* direction, resulting in a three-dimensional (3D) object. Chapters 38 through 49 contain more information about 3D objects.

3. In the upper left viewport, draw the object shown in Fig. 26-2.
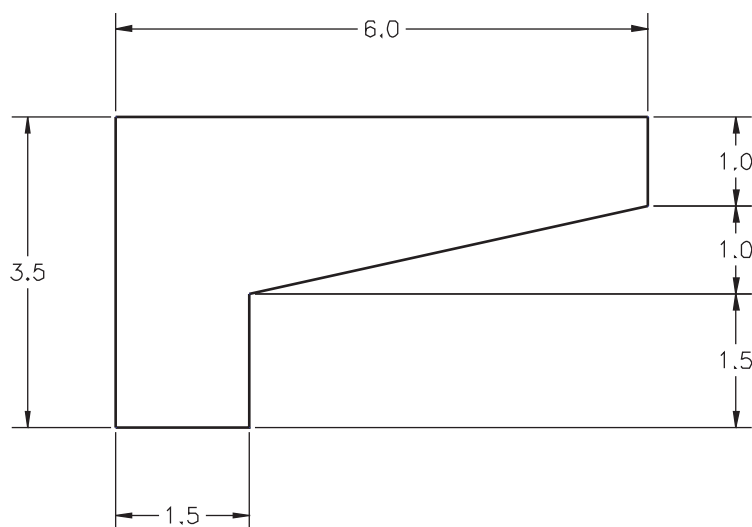
Fig. 26-2

An identical view of the object appears in all four viewports.
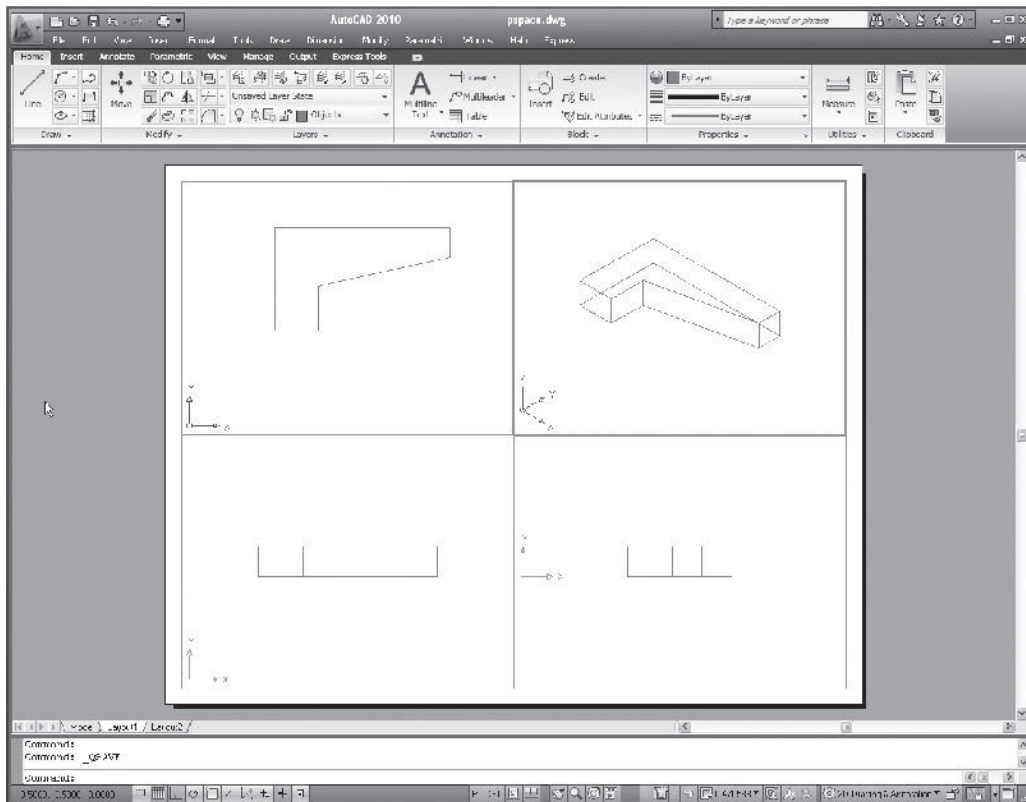
Fig. 26-3

Creating Four Individual Views

Taking advantage of AutoCAD's viewports, let's create four different views of the solid object.

1. Make the lower left viewport the current one.

2. In the **View** menu on the **Menu Bar**, select **3D Views** and **Front**.

3. Enter **ZOOM** and **1**.

This scales the viewport to an apparent size of **1**. Entering **2** would make it twice the size.

4. Make the lower right viewport the current one.

5. View the object from the right side by picking the **Right** button under **3D Views** in the **View** menu.

6. Enter **ZOOM** and **1**.

7. Make the upper right viewport current and view the object from above, from the front, and from the right by picking **SE Isometric** under **3D Views** in the **View** menu.

8. Enter **ZOOM** and **1**, and save your work.

The screen should look similar to the one in Fig. 26-3.

Editing Objects

When you are working in paper space, you cannot edit objects created in model space. Likewise, when you are in model space, you cannot edit objects created in paper space.

1. Double-click outside the four viewports.

Little appears to change except for the coordinate system icons.

2. Attempt to select the object in any of the four viewports.

As you can see, you cannot select it because it was created in model space.

Working with Paper Space Viewports

One of the big advantages of working with paper space viewports is that they allow you to print more than one view of an object on a single sheet of paper. They are also helpful for placing and arranging views on the drawing sheet.

Editing Viewports

Viewports in paper space are treated much like other AutoCAD objects. They can be moved and even erased, but not in model space. Only the views (objects) themselves can be edited in model space.

1. Enter the **UNDO** command and the **Mark** option.
2. Select one of the lines that make up one of the four viewports and move the viewport a short distance toward the center of the screen.
3. Erase one of the viewports.
4. Scale one of the viewports by **.75**.
5. Switch to model space within the layout.
6. Attempt to move, erase, or scale a viewport.

You can't do it, because you are no longer in paper space.

7. Enter **UNDO** and the **Back** option.

NOTE Do not double-click a viewport because this will cause you to enter paper space.
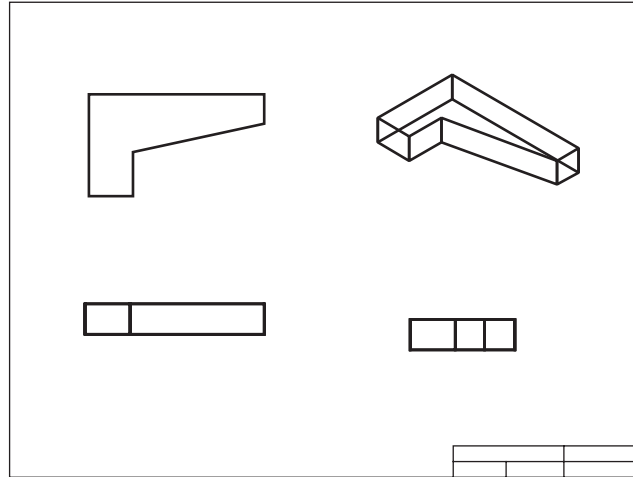
Fig. 26-4

Plotting Multiple Viewports in Paper Space

One of the benefits of paper space is multiple viewport plotting. (You cannot plot more than one model space viewport at a time.)

1. Freeze layer **Vports** and make **Border** the current layer.

The lines that make up the viewports should now be invisible.

2. Switch to model space within the layout.

The viewport lines are invisible in model space also.

3. Switch to paper space and draw a border and basic title block similar to those in Fig. 26-4.

4. Plot the layout at a scale of **1 = 1**.

Positioning Viewports

It is possible that the current position of the border and views did not plot perfectly. Even if they did, make adjustments to the location of the border, title block, and views by following these steps.

1. Thaw layer **Vports**.

2. Move the individual viewports to better position them in the drawing. It is normal for them to overlap.

3. Edit the size and location of the border and title block if necessary.

4. Freeze layer **Vports**, save your work, and replot the drawing.

5. Save your drawing and exit AutoCAD.

Chapter 26 **Review & Activities**

• REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Explain the difference between model space and paper space.
2. How can using multiple viewports in model space help you construct drawings?
3. How do you make a viewport in model space the current viewport?
4. Explain how you would join two viewports in model space.
5. What option should you enter to obtain two viewports in the top half of the screen and one viewport in the bottom half of the screen?
6. If you are working in paper space and you discover that you need to change an object that was created in model space, what must you do before you can make the change? Why?
7. Explain why you may want to edit viewports in paper space.
8. What is the main benefit of plotting in paper space?

• CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Find out how many viewports you can have at one time in AutoCAD. Would you want to use that many? Why? Explain the advantages and disadvantages of using multiple model space viewports in your drawings.
2. Experiment with viewports created in model space and paper space. Is it possible to create more than one viewport in model space, then import the model space viewports into a paper space viewport? Explain.

Review & Activities Chapter 26

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create each of the viewport configurations shown in Fig. 26-5.
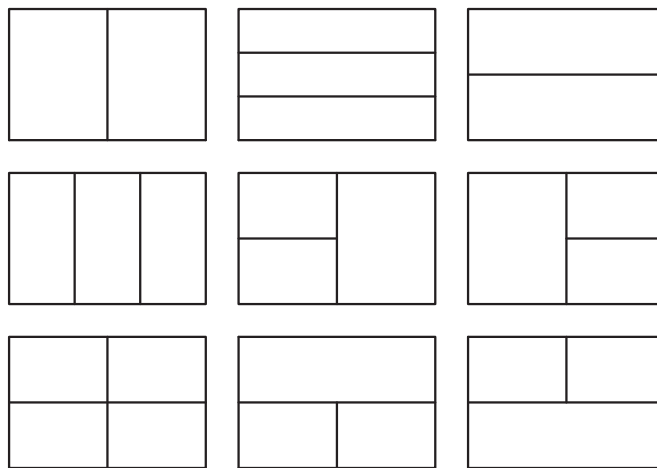
Fig. 26-5

2. Using viewports in paper space, create the top, front, right side, and SE isometric view of the security clip shown in Fig. 26-6. Create a border and title block, and plot the multiple views on a single sheet.
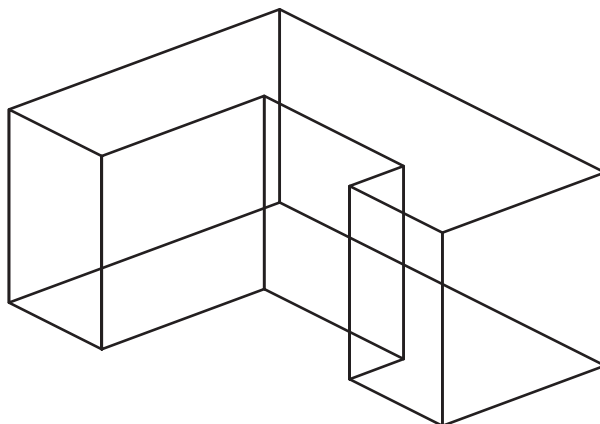
Fig. 26-6

Continued

Chapter 26 **Review & Activities**

3. Open the **db_samp.dwg** file in AutoCAD's **Sample** folder and save it in your named folder as **Fremont.dwg**. The current tenant of this building wants to reconfigure the office by moving a printer island to an area of unused offices.

   a. Create two viewports side by side (use the **Vertical** option).

   b. Make the right viewport active and zoom in so that the building almost fills the viewport. Then zoom in further on the printer island on the right side of the building.

   c. Make the left viewport active and again zoom in so that the building almost fills the viewport. Then zoom in on the empty offices in a horizontal line at the bottom of the viewport (office numbers 6156, 6158, 6160, 6162, and 6164). This will be the new location of the printer island.

   d. Erase the five offices, including the wall that lies against the outside wall.

   e. Using the two viewports as necessary, move the entire printer island to the space formerly occupied by the offices. (Note: When you select the printer island, do not include the gray H or the lines above and below it. It is a structural I-beam that helps support the building.)

   f. Return to a single viewport and save the drawing.

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Draw the locking receptacle (Fig. 26-7) for the alarm system to be installed in the administration building. Create **Three: Right** viewports in paper space, and show the top and front views. In the large right viewport, show the SW isometric view. Plot the views.

2. Draw the hexagonal locking pin that fits into the alarm system receptacle in the administration building from problem 1. The pin, shown in Fig. 26-8, is 2″ long and measures 2.5″ across the flats. Use three viewports in paper space, and show the same views that you used for the locking receptacle in problem 1. Plot the views.

Review & Activities Chapter 26
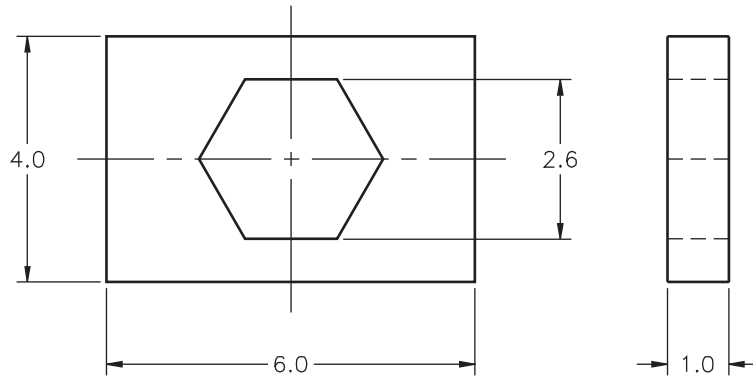
Fig. 26-7
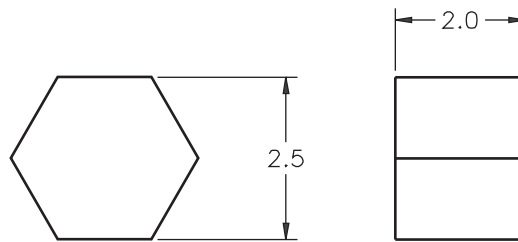
Pin

Fig. 26-8

Part 5 Project | Applying Chapters 23–26

Designing a Bookend

Creating and working with multiple viewports allows the AutoCAD user greater flexibility in preparing, viewing, and plotting drawings. This exercise requires the use of both model space and paper space.

▶ Description

Design a bookend based primarily on one or more initials of your name. Figure P5-1 shows an example of a bookend created using the initials L.B.

Create or modify the design to have a thickness (Z axis) of 2 or more inches. Actual dimensions are not critical, but they should approximate the height and width of a small book.

Keep the following suggestions in mind as you work.

- Begin by drawing everything in model space.
- Switch to paper space and set up four viewports.
- Show the plan view in the lower left viewport, the top view in the upper left viewport, the right view in the lower right viewport, and an isometric view in the upper right viewport.
- Place each view on its own layer. Assign each layer a different color.
- Zoom each viewport appropriately.

▶ Hints and Suggestions

1. Thickness gives the piece a three-dimensional look.
2. Remember as you work that lines and objects created in model space cannot be altered in paper space, and vice versa.
3. Only paper space allows adjustment and manipulation of multiple viewport plotting. Some trial and error may be required to achieve the desired look.
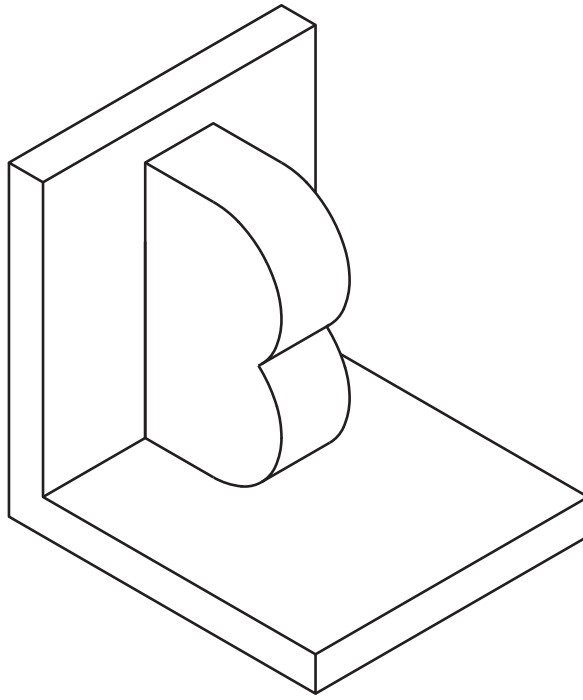
Fig. P5-1

▶ Summary Questions

Your instructor may direct you to answer these questions orally or in writing. You may wish to compare and exchange ideas with other students to help increase efficiency and productivity.

1. What factors determine the drawing setup? (How do you determine the "right" choices?) Why can't all drawings successfully use the same setup?

2. What variables can be modified on each layer?

3. Discuss some advantages of being able to draw on and individually control separate layers.

4. Compare and contrast model space with paper space.

5. What is useful about working in multiple viewports?

Chapter 27

Basic Dimensioning

Objectives

- Set up a text style for dimensions
- Produce linear dimensions using dimensioning commands and shortcuts
- Dimension round shapes, curves, and holes
- Dimension angles
- Determine the need for and use baseline and ordinate dimensioning when appropriate

Vocabulary

baseline dimensions
datum
datum dimensions
dimensioning
dimension line
extension lines
linear dimensions
ordinate dimensions

Technical drawings lack meaning without information that communicates size. Drafters and designers use a method known as **dimensioning** to describe the size of features on a drawing. In this chapter and the two chapters that follow, you will discover the wide range of dimensioning options, settings, and styles that AutoCAD makes available to you for mechanical, architectural, and other types of drafting and design work.

Figure 27-1 shows a drawing of a part that fits into an injection mold (mold insert). We will dimension the drawing, but first we need to prepare the drawing file.

1. Start AutoCAD and use the **Quick Setup** wizard to establish decimal units and a drawing area of 11″ × 8.5″.
2. Select **2D Drafting & Annotation**.
3. Create two new layers using the information in Table 27-1.

Table 27-1. Layers

Layer Name	Color	Linetype	Lineweight
Objects	red	Continuous	0.50 mm
Dimensions	green	Continuous	0.20 mm

4. Set layer **Objects** as the current layer.

5. Set snap at **.25** and grid at **.5**; **ZOOM All**.

6. Produce the drawing shown in Fig. 27-1 on layer **Objects**, but omit the dimensions. From the lower left corner of the object, position the center of the hole **2″** in the positive *x* direction and **1.25″** in the positive *y* direction.

7. Save your work in a file named **dimen.dwg**.
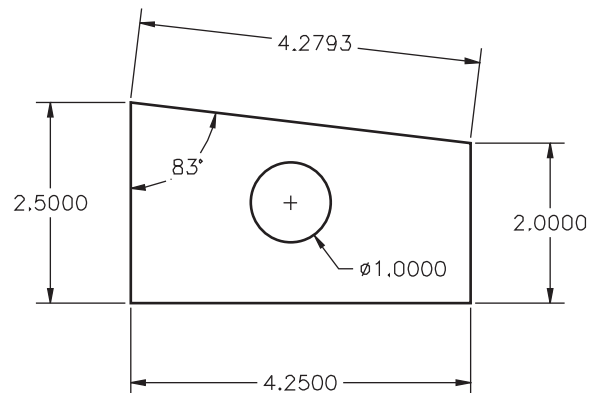
Fig. 27-1

Setting the Dimension Text Style

AutoCAD uses text styles for its dimension text. AutoCAD's **romans** font is suitable for dimension text and is popular among companies that use AutoCAD. This font resembles the Gothic lettering used extensively in hand-produced drawings. For most drafting applications, a text height of .125″ ($^1/_8$″) is standard practice in industry. Some companies use taller text, such as .150″. Whichever height you use, it's important that you use a consistent text height throughout the drawing or set of drawings.

1. Create a new text style named **rom** using the **romans.shx** font. Accept the default settings for the new style.

> **HINT**
>
> Activate the **STYLE** command by entering **ST** or by selecting the **Text Style** pull-down menu on top of the **Text** panel under the **Annotate** tab of the **Ribbon.**

DIMENSIONS

2. On the **Annotate** tab of the **Ribbon,** pick the arrow in the lower right corner of the **Dimensions** panel.

This displays the **Dimension Style Manager** dialog box. We will use it here, but you will learn much more about it in the following two chapters.

3. Pick the **Modify...** button.

4. Pick the **Text** tab.

5. At the right of **Text style**, pick the down arrow and select **rom**.

6. Pick **OK** and then pick **Close**.

Creating Linear Dimensions

DIMENSIONS

Linear dimensions are those with horizontal, vertical, or aligned dimension lines. A **dimension line** is the part of the dimension that typically contains arrowheads at each of its ends, as shown in Fig. 27-2.

1. Make layer **Dimensions** the current layer.

2. Pick the **Linear** button from the **Dimensions** panel on the **Ribbon.**

This enters the **DIMLINEAR** command.

3. In response to **Specify first extension line origin**, pick one of the endpoints of the mold insert's horizontal edge.

4. In response to **Specify second extension line origin**, pick the other end of the horizontal line.

5. Move the crosshairs downward to locate the dimension line 1″ away from the object and press the pick button.

The dimension's **extension lines** are the lines that extend from the object to the dimension line.
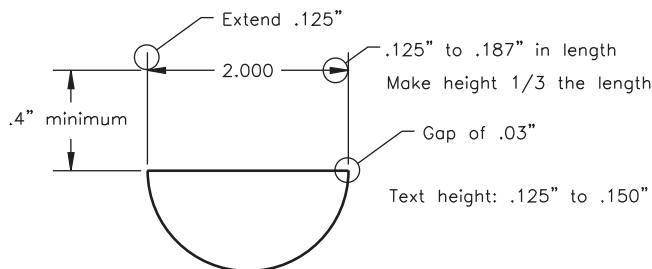
Fig. 27-2

Dimensioning Overall Size

Let's dimension the vertical edges of the object.

1. Press the spacebar to reenter the **DIMLINEAR** command.

2. This time, when AutoCAD asks for the first extension line origin, press **ENTER** or the spacebar.

The crosshairs changes to a pick box.

3. Pick any point on the left edge of the object.

4. Move the crosshairs to the left and pick a point 1″ from the edge to place the dimension.

Let's dimension the other vertical edge, but this time we will use AutoCAD's **Quick Dimension** feature.

5. Pick the **Quick Dimension** button on the **Dimensions** panel or enter the **QDIM** command.

DIMENSIONS

As you can see, a pick box replaces the crosshairs. AutoCAD asks that you select geometry to dimension.

6. Pick the right edge of the object and right-click for **ENTER**.

7. Move the crosshairs to the right and pick a point for the dimension.

Notice that adding the last two dimensions required only three clicks for each dimension.

> **NOTE**
> You can create more than one dimension at a time using the **Quick Dimension** method. However, the results are often unpredictable.

Dimensioning Inclined Edges

Let's dimension the inclined edge by aligning the dimension to the edge.

1. Pick the **Aligned** button from the flyout menu under the **Linear** button on the **Dimensions** panel.

DIMENSIONS

This enters the **DIMALIGNED** command.

2. Press **ENTER**, pick the inclined edge, and place the dimension.

Dimensioning Round Features

DIMENSIONS

Now let's dimension the hole. For mechanical drafting, you should use diameter dimensions for features such as holes and cylinders. Use radius dimensions for features such as fillets and rounds (rounded outside corners).

1. Pick the **Diameter** button from the flyout menu under the **Aligned** button on the **Dimensions** panel.

This enters the **DIMDIAMETER** command.

2. Pick a point anywhere on the hole. (You may need to turn off snap.)

The dimension appears.

3. Move the crosshairs around the circle and watch what happens.

Notice that you have dynamic control over the dimension.

4. Pick a point down and to the right as shown in Fig. 27-1 (page 359).

Dimensioning Angles

DIMENSIONS

Let's dimension the angle, as shown in Fig. 27-1.

1. Pick the **Angular** button from the flyout menu on the **Dimensions** panel.

This enters the **DIMANGULAR** command.

2. Pick both edges that make up the angle.

3. Move the crosshairs outside the mold insert and watch the different possibilities that AutoCAD presents.

4. Pick a location for the dimension arc inside the drawing as shown in the illustration.

5. Save your work. Also, save this file as a template file in your named folder. Name the file **dimen.dwt** and enter **Dimensioning Practice** for the template description.

6. Close the file.

Dimensioning Arcs

Now we will modify our injection mold insert by adding an arc. Then we will dimension the arc in two ways.

1. Pick the **QNew** button on the **Quick Access** toolbar.
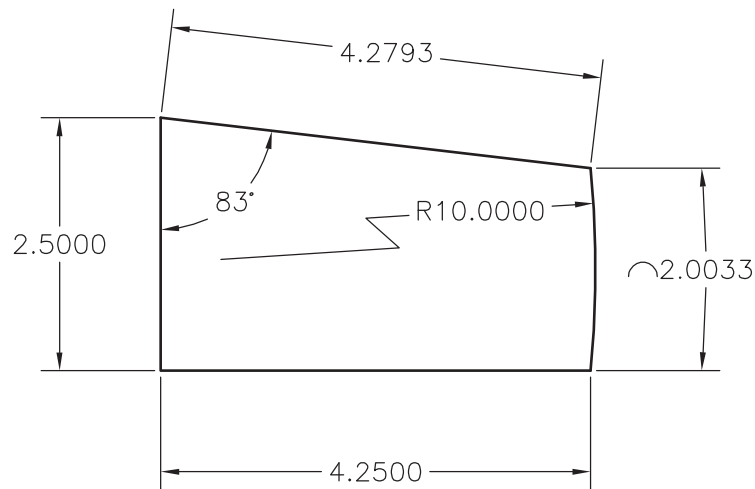
Fig. 27-3

2. Pick **Use a Template** and select **dimen.dwt** as the template file to begin a new drawing.

3. Save the drawing and name it **dimenarc.dwg**.

4. Erase the right edge of the part, the center circle, and the two associated dimensions.

5. Set layer **Objects** as the current layer.

6. Pick the **Arc** button from the **Draw** panel, or enter **A** at the keyboard.

DRAW

7. Create the arc as shown in Fig. 27-3 by following this procedure:

   • Pick the lower right corner of the part as the start point of the arc.

   • Type **E** for **End** and pick the upper right corner of the part as the second endpoint.

   • Type **R** for **Radius**, move the crosshairs to the far left side of the screen, and type **10**.

8. Set layer **Dimensions** as the current layer.

9. Pick the **Arc Length** button from the flyout menu on the **Dimensions** panel.

DIMENSIONS

This enters the **DIMARC** command.

10. Pick the arc you just created.

The dimension appears. Notice the arc symbol at the beginning of the dimension. This indicates that the dimension is the length of an arc and not a linear dimension.

DIMENSIONS

11. Move the crosshairs to the right and pick a location for the dimension.

The radius of this arc is much larger than our part. To dimension large-radius arcs and circles, we use the **DIMJOGGED** command.

12. Pick the **Jogged** button from the flyout menu on the **Dimensions** panel.

13. Pick the arc.

14. In reply to **Specify center location override**, pick a point inside the part near its left edge.

15. Specify a location for the dimension line; then specify a location for the jog. Undo and recreate the jogged dimension until yours looks like the dimension in Fig. 27-3.

The jog in the dimension line indicates that the center point of the dimension is not the true center point of the arc.

16. Save your work and close the file.

Using Other Types of Dimensioning

The dimensioning process you have followed so far in this chapter works for most objects. However, other types of dimensioning are better suited for some objects and engineering drawing applications. In many cases, they produce a cleaner, less confusing drawing.

Baseline Dimensioning

Baseline dimensions are progressive, each starting at the same place, as shown in Fig. 27-4. Baseline dimensioning is most useful on machine drawings, on which precision is critical. Measuring from a single reference dimension reduces the chance of errors that can accumulate from a "stack" of dimensions.

1. Select **dimen.dwt** as the template file to begin a new drawing.

2. Save the drawing and name it **base.dwg**.

3. Erase the drawing and dimensions and create the sheet-metal drawing shown in Fig. 27-4. Place the part on layer **Objects**. Fill the top half of the screen, and estimate the dimensions.

4. Make **Dimensions** the current layer.

DIMENSIONS

5. Pick the **Linear** button and dimension line **A**. It is important that you pick line A's right endpoint first.

DIMENSIONS

6. Pick the **Baseline** button (you need to pick the down arrow next to the **Continue** button on the **Dimensions** panel to see the button).
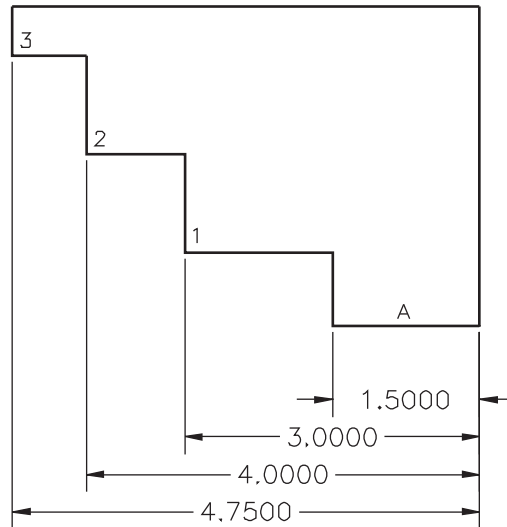
Fig. 27-4

This enters the **DIMBASELINE** command.

7. Pick points **1**, **2**, and **3** in order.

8. Press **ENTER** twice and save your work.

The **Quick Dimension** method also works with baseline dimensioning.

9. Erase the dimensions.

10. Pick the **Quick Dimension** button or enter **QDIM**.

11. Type **ALL** to select the entire drawing and press **ENTER**.

DIMENSIONS

Read the options AutoCAD presents.

12. Enter **B** for **Baseline.**

13. Enter **P** for **datumPoint** and pick the lower right corner of the sheet-metal part.

This establishes the point from which the baseline dimensions start.

14. Place the dimensions.

15. Save your work.

Ordinate Dimensioning

Ordinate dimensions, also known as **datum dimensions**, are similar to baseline dimensions in that both use a datum, or reference, dimension. A **datum** is a surface, edge, or point that is assumed to be exact. One basic difference between baseline dimensions and ordinate dimensions is appearance. Baseline dimensioning uses conventional dimension lines, whereas ordinate

dimensions show absolute coordinates. Both types of dimensioning are especially useful when producing machine parts. Using a datum reduces the chance of error buildup caused by successive dimensions.

1. In the **Menu Browser**, pick **Save As…** to create a new file named **ordinate.dwg**.

2. Create a new layer named **Orddim**, assign the color magenta to it, make **Objects** the current layer, and freeze layer **Dimensions**.

3. Mirror the sheet-metal part to the right (see Fig. 27-5) and delete old objects. If necessary, move the part so it is in the positive *x* and *y* quadrant.

4. On the **Objects** layer, create a hole with a diameter of **.5**. Place the hole **.5** units down and **.5** units to the right of the top left corner.

5. Use **COPY** to create four additional holes. Place the holes as shown in Fig. 27-5.

DIMENSIONS

6. Create a layer named **Center**, and on this layer add center marks to each of the holes. To do this, expand the **Dimensions** panel and pick the **Center Mark** button.
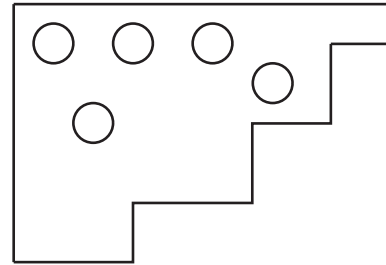
Now we're ready to define the datum, or reference dimension.

DIMENSIONS

7. Make **Orddim** the current layer.

8. Pick the **Ordinate** button from the flyout menu on the **Dimensions** panel to enter the **DIMORDINATE** command.

9. In reply to the **Specify feature location** prompt, pick the upper left corner of the sheet-metal part.

The **Xdatum** option measures the absolute *x* ordinate along the X axis of the drawing. Also, it determines the orientation of the leader line and prompts you for its endpoint. This defines the location of the datum.

10. Enter **X** for **Xdatum**.

11. Pick a point about **1″** above the object to make the ordinate dimension appear.

12. Reenter the command and pick the upper left corner of the sheet-metal part again.

13. Enter **Y** for **Ydatum** and pick a point about **1″** to the left of the object to make the ordinate dimension appear.

The **Ydatum** option measures the absolute *y* ordinate along the Y axis of the drawing. This defines the location of the datum. Your drawing should now look similar to the one in Fig. 27-6, although it will not include the 3.0000 ordinate dimension.

> **NOTE**
>
> The ordinate values may differ in your drawing, depending on where in the drawing area you placed the drawing.

DIMENSIONS

14. Pick the **Ordinate** button.

15. Snap to the center of the hole nearest to the upper left corner, as shown in Fig. 27-6.

16. Pick a point about **1″** from the top of the sheet-metal part. (Snap should be on.)

The X-datum ordinate dimension appears.

17. Repeat Steps 14 through 16 to create X-datum ordinate dimensions for the remaining holes.

18. Create the Y-datum ordinate dimensions on your own using Fig. 27-7 as a guide.

When you finish, your drawing should look similar to the one in Fig. 27-7.

19. Save your work and exit AutoCAD.
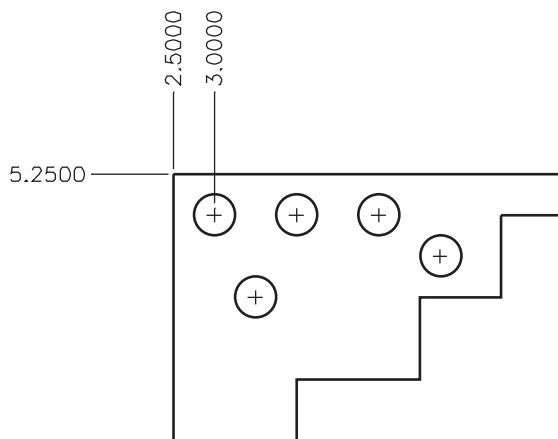
Fig. 27-6

Fig. 27-7
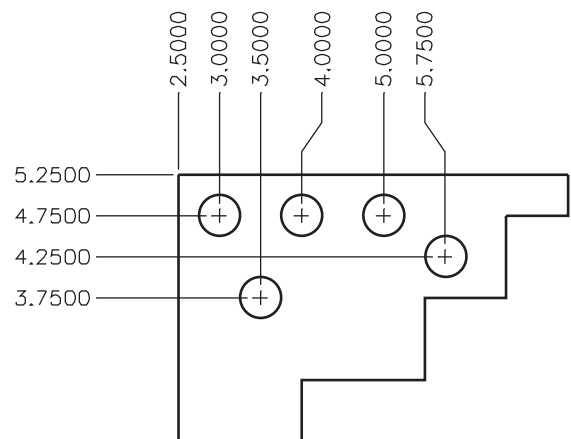
Chapter 27: Basic Dimensioning   **367**

Chapter 27 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. How do you specify a text style for dimension text?
2. Describe the alternative to specifying both endpoints of a line when dimensioning the entire length of an edge.
3. Which dimension button do you use to dimension inclined lines? …to dimension angles?
4. Which dimension buttons do you use to dimension fillets, rounds, and holes? Explain when you would use each button.
5. What does a jog in a radius dimension represent?
6. Describe baseline dimensioning. On what types of drawings is it useful?
7. Describe ordinate dimensioning. On what types of drawings is it useful?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Experiment with altering a linear dimension using its grips. Try to change the location of the dimension line by selecting and moving a grip located at the end of an extension line that is closest to the object. Is it possible? Explain.
2. Drafters generally classify dimensions in two broad categories: size dimensions and location dimensions. What is the difference between them? Are both necessary on every drawing you create? Write a paragraph explaining your findings.
3. What is the relationship between the height of text saved in a text style and the height saved in a dimension style?

Review & Activities   Chapter 27

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter. Begin a new drawing for each problem. Save your work.

1–2. Create the blocks shown in Figs. 27-8 and 27-9. Place object lines on a layer named **Objects** and dimensions on a layer named **Dimensions**. Create a new text style using the **romans.shx** font. Approximate the location of the holes. Refer to **block1.dwg** and **block2.dwg** (These drawings are located on the OLC for this book, in the Drafting & Design Problems section.)
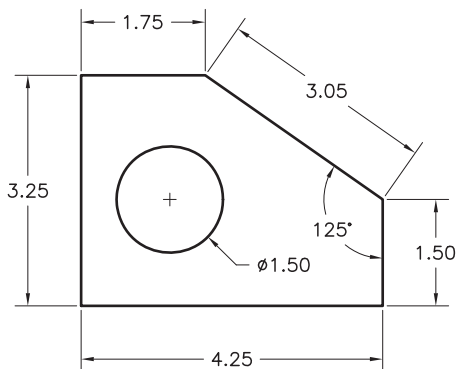
Fig. 27-8
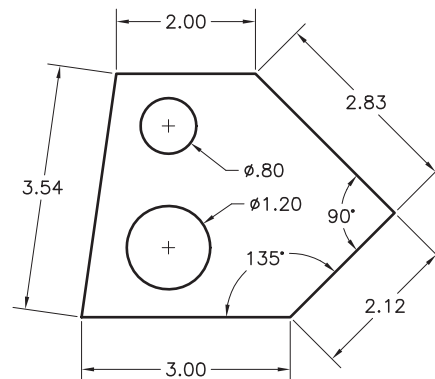
Fig. 27-9

3. Open the **slide.dwg** file that you created at the end of Chapter 24 and dimension it. When you finish, your drawing should look like the one in Fig. 27-10. Save the drawing as **slide2.dwg**.

Fig. 27-10

Continued

Chapter 27: Basic Dimensioning   **369**
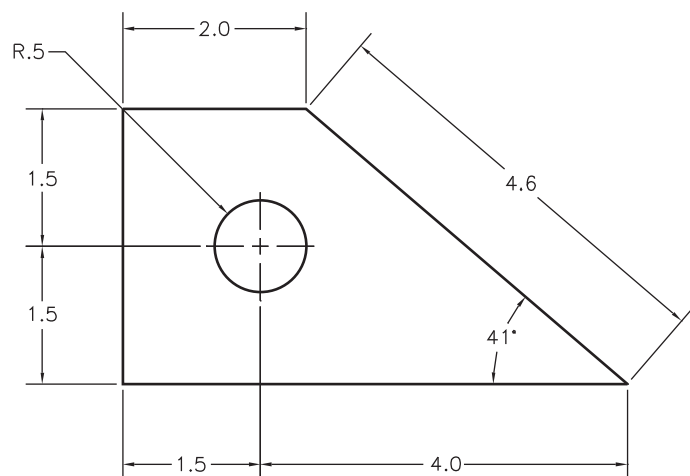
Chapter 27 **Review & Activities**

4. Create the mounting bracket drawing shown in Fig. 27-11. Place the bracket on layer **Objects**. Then dimension the bracket on a layer named **Dimensions**. Refer to the file **bracket1.dwg** (This drawing is located on the OLC for this book, in the Drafting & Design Problems section.)
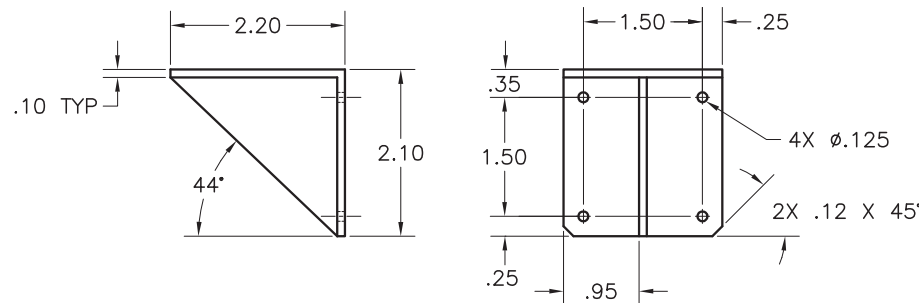
Fig. 27-11

Courtesy of Joseph K. Yabu, Ph.D., San Jose State University

5. Open the **rockerarm.dwg** file that you created at the end of Part 1 on page 112. Place the rocker arm on a layer named **Objects**. Then dimension the part on a layer named **Dimensions**. See Fig. 27-12. Save the file as **rockerarmdim.dwg**.
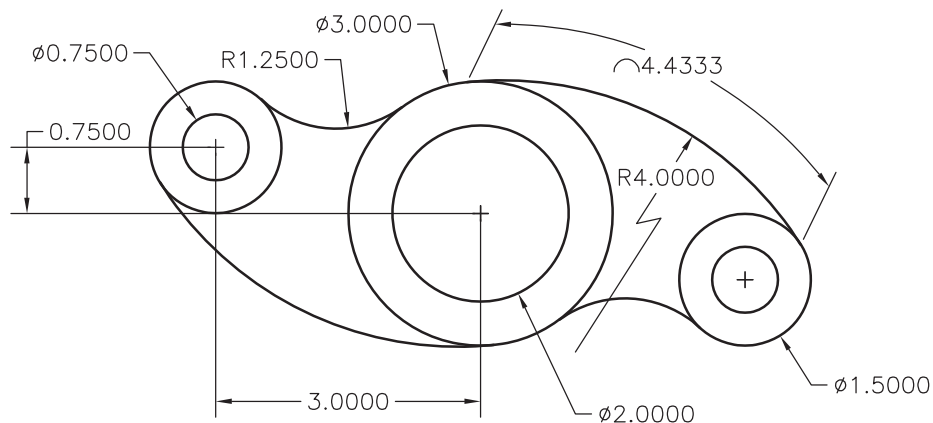
Fig. 27-12

Review & Activities Chapter 27

6. Draw the front view of the shaft lock shown in Fig. 27-13. Create appropriate layers for the dimensions and linetypes. Use the **romans.shx** font to create a new text style named **roms** for the dimension text. Dimension the front view, placing the dimensions on layer **Dimensions**. Show all pertinent dimensions. Also, draw and dimension the right-side view. Resize the drawing area as necessary to accommodate the dimensions of the part. Refer to the file **shaftloc.dwg** (This drawing is located on the OLC for this book, in the Drafting & Design Problems section.)
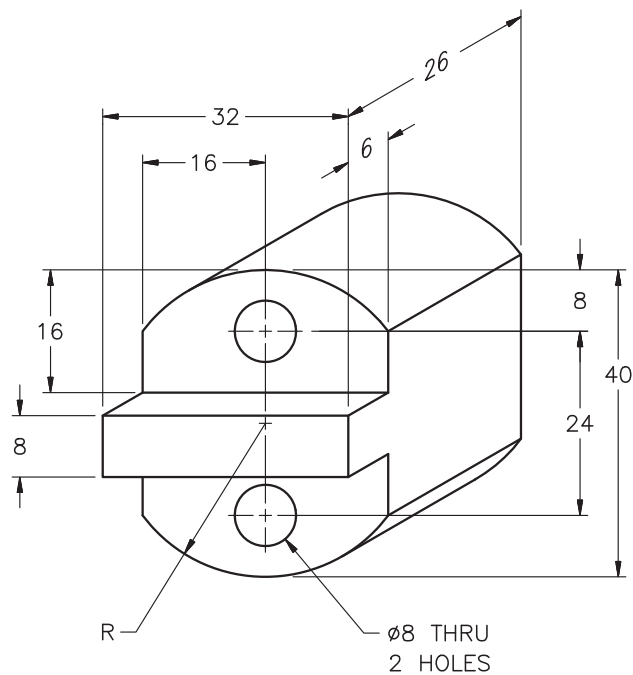
Fig. 27-13

Chapter 27 **Review & Activities**

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. The architect for your firm has given you the elevation sketch shown in Fig. 27-14. Select an appropriate template file, complete the title block, and draw the front elevation; include dimensions. Since the drawing was not made by a drafter, make any changes necessary to comply with correct dimensioning practices. Save the drawing as **elev1.dwg**.
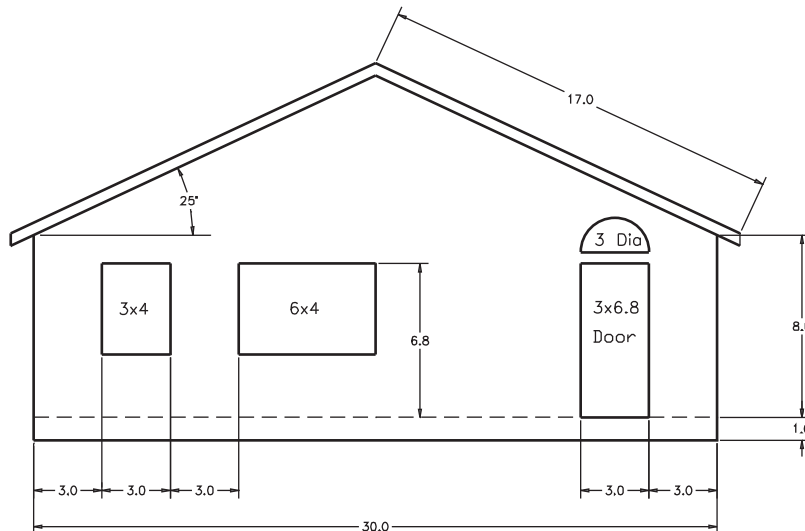
Fig. 27-14

2. The sketch of a shaft support shown in Fig. 27-15 is not to scale; however the following description applies. The base is 2″ square and $1/4″$ thick. The boss (lipped area) is in the center of the base and has an outer diameter of 0.5″, a thickness of $1/4″$, and a through hole with a .35″ radius. The four corner holes are each .15″ in diameter and their center lines are offset from the horizontal and vertical sides by .35″. Draw and dimension two views of the shaft support.
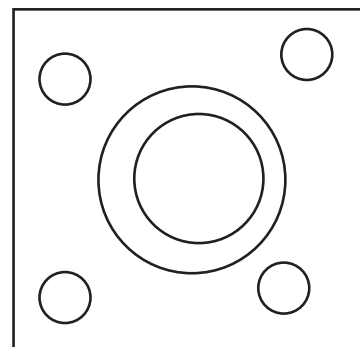
Fig. 27-15

Chapter 28

Advanced Dimensioning

Objectives

- Create a dimension style and apply it to a drawing and template file
- Adjust the dimension text, arrowheads, center marks, and scale in a dimension style
- Edit individual dimensions using a shortcut menu
- Use AutoCAD's associative dimensioning feature to update dimensions on a rescaled drawing
- Apply dimension styles to a drawing template file

Vocabulary

associative dimensions
dimension styles
leaders
spline leader

AutoCAD permits you to create **dimension styles**, which consist of a set of dimension settings. Dimension styles control the format and appearance of dimensions and help you apply drafting standards. They define the format of dimension lines, extension lines, arrowheads, and center marks. They also define the appearance and position of dimension lines and text. Using a dimension style, you can format and define the precision of the linear, radial, and angular units.

Before we can apply a dimension style to something meaningful, we need to produce a drawing. For this chapter, we will work in the **AutoCAD Classic** workspace.

1. Start AutoCAD and select the **tmp1.dwt** file.
2. Select the **AutoCAD Classic** workspace and close all floating toolbars and palettes.
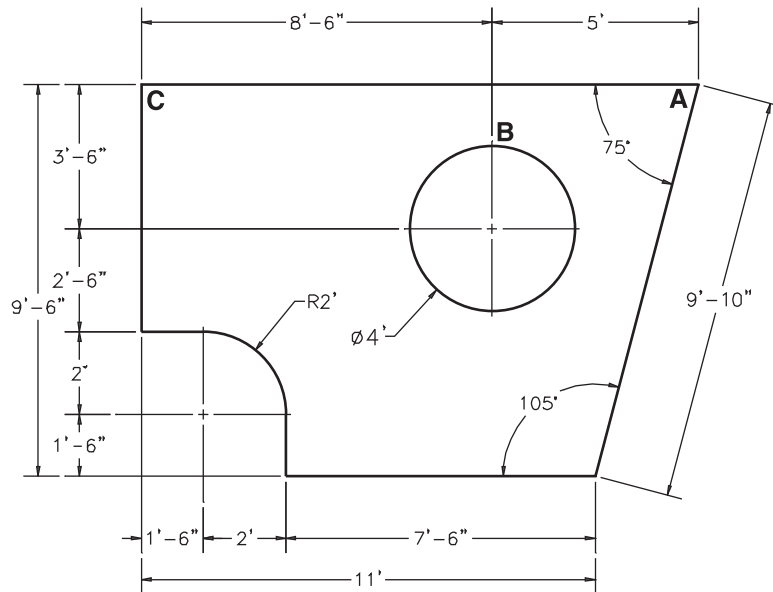
Fig. 28-1

3. On layer **Objects**, create the top view drawing of the redwood deck shown in Fig. 28-1. Do not add dimensions at this time.

The round object represents an outdoor table. It was included in the drawing to ensure that the deck is of sufficient size and shape to accommodate a table of this size.

4. Save your work in a file named **deck.dwg**.

> **HINT**
>
> For best results, begin in the upper right corner and work counterclockwise to draw the edges of the deck. Add the inside rounded corner using the **FILLET** command.

Creating a Dimension Style

Dimension styles provide an easy method of managing the appearance of dimensions. This saves time and effort and helps you produce professional drawings. Note that drafting standards vary from industry to industry and even among companies in the same industry.

1. Display the **Dimension** toolbar (Right-click on any docked toolbar and select **Dimension**).

2. On layer **Dimensions**, place a horizontal dimension along the bottom edge of the deck.

As you can see, the dimension text, arrowheads, etc., are much too small. We need to make some adjustments.

3. Erase the dimension.

4. From the **Dimension** toolbar, pick the **Dimension Style...** button.

DIMENSION

This displays the **Dimension Style Manager** dialog box. As you can see, **Standard** is the default dimension style. The sample drawing shows how the dimensions will appear using this style.

5. Pick the **New...** button.

6. For **New Style Name**, enter **My Style**.

7. Pick the down arrow next to **Start Width** and pick **Standard**.

This selection means that we will start with all of the settings saved in the **Standard** style.

8. After **Use for**, pick the down arrow.

You can apply a dimension style to the types of dimensions included in this list.

9. Select **All dimensions** and pick the **Continue** button.

The **New Dimension Style** dialog box appears, showing several tabs across the top. This is where you change the settings to store in the dimension style.

10. Pick the **OK** button.

11. Pick the **Set Current** button then the **Close** button.

Notice that **My Style** is now the current dimension style.

12. Save your work.

Changing a Dimension Style

Currently, **My Style** is identical to the **Standard** style. We will need to customize the new **My Style** dimension style for our particular drawing. This means changing the size of the arrowheads and center marks, the style and size of the text, and the overall scale of the dimensions.

DIMENSION

Arrowhead Size

1. Pick the **Dimension Style...** button and pick the **Modify...** button.
2. Pick the **Symbols and Arrows** tab unless it is already in front.

You can make adjustments to the arrowheads in the left side of the dialog box.

3. Using the up and down arrows below **Arrow size**, change the value to $1/8''$ and watch closely how the arrowheads in the sample drawing change to reflect the new value. They may be difficult to see.

Sizing Center Marks or Center Lines

In the area titled **Center marks**, you can make changes to the appearance of the centers of circles. Use center marks when space is not available for complete center lines. You may also need to make changes in this area when a company style requires one or the other.

1. Pick the radio button next to **Mark** unless it is already selected.

Notice that the center mark appears on the sample drawing.

2. Click the down arrow until it drops to a value of **0"**.
3. Reselect the radio button next to **Mark** and then set the size to **1/16"**.
4. Select the **Line** radio button and watch to see how the sample drawing changes to reflect the new center lines.

This causes complete center lines to appear at the centers of circles and arcs, as shown in the sample drawing.

Tailoring Dimension Text

Let's create a new text style and height for our dimension style.

1. Pick the **Text** tab.
2. In the upper left area titled **Text appearance**, pick the button containing the three dots.

This causes the familiar **Text Style** dialog box to appear.

3. Create a new style named **roms** using the **romans.shx** font and the default settings.

4. After closing the **Text Style** dialog box, pick the down arrow at the right of **Text style** and select **roms**.

Roms is now the text style for **My Style**, as shown in the sample drawing.

5. In the **Text height** edit box, change the value to $^1/_8''$.

As a reminder, the dimension text should be **.125**″ to **.150**″ in height.

Setting the Units

1. Pick the **Primary Units** tab.

2. At the right of **Unit format**, change the setting to **Architectural**.

3. At the right of **Precision**, change the value to **0′-0**″.

The sample drawing changes to reflect the unit format and precision.

Scaling the Dimensions

1. Pick the **Fit** tab and focus your attention on the area titled **Scale for dimension features**.

This is where you set the scale for dimensions. The value should be the reciprocal of the plot scale. When creating **tmp1.dwt**, we determined that the scale would be $^1/_2'' = 1'$, which is the same as $^1/_{24}$. The reciprocal of $^1/_{24}$ is 24.

2. Double-click the number in the edit box and change it to **24**.

> **NOTE**
>
> You could also enter **DIMSCALE** at the AutoCAD prompt and enter **24**.

There are many other options. We have changed only those needed for the dimension style we will use to dimension the redwood deck. You will learn more about dimension style options in Chapters 29 and 30.

3. Pick **OK**; pick **Close**.

4. Save your work.

Applying a Dimension Style

Now that you have made several adjustments to the dimension style, you are ready to dimension the drawing. Layer **Dimensions** should be the current layer.

DIMENSION

1. Begin by dimensioning the table and rounded corner. Use the **Diameter** button for the table and the **Radius** button for the filleted corner. Place the dimensions as shown in Fig. 28-1 (page 374).

DIMENSION

Now that you have center lines, you can use them to locate the centers of the table and rounded corner using dimensions.

2. Pick the **Linear** button from the **Dimension** toolbar.

DIMENSION

3. For the first and second extension line origins, snap to points **A** and **B**, and then place the dimension **1.5**′ from the edge of the deck.

> **HINT**
>
> Use grid or snap to determine the **1.5**′ distance.

DIMENSION

4. Pick the **Continue** button, snap to point **C**, and press **ENTER** twice to terminate the command.

> **HINT**
>
> When locating the center of the rounded corner with dimensions, use the end of the center lines for the extension line origins, similar to the way you picked point **B**. Separately, using the grips method of moving dimension text, move the **9**′**-6**″ text of the vertical dimension so that it is lower than the **2**′**-6**″ dimension beside it.

Notice that the dimension lines for the two dimensions are aligned when **Continue Dimension** is used. Another way to align dimensions is with the **DIMSPACE** command.

5. With snap off, pick the **Linear** button and place the four short vertical dimensions to the left of the deck. Do not attempt to align the dimension lines.

6. Pick the **Dimension Space** button from the **Dimension** toolbar.

7. In response to **Select base dimension**, pick the **3'-6"** dimension at the top.

8. In response to **Select dimensions to space**, pick the other three dimensions and press **ENTER**.

9. In response to **Enter value**, enter **0** and press **ENTER**.

AutoCAD aligns the four dimensions. **Dimension Space** can also be used to evenly space a series of dimensions that start from a common baseline. One example of this type of dimensioning is Fig. 27-4 on page 365.

10. Using similar methods, add the remaining horizontal and vertical dimensions.

11. Add the aligned and angular dimensions.

Your dimensioned drawing of the redwood deck should now look similar to the one in Fig. 28-1 (page 374).

12. Save your work.

13. If you have access to a plotter or printer, plot the drawing.

DIMENSION

Editing a Dimension

Using AutoCAD's grips, you can change the location of dimension lines and dimension text.

1. Pick one of the linear dimensions.

Five grip boxes appear. The floating **Quick Properties** panel may appear too if **Quick Properties** is toggled on.

2. Pick the grip box located at the center of the dimension text.

3. Move the text up and down, then back and forth, and then pick a new point.

4. Press **ESC**.

By selecting a dimension and right-clicking it, you can make other useful changes.

5. Select one of the linear dimensions and right-click.

A shortcut menu appears.

6. Rest the pointer on **Dim Text position**.

These options permit you to adjust the location of the dimension text.

7. Try each of them, and pick **Home text** last.

As you can see, it's easy to tailor individual dimensions, regardless of the current dimension style.

8. Select one of the linear dimensions and right-click.

9. Rest the pointer on **Dim Style**.

This allows you to save a new style or apply an existing one.

10. Select **Standard**.

The dimension uses settings stored in the **Standard** dimension style. The arrowheads and dimension text are there, but you can't see them because they are so small. The value of the dimension scale in **Standard** is **1**, and we set it to **24** in **My Style**. This means that the text, arrowheads, and other dimensioning features are 24 times larger in **My Style**.

11. Change the dimension's style back to **My Style**.

12. Save your work.

Leaders

Lines with arrowheads at one end that are used to point out a particular feature in a drawing are called **leaders**. Radial and diameter dimensions usually use leaders. The plastic retainer in Fig. 28-2 provides an example of a leader.

Ø.25–6 HOLES
EQ SPACED
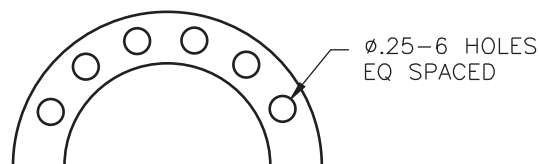
Fig. 28-2

1. Display the **Multileader** toolbar.

2. Pick the **Multileader Style** button on the **Multileader** toolbar.

MULTILEADER

This displays the **Multileader Style Manager** dialog box. **Standard** is the default multileader style.

3. Pick the **New...** button, name the new style **My leader style**, and pick the **Continue** button.

4. In the **Type** area of the **Leader Format** tab, pick the down arrow next to **Straight** and select **Spline**.

Notice how the leader in the sample drawing changes. A **spline leader** is similar to a regular, straight leader, except the leader line can consist of a spline curve, giving you more flexibility with its shape when space is tight in a drawing.

5. Reselect the **Straight** leader type, and change the arrowhead size to **1/8"**.

6. Select the **Leader Structure** tab and specify a leader scale of **24**.

7. Pick **OK**; pick **Close**.

You have created a leader style that matches your dimension style in scale, text size, and arrowhead size. Notice that **My leader style** is now the current leader style.

8. Pick the **Multileader** button on the **Multileader** toolbar.

MULTILEADER

This enters th**e MLEADER** command.

9. Pick point **A** (shown in Fig. 28-1, page 374).

10. Move the crosshairs up and to the right about **1.5'** and pick a point.

The **Text Formatting** toolbar appears, along with a blinking cursor in a blank text editor field.

11. In the first drop-down list, change the text style from **Standard** to **roms**.

12. Enter **USE REDWOOD.**

13. Pick **OK** to close the **Text Formatting** toolbar.

AutoCAD creates the leader. Notice the arrowhead size, the text size, and the text style of the leader match the dimensions.

14. Pick the new leader, pick the down arrow in the **Multileader** toolbar and select **Standard**.

The leader text is now too small to read, because text in the **Standard** leader style is 1/24 the size of text in **My leader style**.

15. Pick the **Undo** button.

16. Save your work.

Working with Associative Dimensions

By default, AutoCAD creates **associative dimensions**—dimensions that update automatically when you change the drawing by stretching, scaling, and so on.

1. Using the **SCALE** command, scale the drawing by **1.1**.

> **HINT**
>
> In response to **Select objects**, enter **All**. Use **0,0** for the base point and **1.1** for the scale factor.

Notice that the dimensions changed automatically. This becomes useful when you need to change a drawing after you've dimensioned it.

2. Using the **STRETCH** command, stretch the rightmost part of the drawing a short distance to the right. (When selecting, use a crossing window.)

The dimensions change accordingly.

3. Undo Steps 1 and 2 and save your work.

Adding Dimension and Leader Styles to Templates

Now you know how to perform the remaining steps (12 through 14) in creating a template file (see Table 23-1 on page 299). Let's apply these steps to the **tmp1.dwt** template.

1. Open **tmp1.dwt**.

> **HINT**
>
> When the **Select File** dialog box appears, you will need to select **Drawing Template File (*.dwt)** at the right of **Files of type**.

2. Create a new dimension style using the following information.
   - Name the dimension style **Preferred**. In the **Create New Dimension Style** dialog box, start with **Standard**. Apply the new style to all dimensions.

- In the **Primary Units** tab, set **Unit format** to **Architectural** and **Precision** to **0′-0″**.

- In the **Symbols and Arrows** tab, set the dimension arrow size to $1/8''$. Pick the **Line** radio button in the **Center marks** area and set the size of the center marks to $1/16''$.

- In the **Text** tab, create a new text style named **roms** using the **romans.shx** font and use the default values.

- Close the **Text Style** dialog box, make **roms** the style for **Preferred**, and set the text height at $1/8''$.

- In the **Fit** tab, set **Scale for dimension features** to **24**.

3. Pick **OK** and close the dialog boxes.

4. Create a new leader style named **Preferred leader**. Set the arrowhead size to **1/8"**, the leader scale to **24**, the text size to **1/8"**, and the text style to **roms**.

5. Save your work and close the file.

The **tmp.dwt** template file is now ready for use with other new drawing files.

6. Close the **Dimension** and **Multileader** toolbars.

7. Exit AutoCAD.

Chapter 28 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Describe the purpose of dimension styles. Why might you need to create one?
2. How do you determine the dimensioning scale?
3. Explain the difference between center marks and center lines. When should you use each?
4. Explain the use of the **Continue** button in the **Dimension** toolbar.
5. When you select a dimension and right-click, AutoCAD displays a menu. Name and briefly describe the three dimension-related options that are available to you.
6. When might you use a spline leader?
7. What is the advantage of using associative dimensions?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. In this chapter, you placed object lines on one layer and dimensions on another. Discuss the advantages of placing dimensions on a separate layer. Are there any disadvantages to placing dimensions on a separate layer? Explain.
2. Describe the changes you would need to make to **tmp1.dwt** to create a template for a B-size sheet at the same drawing scale.

• APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Begin a new drawing and establish the following drawing settings. Store it as a template file named **tmp2.dwt**.

Units: Engineering
Scale: **1″ = 10′** (or **1″ = 120″**)
Sheet size: **17″ × 11″**
Drawing Area: You determine the drawing area based on the scale and sheet size.
Grid: **10′**
(Reminder: Be sure to enter **ZOOM All**.)
Snap: **2′**
Layers: Set up as shown in Table 28-1.

Table 28-1. Layers

Layer Name	Color	Linetype	Lineweight
Thick	red	Continuous	0.50 mm
Thin	green	Continuous	0.25 mm

Create a new dimension style for the template using the following information.

- Name the dimension style **D001**. In the **Create New Dimension Style** dialog box, start with **Standard** and apply the style to all dimensions.
- Set the unit format to **Engineering** and the precision to **0′-0.00″**.
- Set the dimension arrow size to **.125″**.
- Use center marks and set their size to **.1″**.
- Create a new text style named **arial** using the **Arial** font. Use the default settings and make **arial** the current style for **NewStyle**. Set the text height at **.125″**.
- Set the dimension scale to the proper value.

Continued

Chapter 28 **Review & Activities**

2–3. Use the template file you created in problem 1 to create the hotel recreation areas shown in Figs. 28-3 and 28-4.
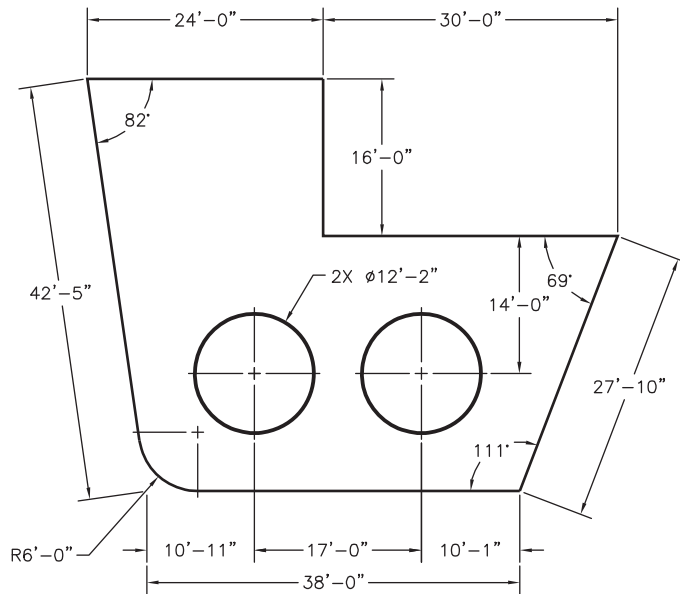
Fig. 28-3
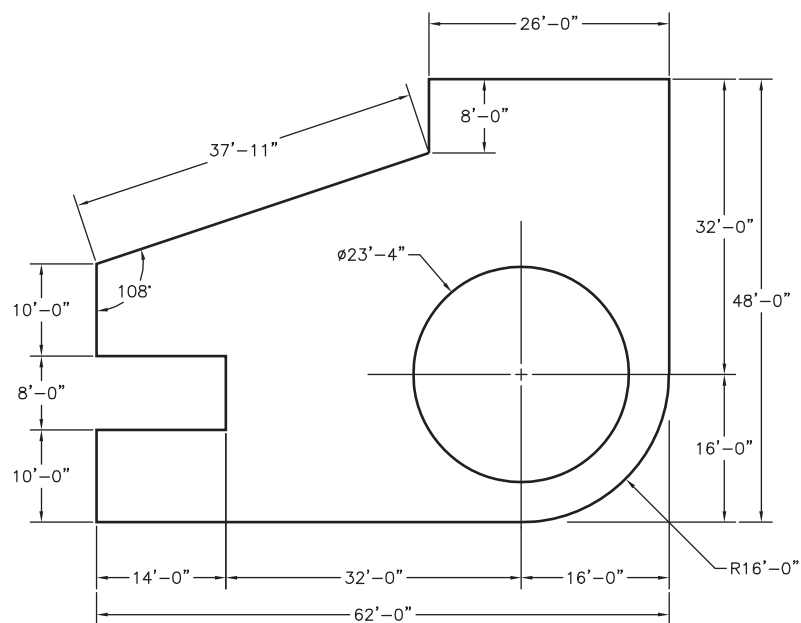
Fig. 28-4

Review & Activities Chapter 28

4. Create and identify the radii of the sheet metal elbow shown in Fig. 28-5. First draw the perpendicular lines, and then create two arcs: one with a radius of 1.5 and the other with a radius of 3. Then dimension the 90° angle and add the notes. Before you create the leaders, open the **Multileader Style Manager** and change the arrowheads to dots. Add the text as shown, and save the drawing as **elbow1.dwg**.
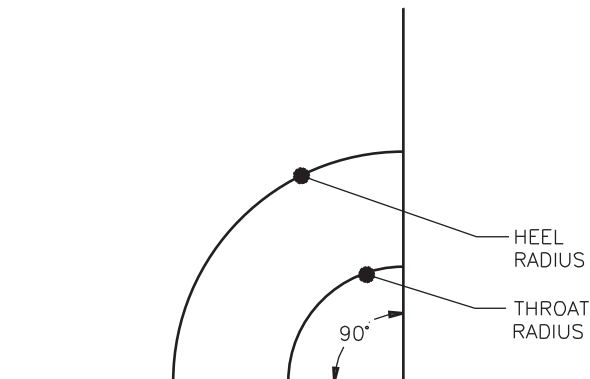
Fig. 28-5

5. Draw the front view of the shaft shown in Fig. 28-6. Use the **Quick Setup** method to change the drawing area to accommodate the dimensions of the drawing. Place the visible lines on a layer named **Visible** using the color green. Create a layer named **Forces** using the color red. Draw and label the four load and reaction forces, using leaders and a text style based on the **romans.shx** font. Create another layer named **Dimensions** using the color cyan. Dimension the front view, placing the dimensions on layer **Dimensions**.
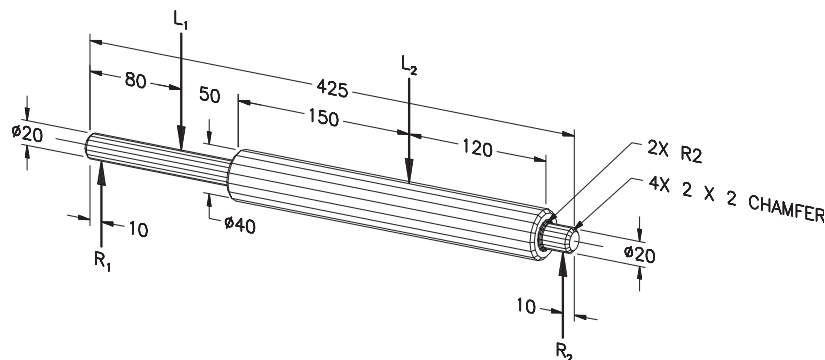
Fig. 28-6

Courtesy of Gary J. Hordemann, Gonzaga University

Continued

Chapter 28 **Review & Activities**

6. Draw and dimension the front view of the metal casting shown in Fig. 28-7. Use the following drawing and dimension settings, and place everything on the proper layer.

Units: Decimal with two digits to the right of the decimal point
Drawing Area: **11 × 8.5**
Scale: **1:1**
Grid: **.1**
Snap: **.05**
LTSCALE: Start with **.5**
Layers: See Table 28-2.
Create a new dimension style using the following information:

• Choose a name for a new dimension style that will help you remember the purpose of this dimension style. Start with **Standard** and apply the style to all dimensions.

• Set the unit format to **Decimal** and the precision to **0.00**.

• Set the dimension arrow size to **.125**.

• Set the size of the center marks to **.1** and use center lines.

• Create and name a new text style. Use **romans.shx**, apply default settings, and make it the current style. Set the text height at **.125**.

• Set the dimension scale to the proper value.
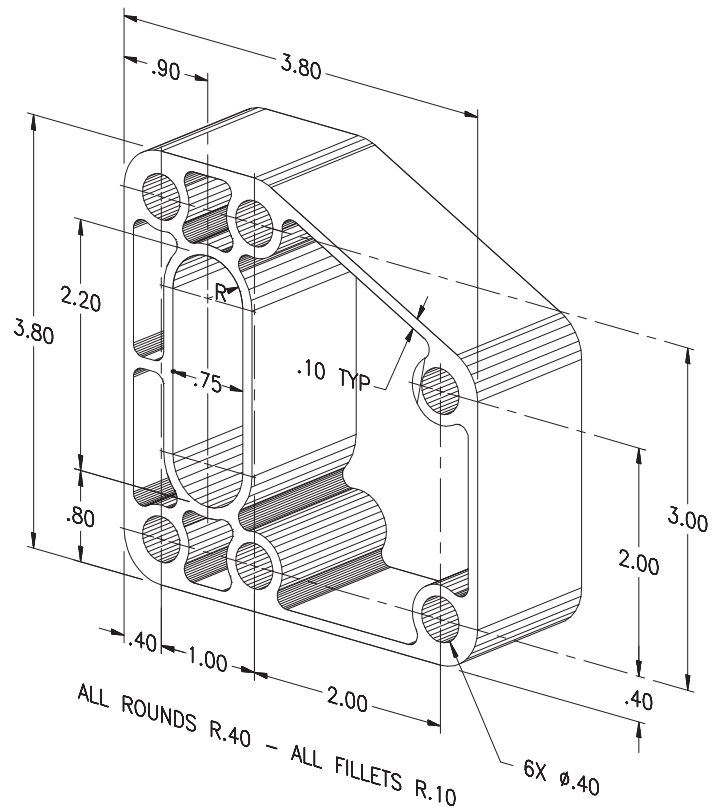
Fig. 28-7

Problem 6 courtesy of Gary J. Hordemann, Gonzaga University

Table 28-2. Layers

Layer Name	Color	Linetype	Lineweight
Visible	red	Continuous	0.50 mm
Dimensions	blue	Continuous	0.25 mm
Text	magenta	Continuous	0.30 mm
Center	green	CENTER	0.20 mm

Review & Activities   Chapter 28
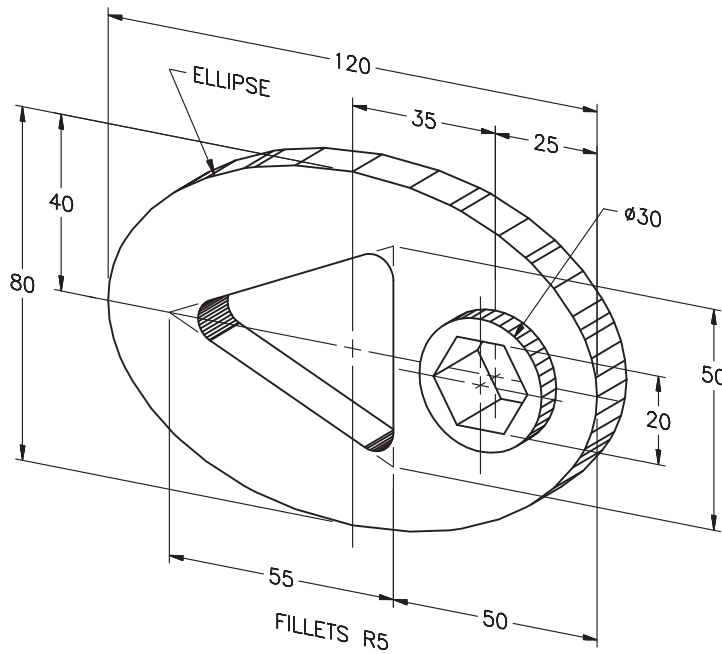
Fig. 28-8

Courtesy of Gary J. Hordemann, Gonzaga University

7. Draw and dimension the front view of the steel plate shown in Fig. 28-8. Use the following drawing, dimension, and leader settings to set up the metric drawing, and place everything on the proper layers. Also draw and dimension the top and right-side views, assuming the plate and boss thicknesses to be 10 and 5, respectively.

Units: Decimal with no digits to the right of the decimal point
Drawing Area: **280 × 216**
Scale: **1:1**
Grid: **10**
Snap: **5**
LTSCALE: Start with **10**
Layers: Set up as shown in Table 28-3 on page 390.
Create new dimension and leader styles using the following information:

• Name the styles using names of your choice. Start with **Standard** and apply the new styles to all dimensions and leaders.

• Set the unit format to **Decimal** and the precision to **0**.

Continued

Chapter 28 **Review & Activities**

Table 28-3. Layers

Layer Name	Color	Linetype	Lineweight
Visible	red	Continuous	0.60 mm
Dimensions	blue	Continuous	0.20 mm
Text	magenta	Continuous	0.20 mm
Center	green	CENTER	0.20 mm
Hidden	magenta	HIDDEN	0.30 mm

- Set the arrow size to **3**.
- Set the size of the center marks to **1.5** and use center lines.
- Create a new text style using a name of your choice. Use **romans.shx**, apply default settings, and make it the current style. Set the text height at **3**.
- Set the scale to the proper value.

● USING PROBLEM-SOLVING SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Your architectural drafting trainee showed you the wall corner shown in Fig. 28-9. Notice that the trainee dimensioned the drawing in a mechanical drafting style. Redraw the corner and set the correct dimension size and units, change the arrowheads to architectural ticks (short diagonal lines), and move the dimensions above the dimension line to conform to your company's style. Save the drawing as **framing1.dwg**.
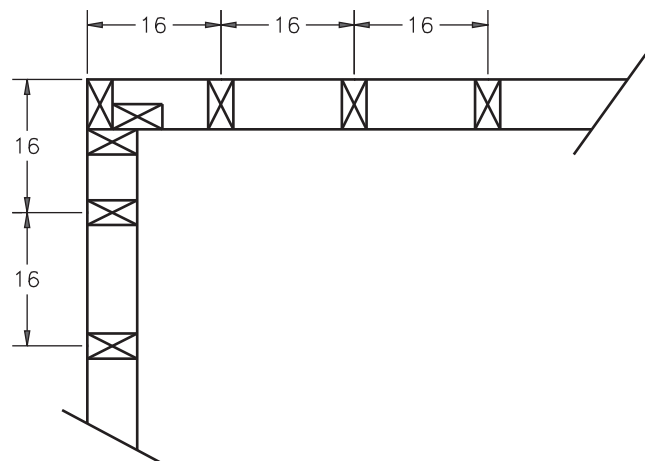
Fig. 28-9

Review & Activities Chapter 28

2. As the set designer for a theatrical company, you are creating a template for a cityscape. The director has given you a sketch of the general building landscape she would like (Fig. 28-10). Draw the template and dimension it from the datum plane. Each square equals 9 inches. Save and plot the drawing using the **Extents** plotting option.
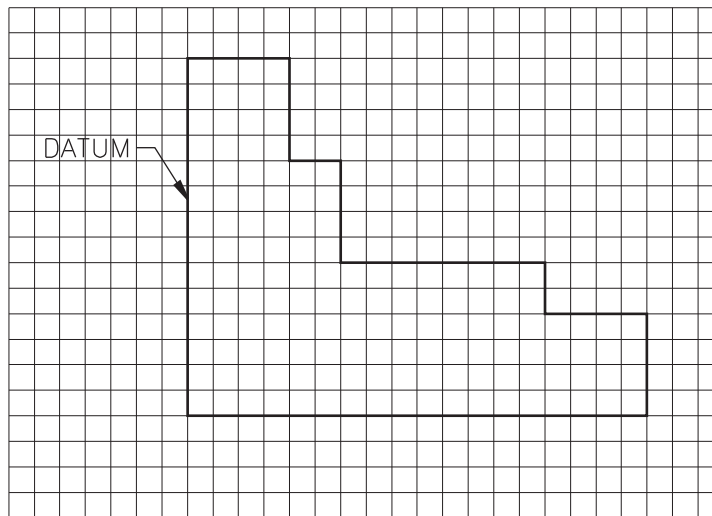
Fig. 28-10

Chapter **29**

Fine-Tuning Dimensions

Objectives

- Adjust the appearance of dimension lines, arrowheads, and text
- Control the placement of dimension text
- Adjust the format and precision of primary and alternate units
- Edit a dimension's properties
- Explode a dimension

Vocabulary

alternate units
primary units

Most industries that use AutoCAD apply dimensioning differently. The architectural industry, for example, uses architectural units and fractions, whereas manufacturers of consumer and industrial products typically use decimal units. Within each industry, companies apply standards, some more rigidly than others. In manufacturing, most use a variation of ANSI or ISO standards and enforce the standard company-wide. Because thousands of companies use AutoCAD worldwide, the software must accommodate a wide range of industry and company standards. This chapter focuses on how you can fine-tune dimensions to meet a particular company's style or standard.

Dimension Style Options

AutoCAD offers a wealth of options for tailoring a dimension style to a specific need.

1. Start AutoCAD and open **deck.dwg**.
2. Select the **2D Drafting & Annotation** workspace and close all floating toolbars and palettes.
3. On the **Menu Browser**, select **Save As...** and enter **deck2.dwg** for the file name.
4. On the **Annotate** tab of the **Ribbon**, pick the arrow in the lower right corner of the **Dimensions** panel to open the **Dimension Style Manager**.

The current dimension style is **My Style**, as shown in the upper left corner of the dialog box. In the last chapter, we covered the purpose of the **New...** and **Modify...** buttons. The **Set Current** button permits you to make another style current. The **Override...** button enables you to override one or more settings in a dimension style temporarily. The **Compare...** button allows you to compare the properties of two dimension styles or view all the properties of one style.

Lines

1. Pick the **Modify...** button and pick the **Lines** tab unless it is already in front.

Focus your attention on the **Dimension lines** area. **Color**, **Linetype**, and **Lineweight** permit you to change the color, linetype, and lineweight of dimension lines. In most cases, it is best not to change them. **Extend beyond ticks** specifies a distance to extend the dimension line past the extension line when you use oblique, architectural tick, integral, and no marks for arrowheads. **Baseline spacing** sets the spacing between the dimension lines when they're outside the extension lines. **Dim line 1** suppresses the first dimension line, and **Dim line 2** suppresses the second dimension line, as shown in Fig. 29-1.
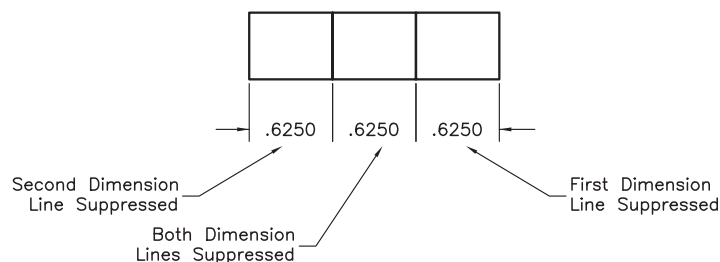
Fig. 29-1

2. Make changes to the settings in the **Dimension lines** area and notice how the sample drawing changes.

Focus on the area labeled **Extension lines**. **Color**, **Linetype**, and **Lineweight** allow you to change the color and lineweight of the extension lines. **Extend beyond dim lines** specifies a distance to extend the extension lines beyond the dimension line. **Offset from origin** specifies the distance to offset the extension lines from the origin points that define the dimension. **Suppress** prevents the display of extension lines. **Fixed length extension lines** allows you to specify a fixed length for all extension lines. **Ext line 1** suppresses the first extension line, and **Ext line 2** suppresses the second extension line. Suppressing an extension line is useful when you want to dimension to an object line. In this case, the object line would serve the same purpose as the extension line.

3. Make changes to the settings in the **Extension lines** area and notice how the sample drawing changes.

Symbols and Arrows

1. Pick the **Symbols and Arrows** tab.

Concentrate on the area titled **Arrowheads**. **First** sets the arrowhead for the first end of the dimension line. **Second** sets the arrowhead for the second end of the dimension line. In most cases, you would use the same arrowhead style for both ends of a dimension, but AutoCAD gives you flexibility in case you need to make them different. **Leader** sets the arrowhead for quick leaders—a leader type that is different than a multileader.

2. To the right of **First**, pick the down arrow and review the types of arrow-heads available.

Some companies and individual architects use ticks instead of arrows at the ends of dimension lines. It would be rare to use **None**, but AutoCAD gives you that option. As you learned in the previous chapter, **Arrow size** displays and sets the size of the arrow. You also learned how to change the appearance of center marks in the **Center marks** area.

3. Make changes to the settings in the **Arrowheads** area and notice how the sample drawing changes.

Text

1. Pick the **Text** tab.

In the **Text appearance** area, you are familiar with **Text style** and **Text height**. **Text color** permits you to change the color of the dimension text. In most

Wohlers: Applying
AutoCAD® 2010

VI. Dimensioning and
Tolerancing

29. Fine–Tuning
Dimensions

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401

circumstances, it is best to leave it alone. **Fraction height scale** sets the scale of fractions relative to dimension text by multiplying the value entered here by the text height. **Draw frame around text** draws a frame around the dimension text. This option would be useful to companies that frame certain notes in a drawing.

2. Make changes to the settings in the **Text appearance** area and notice the changes in the sample drawing.

The **Text placement** area allows you to control the placement of text. **Vertical** controls the vertical justification of dimension text along the dimension line. **Horizontal** controls the horizontal justification of dimension text along the dimension line and the extension line. **Offset from dim line** sets the text gap (distance) around the dimension text when the dimension line is broken to accommodate the dimension text. If the space between the dimension text and the end of the dimension lines becomes too large or small, use this option to adjust the space. It is rare for a company or even a government agency to require a specific distance.

3. Make changes to the settings in the **Text placement** area and notice the changes in the sample drawing.

The **Text alignment** area permits you to change the orientation (horizontal or aligned) of dimension text, whether it is inside or outside the extension lines. **Horizontal** places text in a horizontal position. **Aligned with dimension line** aligns the text with the dimension line. **ISO Standard** aligns text with the dimension line when text is inside the extension lines. If the text is outside the extension lines, it aligns it horizontally.

4. Experiment with the text alignment options.

Fit

1. Pick the **Fit** tab.

Focus on the **Fit options** area. The fit options permit you to control the placement of dimension text and arrows when there is not enough space to put them inside the extension lines.

2. Select each of the fit options and notice how they affect the placement of dimension text and arrows.

The **Text placement** area allows you to control the placement of text when it's not in the default position.

3. Try each of the placement options.

As you discovered in Chapter 28, the **Scale for dimension features** area controls the scaling of dimensions. When you pick **Scale dimension to layout**, AutoCAD determines a scale factor based on the scaling between the current model space viewport and paper space.

In the **Fine tuning** area, if you check **Place text manually,** AutoCAD allows you to place the text at the position you specify. If you check **Draw dim line between ext lines**, AutoCAD draws dimension lines between the measured points even when the arrowheads are outside the extension lines.

4. Experiment with these two options.

Primary Units

The **primary units** are those that appear by default when you add dimensions to a drawing. The term *primary* is used by AutoCAD to distinguish them from alternate dimensions, which are discussed in the next section.

1. Pick the **Primary Units** tab.

This tab establishes the format and precision of the primary dimension units, as well as prefixes and suffixes for dimension text.

You were introduced to the **Linear dimensions** area in Chapter 28, so you know the purpose of **Unit format** and **Precision. Fraction format** controls the format for fractions. **Decimal separator** sets the separator for decimal formats. Options include a period (**.**), comma (**,**), or space ( ). **Round off** sets rounding rules for dimension measurements for all dimension types except angular. **Prefix** and **Suffix** permit you to enter text or use control codes to display special symbols such as the diameter symbol.

2. Experiment with the settings in the **Linear dimensions** area.

Under **Measurement scale**, **Scale factor** establishes a scale factor for all dimension types except angular.

Zero suppression controls the suppression of zeros. Some companies do not use a leading zero, for example. One widely accepted rule of thumb is to include the leading zero in drawings dimensioned using the metric system, but to exclude it when dimensioning using the English system (feet and inches). Space (or lack of it) for the dimension may also be a consideration.

A distance of **0.2000** becomes **.2000** when you check **Leading**. A distance of **2.0000** becomes **2** when you check **Trailing**. A distance of **0′-2¹/4″** becomes **2¹/4″** when you check **0 feet**, and a distance of **2′-0″** becomes **2′** when you check **0 inches**.

Concentrate on the area labeled **Angular dimensions**. **Units format** sets the angular units format, and **Precision** sets the precision for the units. The **Zero suppression** area controls the suppression of zeros in angular dimensions.

   3. Make changes to the settings in the **Angular dimensions** area.

Alternate Units

For some drawings, you may need to show dimensions in more than one type of unit. **Alternate units** are a second set of distances inside brackets in dimensions. The most common reason for using alternate units is to show both inches and millimeters, so millimeters are shown by default when you activate alternate units. However, the conversion factor is customizable, as you will see, so that you can change the alternate units to meet needs for specific drawings.

   1. Pick the **Alternate Units** tab and experiment with the settings in this area.

   2. Pick the **Cancel** button; pick the **Close** button to close the **Dimension Style Manager** dialog box.

Editing a Dimension's Properties

AutoCAD offers additional options for editing a dimension.

   1. Select the **9'-10"** aligned dimension on the right side of the deck, right-click to display the shortcut menu, and review the editing options.

Dim Text position allows you to change the position of the text for the selected dimension. Under the **Precision** option, you can change the linear precision of the dimension. **Flip arrow** will flip the direction of one of the dimension's arrows. With this option, you must first select the side of the dimension nearest the arrow you want to flip. **Dim style** allows you to select from the named styles or to create a new style based on the changes you make to the dimension.

   2. Rest the crosshairs on one of the arrows and right-click.

   3. Pick **Flip Arrow** from the shortcut menu.

   4. Repeat for the other arrow.

   5. Right-click and select **Properties** near the bottom of the shortcut menu.

AutoCAD displays the **Properties** palette.

   6. If information is not displayed under the **General** tab, click the down arrow located to the right of **General**.

AutoCAD provides a list of the dimension's properties.

7. Change the color and lineweight to new values of your choice.

8. Open the **Text** tab by clicking the down arrows located to the right of **Text**.

This displays the settings associated with the dimension text.

9. Change the height to $^3/_{16}''$.

10. Open each of the other tabs, review their contents, and then close them.

As you can see, you can easily tailor the properties of an individual dimension. In most circumstances, all of the dimensions on a drawing should use a consistent style, but AutoCAD gives you the option of changing the individual elements of a particular dimension if necessary.

11. Close the **Properties** palette.

Exploding a Dimension

1. Attempt to erase a single element of any dimension on the drawing, such as the dimension text or an extension line.

AutoCAD selects the entire dimension because it treats an associative dimension as a single object.

MODIFY

2. Press the **ESC** key and pick the **Explode** button on the **Modify** panel of the **Ribbon.**

This enters the **EXPLODE** command.

3. Select any dimension and press **ENTER**.

AutoCAD explodes the dimension, enabling you to edit (move, erase, trim, etc.) the individual parts of the dimension.

4. Select the dimension.

As you can see, it is now possible to select individual parts of the dimension. Sometimes, it is necessary to explode a dimension to edit it because it is impossible to change it by any other way. Use this method, however, as the last alternative because exploded dimensions lose their associativity and you cannot control their appearance using a dimension style.

5. Undo the **Explode,** save your work, and exit AutoCAD.

Chapter 29 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Why is it important for AutoCAD to offer so many settings that control the appearance of dimensions?

2. What are alternate units?

3. Describe a drawing situation in which you might need to rotate dimension text.

4. What is the purpose of the **New** option of the **DIMEDIT** command?

5. How do you adjust the angle of a dimension's extension lines? Why might you choose to make this adjustment?

6. Will the **Home** option of the **DIMEDIT** command work with a dimension that is not an associative dimension? Explain.

7. What is the fastest way of reviewing and changing the properties of an individual dimension?

8. Under what circumstances would you want to suppress one or both dimension lines? Explain.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. A drawing of a complex automotive part has been completed with AutoCAD using the English inch system. Without redimensioning the drawing, how could you change it to show metric units only?

2. Contact two different companies in your area that use CAD and find out what their company dimensioning standards are. How do the two companies compare? How might any differences be explained?

Continued

Chapter 29 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create a new drawing named **prb29-1.dwg**. Plan for a drawing scale of **1″ = 1″** and sheet size of **11″ × 17″**. After you apply the following settings, create and dimension the drawing of an aircraft door hinge component shown in Fig. 29-2. In addition to saving **prb29-1.dwg** as a drawing file, save it as a template file.

   Snap: **.25**
   Grid: **1**
   Layers: Create layers to accommodate multiple colors, linetypes, and lineweights.
   Linetype scale: **.5**
   Font: **romans.shx**
   Dimension style name: **Heather**
   Dimension scale: **1**
   Dimension text height: **.16**
   Arrowhead size: **.16**
   Center mark size: **.08**
   Mark with center lines? Yes
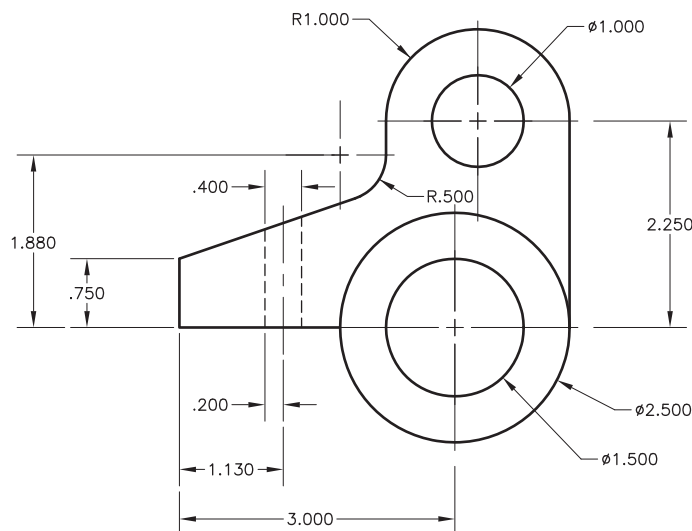
Fig. 29-2

2. Draw and dimension the front view of the oscillating follower shown in Fig. 29-3 on page 402. Use the drawing settings shown below. Draw the top and right-side views, assuming the follower to have a thickness of **8**. This is a metric drawing.

Units: Decimal with no digits to the right of the decimal
Drawing area: **280** × **216**
Grid: **10**
Snap: **5**
Linetype scale: Start with **10**
Text style: Use the **romans.shx** font to create a new text style and set the height at **3**.
Layers: Set up layers as shown in Table 29-1.

Table 29-1. Layers

Layer Name	Color	Linetype	Lineweight
Visible	red	Continuous	0.60 mm
Dimensions	blue	Continuous	0.20 mm
Text	magenta	Continuous	0.35 mm
Center	green	CENTER	0.20 mm
Hidden	magenta	HIDDEN	0.35 mm

Create two dimension styles named **Textin** and **Textout**. Use **Textin** for those dimensions in which the text is inside the dimension lines, and **Textout** for the few radial dimensions in which the text should be outside the extension lines. Use the **Standard** settings except for the following:

Arrowhead size: **3**
Text size: **3**
Distance to extend extension line beyond dimension line: **1.5**
Distance the extension lines are offset from origin points: **1.5**
Distance around the dimension text: **1.5**
Color of dimension text: magenta

Set the dimension style to **Textin** and **Textout** as appropriate for each of the two styles.

Continued

Chapter 29 **Review & Activities**
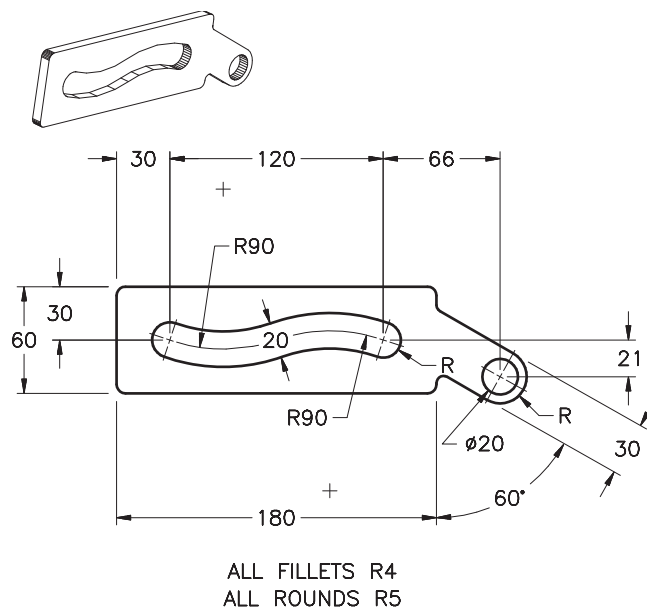
Fig. 29-3

Courtesy of Gary J. Hordemann, Gonzaga University

3. Dimension the gasket as shown in Fig. 29-4. Set the arrowheads to **Integral** and the text font to **Times New Roman**. Display alternate units below the primary values and in the same unit format. Save the drawing as **ch29gasket.dwg**.
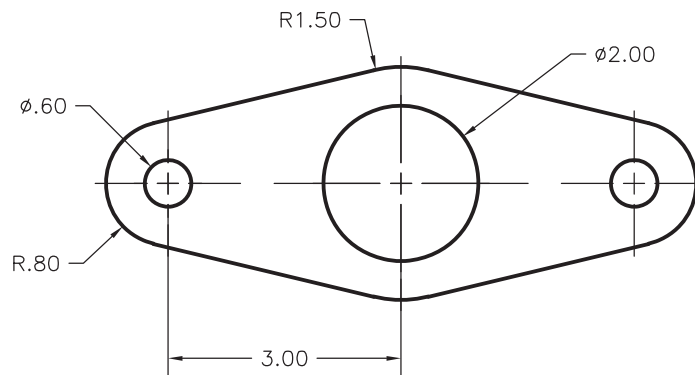
Fig. 29-4

Review & Activities Chapter 29

4. Create a new drawing using the **prb29-1.dwt** template file and make the changes shown in Fig. 29-5. Note that the **2.250** diameter is now **2.500**. Stretch the top part of the hinge component upward **.250** units and let the associative dimension text change on its own.
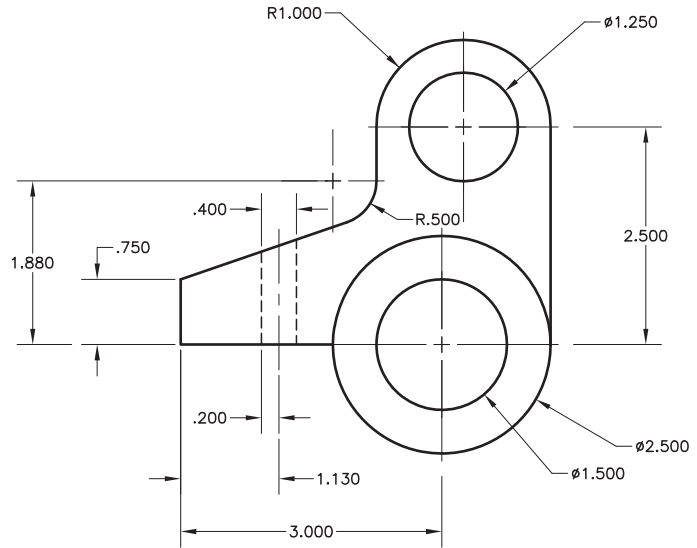
Fig. 29-5

USING PROBLEM-SOLVING SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Architects are noted for their individuality. Open **elev1.dwg** from Chapter 27. Redimension the elevation, including text, with the following style modifications. Save the drawing as **ch29elev-rev1.dwg**.

Text font name: **Trekker**
Arrowheads: Architectural tick
Dimension lines: Extend beyond ticks $1/8$
Vertical text placement: **Outside**

2. Dimension the shim shown in Fig. 29-6. Set the font for **Times New Roman**. Apply **Override** as necessary. Save the drawing as **ch29shim.dwg**.

Fig. 29-6

Chapter 29: Fine-Tuning Dimensions   **403**
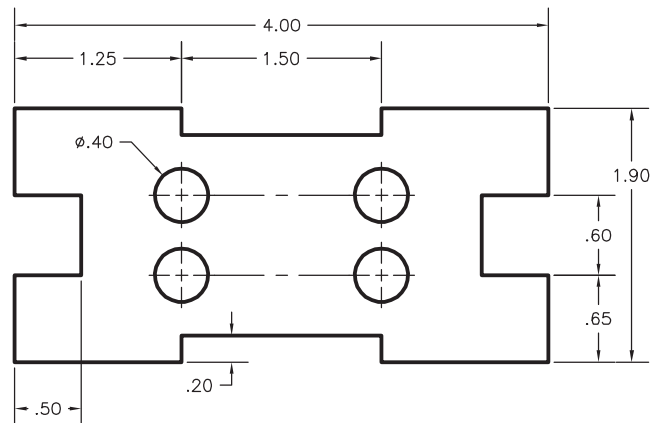
Chapter **30**

Tolerancing

Objectives

- Apply the symmetrical, deviation, limits, and basic methods of tolerancing
- Insert surface controls on a drawing
- Create geometric characteristic symbols and feature control frames that follow industry standard practices for geometric dimensioning and tolerancing
- Add material condition symbols, datum references, data identifiers, and projected tolerance zones to drawings in accordance with industry standards

Vocabulary

basic dimension
deviation tolerancing
feature control frames
geometric characteristic symbols
geometric dimensioning and tolerancing (GD&T)
limits tolerancing
material condition symbols
Maximum Material Condition (MMC)
projected tolerance zone
surface controls
symmetrical deviation
symmetrical tolerancing
tolerance

AutoCAD permits you to add tolerances to dimensions. A **tolerance** specifies the largest variation allowable for a given dimension. Tolerances are necessary on drawings that will be used to manufacture parts because some variation is normal in the manufacturing process.

This chapter applies AutoCAD's tolerancing capabilities and provides a basic introduction to **geometric dimensioning and tolerancing (GD&T)**. This is a system by which drafters and engineers define tolerances for the location of features (such as holes) of a part. The guidelines in this chapter reflect the standards that have been adopted by the American National Standards Institute (ANSI).

Methods of Tolerancing

Tolerances can be shown in more than one way, so AutoCAD provides many options for different kinds and styles of tolerances. The method used depends on the individual drawing. In some cases, you can place the tolerance information in a note or title block. However, when many different tolerances apply to different parts of the drawing, you may need to specify them individually with the dimension information.

1. Start AutoCAD and use the **dimen.dwt** template file to create a new drawing.

2. Select the **AutoCAD Classic** workspace, close all floating toolbars and palettes, and display the **Dimension** toolbar.

3. Pick the **Dimension Style** button from the toolbar and pick the **New...** button.

4. Create a new dimension style named **toler**. Start with **Standard** and use it for all dimensions.

DIMENSION

5. Pick the **Text** tab and change the text height to **0.125**.

6. Pick the **Primary Units** tab and set **Precision** to **0.000**.

7. Pick the **OK** button; pick the **Close** button.

8. Make **toler** the current dimension style.

9. Save your work in a file named **toler.dwg**.

10. Pick the **Dimension Style** button from the toolbar and pick the **Modify...** button.

11. Pick the **Tolerances** tab to display the tolerance options.

Focus your attention on the area titled **Tolerance format**. Notice that many of the settings are grayed out and not available. This is because **Method** is set to **None**. **Method** refers to the approach used to calculate the tolerance. The four methods that AutoCAD makes available, along with an example, are shown in Fig. 30-1 on page 406.
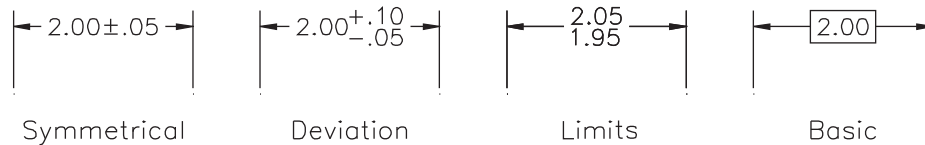
Fig. 30-1

The **Symmetrical** option permits you to create a plus/minus expression tolerance. AutoCAD applies a single value of variation to the dimension measurement. In other words, the upper and lower tolerances are equal. **Deviation** allows you to add different values for the upper and lower tolerance for cases in which the two are not equal. **Limits** adds a limit dimension with a maximum and a minimum value—one on top of the other. **Basic** creates a plain dimension inside a box. A **basic dimension** is one to which allowances and tolerances are added to obtain limits.

12. After **Method**, pick each of the options and notice how the sample drawing changes.

Precision sets the number of decimal places for tolerances. Precision varies with the tolerance precision and the numbering system you use. If you are using the Imperial system (inches, feet, etc.), the basic dimension should have the same number of decimal places as the tolerance. For example, a 1.5-inch hole with a tolerance of ±.005 should be written as 1.500 ±.005. Note that leading zeros (zeros to the left of the decimal point) are often not used with Imperial units.

Upper value sets the maximum or upper tolerance value, and **Lower value** sets the minimum or lower value. **Scaling for height** sets the current height for tolerance text. **Vertical position** controls text justification for symmetrical and deviation tolerances. **Zero suppression** works the same way as it does for primary and alternate units.

The settings in the **Alternate unit tolerance** area sets the precision and zero suppression rules for alternate tolerance units. **Zero suppression** works the same way for tolerancing as it does for primary and alternate units.

13. Experiment with each of the options in the **Tolerance** tab and notice how their settings affect the sample drawing.

14. Pick the **Cancel** button to clear all changes that you made.

Symmetrical Tolerancing

In **symmetrical tolerancing** (also called **symmetrical deviation**), the dimension specifies a plus/minus (±) tolerance in which the upper and lower limits are equal.

1. Pick the **Modify...** button and pick the **Tolerances** tab.
2. Set **Method** to **Symmetrical** and **Precision** to **0.000**.
3. Set **Upper value** to **0.005**.

Lower value is not available because it is the same as the upper value by definition in symmetrical tolerancing.

4. Under **Zero suppression**, check both **Leading** and **Trailing**, and do not change anything else.
5. Pick **OK**; pick **Close**.

Tolerances do not appear yet because the **toler** dimension style was created after the drawing was completed, and **toler** has not yet been applied to the drawing.

6. Select the drawing and dimensions.
7. Make **toler** the current dimension style and press **ESC** to remove the grips.

Your drawing should look similar to the one in Fig. 30-2.
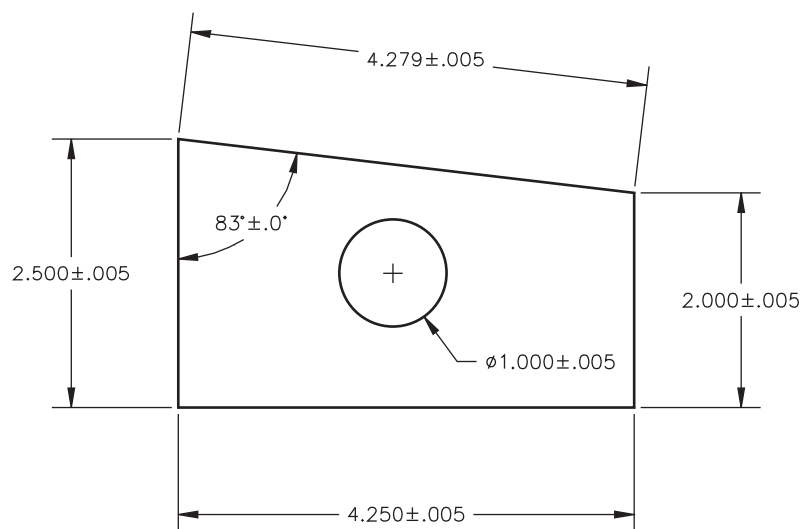
Fig. 30-2
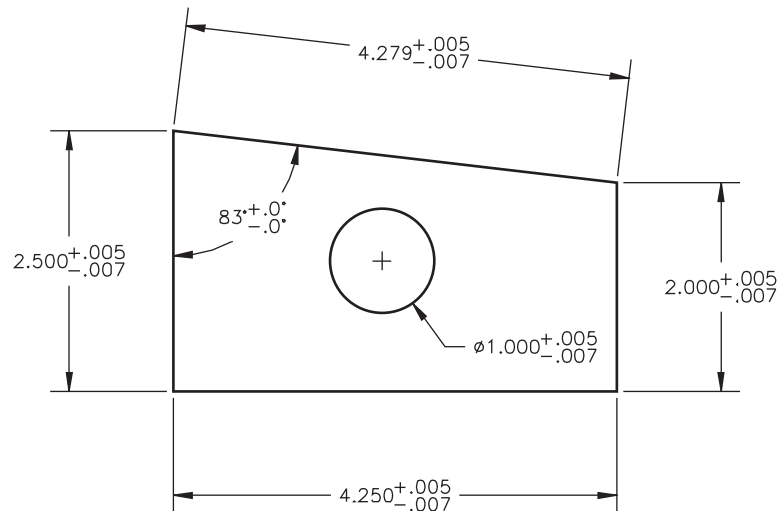
Fig. 30-3

Deviation Tolerancing

Sometimes the upper and lower tolerances for an object are unequal. AutoCAD's **deviation tolerancing** method allows you to set the upper and lower values separately.

1. Open the **Dimension Style Manager**, pick the **Modify...** button, and display the **Tolerances** tab of the **Modify Dimension Style** dialog box.
2. Change **Method** to **Deviation**, set **Lower value** to **0.007**, and leave everything else the same. Pick **OK**.
3. Close the **Dimension Style Manager** and notice how the drawing changes.

Your drawing should now look similar to the one in Fig. 30-3.

Limits Tolerancing

In **limits tolerancing**, only the upper and lower limits of variation for a dimension are shown. The basic dimension is not shown.

1. Open the **Dimension Style Manager** and display the **Tolerances** tab in the **Modify Dimension Style** dialog box.
2. Change **Method** to **Limits** and pick **OK**.
3. Close the **Dimension Style Manager** and notice how the drawing changes.

Your drawing should now look similar to the one in Fig. 30-4.
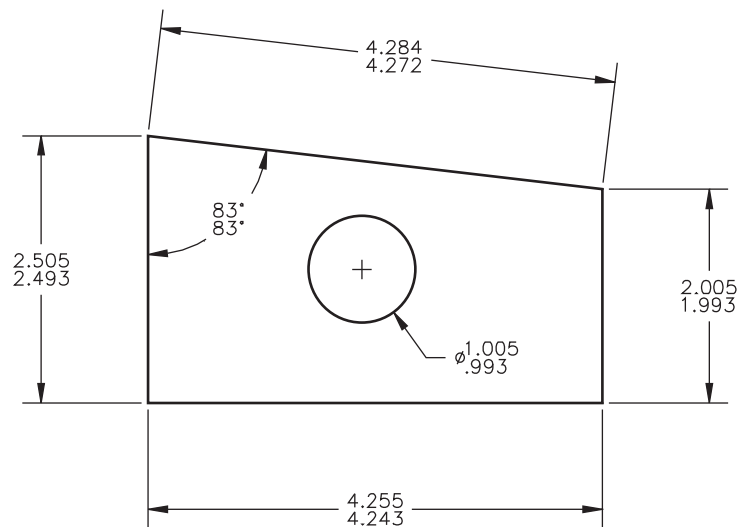
Fig. 30-4

Basic

When you select the **Basic** method, AutoCAD draws a box around the dimension. As you may recall, a basic dimension is one to which allowances and tolerances are added to obtain limits. Therefore, you should use the **Basic** method to present theoretically exact dimensions.

1. In the **Dimension Style Manager**, display the **Tolerances** tab of the **Modify Dimension Style** dialog box.
2. Change **Method** to **Basic** and pick **OK**.
3. Close the **Dimension Style Manager** and notice how the drawing changes.

GD&T Practices

With AutoCAD, you can create geometric characteristic symbols and feature control frames that follow industry standard practices for geometric dimensioning and tolerancing (GD&T). **Geometric characteristic symbols** are used to specify form and position tolerances on drawings. **Feature control frames** are the frames used to hold geometric characteristic symbols and their corresponding tolerances.

Surface Controls

When a feature control frame is not associated with a specific dimension, it usually refers to a surface specification regardless of feature size. For this reason, such frames are often known as **surface controls**.

In Chapter 28, you created leaders using the **MULTILEADER** command. To create leaders for surface controls, you must use the **LEADER** command.

1. Create a new layer named **GDT** and assign the color magenta to it. Make it the current layer and freeze layer **Dimensions**.

2. Enter **LEADER** at the keyboard.

3. In reply to **specify leader start point**, pick the midpoint of the inclined line.

> **HINT** Use the **Midpoint** object snap.

4. Pick a second point about 1 unit up and to the right of the first point to specify the second point of the leader line.

5. Pick a third point to the right of the second point.

6. Press **ENTER** twice to reach the **Annotation Options** and then type **T** for Tolerance.

The **Geometric Tolerance** dialog box appears, as shown in Fig. 30-5.
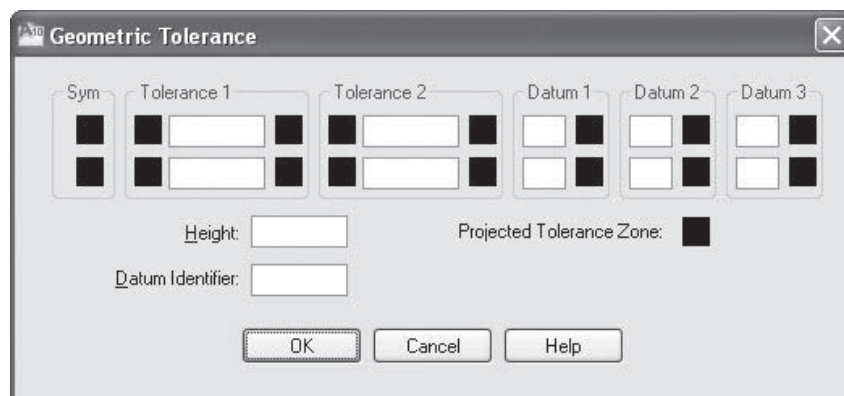
Fig. 30-5

This dialog box permits you to create complete feature control frames by adding tolerance values and their modifying symbols.

Focus your attention on the area labeled **Sym**.

7. Pick the top black box under **Sym**.

This displays the **Symbol** dialog box, as shown in Fig. 30-6. The dialog box contains 14 geometric symbols that are commonly used in GD&T.

8. Pick the flatness symbol (the third symbol on the second row).

AutoCAD adds the symbol to the black box you picked under **Sym**.

9. Under **Tolerance 1**, in the top white box, type **.020** for the tolerance value.

10. Pick **OK**.

The feature control frame appears as a part of the leader, as shown in Fig. 30-7. The feature control frame specifies that every point on the surface must lie between two parallel planes .020 apart.
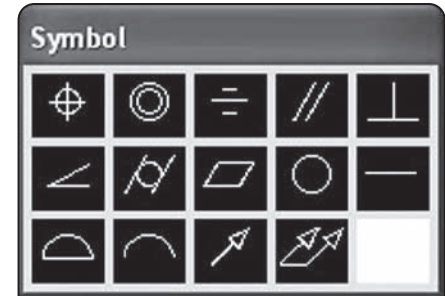
Fig. 30-6
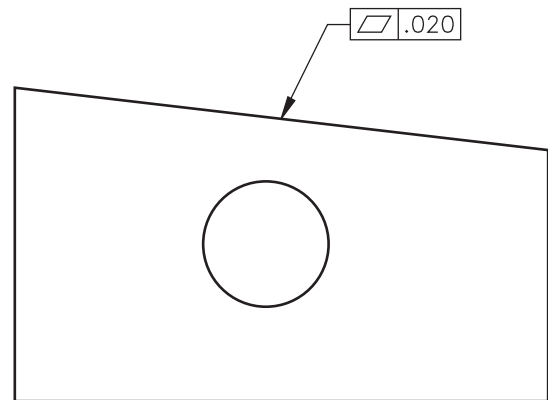
Fig. 30-7

Geometric Characteristic Symbols

To associate geometric symbols with specific dimensions, you must first create the dimension using the appropriate dimensioning command. Then you use the **TOLERANCE** command to add the tolerancing information.

1. Pick the **Diameter Dimension** button from the toolbar and pick a point anywhere on the hole.

2. Pick a second point to position the dimension as shown in Fig. 30-8 on page 412.

Let's change the method of tolerance from **Basic** to **Limits**.

3. Select the new dimension, right click, and pick **Properties** from the shortcut menu.

DIMENSION
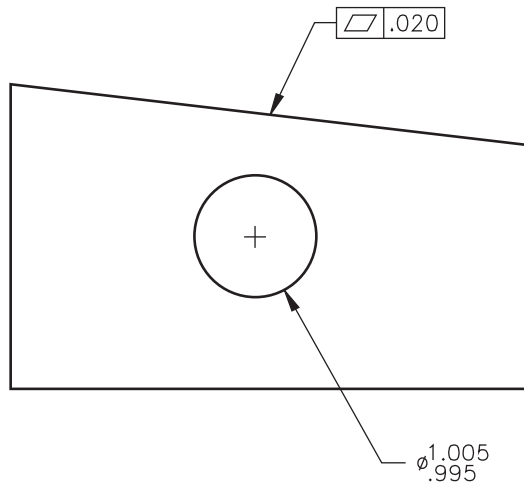
Fig. 30-8

4. In the **Properties** palette, click the down arrow at the right of **Tolerances**, change **Tolerance display** to **Limits**, and close the **Properties** palette.

5. Press **ESC** to remove the selection.

Your drawing should now look similar to the one in Fig. 30-8.

DIMENSION

6. Pick the **Tolerance...** button from the **Dimension** toolbar.

This enters the **TOLERANCE** command and displays the **Geometric Tolerance** dialog box.

7. Under **Sym**, pick the top black box and pick the true position symbol located in the upper left corner of the **Symbol** box.

8. Under **Tolerance 1**, pick the black box located in the upper left corner.

The diameter symbol appears.

9. Pick it again, and again, causing it to disappear and reappear.

As you can see, this serves as a switch, toggling the diameter symbol on and off.

10. In the top white box under **Tolerance 1**, type **.010** for the tolerance value.

Material Condition Symbols

The black box located at the right of the white text box specifies material condition. **Material condition symbols** are used in GD&T to modify the geometric tolerance in relation to the produced size or location of the feature.

In this case, we will use the symbol for **Maximum Material Condition (MMC)**. MMC specifies that a feature, such as a hole or shaft, is at its maximum size or contains its maximum amount of material.

11. At the right of the same white box, pick the black box.

This displays the **Material Condition** dialog box, as shown in Fig. 30-9.

12. Pick the first symbol, which is the symbol for Maximum Material Condition.

The symbol appears in the **Geometric Tolerance** dialog box.

Fig. 30-9

Datum References and Datum Modifiers

AutoCAD allows you to specify datum reference information on three different levels (primary, secondary, and tertiary).

1. In the top white boxes under **Datum 1**, **Datum 2**, and **Datum 3**, enter **A**, **B**, and **C**, respectively.

You can also associate a material condition with any or all of the datum references. To do this, you would pick the black box to the right of the datum boxes and select the appropriate material condition symbol. For the current drawing, do not use material condition symbols.

Datum Identifier near the bottom left of the dialog box allows you to enter a feature symbol to identify the datum.

2. In the box located to the right of **Datum Identifier**, enter **-D-**.

The **Geometric Tolerance** dialog box on your screen should now look like the one in Fig. 30-10.
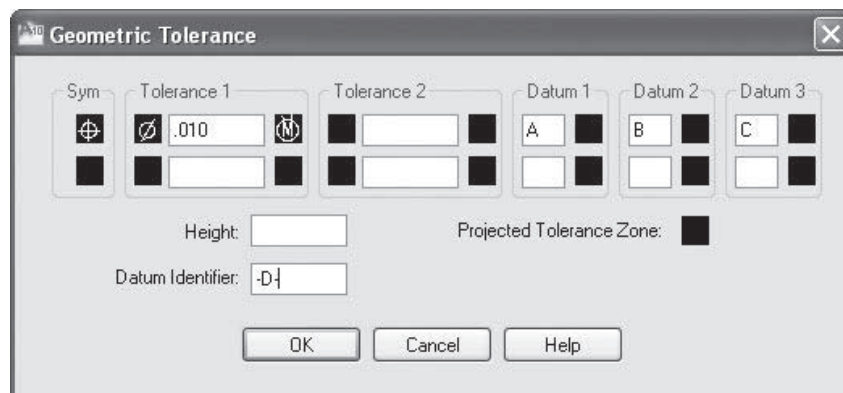
Fig. 30-10

The black and white boxes in the second row permit you to create a second feature control frame. You would need to use a second feature control frame when the feature has two geometric characteristics that are of dimensional importance.

3. Pick the **OK** button and position the feature control frame as shown in Fig. 30-11.

4. Save your work as **tolerance.dwg**.
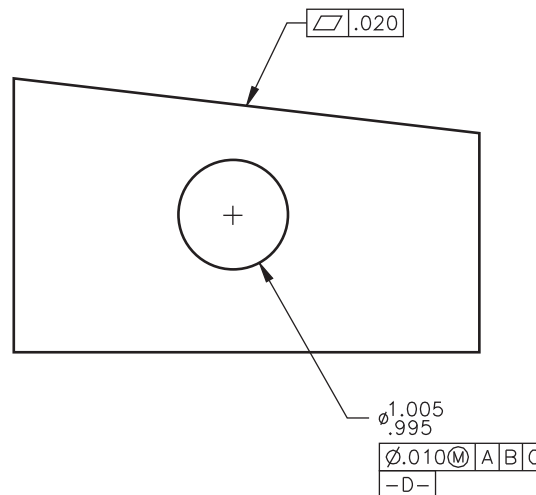
See Appendix D for more information about geometric symbols.

Fig. 30-11

Projected Tolerance Zone

You can specify projected tolerances in addition to positional tolerances to make a tolerance more specific. The **projected tolerance zone** controls the height of the extended portion of a perpendicular part. To display a projected tolerance, you would pick the black box next to **Projected Tolerance Zone** in the **Geometric Tolerance** dialog box. Then, under the **Tolerance 1** area of the dialog box, enter a value for the height. Doing so creates a projected tolerance zone in the feature control frame.

Experiment with placing a projected tolerance zone in a feature control frame by following the steps below.

1. Enter the **UNDO** command and **Mark** option.

2. Create another feature control frame.

3. Pick the black box next to **Projected Tolerance Zone** and enter a height in the **Height** box.

4. Place a dimension to see how the projected tolerance appears.

5. Enter the **UNDO** command and specify the **Back** option.

6. Close the **Dimension** toolbar and exit AutoCAD.

Chapter 30 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is the purpose of adding tolerances to a drawing?
2. When is it necessary to include tolerances on a drawing?
3. Give an example for each of the following tolerancing methods.
   a. **Symmetrical**
   b. **Deviation**
   c. **Limits**
4. Which tolerancing method should you choose in AutoCAD if the upper tolerance value of a dimension is different from the lower value?
5. What is a surface control? How can you create a surface control in AutoCAD?
6. What is a geometric characteristic symbol?
7. Explain how you would include material condition symbols in feature control frames.
8. How would you include a second feature control frame below the first one?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. With AutoCAD, you can quickly produce feature control frames, complete with geometric characteristic symbols. Investigate ways of editing them.
2. When a second feature control frame is added to a dimension, how does this affect the meaning of the first feature control frame?
3. Investigate material condition symbols and their meanings. In what way do material conditions affect dimensions?

Continued

Chapter 30 **Review & Activities**

● **APPLYING AUTOCAD SKILLS**

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create the two-view drawing of the plug according to the dimensions shown in Fig. 30-12. Use a drawing area of **100,80**, and name the drawing **plug.dwg**. Dimension it as shown in Fig. 30-13 to show the runout of the plug. (*Runout* is the form and location of a feature relative to a datum.) Notice that the limits are different on the two holes.
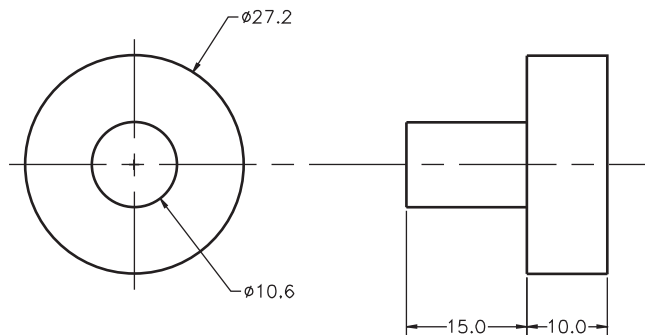
Fig. 30-12
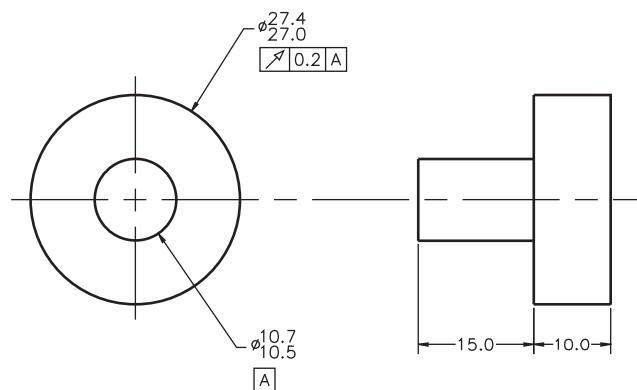
Fig. 30-13

Review & Activities Chapter 30

2. Create a new drawing named **ch30gasket.dwg** and draw the gasket shown in Fig. 30-14. Dimension the gasket using limits tolerancing, and specify an upper value of **.02**. Create a feature control frame to show a true position diameter tolerance of **.025** at maximum material condition.

4X ø0.500 EQUALLY SPACED
R2.250
R0.500 TYP
R .375 TYP
ø3.000
45°
ø4.0000
MATL .030 THK

UNLESS OTHERWISE SPECIFIED ALL
DIMENSIONS ARE IN INCHES
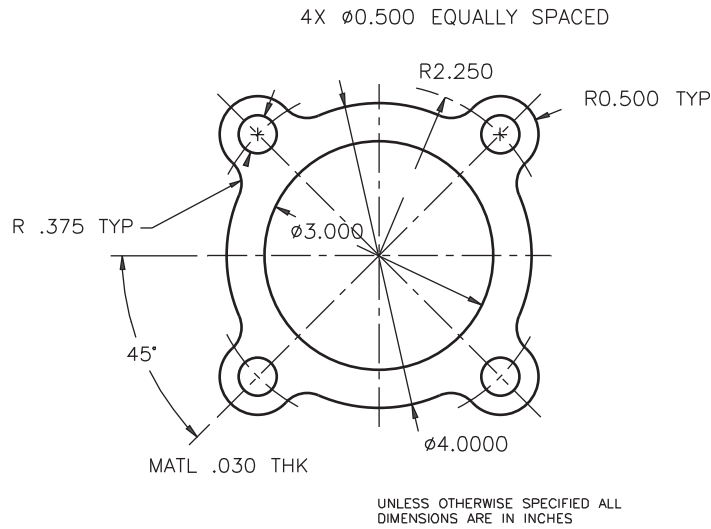
Fig. 30-14

3. Create a new drawing named **hinge.dwg** and draw the aircraft door hinge component shown in Fig. 30-15. Dimension the hinge using symmetrical tolerancing and specify tolerances as shown.
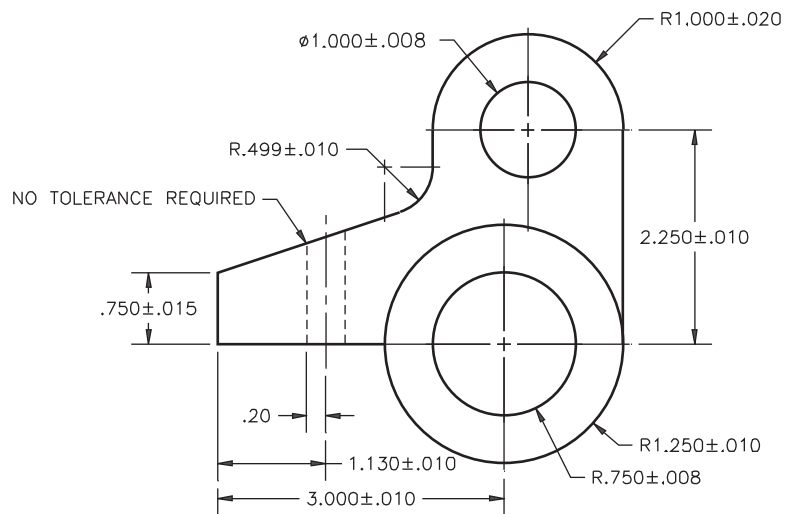
ø1.000±.008
R1.000±.020
R.499±.010
NO TOLERANCE REQUIRED
2.250±.010
.750±.015
.20
R1.250±.010
1.130±.010
R.750±.008
3.000±.010

Fig. 30-15

Continued

Chapter 30: Tolerancing   **417**

Chapter 30 **Review & Activities**

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. The drawing you dimensioned in Chapter 27, **slide2.dwg**, will be used in a new CAD textbook to illustrate dimensioning styles. Save the drawing with a new name of **slide3.dwg**. Create two new layers named **Symmetrical** and **Deviation**. Freeze the layer on which the current dimensions were drawn. Make the **Symmetrical** layer current and redimension the drawing with tolerances expressed in the symmetrical style. Then freeze **Symmetrical**, make **Deviation** current, and redimension using the deviation style. The tolerance on the angle is **±15** units, on the radius **+0.02°** and **−0.00°**, and all others **±.05** unit. Plot the drawing twice to show the two different dimensioning methods.

2. Your editor also wants an example of GD&T with limits dimensioning. Save **slide3.dwg** as **slide4.dwg**. Redimension the drawing to maintain the **3.0**-unit vertical side perpendicular to the **5.5**-unit base within **0.02** units. Save the drawing.

AutoCAD at Work

Prototype Testing Expert

Traditional prototype testing is a long and expensive proposition. First, the physical prototype has to be built at full size using the materials to be used in the final product. Then the prototype has to be tested—a process that often involves expensive testing devices and sometimes the actual destruction of the prototype.

Now, using computers and high-end CAD modeling and testing software, companies can carry out much of their testing on virtual prototypes before a physical prototype is created.

Advantages of Virtual Prototypes

Courtesy of Objet Geometries Ltd.

Testing a virtual prototype has many advantages. It reduces costs and shortens design time by allowing designers to eliminate design flaws without having to build another prototype each time a major change is made to the product.

But testing virtual prototypes also increases safety. For example, in the illustration, air flow around a hovering fighter plane is being analyzed using a computer simulation. The pattern of air flow caused by the jet's movable exhaust nozzles is critical. If hot air from the exhaust is sucked into the plane's engines, the plane may crash. Using a physical prototype for this test would put pilots at serious risk.

Testing Experts

In addition to understanding the software, prototype testers should have a solid background in math and perhaps even engineering in the field in which they are testing. They must have an eye for detail to catch the slightest indication of possible design flaws.

▶ Career Activities

1. Choose a manufacturer of a vehicle (car, plane, train, space vehicle, etc.) and find out how the company tests prototypes for new designs.

2. Research job openings in this field. What skills would you need to become a prototype tester?

Chapter **31**

Calculations

Objectives

- Find the exact coordinates of points
- Calculate the distance between specific points in a drawing
- Calculate area and circumference of objects in a drawing
- Use AutoCAD's online geometry calculator to place objects at precise points in a drawing
- Use **QuickCalc** to perform arithmetic, algebraic, and trigonometric calculations
- Display information from AutoCAD's drawing database on objects and entire drawings
- Divide an object into equal parts
- Place markers or points at specified intervals on an object

Vocabulary

area
circumference
delta
drawing database
integer
perimeter
vector

With the power of the computer, AutoCAD can perform measurement and calculation tasks that would be too time consuming to do manually. An example is calculating miles of a chain-link fence on a drawing.

The drawing in Fig. 31-1 shows an apartment complex with parking lots, streets, and trees. With AutoCAD, you can calculate the square footage of the parking lot and the distance between parking stalls on such a drawing.
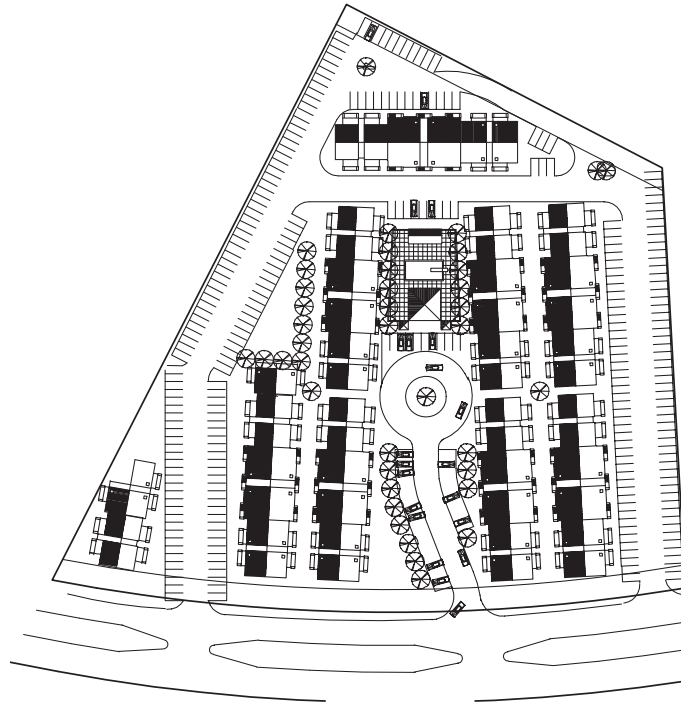
Fig. 31-1

Performing Calculations

AutoCAD includes commands that help you find the coordinates of specific points on a drawing, calculate the distance between two points, calculate the area of an object, and perform other geometric calculations.

1. Start AutoCAD and start a new drawing using the **dimen.dwt** template file.

2. Select the **2D Drafting & Annotation** workspace and close all floating toolbars and palettes.

3. Erase the drawing and dimensions.

4. Create the drawing of the end view of a shaft with a square pocket machined into it (Fig. 31-2). Use the following guidelines:

   • Create the end view on the layer named **Objects**.
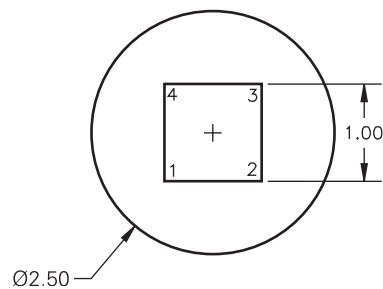
Fig. 31-2

- Use the sizes shown, but omit dimensions.
- The shaft and pocket share the same center point.
- Use **RECTANG** to create the square pocket.

5. Save your work in a file named **calc.dwg**.

Locating Points

UTILITIES

Let's find the coordinates of point 1.

1. Pick the **ID Point** button on the expanded **Utilities** panel or type **ID** at the keyboard to enter the **ID** command.

2. In response to **Specify point**, pick point **1**.

AutoCAD displays the coordinates of point 1.

3. Determine the coordinates of point **2**.

Calculating Distances
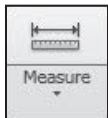
UTILITIES

The **DIST** command calculates the distance between two points.

1. On the **Utilities** panel of the **Home** tab of the **Ribbon**, select the **Distance** button.

AutoCAD enters the **DIST** command.

2. Pick points **1** and **3**.

In addition to the distance, AutoCAD calculates angle and delta information. If you have studied geometry, you may be familiar with the use of the term **delta** to mean "change." **Delta X** is the distance, or change, in the X direction, and **Delta Y** is the distance in the Y direction from one point to another.

Calculating Area

The **AREA** command determines the area and perimeter of several different object types. **Area** is the number of square units (inches, acres, miles, etc.) needed to cover an enclosed two-dimensional shape or surface. **Perimeter** is the distance around a two-dimensional shape or surface.

INQUIRY

1. Select the **Area** button from the flyout menu of the **Utilities** panel.

AutoCAD enters the **AREA** command.

2. Enter **O** for **Object** and pick the square pocket.

AutoCAD displays the area and perimeter of the pocket. Suppose we want to know the area of the end of the shaft minus the pocket.

3. Reenter **AREA** and enter **A** for **Add**.

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VI. Dimensioning and
Tolerancing

31. Calculations

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429

This puts the command in **Add** mode.

4. Enter **O** for **Object** and pick the shaft.

AutoCAD displays **4.9087** for the area and **7.8540** for the circumference. (**Circumference** is the distance around a circle.) Notice that AutoCAD is still in **Add** mode and is asking you to select objects.

5. Press **ENTER**.

6. Enter **S** for **Subtract**.

Now the command is in **Subtract** mode.

7. Enter **O** for **Object** and select the pocket.

AutoCAD subtracts the area of the pocket from the area of the end of the shaft and displays the result (**3.9087**).

8. Press **ENTER** twice to terminate the command.

Using the Geometry Calculator

With the **CAL** command, AutoCAD offers a geometry calculator that evaluates vector, real, and integer expressions. A **vector** is a line segment defined by its endpoints or by a starting point and a direction in 3D space. An **integer** is any positive or negative whole number or 0. You can use the expressions in any AutoCAD command that requests points, vectors, or numbers.

1. Enter the **CAL** command.

2. Type **(3*2)+(10/5)** at the keyboard and press **ENTER**.

As in basic algebra, the parts in parentheses are calculated first. The asterisk (*) indicates to multiply and the forward slash indicates to divide the numbers.

AutoCAD calculates the answer as **8.0**. You can also use **CAL** with object snaps to calculate the exact placement of a point. For example, suppose you wanted to add a feature to the shaft that required starting a polyline halfway between the center of the shaft and the lower left corner of the machined pocket.

3. Enter the **PLINE** command.

4. Enter **'CAL**. (Note the leading apostrophe.)

By preceding the command with an apostrophe, you have entered the command transparently. In other words, when you complete the **CAL** command, the **PLINE** command will resume.

5. Type **(cen+end)/2** and press **ENTER**.

6. Pick any point on the shaft.

7. Pick either line near point **1**.

AutoCAD calculates the midpoint between the shaft's center and point **1** and places the first point of the polyline at that location.

8. Press **ENTER** to terminate the **PLINE** command.

9. Undo the polyline and save your work.

Using QuickCalc

The **QUICKCALC** command allows you to perform a full range of mathematical and trigonometric calculations, as well as determine graphic information such as the location of a point or the length of a line.

1. Select the **Open...** button from the **Quick Access** toolbar and open the **deck.dwg** file.

2. Pick the **QuickCalc** button from the **Utilities** panel, or type **QUICKCALC** at the keyboard.

UTILITIES

This displays the **QuickCalc** palette, as shown in Fig. 31-3. The buttons across the top of the palette allow you to find the coordinates of a point, the distance between two points, the angle of a line, and the intersection point of two lines.

3. Pick the **Get Coordinates** button and select the upper right corner of the deck.

The *x,y,z* coordinates of the point are displayed.

4. Pick the **Clear** button to clear the calculator's display.

5. Pick the **Distance Between Two Points** button.

6. Pick the upper left and lower right corners of the deck.
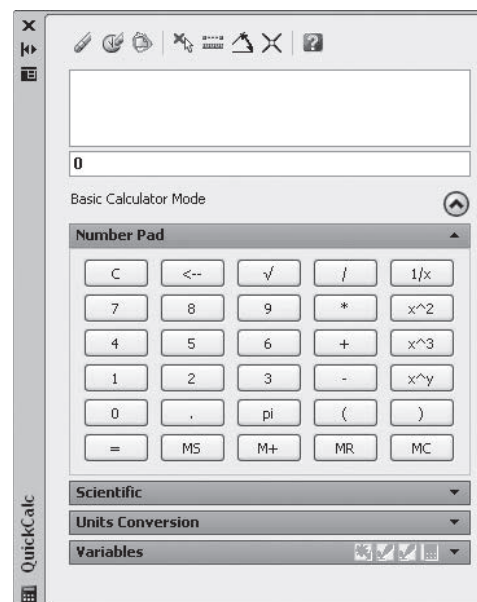
Fig. 31-3

The distance between the two corners of the deck is displayed: **14′-6-3/8″**.

(AutoCAD may display the decimal inch equivalent: **174.413302**.)

7. Pick the **Clear** button.

Now focus your attention on the **QuickCalc Number Pad**.

8. Experiment with the calculator by entering mathematical equations, just as you would with an ordinary calculator.

> **NOTE**
> You may have to drag the bottom line of the calculator down to enlarge the window so that you can see the entire calculator.

Notice that the equations and solutions are stored in a history list at the top of the **QuickCalc** palette. More functions are located below the **Number Pad** in the **Scientific** area.

9. Pick the arrow to the right of **Scientific** and review the trigonometric and algebraic functions available there.

Now focus your attention on the **Units Conversion** area. **QuickCalc** allows you to obtain values for different units of measurement.

10. Pick the arrow to the right of **Units Conversion** to display the **Units Conversion** area.

11. Convert an area of 15 square feet to square meters.

In the **Variables** section, you can use existing values and functions or create your own for use in calculations.

> **HINT**
> To access the **Square feet** and **Square meters** conversion options, you must first select **Area** in the **Units type** drop-down box.

12. Close the **QuickCalc** palette and close the **deck.dwg** file without saving.

Displaying Database Information

AutoCAD maintains an internal database on every drawing. The **drawing database** contains information about the types of objects in the drawing, defined layers, and the current space (model or paper). It also contains specific numerical information about individual objects and their placement in the drawing. This information can be useful to drafters and programmers using AutoCAD for a variety of tasks.

INQUIRY

Listing Information About Objects

1. Pick the **List** button from the expanded **Properties** panel.
2. Pick any point on the shaft and press **ENTER**.

The **AutoCAD Text** window displays the object type, layer, space, center point, radius, circumference, and area.

3. Close the window.
4. Select the shaft.
5. Right-click and pick **Properties** from the shortcut menu.

AutoCAD displays most of the same information, but does not give the space. The benefit of this list is that you can make changes to the object, such as changing its circumference and area.

6. Pick the text field to the right of **Area**.

Notice the calculator button that appears on the right side of the **Area** line. This provides you with another way to open **QuickCalc**.

7. Pick the calculator button to open the **QuickCalc** palette, then close it.
8. In the **Properties** palette, change the area of the shaft to **3**.

AutoCAD redraws the shaft with an area of 3 units.

9. Close the **Properties** palette and undo Step 8.

Editing Objects Mathematically

You can apply the same power that AutoCAD uses to calculate coordinates, areas, and circumferences to edit objects with mathematical precision. You can use this power to divide objects into equal parts and to measure exact intervals along a line or arc.

Dividing an Object into Equal Parts

The **DIVIDE** command divides an object, such as the end of the shaft, into a specified number of equal parts.

1. Expand the **Draw** panel in the **Home** tab of the **Ribbon,** then pick the **Divide** button from the flyout below **Multiple Points.**

This enters the **DIVIDE** command.

DRAW

2. Select the shaft.
3. Enter **20** for the number of segments.

It appears as though nothing has happened. Something did happen: the **DIVIDE** command divided the end of the shaft into 20 equal parts using 20 points; you just can't see them. Here's how to use them.

4. Turn off snap and object snap and open the **Object Snap** toolbar.
5. Enter the **LINE** command and pick the **Node** object snap. (**Node** is used to snap to the nearest point.)
6. Move the crosshairs along the shaft and snap to one of the nodes.
7. Snap to the center of the shaft.
8. Enter the **Node** object snap again and snap to another point on the shaft.
9. Terminate the **LINE** command.
10. Save your work.

Displaying the Points

AutoCAD's **DDPTYPE** command is used to control the appearance of points. Let's use it to make the points on the shaft visible.

1. In the **Format** menu of the **Menu Bar,** pick the **Point Style...** option, or enter **DDPTYPE.**

AutoCAD displays the dialog box shown in Fig. 31-4.

2. Pick the first point style in the second row—the one with a circle and a point at its center—and pick **OK**.
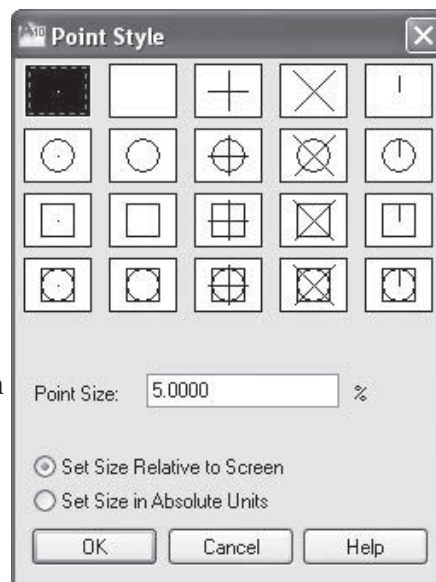
Fig. 31-4
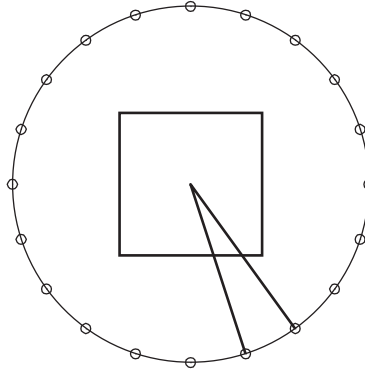
Fig. 31-5

AutoCAD places a small circle at each of the 20 equally spaced points, as shown in Fig. 31-5.

3. Display the same dialog box (press the spacebar).

4. Experiment with other point styles and adjust the point size.

5. Display the **Point Style** dialog box again and set the style to the last one in the top row.

6. Pick **OK** to close the dialog box.

Placing Markers at Specified Intervals

The **MEASURE** command is similar to **DIVIDE** except that **MEASURE** does not divide the object into a given number of equal parts. Instead, the **MEASURE** command allows you to place markers along the object at specified intervals.

1. On the expanded **Draw** panel, pick the **Measure** button from the flyout menu.

DRAW

This enters the **MEASURE** command.

2. Select one of the two lines.

3. Enter **.3** units.

AutoCAD adds points spaced .3 units apart.

4. Further experiment with **MEASURE**.

5. Set the point style to a single dot.

6. Save your work, close all floating toolbars, and exit AutoCAD.

Chapter 31 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Which AutoCAD commands are used to find coordinate points?
2. What information is produced with the **AREA** command?
3. What information is produced with the **LIST** command?
4. How do you calculate the perimeter of a polygon?
5. How do you find the circumference of a circle?
6. Explain how you control the appearance of points.
7. Explain the difference between the **DIVIDE** and **MEASURE** commands. Under what conditions would you use each of these commands?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Describe one drafting situation in which you might need to use the **DIVIDE** command and one in which you would prefer to use the **MEASURE** command.
2. Investigate the difference between AutoCAD's points created when you use the **POINT** command and those created when you use the **DIVIDE** and **MEASURE** commands. Write a paragraph comparing and contrasting the two methods.

Continued

Chapter 31 **Review & Activities**

• APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

Draw the views of the fasteners in Figs. 31-6 through 31-8 at the sizes indicated. Omit text and dimensions. Use the appropriate commands to find the information requested. Write their values on a separate sheet of paper. Refer to the files **calcprb1.dwg**, **calcprb2.dwg**, and **calcprb3.dwg** located on the OLC in the Drafting & Design Problems section. These files correspond to Figs. 31-6, 31-7, and 31-8, respectively.

1. Draw the top view of the Phillips-head screw shown in Fig. 31-6 according to the dimensions shown. Then find the following information:
   - location of point **A**
   - distance between points **A** and **B**
   - area of the polygon
   - perimeter of the polygon

2. Draw the top view of the nut shown in Fig. 31-7 according to the dimensions shown. Then find the following information:
   - distance between **A** and **B**
   - distance between **B** and **C**
   - area of hole
   - circumference of hole
   - area of nut
   - perimeter of nut

3. What information does AutoCAD list for arc **A** on the screw shown in Fig. 31-8?

4. List the information AutoCAD provides for line **B** on the screw.

5. On the screw shown in Fig. 31-8, divide line **B** into five equal parts. Make the points visible.
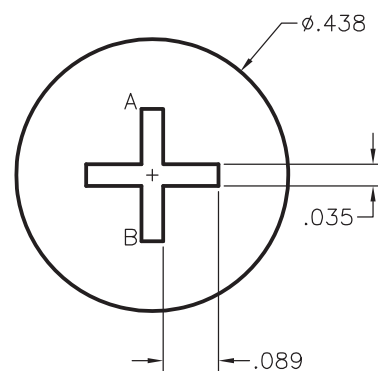
Fig. 31-6
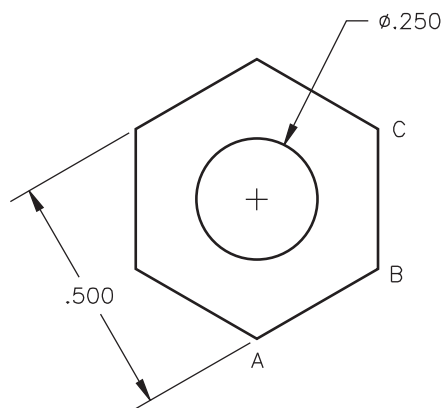
Fig. 31-7

Review & Activities Chapter 31
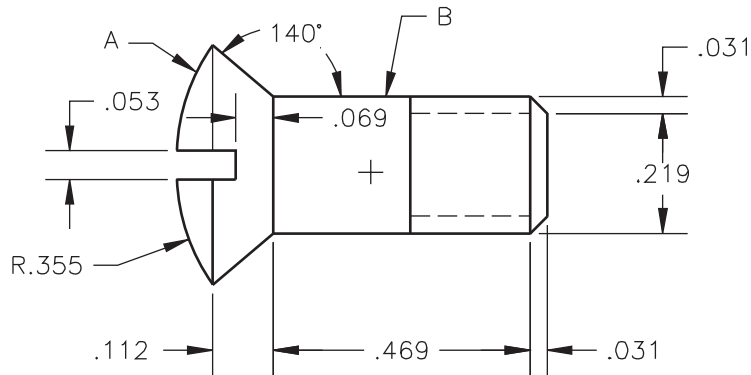
Fig. 31-8

6. On arc **A** in Fig. 31-8, place markers along the arc at intervals of **.02** units. If the markers are invisible, make them visible.

7. Create the floor plan shown in Fig. 31-9 on page 432 according to the dimensions given. Then find the information requested below. This floor plan is also available in the Drafting & Design Problems section of the OLC for this book as **flplan.dwg**.

Find the following information in square feet:
- area of showroom carpet
- area of entry clay tile
- area of bathroom roll tile

As you may know from your math courses, 1 square yard contains 9 square feet. Calculate the area of the carpet, entry clay tile, and bathroom roll tile as square yards using the **Units Conversion** area of the **QuickCalc** palette.
- square yards of carpet
- square yards of clay tile
- square yards of roll tile

Calculate the distance between opposite corners of the following areas:
- showroom
- entry area
- bathroom

Continued

Chapter 31 **Review & Activities**
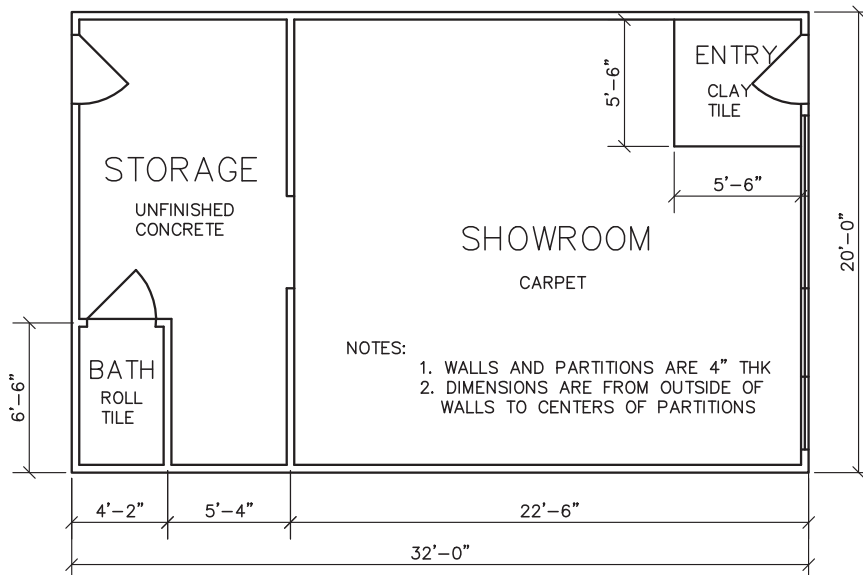
Fig. 31-9

Problem 7 courtesy of Mark Schwendau, Kishwaukee College

8. Create the top view of the nut shown in Fig. 31-10 according to the dimensions given. Then find the information requested below. This drawing is also available as **1-8UNC2B.dwg** in the Drafting & Design Problems section of the OLC for this book.

Find the following information:

- distance between **A** and **B**
- area of the minor diameter of the thread
- area of the major diameter of the thread
- circumference of the minor diameter
- circumference of the major diameter
- area of the top surface of the nut
- area of the top surface of the nut minus the minor diameter
- area of the top surface of the nut minus the major diameter
- Calculate the average of the last two answers using the **QUICKCALC** command.
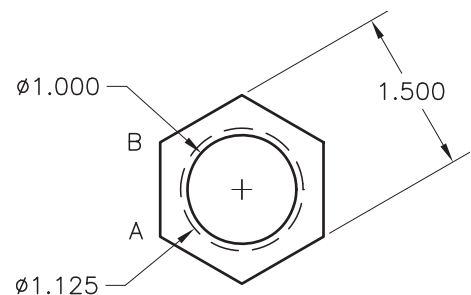
Fig. 31-10

Review & Activities Chapter 31

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Create a new drawing using the **Quick Setup** wizard. Specify a drawing area of **300′ × 200′**. Draw two horizontal lines of different lengths. Place one near the top of the drawing area and the other near the bottom. Then connect their ends to form a four-sided polygonal area. Determine the area and perimeter of the polygon.

The area you created represents a real-estate plot. As a paralegal for an attorney, you have been asked to divide this plot into five equal lots. Determine how to do this and carry out your plan. (*Hint:* consider using the **DIVIDE** command and the **Node** object snap.) To avoid future conflict, the attorney also wants you to verify the size of each lot. Determine the area and perimeter of each lot, and prepare a statement for the attorney explaining why the areas are the same, but the perimeters are not. Save the drawing as **realestate.dwg**.

2. You work for a company that designs and manufactures custom clocks for wealthy international clients. You have been assigned to design a clock for a client whose office is decorated in ultramodern style. Research the ultramodern style to determine what characteristics the clock should have to fit in with the client's current decor. Then design the clock in an ultramodern style using the following customer specifications:

- The client does not care whether the clock is a floor-standing or desk model, but she does not want a wall-mounted clock.

- The clock must have a mark at each minute in the hour as well as at each hour.

- The clock must have a large face so that it can be seen easily from any point in the client's 20′ × 30′ office.

Part 6 Project | Applying Chapters 27–31

Designer Eyewear

Part 6 reviewed the extensive dimensioning and calculating abilities of AutoCAD. In this exercise, you will be asked to design and draw an object and then to dimension it as specified. To review, traditional dimensions generally inform of size and location. Tolerance dimensions also yield information about characteristics or features of the item (*e.g.,* How round is the hole? How flat is the surface?).

▶ Description

Your task is to design a pair of eyeglasses. The actual style (appearance) and size are up to you. Whether latest fashion fad or very old-fashioned, the eyeglasses should be designed to be appropriate for the human face. (Feel free to inquire whether your instructor will allow a design for some other creature, real or imagined.) In any event, the chosen design must be revealed in two or three views and fully dimensioned. Of particular interest will be dimensioning information concerning the minimum and maximum tolerance permitted for the frame and lenses (such as ±.01). The nose pads or nosepiece may be separate or part of the frame itself. You may wish to measure your own eyeglasses or those of a family member or friend, or you may have someone actually measure your face to gain some insight with respect to realistic sizes. Figs. P6-1 and P6-2 may also give you some design ideas.
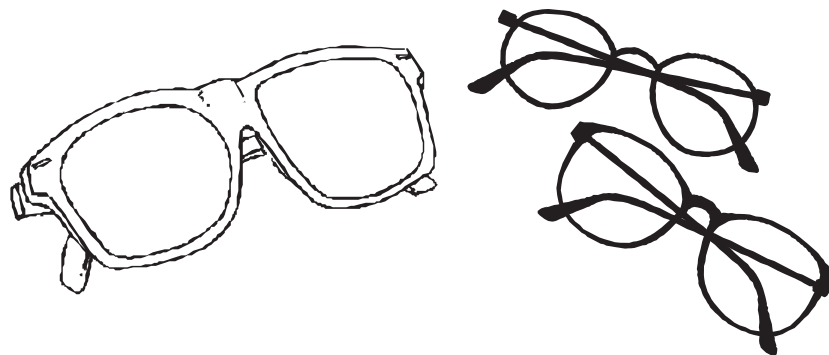
Fig. P6-1

Fig. P6-2

▶ Hints and Suggestions

1. Consider showing a sketch of your idea to the instructor for feedback before investing significant time and effort into the three-view CAD version. Guidance gained early is likely to save more time than constructive criticism given upon completion.

2. Don't spend too much time on exact sizes for the eyeglasses, but once drawn, dimension details thoroughly and accurately.

3. Include dimensions for the angles of the earpieces (end pieces) as well as radii for the curvature of the lenses.

4. Consider drawing one half of the eyeglasses; then mirror the drawing to complete the pair.

▶ Summary Questions

Your instructor may direct you to answer these questions orally or in writing. You may wish to compare and exchange ideas with other students to help increase efficiency and productivity.

1. Why might accurate dimensions be needed for one drawing and not for another?

2. Name some purposes for which tolerance dimensioning (min/max) might be considered of vital importance.

3. Once created, a dimension style can be a great timesaver. In what ways might it save time?

4. How do you determine which dimensions are needed on a drawing?

Chapter **32**

Groups

Objectives

- Create a group
- Add and delete objects from a group
- Edit the group name and description
- Make a group selectable or unselectable
- Reorder the objects in a group

Vocabulary

group
selectable

A **group** in AutoCAD is a named set of objects. A circle with a line through it, for instance, could become a group. Groups save time by allowing you to select several objects with just one pick. This speeds up editing operations such as moving, scaling, and erasing these objects.

Creating a Group

The **GROUP** command permits you to create a named selection set of objects.

1. Start AutoCAD and open the drawing named **calc.dwg**.
2. Select the **AutoCAD Classic** workspace and close all floating toolbars and palettes.
3. Using **Save As...**, create a new drawing named **groups.dwg** and erase all objects.
4. Create the drawing of the pulley wheel shown in Fig. 32-1 using the following guidelines:

- Create a layer named **Hidden** and assign the **HIDDEN** linetype and a light gray color to it. Place the hidden line on this layer.
- Place the remaining objects on layer **Objects**.
- Omit the dimensions, including the center line.
- Set **LTSCALE** to **.5**.

5. Enter the **GROUP** command.

This displays the **Object Grouping** dialog box, as shown in Fig. 32-2.

6. For **Group Name**, enter **wheel** using upper- or lowercase letters.

7. Enter **Cast aluminum** for **Description**.

Focus your attention on the **Create Group** area of the dialog box. Notice that the **Selectable** check box is checked.

8. Pick the **New** button, select all four objects, and press **ENTER**.

When the dialog box returns, notice that **WHEEL** is listed under **Group Name**. **Yes** appears under **Selectable** because the **Selectable** box was checked when you picked the **New** button. **Selectable** means that when you select one of the group's members (*i.e.*, one of the circles), AutoCAD selects all members in the group.

9. Pick the **OK** button.

10. Pick any one of the objects.

Notice that AutoCAD selected all four circles.

11. Move the group a short distance and then press **ESC**.

12. Lock layer **Hidden**.

13. Pick one of the objects again and then move the group a short distance.

The gray hidden-line object did not move because it is on a locked layer.

14. Undo the move and unlock layer **Hidden**.
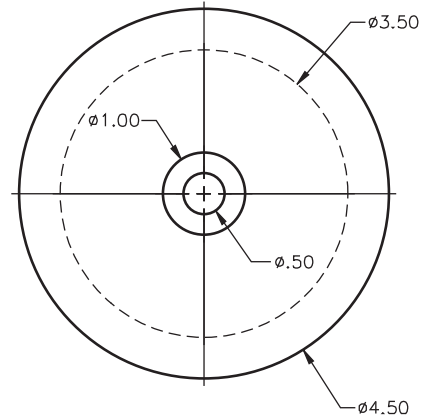
Fig. 32-1
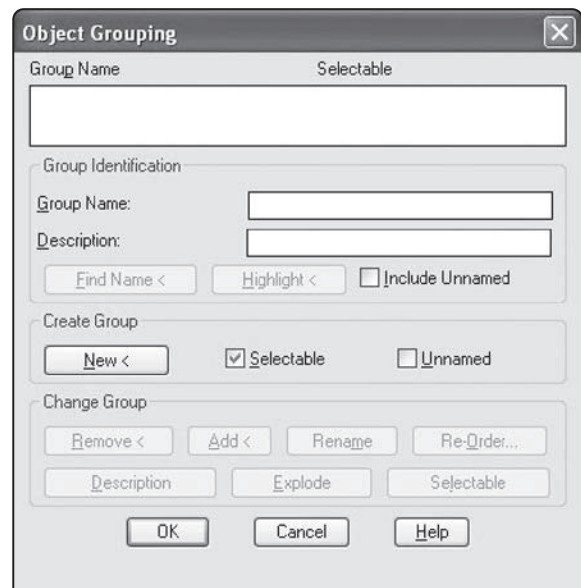
Fig. 32-2

Changing Group Properties

The **Object Grouping** dialog box offers several features for changing a group's definition and behavior. For example, you can add and delete objects from the group, change the group's name or description, and re-order the contents (objects) in the group. The dialog box also allows you to explode the group, which deletes the group definition from the file.

Adding and Deleting Objects

1. Enter the **GROUP** command.
2. Select **WHEEL** in the **Group Name** list box.
3. Pick the **Highlight** button.

This highlights all members of the group to show you at a glance which objects are members of the group. This is particularly useful when you are editing a complex drawing containing many objects and groups.

4. Pick the **Continue** button.

Focus your attention on the **Change Group** area of the dialog box.

5. Pick the **Remove** button and pick the gray hidden line.

AutoCAD removes this object from the group.

6. Press **ENTER**.
7. Select the **Highlight** button again.

Notice that the hidden-line circle is not highlighted; it is no longer part of the group.

8. Pick **Continue**.
9. Pick the **Add** button, pick the gray hidden-line circle, and press **ENTER**.

This adds the circle back to the group.

Changing the Group Name and Description

1. In the **Group Name** edit box, change the group name to **pulley** and pick the **Rename** button.

PULLEY appears in the **Group Name** list box near the top of the dialog box.

2. Rename the group back to **wheel**.

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VII. Groups and Details

32. Groups

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445

3. In the **Description** edit box, change the group description from **Cast aluminum** to **Cast magnesium**.

4. Pick the **Description** button (just above the **OK** button).

This causes the group description to update, as indicated at the bottom of the dialog box.

Making a Group Selectable

1. In the **Change Group** area, pick the **Selectable** button.

In the upper right area under **Selectable**, notice that **Yes** changed to **No**.

2. Pick the **OK** button and select one of the objects in the group.

As you can see, the group is no longer selectable. Grips appear only for the specific object you select.

3. Reenter the **GROUP** command, pick **WHEEL** in the **Group Name** list box, pick the **Selectable** button again, and pick **OK**.

The group is selectable once again.

Reordering the Objects

AutoCAD numbers objects in the order in which you select them when you create the group. In the **Order Group** dialog box, AutoCAD allows you to change the numerical sequence of objects within a group. Reordering can be useful when you are creating tool paths for computer numerical control (CNC) machining.

1. Enter the **Group** command.

2. Pick the **Re-Order...** button to make the **Order Group** dialog box appear, as shown in Fig. 32-3.

3. Select **WHEEL** under **Group Name**.

This makes the grayed-out items available.

4. Pick **OK**; pick **OK** again.

5. Save your work and exit AutoCAD.
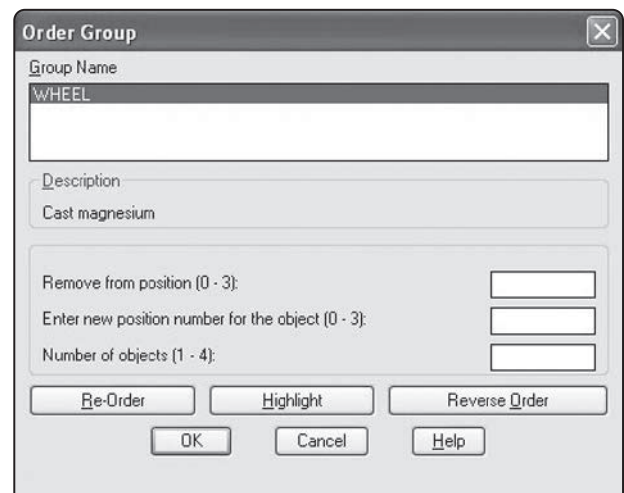
Fig. 32-3

Chapter 32 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is a group?
2. How can groups help AutoCAD users save time?
3. When creating a new group, what is the purpose of entering the group description?
4. Explain how you would add and delete objects from a group.
5. Is it possible to remove the selectable property from a group but still retain the group definition in the drawing? Explain.
6. Under what circumstances might you need to re-order the objects in a group?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Experiment with object selection using a group name. Note that you can enter **group**, followed by the group name, at any **Select objects** prompt for moving, copying, scaling, etc. When might this feature be useful? Explain.
2. Under what circumstances might you want to make a group unselectable? Explain.

● APPLYING AUTOCAD SKILLS

Work the following problem to practice the commands and skills you learned in this chapter.

1. Perform the following tasks.
   a. Load **calc.dwg** from the previous chapter and use **Save As…** to create a file named **stop.dwg**. If the points are visible, make them invisible.

Review & Activities   Chapter 32

b. Create a group named **WIDGET**. Enter **New product design** for the description. The group should consist of all four objects.

c. Change the name of the group to **WATCH** and the group description to **Stop watch design**. Remove the two lines from the group.

d. Move the group to a new location, away from the two lines. Set the text height to **.2** and add **12:30** inside the square. Add this text to the group. Your drawing should look similar to the one in Fig. 32-4.
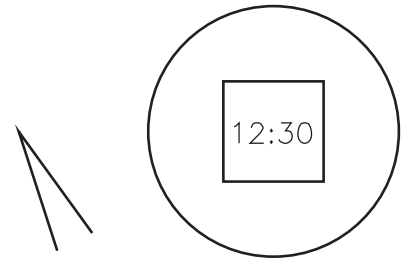
Fig. 32-4

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. The exterior trim design of your company's administration building has not yet been approved. Open **db_samp.dwg** in AutoCAD's **Sample** folder and save it as **AdmBldg.dwg** in the folder with your name. Make a group of each of the three exterior entrances (two sides in turquoise and the front in yellow). Move these groups to various locations to find a better arrangement. You may also remove the five-sided structure at the rear of the building to be replaced by a possible entry. Save the design you think is the best, and be prepared to explain your choice.

2. Your company has been hired to create a sales brochure for a new bicycle. The bicycle will be available in several colors, and your customer wants the brochure to show the bicycle in each color. Research typical bicycle shapes and designs. Then create a bicycle design of your own. Use commands such as **PLINE**, **SOLID**, and **FILL** to make the body (frame) of the bicycle solid green. Save the drawing as **bikesales.dwg**. Then, using the **GROUP** and **COPY** commands, create additional copies of the bicycle and change the body colors to show a version in six colors that you think would look good on a bicycle. Use the entire **Color** palette to make your color selections.

Blocks

Objectives

- Create and insert blocks
- Rename, explode, and purge blocks
- Organize and insert blocks using the **DesignCenter** and **Tool Palettes Window**
- Add commands to a tool palette
- Insert a drawing file into the current drawing
- Create a drawing file from a block
- Copy and paste objects from drawing to drawing

Vocabulary

block
copy and paste
purge

With AutoCAD, you seldom need to draw the same object twice. Using blocks, you can define, store, retrieve, and insert symbols, components, and standard parts without the need to recreate them. A **block** is a collection of objects that you can associate together to form a single object. Blocks are especially useful when you are creating libraries of symbols and components.

Blocks are similar to groups, but the two are different. A block definition is stored as a single, selectable object in a drawing file that you can insert, scale, and rotate. A group is a named selection set of objects.

Working with Blocks

The **BLOCK** command allows you to combine several objects into one and then store and retrieve it for later use.

1. Start AutoCAD and start a new drawing from scratch using **Imperial** units.
2. Select the **2D Drafting & Annotation** workspace and close all floating toolbars and palettes.
3. Draw the top view of a flathead screw as shown in Fig. 33-1. Omit the dimensions.
4. Save your work in a file named **blks.dwg**.

.347

.106

ø.882
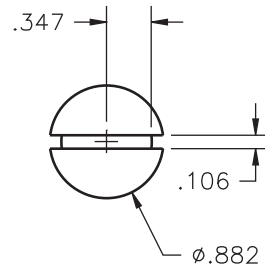
Fig. 33-1

Creating a Block

1. Pick the **Create** button from the **Block** panel on the **Home** tab of the **Ribbon,** or enter the **BLOCK** command.

BLOCK

This displays the **Block Definition** dialog box, as shown in Fig. 33-2.
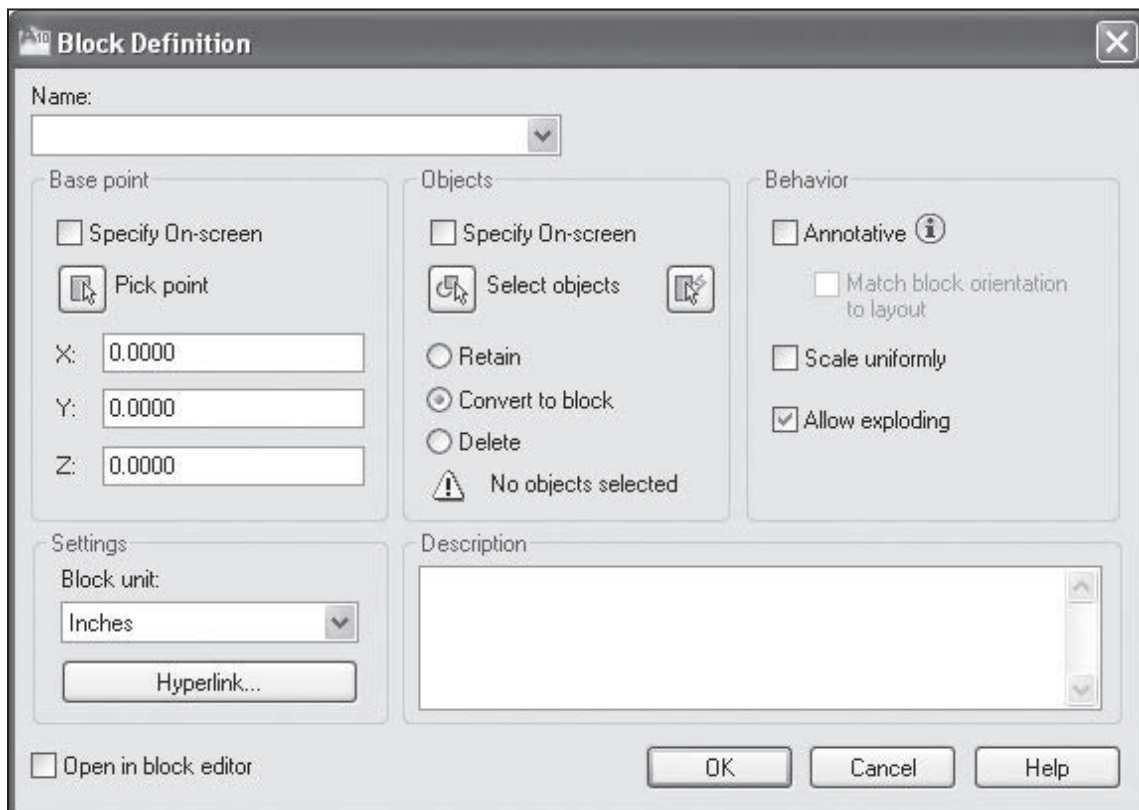
Fig. 33-2

2. In the **Base Point** area, pick the **Pick point** button.

3. Snap to the center point of the head of the screw.

This defines the base point for subsequent insertions of the screw head.

4. In the **Objects** area, pick the **Convert to block** radio button and then pick the **Select objects** button.

5. Select the entire screw head and press **ENTER**.

In the **Objects** area, AutoCAD displays **6 objects selected**. The **Retain** option retains the selected objects as distinct objects in the drawing after you create the block. **Convert to block** converts the selected objects to a block instance in the drawing after you create the block. **Delete** deletes the selected objects from the drawing after you create the block. (The block definition remains in the drawing database.)

6. In the **Settings** area, review the options available for the **Block unit**.

7. Set **Block unit** at **Unitless;** do not add a description.

You have just created a new block definition.

8. In the **Name** text field, type **flat** and pick the **OK** button.

Inserting a Block

BLOCK

1. Pick the **Insert** button from the **Block** panel on the **Ribbon,** or enter the **INSERT** command.

This displays the **Insert** dialog box, as shown in Fig. 33-3. **Flat** is listed after **Name** because it is the only block definition in the drawing. Picking the down arrow would list additional blocks, if any existed, in alphabetical order.
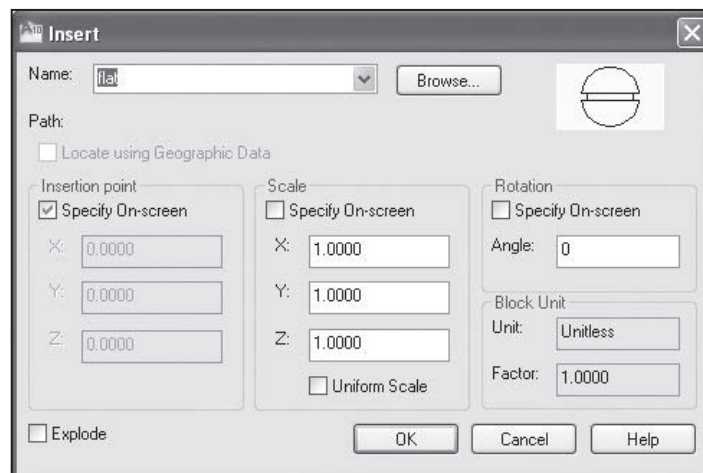
Fig. 33-3

The **Insertion point** area permits you to enter specific coordinates for the block's insertion. **Specify On-screen**, which should be checked, allows you to specify the insertion point on screen. The **Scale** area allows you to scale the block now or on the screen when you insert it. The **Rotation** area enables you to specify a rotation angle in degrees now or when you insert the block. The **Block Unit** area is not selectable; it displays the unit and scale factor used when the block was created. The **Explode** check box explodes the block and inserts the individual parts of the block.

2. Under **Scale**, pick the **Specify On-screen** check box.

3. Under **Rotation**, pick the **Specify On-screen** check box.

4. Pick the **OK** button.

AutoCAD locks the insertion point of the block onto the crosshairs.

5. Pick an insertion point anywhere on the screen.

6. Move the crosshairs up and down, then back and forth.

As you can see, you can drag the X and Y scale factors.

7. Enter **1.5** for the X scale factor and **1.5** for the Y scale factor.

You can also drag the rotation into place, or you can enter a specific degree of rotation.

8. Enter **45** for the rotation angle.

9. Insert additional instances of the block using different settings in the **Insert** dialog box.

Editing an Inserted Block

1. Try to erase one of the lines from the first inserted block; then press **ESC**.

You cannot erase part of a block because a block is a single object.

2. Pick the **Explode** button from the **Modify** panel, pick the inserted block, and press **ENTER**.

3. Now try to erase a line from the inserted block.

MODIFY

As you can see, exploding a block returns it to its component parts. Exploding a block with the **EXPLODE** command is the same as checking the **Explode** check box in the **Insert** dialog box when inserting the block.

Renaming a Block

The **RENAME** command lets you rename previously created blocks.

1. Pick **Rename...** from the **Format** menu on the **Menu Bar,** or enter **RENAME** at the keyboard.

This enters the **RENAME** command and displays the **Rename** dialog box. Under **Named Objects**, the dialog box lists all of the object types in AutoCAD that you can rename.

2. Pick **Blocks**.

AutoCAD lists the only block, **flat**, under **Items**.

3. Pick **flat** and then enter **flathead** in the box at the right of **Rename To**.

4. Pick the **Rename To** button.

Flathead is now listed under **Items**.

5. Pick the **OK** button.

6. Save your work.

In the future, if you need to rename blocks, dimension styles, layers, linetypes, text styles, UCSs, viewports, or views, use the **RENAME** command.

Purging Blocks

The **PURGE** command enables you to selectively delete, or **purge**, any unused named objects, such as blocks. Purging named objects reduces drawing file size.

1. Enter the **PURGE** command.

AutoCAD displays the **Purge** dialog box and shows the different object types that you can purge. If the drawing database contained unused blocks, the **Purge** button would not be grayed out. In the future, you may choose to purge unused blocks, dimension styles, layers, and other object types.

2. Pick the **Close** button.

DesignCenter and Tool Palettes

The **DesignCenter** is a palette that allows you to browse, find, and insert content, such as blocks and hatches. The **Tool Palettes Window** offers tabbed areas that provide an easy method of organizing, sharing, and placing blocks and hatches.

PALETTES

Inserting Blocks from DesignCenter

1. Pick the **DesignCenter** button from the **Palettes** panel in the **View** tab of the **Ribbon.**

This displays the **DesignCenter** palette.

2. Click each of the tabs to familiarize yourself with **DesignCenter**.

3. Pick the **Open Drawings** tab and double-click **Blocks**.

Notice that the **flathead** block was added automatically to **DesignCenter**.

4. Click and drag the block named **flathead** from **DesignCenter** into the drawing area.

As you can see, **DesignCenter** offers a quick and easy alternative for inserting blocks.

5. Right-click and drag the **flathead** block into the drawing area and pick **Insert Block...** from the shortcut menu.

This displays the **Insert** dialog box.

6. Make sure that **Specify On-screen** is checked under **Insertion point**, **Scale**, and **Rotation** and pick **OK**.

7. Pick an insertion point.

8. Enter **.75** for the X and Y scale factors.

9. Enter **45** for the rotation angle.

10. After inserting the block, auto-hide **DesignCenter** to make it smaller, but leave it open.

Inserting Blocks from a Tool Palette

1. Pick the **Tool Palettes Window** button from the **Palettes** panel.

2. Right-click the title bar of the **Tool Palettes Window** and select **New Palette** from the menu.

3. In the text box, type **Fasteners** and press **ENTER**.

This creates a new (but empty) tool palette named **Fasteners**.

4. From **DesignCenter**, click and drag the block named **flathead** to the new palette.

The **flathead** block appears in the tool palette.

5. Close **DesignCenter**.

6. From the new tool palette, click and drag the **flathead** block into the drawing area and release the pick button.

As you can see, this is a fast and easy way of inserting blocks.

7. Insert another **flathead** block into the drawing area from the tool palette; then insert another.

8. Attempt to right-click and drag the **flathead** block from the palette into the drawing area.

PALETTES

As you can see, this is not an option. The **Tool Palettes Window** does not give you the option of inserting blocks using the **Insert** dialog box, similar to right-clicking and inserting them from **DesignCenter**.

Adding Commands to a Tool Palette

AutoCAD makes it easy to add commands to a tool palette.

1. Display the **Draw** toolbar by selecting **Tools** on the **Menu Bar,** then **Toolbars, AutoCAD,** and **Draw.**

2. Right-click in the **Fasteners** tool palette and pick **Customize Palettes...** from the menu.

This displays the **Customize** dialog box.

3. With the **Customize** dialog box open, click and drag the **Polyline** button from the **Draw** toolbar to the **Fasteners** tool palette.

The **Polyline** button becomes part of the palette. Notice that a small black triangle appears next to it. The button is now a flyout button—a set of **Draw** commands are nested under this single button.

4. Click the black triangle next to the **Polyline** button on the **Fasteners** tool palette.

As you can see, the buttons for eight of the commands from the **Draw** toolbar are displayed on the flyout button.

5. Pick the **Close** button in the **Customize** dialog box.

6. Display the **Dimension** and **Modify** toolbars.

7. Right-click in the **Fasteners** tool palette and pick **Customize Palettes...** from the menu once again.

8. Click and drag the **Linear Dimension** button from the **Dimension** toolbar to the **Fasteners** tool palette.

9. Click and drag the **Erase** button from the **Modify** toolbar to the **Fasteners** tool palette.

10. Close the **Customize** dialog box.

11. Try each of the commands that you added to the **Fasteners** tool palette.

You can also arrange the order of tools on a tool palette and rearrange the order of tabs in the **Tool Palettes Window.**

12. On the **Fasteners** tool palette, click and drag the **Erase** button up to the top of the palette.

13. Right-click on the **Fasteners** tab and pick **Move Up** or **Move Down** to rearrange the order of tabs in the **Tool Palettes Window.**

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VII. Groups and Details

33. Blocks

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455

14. Right-click in an open area of the **Fasteners** tool palette, select **Delete Palette** from the menu, and pick **OK** to confirm that you want to delete the tool palette.

AutoCAD deletes the tool palette.

15. Close the **Tool Palettes Window,** close the **Draw**, **Dimension**, and **Modify** toolbars, and save your work.

Blocks and Drawing Files

It is possible to insert a drawing (**DWG**) file as if it were a block. This can be useful when you want to use part or all of another drawing in the current drawing. Also, it's possible to create a drawing file from a block. This can be especially helpful when you need to transport a block to another computer.

Inserting a Drawing File

1. Begin a new drawing from scratch using **Imperial** units.

2. Pick the **Insert** button from the **Block** panel and pick the **Browse...** button from the dialog box.

3. Find and open the folder with your name.

4. Open the file named **toler.dwg**.

Toler appears in the **Name** box.

5. Uncheck **Specify On-screen** under **Insertion point**, **Scale**, and **Rotation**.

6. Pick the **Explode** check box and pick **OK**.

Toler.dwg appears.

7. Pick one of the objects from the **toler.dwg** drawing.

This is possible because we selected the **Explode** check box to make the components of the block selectable individually.

8. Close the current drawing without saving.

The drawing named **blks.dwg** (containing the flathead screws) is now the current drawing.

BLOCK

Creating a Drawing File from a Block

WBLOCK, short for **Write BLOCK**, writes (saves) objects or a block to a new drawing file.

1. Enter the **WBLOCK** command.
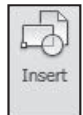
AutoCAD displays the **Write Block** dialog box.

2. Under **Source**, pick the **Block** radio button.

3. Click in the box at the right of **Block**.

4. Pick **flathead**.

The **Base point** and **Objects** areas are grayed out because both base point and objects are a part of the block definition.

5. Review the information listed under **Destination**, but do not change it.

Notice that the proposed file name is **flathead**.

6. At the right, pick the button containing the three dots, select the folder with your name, and pick **Save**.

7. Pick the **OK** button to close the **Write Block** dialog box.

AutoCAD creates a new file named **flathead.dwg**.

8. Pick the **Open** button from the docked **Quick Access** toolbar.

9. Find **flathead.dwg** and preview it, but do not open it.

10. Pick the **Cancel** button.

Copying and Pasting Objects

AutoCAD's Windows-standard **copy and paste** feature provides an alternative to using the **WBLOCK** and **INSERT** commands. Copying and pasting can be a faster approach when you want to transfer a block or set of objects to another drawing on the same computer. This approach does not work when you need to transfer a block or set of objects to another computer.

CLIPBOARD

1. On the **Clipboard** panel, pick the **Copy Clip** button to enter the **COPYCLIP** command.

2. Pick a couple of objects and press **ENTER** to copy the objects to the Windows Clipboard.

3. Save your work.

4. Create a new drawing from scratch using **Imperial** units.

CLIPBOARD

5. On the **Clipboard** panel, pick the **Paste** button to enter the **PASTECLIP** command.

6. Move the crosshairs and notice that the objects are attached to it.

7. Pick an insertion point.

8. Exit AutoCAD without saving the current drawing.

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VII. Groups and Details

33. Blocks

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457

Chapter 33 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Briefly describe the purpose of blocks.
2. Explain how the **INSERT** command is used.
3. How can you list all defined blocks contained within a drawing file?
4. A block can be inserted with or without selecting the **Explode** check box. Describe the difference between the two.
5. Describe the function of the **WBLOCK** command.
6. When would **WBLOCK** be useful?
7. How can you rename blocks?
8. Why would you want to purge unused blocks from a drawing file?
9. What is an advantage to using the **Tool Palettes Window** to organize and insert blocks?
10. When copying an object from one drawing to another, why might you prefer the copy and paste method over the **WBLOCK** and **INSERT** approach?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. An electrical contractor using AutoCAD needs many electrical symbols in his drawings. He has decided to create blocks of the symbols to save time. Describe at least two ways the contractor can make the blocks easily available for all his AutoCAD drawings. Which method would you use? Why?
2. If you were to copy a block from one drawing and paste it into another, would AutoCAD recognize the pasted object as a block? Explain.

Chapter 33 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Open **range.dwg**, which you created in the "Applying AutoCAD Skills" section of Chapter 18. Make a block of the range and reinsert it into the drawing at a scale factor of **.25**, with a rotation of **90°**. Save the drawing as **range2.dwg**.

2. Begin a new drawing named **livroom.dwg**. Draw the furniture representations shown in Fig. 33-4 and store each as a block. Then draw the living room outline. Don't worry about exact sizes or locations, and omit the text. Insert each piece of furniture into the living room at the appropriate size and rotation angle. Feel free to create additional furniture and to use each piece of furniture more than once. This file is also available on the OLC for this book, in the Drafting & Design Problems section, as **livroom.dwg**.

3. After creating the blocks in problem 2, write two of them (of your choice) to a file using **WBLOCK**. Store them in the folder with your name.

4. Copy the block of the easy chair and paste it into another drawing.

5. Explode the **PLANT** block and erase every fourth arc contained in it. Then store the plant again as a block.

6. Rename two of the furniture blocks.

7. Purge unused objects.
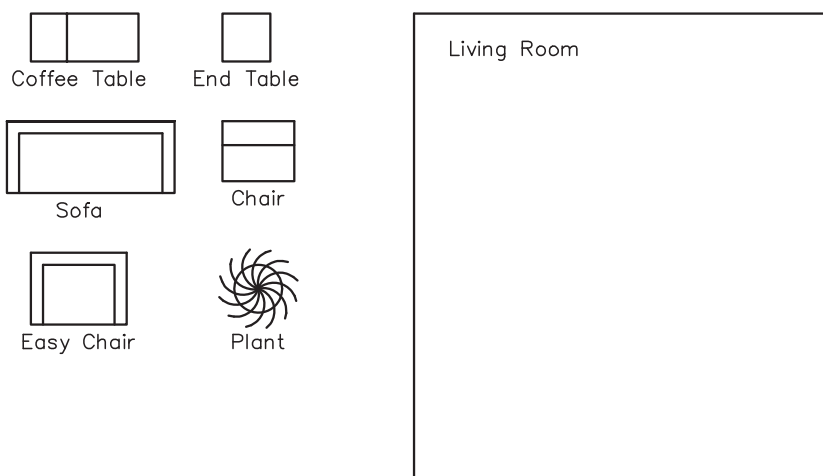
Fig. 33-4

8. Create a new drawing named **revplate.dwg**. Create the border, title block, and revisions box using the dimensions shown in Fig. 33-5. (Do not include the dimensions.) Make a block of the revisions box using the insertion point indicated. Insert the block in the upper right corner of the drawing. This drawing is also available as **revplate.dwg** on the OLC for this book, in the Drafting & Design Problems section.
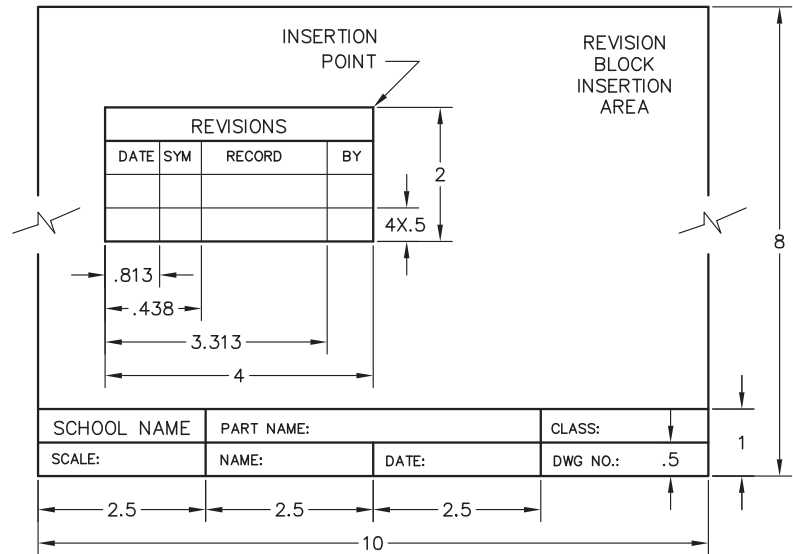
Fig. 33-5

Courtesy of Mark Schwendau, Kishwaukee College

● USING PROBLEM-SOLVING SKILLS

Complete the following problems using problem-solving skills and your knowledge of AutoCAD.

1. Your company wants to make the revisions box you created in problem 8 available to all the designers in the company. Open **revplate.dwg** from your named folder, create a block of the revisions box, and save it as a **DWG** file. Be sure to give the file a descriptive name so that the designers will know what is in the file.

2. Draw the electric circuit shown in Fig. 33-6 as follows: Draw the resistor using the mesh shown; then save it as a block. Draw the circuit, inserting the blocks where appropriate. Grid and snap are handy for drawing the resistor and circuit. Finish the circuit by inserting small donuts at the connection points. Add the text. The letter omega (Ω), which is used to represent the resistance in ohms, can be found under the text style **GREEKC** (character W).
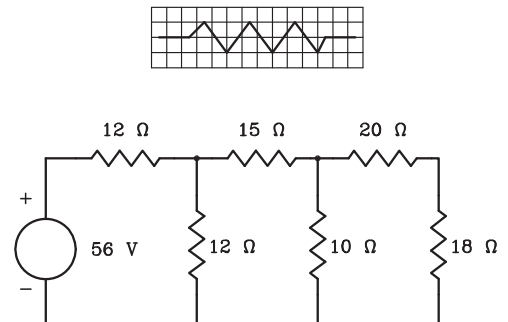
Fig. 33-6

Chapter **34**

Dynamic Blocks

Objectives

- Create a dynamic block
- Assign actions to parameters
- Insert a dynamic block in a drawing
- Create a set of constraining values for a dynamic block
- Use a constrained dynamic block in a drawing

Vocabulary

constrain
dynamic blocks
parameters
value set

In the last chapter, you were introduced to blocks. By creating and using blocks, AutoCAD users avoid drawing the same object more than once. This can be extremely useful, because many objects in architecture, manufacturing, and engineering are used repeatedly and need to be drawn again and again in many drawings.

AutoCAD allows you to take advantage of the convenience of blocks even further with dynamic blocks. A **dynamic block** is a block with certain characteristics or **parameters**, such as size or location, that can be changed after you insert the block into a drawing. With dynamic blocks, you first define the parameters that you want to be changeable using AutoCAD's block editor. Once the block has been created, you only need the one dynamic block to draw a range of sizes or lengths of a part. For example, you could create a dynamic block for drawing a commonly used fastener, such as a hex nut, and vary the size of the nut when you insert the block into the drawing.

Creating a Dynamic Block

Let's create a dynamic block for a simple shape: a bed. We'll start with the shape of a twin-size bed and add parameters and actions that will allow its length and width to be stretched into a bed of any size.

1. Start AutoCAD and start a new drawing from scratch using **Imperial** units.

2. Select the **2D Drafting & Annotation** workspace.

3. Using the **LIMITS** command, set the drawing area to **120 × 100** and **ZOOM All**.

4. Use the **LINE** command to create the top view of a twin-size bed using the dimensions shown in Fig. 34-1. Do not include the dimensions in your drawing. These are the standard dimensions (in inches) for a twin-size bed.

5. Name the drawing **bed.dwg**.

6. Pick the **Create** button from the **Block** panel or enter the **BLOCK** command to display the **Block Definition** dialog box.

7. Name the block **Bed**, pick the lower left corner of the bed for the base point, and select the entire bed to be included in the block.

8. Near the bottom of the dialog box, check the **Open in block editor** box.

9. Pick **OK** to close the dialog box and open the new block in the block editor.
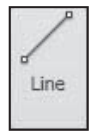
DRAW
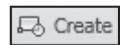
BLOCK
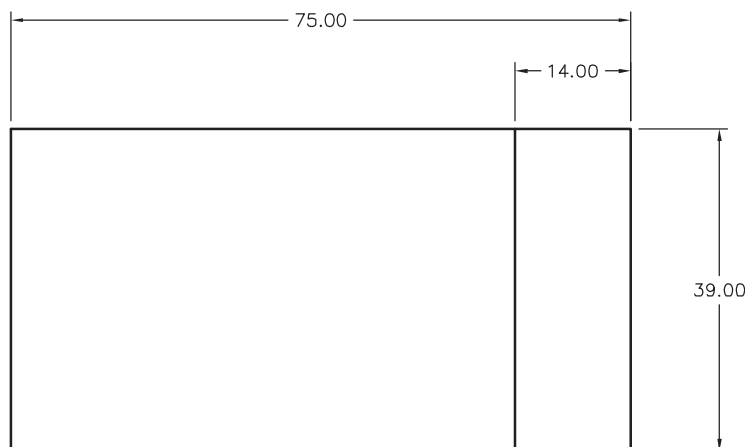
Fig. 34-1

Assigning Parameters to the Block

There are two ways to work with blocks in the block editor. You can use either the **Block Authoring Palettes** window, which appears on the left side of the block editor, or the buttons on the **Block Editor** tab that is automatically displayed on the **Ribbon** when the **Block Editor** is opened. The following steps use both methods.

1. In the **Block Authoring Palettes** window, select the **Parameters** tab if it is not already selected.

2. From the available options, pick **Linear Parameter**.

3. Pick the bottom left corner of the bed as the start point and the bottom right corner of the bed as the endpoint. Be sure to snap accurately to these two points.

4. Pick a point inside the bed for the label location, as shown in Fig. 34-2.

Notice that the parameter is automatically labeled **Distance1**. Also notice the yellow icon with an exclamation mark near the lower left corner of the bed. This symbol means that the parameter is not currently associated with any action. You will set up the actions later. First, let's create a second parameter.

5. From the **Block Authoring Palettes** window, pick **Linear Parameter** again. This time, pick the lower left corner for the start point and the upper left corner for the endpoint. Place the label inside the bed, as shown in Fig. 34-3.

The second parameter is automatically named **Distance2**. However, you can rename parameters to make them easier to identify.

6. Select the first **Distance1** parameter you created, right-click to display the shortcut menu, and pick **Rename Label**.

7. Highlight the word **Distance1**, enter **Length** for the new name, and press **ENTER**.
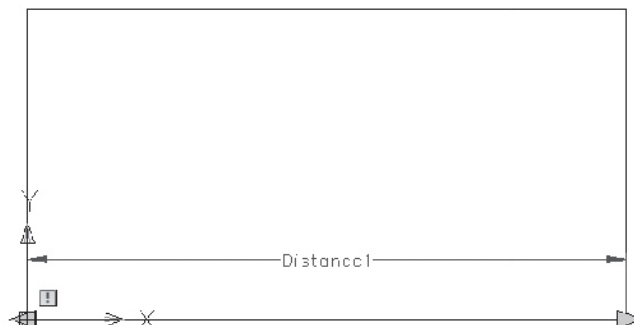
Fig. 34-2
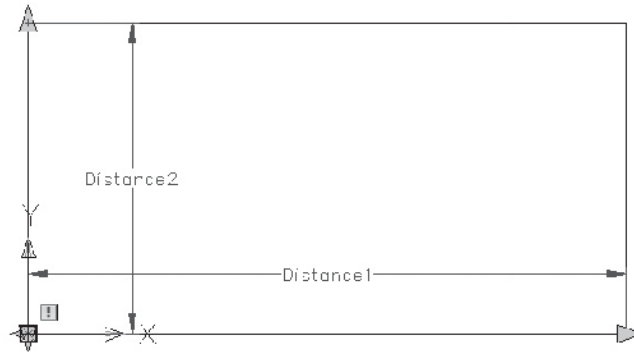
Fig. 34-3

8. Select the **Distance2** parameter and rename it **Width**.

Because we will stretch only one end of each parameter, let's change the grip display.

9. Pick the **Length** parameter, right-click, and highlight **Grip Display**.

10. Pick a **Grip Display** of **1** from the cascading menu.

Notice that the closed blue arrow on the left endpoint disappears.

11. Change the **Grip Display** of the **Width** parameter to **1**.

12. Save your work.

Assigning Actions to the Parameters

The two parameters that you have established will allow you to adjust both the length and the width of the dynamic block named **Bed**. Next, we will set up the actions to be associated with each parameter.

1. Expand the **Action Parameters** panel on the **Block Editor** tab of the **Ribbon** and select the **Show All Actions** button.

> **NOTE**
> You can also define an action using the **Actions** tab of the **Block Authoring Palettes** window.

2. Pick the down-facing arrow below the **Move** button on the **Action Parameters** panel, and select the **Linear** action button from the flyout menu.

3. Select the **Length** parameter.

Notice that a red symbol appears at the lower right corner of the bed.

ACTION PARAMETERS

Chapter 34: Dynamic Blocks   **457**
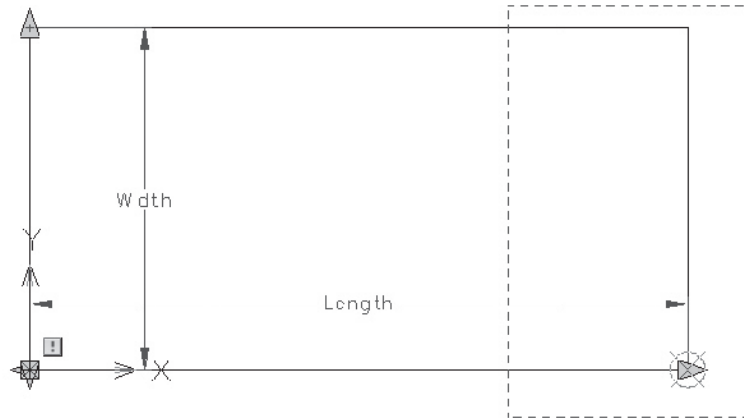
(**Fig. 34-4**)

4. Pick the point in the lower right corner of the bed as the parameter point to associate with the action.

5. In reply to **Specify first corner of stretch frame**, use a standard crossing window (right to left selection). Place the window as shown in Fig. 34-4. Be sure to include the interior line that represents the bedcovers in the crossing window.

6. At the **Select Objects** prompt, use another crossing window to select roughly the same area. Press **ENTER** when you are finished.

A **Stretch** action icon appears to the right of the parameter point. Let's rename the label for our new action.

7. Pick the **Stretch** action icon, right-click, and pick **Rename Action**.

8. Rename the action **Stretch Length**.

Next, we will define the action needed to change the width of the bed.

9. Pick the **Stretch** button again and select the **Width** parameter.

ACTION PARAMETERS

10. Pick the endpoint at the upper left corner as the parameter point to associate with the action.

11. For the stretch frame, place a crossing window across the upper half of the bed, as shown in Fig. 34-5.
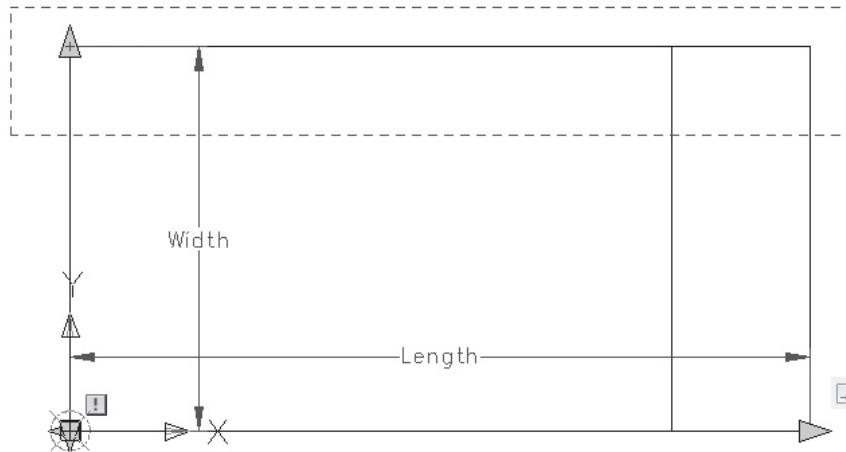
Fig. 34-5

12. At the **Select objects** prompt, use another crossing window to select roughly the same area. Press **ENTER** when you are finished.

13. Rename the action to **Stretch Width**.

The **Length** and **Width** parameters are now completely defined, so the alert icons for the parameters have disappeared. Your block should now look similar to the one in Fig. 34-6.
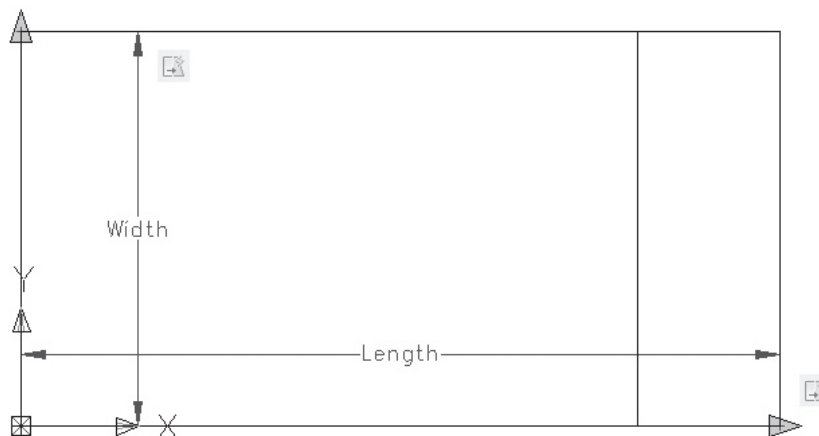
Fig. 34-6

14. Pick the **Close Block Editor** button on the **Close** panel of the **Ribbon** to return to the AutoCAD window.

15. Pick **Yes** when prompted to save changes to the **Bed** block.

Using Dynamic Blocks

The **Bed** block should now appear in AutoCAD's drawing area. Next, we will change the parameters of the block to change the size of the inserted block.

1. Select the **Bed** block.

Notice the arrows that appear instead of grip boxes at the upper left and lower right corners of the bed. These arrows indicate changeable parameters.

2. Pick the arrow at the lower right corner of the bed and notice that it becomes active (red) just as a grip would.

The **Length** parameter appears and an edit box appears to show the current length of the bed, which is 75 inches.

3. Move the crosshairs and notice that the length of the bed changes dynamically.

4. At the keyboard, enter a new length of **80** and press **ENTER** to change the length of the bed to 80 inches.

The bed updates automatically. This is the standard length for a queen-size, king-size, or extra-long twin-size bed. Let's change the **Width** dimension to create a queen-size bed, which has a standard width of 60 inches.

5. Select the bed again.

6. Pick the arrow at the upper left corner of the bed.

7. Enter a new width of **60**.

Once again, the **Bed** block automatically updates when you press **ENTER**.

Constraining Parameters

It may have occurred to you that it is possible to change the dynamic **Bed** block to any length and width using the dynamic block defined in the previous procedure. In real life, beds are usually made to standard dimensions, as shown in Fig. 34-7.

TWIN
39 X 75

KING
76 X 80

EXTRA-LONG TWIN
39 X 80

SPLIT KING
78 X 80

FULL
54 X 75

CALIFORNIA KING
72 X 84

QUEEN
60 X 80
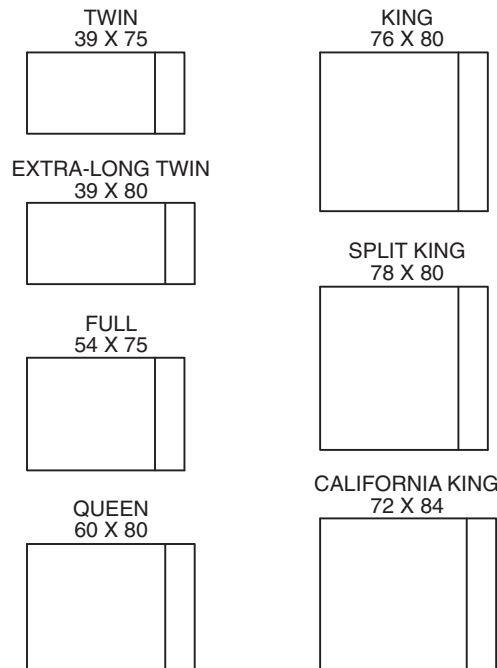
Fig. 34-7

Creating a Value Set

AutoCAD makes it easy to **constrain**, or limit, the choices that can be used for a given parameter. You can do this by creating a **value set** for the parameter. For example, the three standard lengths for a bed are 75, 80, and 84 inches. The following steps constrain the **Length** parameter of the **Bed** block using a value set that contains these three values.

1. Double-click the block to display the **Edit Block Definition** dialog box.
2. Select **Bed** and pick **OK** to enter the block editor.
3. Select the **Length** parameter.
4. Select the **Properties** button from the **Palettes** panel on the **View** tab of the **Ribbon**.

PALETTES

This displays the properties of the linear parameter in the **Properties** palette.

5. Near the bottom of the **Properties** palette, find the **Value Set** section and click to activate the box next to **Dist type**. (You may need to scroll down to find this section.)
6. Pick the down arrow and then pick **List** from the options that appear.

Notice that the next row, **Dist value list**, now shows **75.0000**. This is the value you used when you originally created the block, and it becomes the default value.

7. Pick to activate the **Dist value list** row and pick the three dots to the right of the edit box.

This displays the **Add Distance Value** dialog box.

8. In the **Distances to add** edit box, enter **80** and pick the **Add** button.

The new value of **80.0000** appears in the window below the edit box.

9. With the cursor still active in the **Distances to add** edit box, enter **84** and pick the **Add** button to add **84.0000** to the list.

10. Pick **OK** to close the dialog box.

11. Press **ESC** to deselect the **Length** parameter.

The standard widths for beds are 39, 54, 60, 72, 76, and 78. Let's constrain the **Width** parameter to these values.

12. With the **Properties** palette still open, pick to select the **Width** parameter.

13. In the **Dist type** row in the **Properties** palette, change the value to **List**.

14. Click to activate the **Dist value list** row and pick the button with three dots to display the **Add Distance Value** dialog box.

Once again, the default value appears in the window.

15. Add the other standard width values: **54**, **60**, **72**, **76**, and **78**. Pick **OK** when you are finished.

16. Close the **Properties** palette and save the file. If AutoCAD displays a message asking whether you want to save the drawing and update the existing block definition in the drawing, pick **Yes**.

17. Close the block editor.

Using a Constrained Block

Constrained blocks can be altered on-screen in the same way as unconstrained dynamic blocks. The only difference is that, when you pick one of the parameter arrows, small tick marks appear to mark the valid values for the parameter.

1. Select the **Bed** block to display the parameter arrows.

2. Pick the lower right parameter arrow so that it turns red.

Just below the arrow, notice the faint tick marks on either side of the current bed length. See Fig. 34-8.

You may recall that the bed is currently configured with dimensions for a queen-size bed: 60 × 80 inches. The 80 inches is one of the three standard values. The two ticks mark the other two values allowed for the **Length**
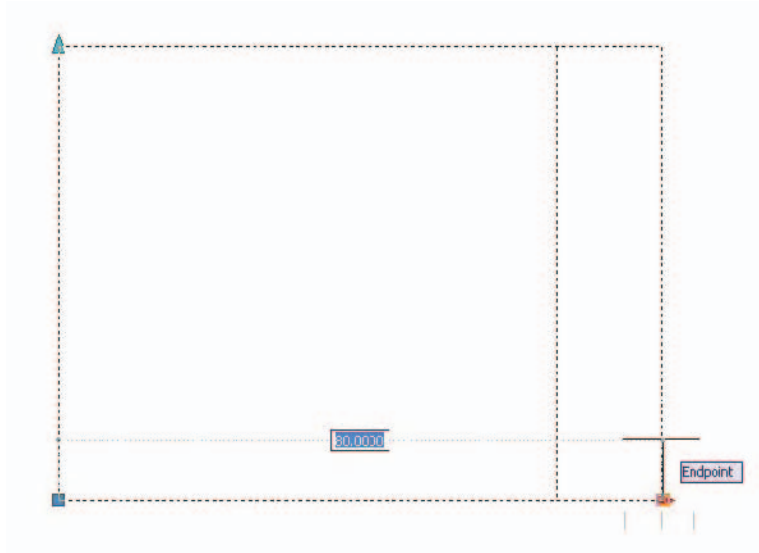
Fig. 34-8

parameter: 75 and 84. Let's change the **Bed** block to represent a full-size bed (54 × 75 inches).

3. Move the cursor to the left and notice that a tentative line appears at the 75-inch tick mark.

4. Pick any point while that line is present to reset the length of the bed to 75 inches.

5. Pick the upper left parameter arrow to activate the **Width** parameter and notice the tick marks representing allowed values.

The current value of **60** shows up in the edit box for reference.

6. Move the cursor down until the edit box shows a value of **54** and pick any point to change the width of the bed to 54 inches.

The block is now configured to represent a full-size bed.

If you enter an undefined value into the edit box for a constrained parameter, AutoCAD automatically chooses the defined value that is closest to the value you entered. For example, if you entered a value of **63** for the **Width** parameter, the width of the bed would change to **60**, which is the closest value defined in the value set.

7. Save your work and exit AutoCAD.

Chapter 34 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Define *dynamic block* in your own words.
2. Briefly describe the process of creating a dynamic block.
3. How can you rename a parameter or an action?
4. What is a value set? How would you create a value set to constrain a parameter for a dynamic block?
5. Why is careful selection of a dynamic block's base point important?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. One dynamic block action is visibility, which allows you to make a block invisible. This is somewhat similar to freezing a layer to make it invisible. When would each method of hiding drawing elements be preferable?
2. Review the **Parameter Sets** tab of the **Block Authoring Palettes**. Which parameter set would we have used to create the adjustments of the bed's length?

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Open the **bed.dwg** file and save it as **hotelplan.dwg**. Create a hotel floor plan of your choosing, making a variety of rooms and room sizes. Place beds in each of the rooms. In about half of the rooms, insert two twin-size beds. In most of the others, insert a single king-size bed. In the remaining rooms, install two queen-size beds. Save your work.

Review & Activities   Chapter 34

2. Open the file named **flathead.dwg** that you created in Chapter 33. Create a block named **dynflathead** with the screw head's center as the base point. In the block editor, create a polar parameter with the center of screw head as the base point. Specify a **Grip Display** of **1**. Assign a scale action to the parameter. Your dynamic block should look similar to the one in Fig. 34-9. Save the dynamic block and close the block editor. Insert several of the blocks into the drawing area. Scale the blocks to a variety of sizes. Use the keyboard to enter numeric values for the diameter on several, then scale the others by dragging the grip with the cursor. Your finished drawing should look similar to Fig. 34-10. Save your work as **dynflathead.dwg**.
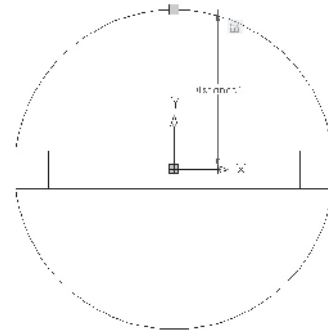
Fig. 34-9
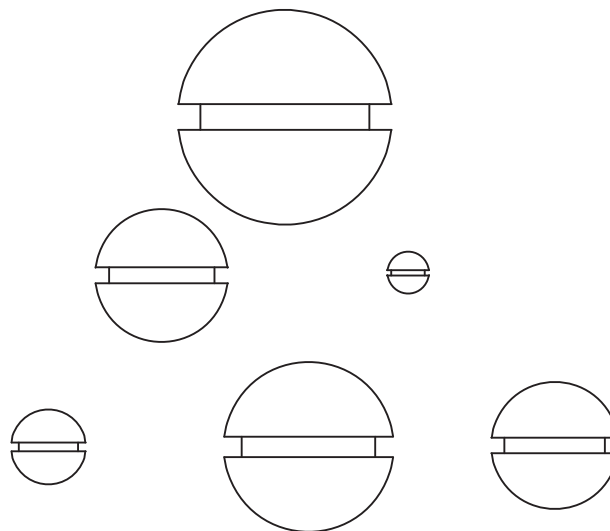
Fig. 34-10

Chapter 34 **Review & Activities**

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Open the **chair.dwg** file, which you created in Chapter 17. Delete all but one of the chairs. Make a block named **chair**, using the center of the circles as the base point. Save the drawing as **dynchair.dwg**. Create a rotation parameter and action. Rename the parameter **Chair Angle**. Your dynamic block should look like the one in Fig. 34-11. Insert six of the chair blocks into the drawing. Use dynamic rotation to rotate the individual chairs to the angles specified in Fig. 34-12.

Fig. 34-11

270°   180°   135°

0°   45°   90°
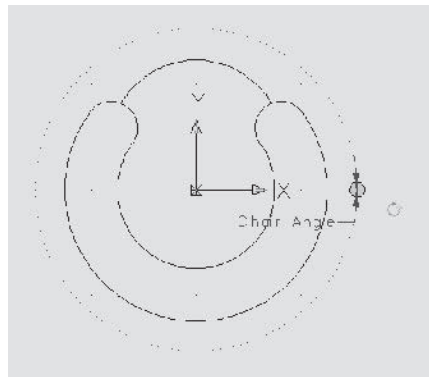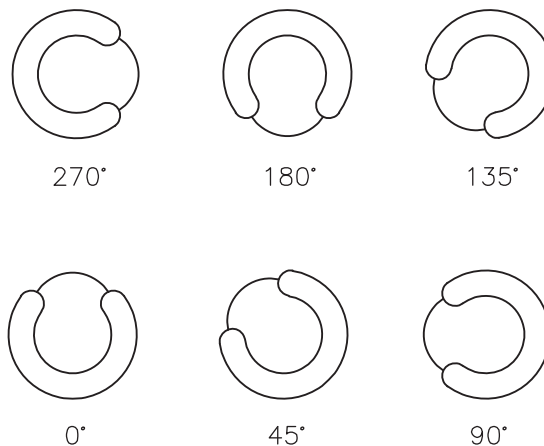
Fig. 34-12

Review & Activities Chapter 34

2. Create the hex bolt shown in Fig. 34-13. Approximate the size and shape of the hex head. Then make a dynamic block of the bolt, creating a parameter and action to stretch the length of the bolt's shank. Create a value set of the following shank lengths: 1.00, 1.25, 1.50, 1.75, 2.00, 2.50, and 3.00 inches. Place seven blocks in the drawing, and use the constraints of the value set to create one bolt of each length. Your finished drawing should look similar to Fig. 34-14. Save the file as **dynbolt.dwg**.
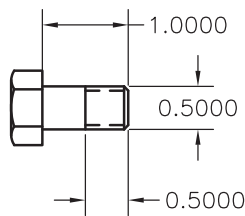
Fig. 34-13
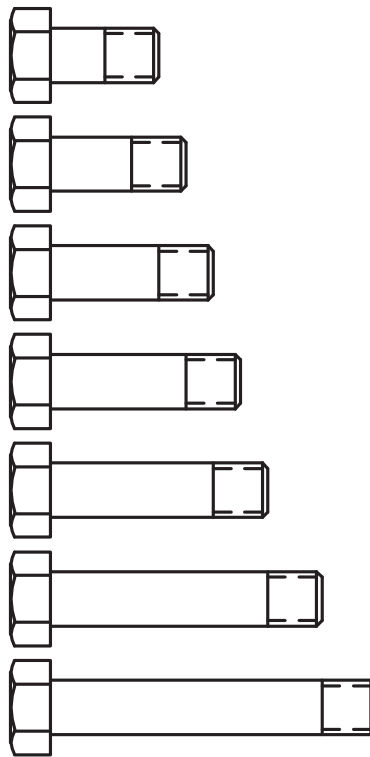
Fig. 34-14

AutoCAD at Work

Aquatic Designer

You're vacationing at a luxurious hotel in the Caribbean. You look around and see yourself surrounded by an oasis of pools and spas. Minutes later you wander outside and see fascinating aquatic structures, including a large swimming pool with many curves and bends, a waterfall, and winding water slide. Did it once cross your mind that hours of time and creativity went into designing and constructing such mesmerizing results?

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A Swarm of Water

Commercial aquatic construction takes place worldwide varying from apartment complexes, gigantic hotel water parks, to even your neighborhood water park. The design process for these structures is crucial and in-depth. The client works with the contractor and designer to establish a design within budget. The engineers and designers of such projects have to pay very close attention to detail and effectively communicate with their client and other contractors working on the project.

AutoCAD Delivers Results

Engineers of aquatic design depend on AutoCAD software throughout their workflow. AutoCAD helps to reduce the risk of interference and construction errors, eliminate the need to redraw objects, and increase client satisfaction with easy to visualize designs. It also allows them to complete hydraulic package calculations in time to keep up with the ever-changing and very competitive leisure resort business.

▶ Career Activities

1. Research the types of firms that would employ a person skilled in aquatic design.

2. What kinds of skills would these firms expect you to have?

Chapter **35**

Symbol Libraries

Objectives

- Create a library of symbols and details
- Insert symbols and details using a symbol library
- Insert layers, dimension styles, and other content from drawings using **DesignCenter**

Vocabulary

symbol library

Figure 35-1 shows a collection of electrical substation schematic symbols in an AutoCAD drawing file. Each of the symbols was stored as a single block and given a block name. (In this particular case, numbers were used for block names rather than words.) The crosses, which show the blocks'

Symbol library named lib.dwg

Note: The number below each device is the device block name.
The crosses indicate the insertion base point of each device.
The scale is full (1:1).
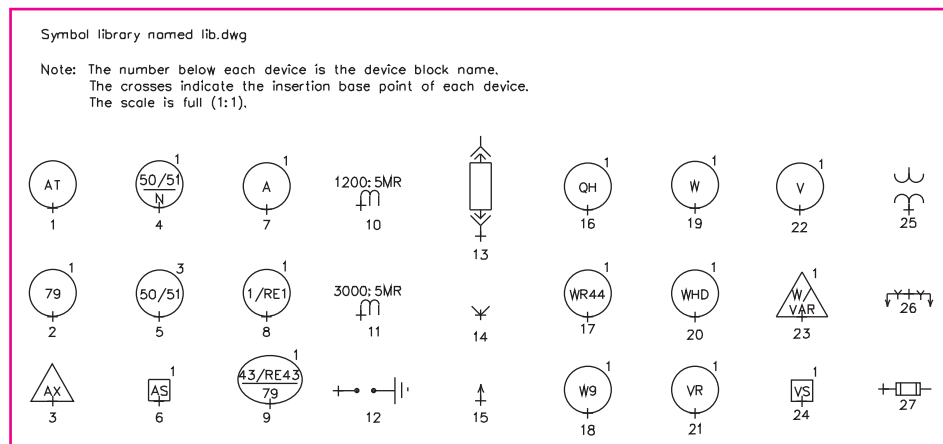
Fig. 35-1

insertion base points, and the numbers were drawn on a separate layer and frozen when the blocks were created. They are not part of the blocks; they are used for reference only. A drawing file such as this that contains a series of blocks for use in other drawings is known as a **symbol library**.

After the symbols were developed and stored in a drawing file, **DesignCenter** was used to insert the symbols into a new drawing for creation of the electrical schematic shown in Fig. 35-2.

Fig. 35-2

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VII. Groups and Details

35. Symbol Libraries

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477
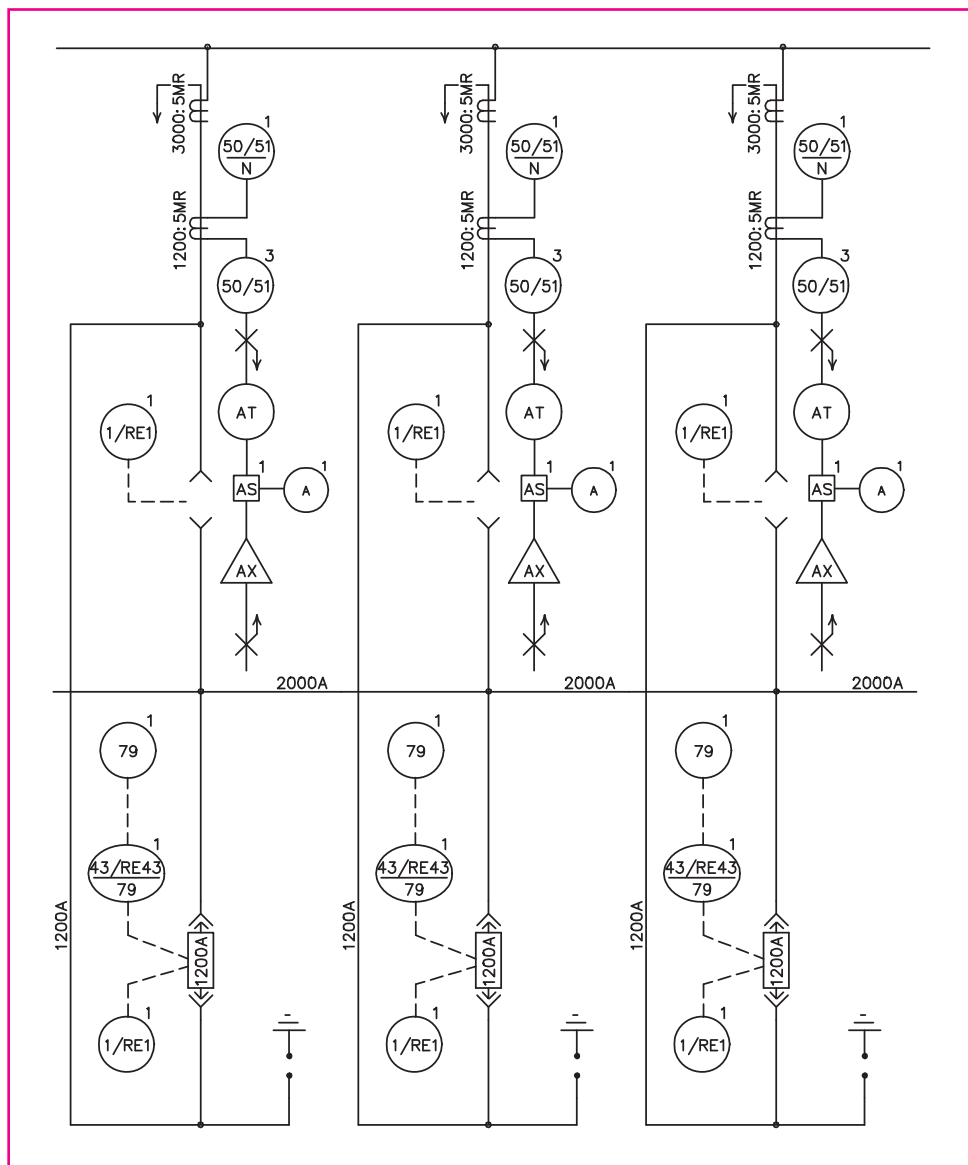
In this example, the blocks were then inserted into their proper locations, and lines were used to connect them. Because these blocks containing the symbols were available, about 80% of the work was complete before the drawing was started. This is the primary advantage of grouping blocks in symbol libraries.

Creating a Library

Let's step through a simple version of the procedures just described. Keep in mind that a symbol library may be nothing more than a drawing file that contains a collection of blocks for use in other drawings.

1. Start AutoCAD and start a new drawing using the **tmp1.dwt** template file.

2. Select the **2D Drafting & Annotation** workspace and close all floating toolbars and palettes.

3. Using the **LIMITS** command, set the drawing area to **24′ × 14′** and **ZOOM All**.

4. Create the schematic representations of tools shown in Fig. 35-3. Set snap at **3″**. Construct each tool on layer **0**. Omit the text.

5. Save your work in a file named **lib1.dwg**.

6. Create a block from each of the tools. Use the following information:

   • Use the names shown in Fig. 35-3.

   • Pick the lower left corner of each tool for the insertion base point.

   • Under **Objects**, pick the **Convert to block** radio button.

   • For **Block unit**, use inches.
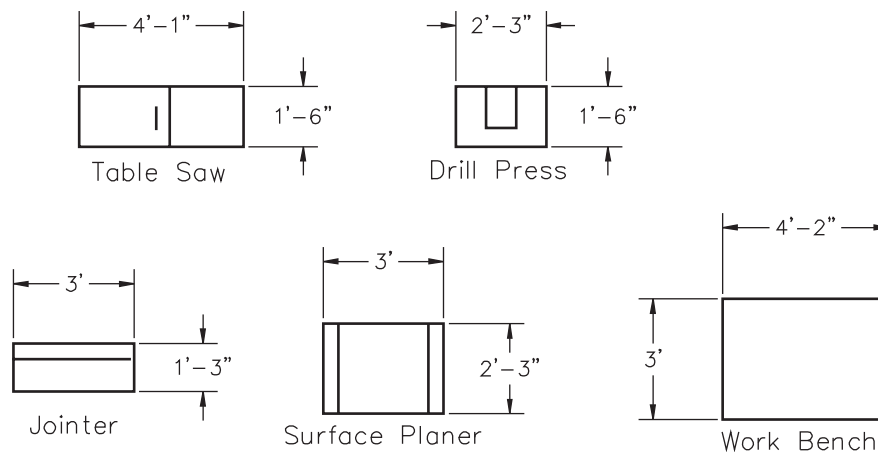
Fig. 35-3

- The block names adequately describe the blocks, so do not add a description for each block.
- Uncheck the **Open in block editor** box.

7. Save your work and close **lib1.dwg**.

> **NOTE**
>
> A block can be made up of objects from different layers, with different colors and linetypes. The layer, color, and linetype information of each object is preserved in the block. When the block is inserted, each object is drawn on its original layer, with its original color and linetype, no matter what the current drawing layer and object linetype are.
>
> A block created on layer **0** and inserted onto another layer inherits the color and linetype of the layer on which it is inserted and resides on this layer. Therefore, it is important to create blocks on layer **0** in most cases. Other options exist, but they can cause confusion.

Using a Symbol Library

We're going to use the new **lib1.dwg** file to create the workshop drawing shown in Fig. 35-4. With **DesignCenter**, we can review and insert blocks efficiently.

1. Start a new drawing using the **Quick Setup** wizard and the following information:

- Use **Architectural** units.
- Make the drawing area **22′** × **17′**.
- Set snap at **6″** and grid at **1′**.
- Be sure to **ZOOM All**.

2. Create a new layer named **Objects**, assign the color red to it, and make it the current layer.

3. Using **PLINE**, create the outline of the workshop as shown in Fig. 35-4. Make the starting and ending width **4″** and make it nearly as large as possible.

4. Save your work in a file named **workshop.dwg**.

CONTENT

5. In the **Insert** tab of the **Ribbon,** pick the **DesignCenter** button on the **Content** panel.
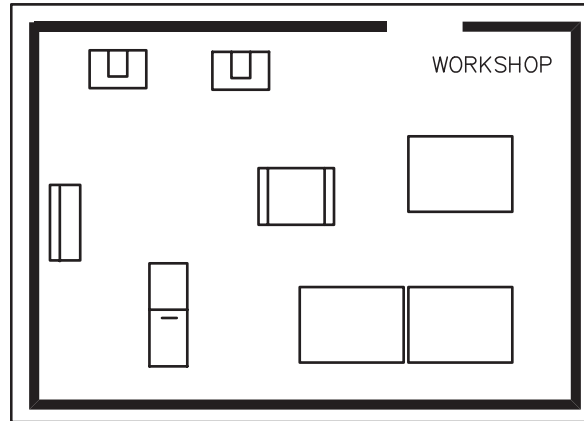
Fig. 35-4

DesignCenter appears. Notice the buttons along the top of the **DesignCenter** window.

6. Pick the **Load** button located in the upper left corner.

The **Load** dialog box appears.

7. Find the folder with your name and open it unless it is already open.

8. Find and open the drawing file named **lib1.dwg**.

This displays several types of **lib1.dwg** content in **DesignCenter**. As you can see, one of them is **Blocks**.

9. Pick the **Open Drawings** tab.

10. Double-click the **Blocks** icon.

The icons of the five blocks you created appear.

Inserting Blocks

1. Right-click and drag the jointer into the drawing area and release the right button.

2. Pick **Insert Block…** from the shortcut menu.

The **Insert** dialog box appears.

3. Under **Insertion point** and **Rotation**, check **Specify On-screen**, but uncheck this option under **Scale**.

4. Uncheck the **Explode** check box and pick the **OK** button.

5. Insert the block in the position shown in Fig. 35-4.

6. Insert the remaining blocks and save your work.

> **NOTE** If you do not need to rotate the block when you insert it, you can save steps by left-clicking and dragging the block into position in the drawing.

You can insert symbols and details that are stored as blocks from any drawing file. AutoCAD provides several good examples.

1. Pick the **Load** button in **DesignCenter** to display the **Load** dialog box.

2. Find and open AutoCAD's **Sample** folder, and then open the **DesignCenter** folder.

3. Find the file named **House Designer.dwg** and open it.

4. Double-click **Blocks** in **DesignCenter**.

5. Insert a **36″** right-swing door into the doorway of the workshop. (You may need to edit the size of the opening.)

6. Experiment with the blocks in some of the other drawing files located in the **DesignCenter** folder, such as **Home - Space Planner.dwg**.

7. Erase any blocks that are not appropriate for this drawing, and save your work.

Inserting Other Content

As you may have noticed, **DesignCenter** permits you to drag other content into the current drawing. Examples include layouts, text styles, layers, and dimension styles.

1. Pick the **Load** button in **DesignCenter** and open the **lib1.dwg** file.

2. Double-click **Dimstyles**.

The **Preferred** and **Standard** dimension styles appear because they are stored in **lib1.dwg**.

3. Left-click and drag **Preferred** into the drawing area.

AutoCAD adds the **Preferred** dimension style to the current drawing.

4. Review the dimension styles in the **Dimensions** panel of the **Annotate** tab on the **Ribbon.**

As you can see, **Preferred** was indeed added to the current drawing.

  5. Pick the **Up** button (the button that has a folder with an **Up** arrow in it) once in **DesignCenter** and double-click **Layers**.

This displays the layers contained in **lib1.dwg**.

  6. Click and drag **Center** to the drawing area.

  7. Check the list of layers in the current drawing.

Center is now among them. So, you see, it is very easy to add content such as blocks, dimension styles, layers, and text styles from another drawing to the current drawing.

Other Features

The buttons at the top of the **DesignCenter** window offer several Windows-standard features. The **Tree View Toggle** button displays and hides the tree view.

  1. Pick the **Tree View Toggle** button (fourth button from the right).

When this button is selected, it splits the window, with the left side displaying an Explorer-style tree view of your computer's files, folders, and devices.

  2. Pick the **Tree View Toggle** button again and again until the display of the tree view is present.

The **Favorites** button displays the contents of the **Favorites** folder. You can select a drawing or folder or another type of content and then choose **Add to Favorites**. AutoCAD creates a shortcut to that item, which is then added to the **AutoCAD Favorites** folder. The original file or folder does not actually move. Note that all of the shortcuts you create using **DesignCenter** are stored in the **AutoCAD Favorites** folder.

The **Search** button displays the **Search** dialog box. You can enter search criteria and locate drawings, blocks, and nongraphic objects within drawings.

The **Preview** button (third selection from the right) displays a preview of the selected item in a pane below the content area if a preview image has been saved with the selected item. If not, the **Preview** area is empty.

The **Description** button displays a description of the selected item in a pane below the content area. When creating a new block, you are given the opportunity to enter a description; this is where you can view this information.

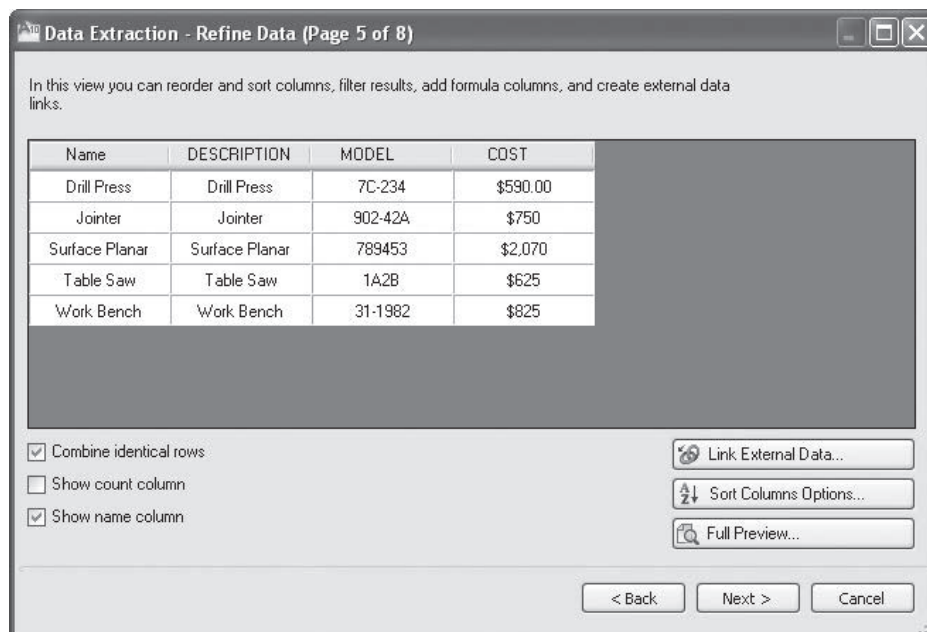 **3.** Save your work, close **DesignCenter**, and exit AutoCAD.

Chapter 35 **Review & Activities**

• REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is the primary purpose of creating a library of symbols and details?
2. When you create a symbol library, on which layer should you create and store the blocks? Why?
3. What types of content does **DesignCenter** display?
4. From **DesignCenter**, how do you insert a dimension style into the current drawing?
5. Is it possible to use **DesignCenter** to insert content from other AutoCAD drawings into the current drawing? Explain.

• CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Identify an application for creating and using a library of symbols and details. Decide what symbols the library should include. Discuss your idea with others and make changes according to their suggestions.
2. Why are blocks, rather than groups, used in symbol libraries? Can you think of any applications for which you could use groups as well as blocks in a symbol library? Explain.
3. Explore reasons it might be useful to add shortcuts to the **Favorites** folder that appears in **DesignCenter**. How might adding shortcuts to libraries of symbols and details be especially helpful?

Review & Activities Chapter 35

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Based on steps described in this chapter, create an entirely new symbol library specific to your area of interest. For example, if you practice architectural drawing, create a library of doors and windows. First create a drawing template. If you completed the first "Challenge Your Thinking" item, you may choose to create the symbol library you planned.

2. After you have completed the library of symbols and details, begin a new drawing and insert the blocks. Create a drawing using the symbol library.

3. The logic circuit for an adder is shown in Fig. 35-5. Draw the circuit by first constructing the inverter and the AND gate as blocks. Use **DONUT** for the circuit connections.
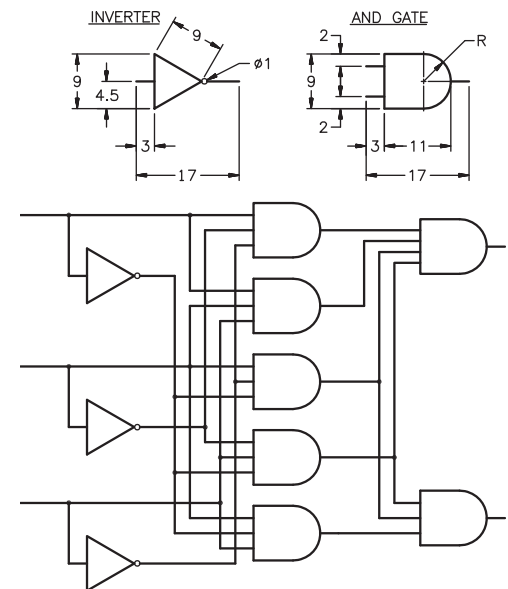
Fig. 35-5

Courtesy of Gary J. Hordemann, Gonzaga University

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. As a building contractor, you want to make a customized library from AutoCAD's existing libraries to suit your own needs. Open **DesignCenter**, select the **Sample** folder, and then pick the **DesignCenter** folder. Find out what blocks are most suitable for a building contractor. Browse through AutoCAD's drawing libraries and select the blocks that are most appropriate. Adjust the scale so that they fit on the screen. Save the drawing as **contractor.dwg**.

2. As a drafter for an electronics firm, you need a library of electronic symbols. Consult reference books to determine the correct schematic symbols for a PNP transistor, NPN transistor, resistor, capacitor, diode, coil, and transformer. Draw the symbols and make blocks of them to create an electronics symbol library. Save it as **electsymbol.dwg**.

Chapter 35: Symbol Libraries **477**

Chapter **36**

Attributes

Objectives

- Create fixed and variable attributes
- Store attributes in blocks
- Edit individual attributes

Vocabulary

attributes
attribute tag
attribute values
fixed attributes
variable attributes

In AutoCAD, **attributes** consist of text information stored within blocks. The information describes certain characteristics of a block, such as size, material, model number, cost, etc., depending on the nature of the block. The advantage of adding attribute information to a drawing is that it can be extracted to form a report such as a bill of materials. The attribute information can be made visible, but in most cases, you do not want the information to appear on the drawing. Therefore, it usually remains invisible, even when you plot the drawing.

The electrical schematic shown in Fig. 36-1 contains attribute information, even though you cannot see it. (The numbers you see in the components are not the attributes.)

Figure 36-2 shows a zoomed view of one of the components after the attribute information has been made visible. In this example, the attribute information is displayed near the top of the component.

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VII. Groups and Details

36. Attributes

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485
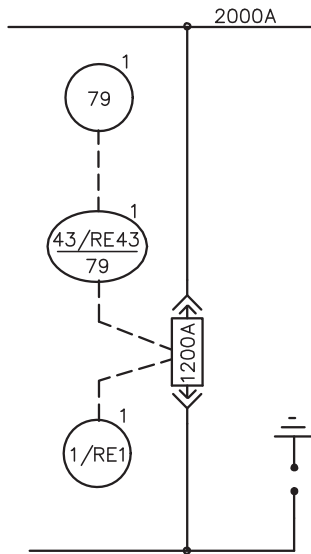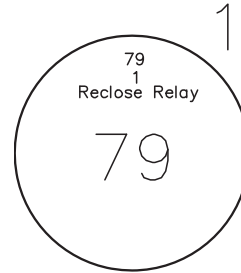
Fig. 36-1

Fig. 36-2

DESCRIPTION	DEVICE	QUANTITY/UNIT
Recloser Cut-out Switch	43/RE43/79	1
Recloser Relay	79	1
Lightning Arrester	--	3
Breaker Control Switch	1/RE1	1
1200 Amp Circuit Breaker	52	1

Fig. 36-3

All of the attributes contained in this schematic were compiled into a file and placed into a program for report generation. The report (bill of materials) in Fig. 36-3 was generated directly from the electrical schematic drawing.

Fixed Attributes

Attributes can be fixed or variable. **Fixed attributes** are those whose values you define when you first create the block. You will learn about variable attributes later in this chapter.

Creating Attributes

Attributes are created, or defined, using the **ATTDEF** command.

1. Start AutoCAD and open the drawing named **lib1.dwg**.
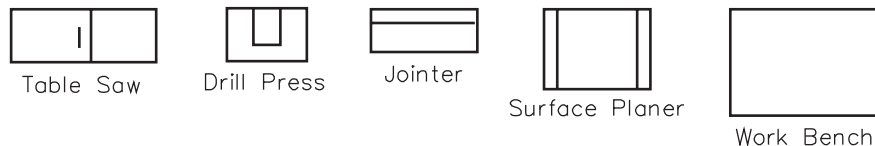
Fig. 36-4

It should look similar to the one in Fig. 36-4, but without the text.

> **NOTE** If you do not have **lib1.dwg** on file, create it using the steps outlined in Chapter 35.

2. Select the **AutoCAD Classic** workspace and close all floating toolbars and palettes.

Let's assign information (**attribute values**) to each tool so that we can later generate a bill of materials. We'll design the attributes so that the report will contain a brief description of the component, its model, and the cost.

3. Zoom in on the table saw. It should fill most of the screen.

4. From the **Draw** pull-down menu, select **Block** and **Define Attributes...**, or enter **ATTDEF** (short for "attribute definition") at the keyboard.

This enters the **ATTDEF** command and displays the **Attribute Definition** dialog box, as shown in Fig. 36-5.

5. In the **Mode** area, check **Invisible** and **Constant**.

Invisible specifies that attribute values are not displayed when you insert the block. **Constant** gives attributes a fixed value for all insertions of that block. You will learn more about these two items when you insert blocks that contain attributes.

6. In the **Attribute** area, type the word **description** (in upper- or lowercase letters) in the box at the right of **Tag**.

The **attribute tag** identifies each occurrence of an attribute in the drawing. Once again, this will become clearer after you create and use attributes.

7. In the box at the right of **Default**, type **Table Saw** (exactly as you see it here).

Table Saw becomes the default attribute value.

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VII. Groups and Details

36. Attributes

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487
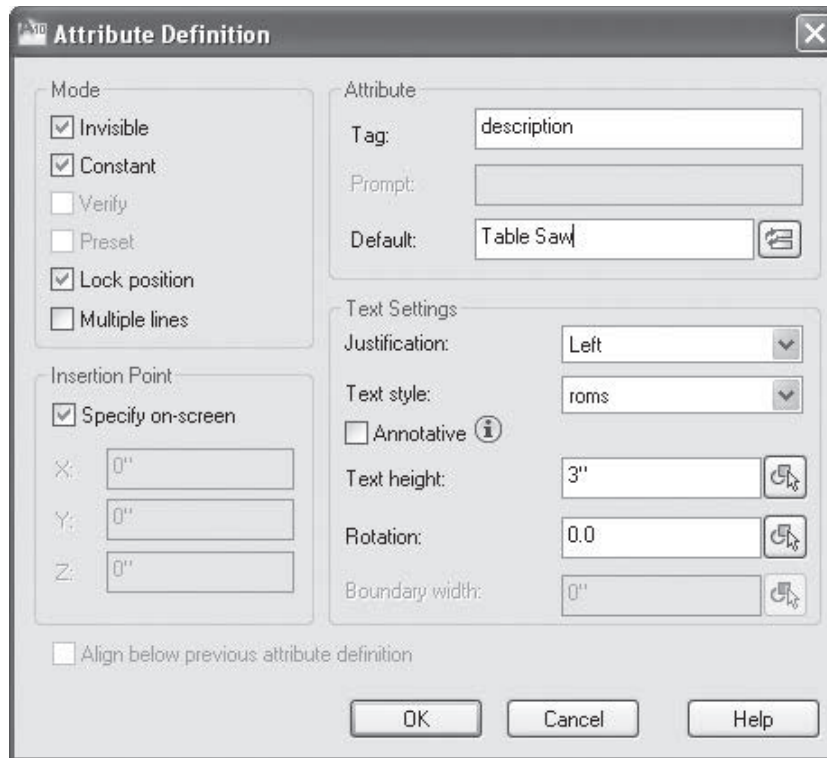
Fig. 36-5

8. Under **Text Settings**, enter the following information:
   - Justification: **Left**
   - Text Style: **roms**
   - Height: **3″**
   - Rotation: **0.0**

9. Pick the **OK** button.

10. In reply to **Specify start point**, pick a point inside the table saw, near the top left.

The word **DESCRIPTION** appears on the table saw.

11. Press the spacebar to repeat the **ATTDEF** command.

12. This time, enter **model** for the tag and **1A2B** for the **Default** value.

13. Pick the **Align below previous attribute definition** check box in the lower left corner of the dialog box and pick **OK**.

The word **MODEL** appears below **DESCRIPTION**.

14. Repeat Steps 11 through 13, but enter **cost** for the tag and **$625.00** for the value. Be sure to pick the **Align below previous attribute definition** check box.

15. Save your work.

Chapter 36: Attributes   481

Table 36-1. Attribute Information

Description	Model	Cost
Drill Press	7C-234	$590.00
Jointer	902-42A	$750.00
Surface Planer	789453	$2070.00
Work Bench	31-1982	$825.00

You are finished entering the table saw attributes. Let's assign attributes to the remaining tools.

16. Zooming and panning as necessary, assign attributes to the remaining tools. Use the information shown in Table 36-1.

17. **ZOOM All** and save your work.

HINT Be sure to pick the **Align below previous attribute definition** check box in the **Attribute Definition** dialog box when adding the second and third attributes to each tool. This will save time and will automatically align the second and third attributes under the first one.

Storing Attributes

Let's store the attributes in the blocks.

MODIFY

1. Pick the **Explode** button from the docked **Modify** toolbar and explode each of the tools.

Exploding the blocks permits you to redefine them using the same name.

DRAW

2. Pick the **Make Block** button from the docked **Draw** toolbar.

3. Pick the down arrow at the right of **Name** and pick **Table Saw**.

4. Pick the **Pick point** button and pick the lower left corner of the table saw for the insertion base point.

5. Pick the **Select objects** button, select the table saw and its attributes, and press **ENTER**.

6. Remove the checkmark in the **Open in Block editor** box.

7. Pick the **OK** button.

8. When AutoCAD asks if you want to redefine **Table Saw**, pick **Redefine**.

This redefines the **Table Saw** block.

9. Repeat Steps 2 through 8 to redefine each of the remaining tool blocks.

10. Save your work.

All of the tools in **lib1.dwg** have been redefined and now contain attributes. When you insert these tools into another drawing, the attributes will insert also. You will learn more about attribute extraction and bills of materials in Chapter 37.

Displaying Attributes

Let's display the attribute values using the **ATTDISP** (short for "attribute display") command.

1. Enter **ATTDISP** and specify **On**.

You should see the attribute values, similar to those shown in Fig. 36-6.

2. Reenter **ATTDISP** and enter **N** for **Normal**.

The attribute values should again be invisible.


```
Table Saw
1A2B
$625.00
```

Fig. 36-6

Variable Attributes

Thus far, you have experienced the use of fixed attribute values. With **variable attributes**, you have the freedom of changing the attribute values as you insert the block. Let's step through the process.

1. Using the **tmp1.dwt** template file, begin a new drawing.

2. Zoom in on the lower left quarter of the display, make layer **0** the current layer, and set the snap resolution to **2″**.

3. Draw the architectural window symbol shown in Fig. 36-7. The dimensions not given are **2″** in length. Do not place dimensions on the drawing. (The symbol represents a double-hung window for use in architectural floor plans.)

4. Save your work in a file named **window.dwg**.

Fig. 36-7

Chapter 36: Attributes   **483**
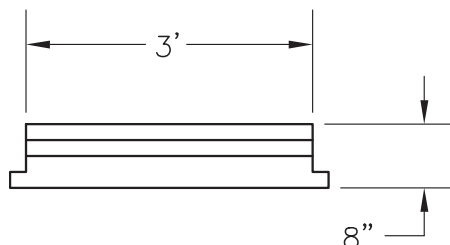
5. Enter the **ATTDEF** command and check **Invisible**, but **Constant**, **Verify**, and **Preset** should be unchecked.

6. For the tag, type the word **type**.

7. In the **Prompt** edit box, type **What type of window?**

8. In the **Default** edit box, type **Double-Hung**.

9. Under **Text Settings**, enter the following information.
- Justification: **Center**
- Text Style: **roms**
- Height: **3″**
- Rotation: **0.0**

10. Pick the **OK** button and pick a point over the top center of the window, leaving space for two more attributes.

The word **TYPE** appears.

11. Redisplay the dialog box, and leave the attribute modes as they are.

12. Type **size** for the attribute tag, **What size?** for the attribute prompt, and **3′ × 4′** for the attribute value.

13. Pick the **Align below previous attribute definition** check box, and pick **OK**.

The word **SIZE** appears on the screen below the word **TYPE**.

14. Repeat Steps 11 through 13 using the following information.
- Tag: **manufacturer**
- Prompt: **What manufacturer?**
- Value: **Andersen**

15. Store the window symbol and attributes as a block. Name it **DH** (short for "double-hung") and pick the lower left corner for the insertion base point.

16. Pick **OK** to close the **Block Definition** dialog box.

The **Edit Attributes** dialog box appears.

17. We do not want to edit the attributes at this time, so pick the **Cancel** button and save your work.

18. Enter **ATTDISP** and **On**.

Did the correct attribute values appear?

Inserting Variable Attributes

1. Enter the **INSERT** command and select the block named **DH** in the **Insert** dialog box.

2. Uncheck **Explode** if it is checked.

3. Make sure **Specify On-screen** is checked under **Insertion point** (**Scale** and **Rotation** should be unchecked) and pick **OK**.

4. Pick an insertion point at any location.

5. In reply to **What manufacturer?**, press **ENTER** to accept **Andersen**.

6. In reply to **What size?**, enter **3′ × 5′**.

7. In reply to **What type of window?**, press **ENTER** to accept **Double-Hung**.

The block and attributes appear.

Editing Attributes

ATTEDIT, short for "attribute edit," allows you to edit attributes in the same way as you edited the size attribute of the double-hung window. Your drawing should currently look similar to the one in Fig. 36-8. The exact location of the attributes is not important.

1. Enter the **ATTEDIT** command and pick one of the two blocks.

This displays the **Edit Attributes** dialog box.

2. Change **Andersen** to **Pella** and pick **OK**.

Andersen changes to **Pella**.

3. Save your work and close the file named **window.dwg**.

The **lib1.dwg** file again becomes the current drawing.
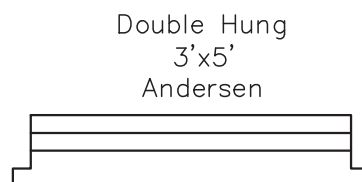
Fig. 36-8
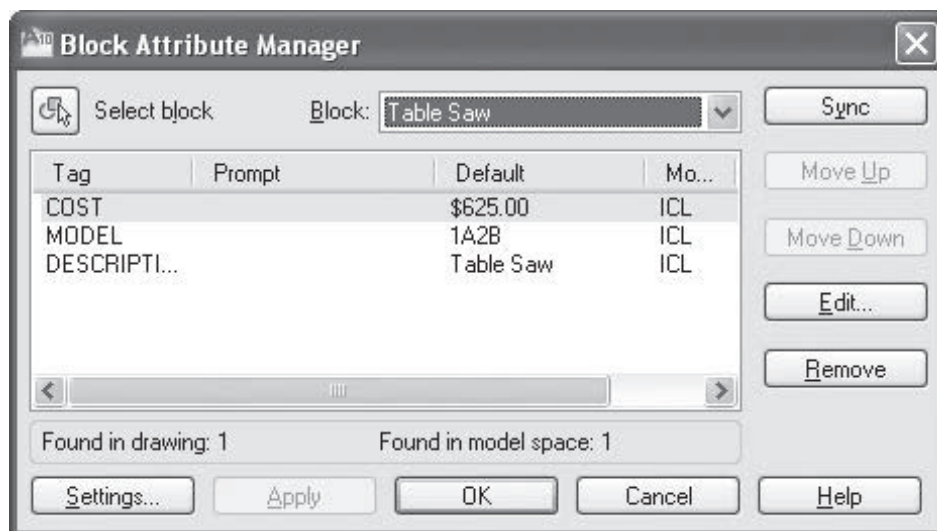
Block Attribute Manager

AutoCAD's **Block Attribute Manager** permits you to edit the attribute definitions in blocks, remove attributes from blocks, and change the order in which you are prompted for attribute values when inserting a block.

1. Enter the **BATTMAN** command.

This displays the **Block Attribute Manager** dialog box, as shown in Fig. 36-9.

2. Pick the down arrow located at the right of **Block** near the top of the dialog box and pick **Surface Planer**.

3. Select the line that includes **COST** and **$2070.00** and pick the **Edit...** button.

This displays the **Edit Attribute** dialog box.

4. Pick each of the tabs and review the information and options in each one and then pick the **Cancel** button.

5. Pick **Cancel** and exit AutoCAD without saving.

Chapter 36 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Explain the purpose of creating and storing attributes.
2. Briefly define each of the following commands.
   a. **ATTDEF**
   b. **ATTDISP**
   c. **ATTEDIT**
   d. **BATTMAN**
3. What are attribute tags?
4. What are attribute defaults?
5. Explain the attribute modes **Invisible** and **Constant**.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Discuss the advantages and disadvantages of using variable and fixed attribute values. Why might you sometimes prefer one over the other?
2. Brainstorm a list of applications for blocks with attributes. Do not limit your thinking to the applications described in this chapter. Compare your list with lists created by others in your group or class. Then make a master list that includes all the ideas.

Continued

Chapter 36 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Load the **livroom.dwg** drawing containing the furniture representations you created in Chapter 33. If this file is not available, create a similar drawing. Outline a simple plan for assigning attributes to each of the components in the drawing. Create the attributes and redefine each of the blocks so the attributes are stored within the blocks.

2. The small hardware shop in which you work is computerizing its fasteners inventory. The shop carries both U.S. and metric sizes. All of the available fasteners are saved as blocks in the **DesignCenter** folder in AutoCAD's **Sample** folder. Place the blocks from **Fasteners – Metric.dwg** and **Fasteners – US.dwg** into a drawing named **hardware.dwg**. Place them in an orderly and logical fashion. Then assign attributes to each fastener to describe it. Use attribute tags of **Type** and **Size**. For the individual attribute values, use the information given in the name of each block.

3. Look in a computer hardware catalog or on the Internet to find the sizes and basic shapes of several printers from different manufacturers. Draw the basic shape of at least five different printers. Create attributes to describe the printers in terms of type (inkjet, laser, etc.), brand, and cost. Save the drawing as **printers.dwg**.

4. Refer to **schem.dwg** and **schem2.dwg** located on the OLC for this book, in the Drafting & Design Problems section. Display and edit the attributes contained in the drawing files.

● USING PROBLEM-SOLVING SKILLS

Complete the following problems using problem-solving skills and your knowledge of AutoCAD.

1. Attributes are used for many reasons besides creating bills of materials. For example, you can use attributes to track items in a collection of anything from model cars to unique bottles to action figures. For this problem, suppose you are a dealer in and collector of Avon® bottles that have a transportation theme. You have been asked to participate in a display of various collections being sponsored by the local library. Display space is limited, so you need to make a layout to determine your specific needs. Create a library of schematic symbols for the car-shaped bottles shown

Review & Activities Chapter 36

Table 36-2. Collectable Attributes

Bottle	Contents	Price
1937 Cord	Wild Country	$12.00
1951 Studebaker	Spicy	$8.00
1988 Corvette	Spicy	$8.00
Silver Deusenberg	Oland	$13.00
1933 Pierce Arrow	Deep Woods	$10.00

in Table 36-2. The cars vary in size from **2″ × 5″** for the Corvette and Studebaker to **2½″ × 8″** for the Pierce Arrow and **3″ × 9″** for the Cord and Deusenberg. Assign the attributes as shown in Table 36-2. Save the drawing as **cars.dwg**.

2. You are setting up sprinklers for new landscaping for a residential plot. Create a new drawing with architectural units and a drawing area of **300′ × 200′**. Draw the plot boundaries with corners at the following absolute coordinates:

45′,170′

235′,140′

210′,45′

35′,25′

The house will be a rectangle of **60′ × 32′**. The northwest corner of the house is at absolute coordinates **90′, 112′**. Your client wants a sprinkler system with enough sprinkler heads to cover the entire property. Each sprinkler head can be set to turn a full 360° or any number of degrees greater than 30°. When set to turn 360°, each sprinkler can cover a radial area of approximately 30′. Decide on the number of sprinklers needed, their placement on the property, and the angle (number of degrees) that each should be set to cover. Then create the sprinkler heads, load them from your library, or load them from **DesignCenter**. Place them in the locations you have determined. On a separate layer, draw the range of each sprinkler to show the client that the sprinkler heads will indeed cover the entire property.

Finally, define attributes with the following tags: **Location**, **Angle**, and **Direction**. Assign these attributes to each sprinkler head in your drawing and give them the appropriate values according to the plan you have developed. Save the drawing as **sprinklers.dwg**.

AutoCAD at Work

Architectural Drafter

What type of house do you live in? Perhaps it's an older home, where floorboard creaks and groans are part of its charm. Or maybe it is tall, narrow, and practically bumps shoulders with your neighbor's house, like many homes located in the city. If you live in the country, your home may sprawl across your property, surrounded by plenty of grass and shade trees.

Although many people are responsible for the actual construction of homes and other buildings, architectural drafters play an important role in the planning stages of a building's design.
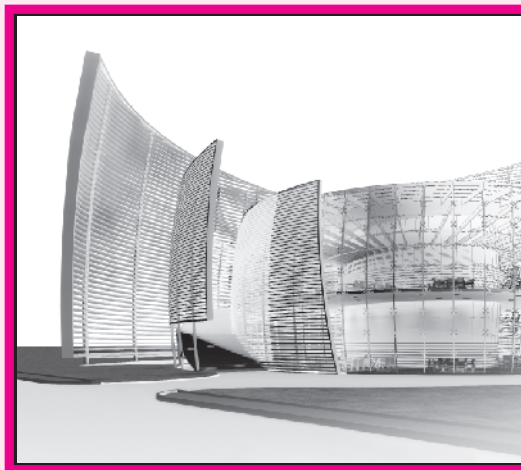
Courtesy of Autodesk, Inc.

Climbing the Architectural Ladder

Architectural drafters are employed by engineering firms, corporate development departments, and architectural and construction companies. Apprenticeships are sometimes offered for students interested in entering this field. Architectural drafters may begin as tracers or junior drafters. After gaining experience, they may advance to senior drafting or checking positions.

Paying Attention to Detail

Architectural drafters must be able to visualize objects in two and three dimensions. They must also exhibit good eye-hand coordination, which is a skill necessary to create fine, detailed drawings. The ability to pay attention to detail is a critical characteristic for an architectural drafter to possess; just one minor design flaw can result in a major problem, such as a lopsided doorway or an imperfect foundation. For information on student involvement in architecture, visit the American Institute of Architecture Students (AIAS) Web site.

► Career Activities

1. What types of detail do you think architectural drafters must be aware of when creating drawings? Compare your list with a classmate's.

2. Has employment in this field been steady over the past few years? What are the forecasts for future employment?

Chapter **37**

Bills of Materials

Objectives

- Extract attribute data from blocks for report generation
- Create a bill of materials

Vocabulary

data extraction
comma-delimited file
xref

After finishing the attribute assignment process (Chapter 36), you are ready to create a report such as a bill of materials. The first step in this process involves **data extraction** (gathering the attribute data and placing it into an AutoCAD table or an electronic file that can be read by a computer program).

1. Start AutoCAD and begin a new drawing using the **tmp1.dwt** template file.

2. Select the **2D Drafting & Annotation** workspace and close all floating toolbars and palettes.

3. Create the four walls of a workshop (fill the upper two-thirds of the drawing area) and insert each of the blocks on layer **Objects** as shown in Fig. 37-1 on page 492.

4. Save your work in a file named **extract.dwg**.
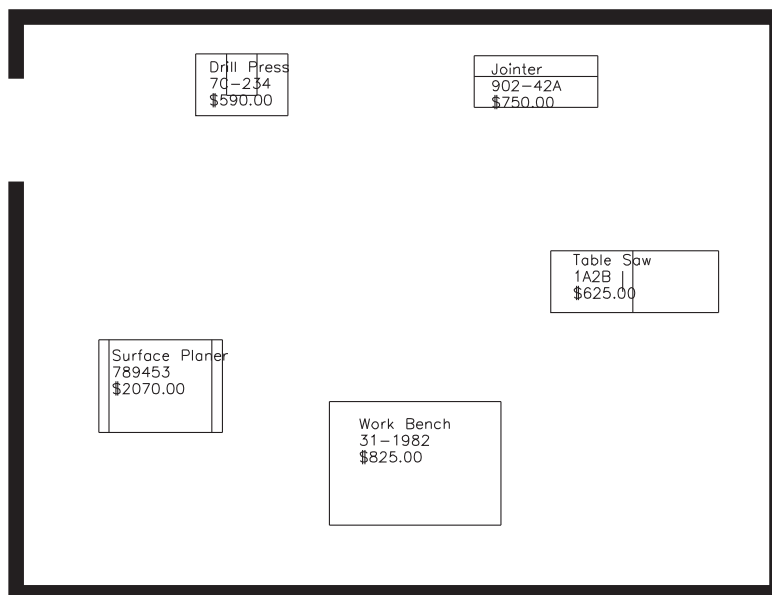
Fig. 37-1

5. Enter **ATTDISP** and **On**.

Each of the tools contains attributes, as shown in Fig. 37-1.

Data Extraction

The **EATTEXT** command permits you to extract the attributes from the **extract.dwg** file using the **Data Extraction** wizard.

1. From the **Tools** menu on the **Menu Bar**, select **Data Extraction…** or enter the **EATTEXT** command at the keyboard.

This starts the **Data Extraction** wizard, as shown in Fig. 37-2. There are eight pages in the **Data Extraction** wizard.

2. Make sure the radio button for **Create a new data extraction** is selected and pick **Next**.

3. Save your data extraction as **extract.dxe** in the file with your name on it.

4. On page 2 of the wizard, pick the **Select objects in the current drawing** radio button, because we want to extract attributes from blocks in the current drawing. The current drawing, **extract.dwg**, should be listed in the **Drawing files and folders** area.

5. Pick the **Settings...** button before continuing. Review the options available in the **Additional Settings** dialog box.

The **Extraction settings** area allows you to select different blocks to be included in the extraction. The **extract. dwg** file does not contain any external references (xrefs), so it does not matter whether the second and third boxes are checked. An **xref** is a drawing that is attached (linked) to the current drawing. Any changes made to the attached drawing are reflected in the current drawing.
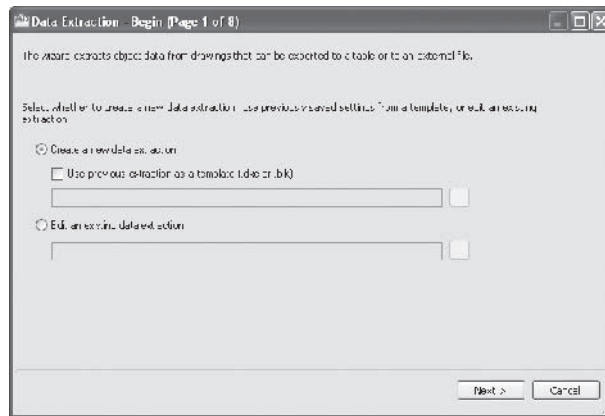
Fig. 37-2

The **Extract from** area allows you to extract blocks from the entire drawing or only blocks from the model space. Again, it does not matter which radio button is selected, because **extract.dwg** contains no blocks in layout space.

6. Pick **Cancel** to exit the **Additional Settings** dialog box.

7. Pick the Select objects button to the right of the **Select objects in the current drawing** radio button, select all five blocks in the drawing, and press **ENTER**.

8. Pick **Next** to go to the next page of the wizard.

Selecting and Reviewing the Attributes

Page 3 of the Data Extraction wizard displays the object names that you selected, as shown in Fig. 37-3 on page 494. The **Objects** area on top lists all the items you selected. The **Display options** at the bottom of the dialog box allows you to choose the object types to display.

1. Uncheck **Display all object types** and pick the radio button next to **Display blocks only**.

The object list now contains only the five blocks.

2. Check the boxes for all five blocks and pick **Next**.

Page 4 of the wizard asks which properties of the selected blocks you want to extract.
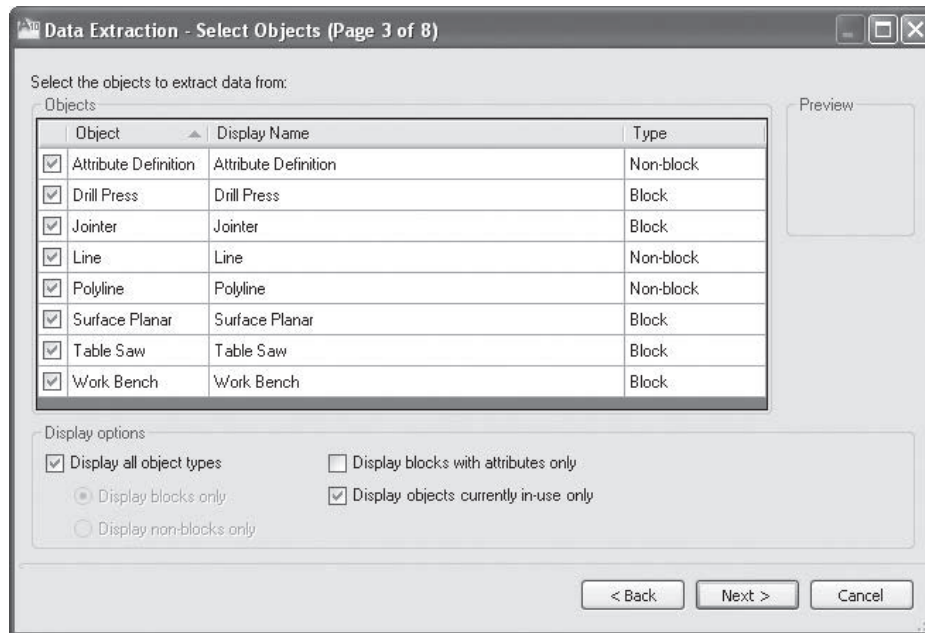
Fig. 37-3

First, focus on the **Category filter** area on the right of the dialog box. All the categories are checked, so all the properties available are listed on the left.

3. Uncheck all the checkboxes under **Category filter** except the box in front of **Attribute**.

Now only the three attribute properties are listed under **Properties**.

4. Select (highlight) the three attributes and pick **Next**.

Page 5 of the wizard allows you to refine the data in the data extraction set. We want to rearrange the columns of the table into a more logical order for a bill of materials.

5. Click in the **COST** column heading and drag it to the far right side of the table.

6. Right-click on the **DESCRIPTION** column heading and select **Sort Ascending**.

7. Uncheck the box in front of **Show count column**.

The preview of your table should now look similar to the table in Fig. 37-4. As you can see, you have many options for arranging the rows and columns of your table.

8. Pick the **Next** button.

Page 6 of the wizard allows you to choose output options for the data extraction. You can insert the table in a drawing, output the data to an external

file, or both. You have the option of creating a Microsoft Excel® (**XLS**) file, a comma-delimited (**CSV**) file, a Microsoft Access® (**MDB**) file, or a text extract (**TXT**) file. A **comma-delimited file** is a popular format for importing data into databases and other programs.

9. Check the box next to **Insert data extraction table into drawing** and pick **Next**.

Page 7 of the **Data Extraction** wizard allows you to select a style for the attribute table. You can modify an existing table style or create a new one.

10. Pick **Next** to accept the **Standard** table style.

Page 8 of the **Attribute Extraction** wizard is displayed.

Placing the Attribute Table in a Drawing

1. Pick **Finish**.

The table attaches to the crosshairs so that you can place it in the drawing.

2. Pick a point below the workshop in response to **Specify insertion point**.

As you can see, our bill of materials is much too small for the drawing.

3. Type **SCALE**, select the table, and use a scale factor of **20**.

4. Move the table so it is centered below the walls of the workshop.

5. Save your work and exit AutoCAD.

Fig. 37-4

Chapter 37 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Describe the purpose of AutoCAD's **EATTEXT** command.
2. Describe the process of creating a report or bill of materials from a drawing that contains blocks with attributes.
3. Name the file types into which you can store the attributes using the **Attribute Extraction** wizard.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Find out more about comma-delimited files. After you have extracted a **CSV** file, explore and list options for opening and printing its contents.
2. Research template (**BLK**) files. Write a short paragraph explaining what they do, what they contain, and why they are useful.

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Open the **livroom.dwg** file from Chapter 33, which contains several pieces of furniture. Add attributes to the furniture and create blocks. Using the **Data Extraction** wizard, create a bill of materials and place it in the drawing.
2. Using the attribute data file created in problem 1, insert its contents into Microsoft Excel® or another spreadsheet if you have access to one.
3. Open **sprinklers.dwg** from Chapter 36. Create a list of the sprinkler heads for the landscaping contractor to use as reference when the sprinkler system is installed.

Review & Activities Chapter 37

4. Refer to **schem.dwg** and **schem2.dwg** located on the OLC for this book, in the Drafting & Design Problems section. Using the procedure outlined in this chapter, create a bill of materials from each of the two drawings.

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Open **cars.dwg** from Chapter 36. Using the schematic blocks, create a display to fit in a $2^1/2' \times 3'$ display space. Before you begin, you may want to consult antique car references if you are unfamiliar with any of the car models. Knowing what the cars look like will help you position them attractively in the display. Then extract the attribute data and create a list to distribute to potential customers.

2. Open **printers.dwg** from "Applying AutoCAD Skills" problem 3 in Chapter 36. Extract the attribute data from the printer blocks you created and create a table of the printer attributes. Display the table and all of the blocks in the drawing.

Continued

Computer Assembly Library

One rule of thumb for CAD drafters and designers is not to draw anything twice. Although that may sound strange at first, what it really means is to use CAD commands and functions to reproduce the same details again and again without having to redraw them. You are already familiar with the **MIRROR**, **COPY**, and **ARRAY** commands. To extend the capabilities of these and other commands, AutoCAD also permits you to block two or more objects so that you can manipulate them as one. These blocks of objects may also be assigned characteristics or attributes, which can be retrieved and listed later if desired.

▶ Description

A business that assembles computers has decided that it wishes to automate its advertising and billing practices. To do so, the CAD drafter has been assigned the task of drawing basic symbols to represent the various parts of the computer. Figure P7-1 shows possible configurations for CPUs, hard disk drives, and DVD drives as an example, and Fig. P7-2 shows a possible symbol library based on these configurations.

CPUs	
2.2 Gigahertz	$300
2.5 Gigahertz	$350
3.0 Gigahertz	$400
3.3 Gigahertz	$450
3.8 Gigahertz	$550
Hard Drives	
320 Gigabyte	$125
900 Gigabyte	$175
1.0 Terabyte	$225
7.5 Terabyte	$450
DVD-ROM Drives	
24x	$35
48x	$75
Read/Write	$300

Fig. P7-1

Creating the Symbol Library

Using the examples shown in Figs. P7-1 and P7-2, create a symbol library that contains various components of a computer. Do not limit the library to

Wohlers: Applying
AutoCAD® 2010

VII. Groups and Details

Part 7 Project: Applying
Chapters 32–37

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505

just the three parts shown in Figs. P7-1 and P7-2. Do research if necessary to find out what other components should be included in the symbol library and the configurations for each component. After you have created the symbols, assign appropriate attributes to them, as shown in Fig. P7-1, block the individual symbols, and save the symbol library.

Creating the Drawing

Create a new drawing and use the symbol library to configure three computers—a low-cost model, a high-end "supermodel," and a middle-of-the-road model. Extract the attributes and create a list of materials needed to assemble the three computers.
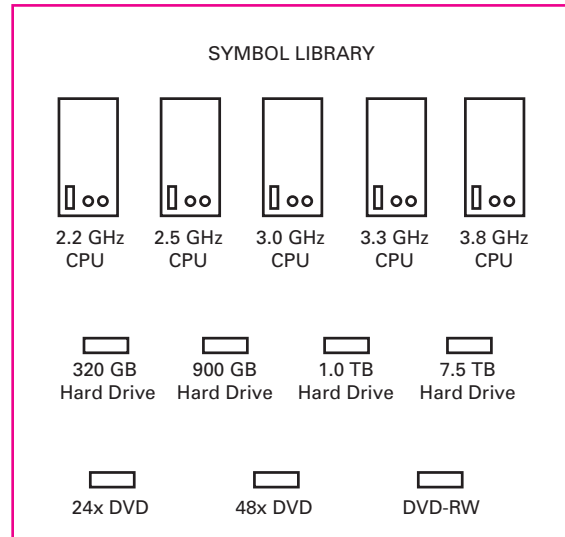
Fig. P7-2

▶ Hints and Suggestions

1. Create all blocks on layer **0**.
2. Create only one symbol for each type of component (CPU, hard disk drive, etc.). Assign attributes to it. Then assign parameters to create a dynamic block that can be used to represent all of the models in that category.

▶ Summary Questions

Your instructor may direct you to answer these questions orally or in writing. You may wish to compare and exchange ideas with other students to help increase efficiency and productivity.

1. Distinguish between groups and blocks.
2. Describe the function of a symbol library.
3. What might be an advantage of maintaining a library of dynamic blocks?
4. What factors might determine the attributes assigned to a block?
5. Why is creating a bill of materials from a set of blocks a timesaving technique? Explain.

Chapter 38

Isometric Drawing

Objectives

- Set up a drawing file for an isometric drawing
- Create an isometric drawing
- Dimension an isometric drawing correctly

Vocabulary

isometric drawing
isometric planes
pictorial representation

An **isometric drawing** is a type of two-dimensional drawing that gives a three-dimensional (3D) appearance. Isometric drawings are often used to produce a **pictorial representation** of a machined part, injection-molded part, architectural structure, or some other physical object. Pictorial representations are useful because they provide a realistic view of the object being drawn, compared to orthographic views. The drawing in Fig. 38-1 is an example of an isometric drawing created with AutoCAD.
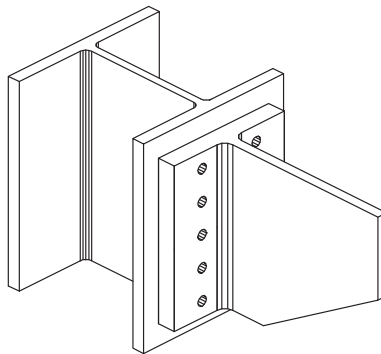
Fig. 38-1

Setting Up an Isometric Drawing

Because isometric drawings are two-dimensional, all of the lines in an isometric drawing lie in a single plane parallel to the computer screen. The drawing achieves the 3D effect by using three axes at 120° angles, as shown in Fig. 38-2. In AutoCAD, isometric drawing is accomplished by changing the **SNAP** style to **Isometric** mode.
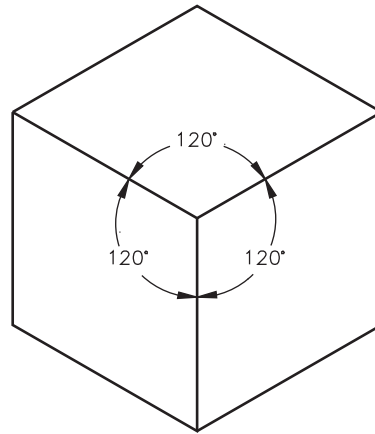
Isometric Planes

As you can see from Fig. 38-2, the 120° angles produce left, right, and top imaginary drawing planes. These three planes are known as the **isometric planes**.

Fig. 38-2

1. Start AutoCAD and begin a new drawing from scratch using **Imperial** units.

2. Select the **AutoCAD Classic** workspace and close all floating toolbars and palettes.

3. Set the drawing limits to **15,12**.

4. Right-click on the **Snap** or **Grid** button on the status bar and select **Settings...** from the shortcut menu.

5. Set the grid to **1** unit and the snap resolution to **.5** unit.

6. In the **Snap type** area of the dialog box, set the **Grid snap** to **Isometric snap.**

7. Pick **OK** to close the **Drafting Settings** dialog box.

You should now be in **Isometric** drawing mode, with the crosshairs shifted to one of the three isometric planes.

8. Move the crosshairs and notice that it runs parallel to the isometric grid.

> **NOTE**
>
> You can also enter isometric snap mode with the **SNAP** command. Enter **S** for **Style** and **I** for **Isometric.**

Toggling Planes

When you produce isometric drawings, you will use more than one of the three isometric planes. To make your work easier, you can shift the crosshairs from one isometric plane to the next.

1. Press the **CTRL** and **E** keys and watch the crosshairs change.
2. Press **CTRL** and **E** again, and again.

As you can see, this toggles the crosshairs from one isometric plane to the next.

NOTE

You can also toggle the crosshairs using the **ISOPLANE** command. Enter **ISOPLANE** and then enter **L**, **T**, or **R** to change to the left, top, and right planes, respectively. The **F5** key also toggles among the three planes.

Creating an Isometric Drawing

Let's construct a simple isometric drawing.

1. Create a layer named **Objects**, assign a color of your choice to it, and make it the current layer.
2. Enter the **LINE** command and draw the aluminum block shown in Fig. 38-3. Make it **3 × 3 × 5** units in size.
3. Save your work in a file named **iso.dwg**.
4. Alter the block so that it looks similar to the one in Fig. 38-4 using the **LINE**, **BREAK**, and **ERASE** commands.
5. Further alter the block so that it looks similar to the one in Fig. 38-5. Use the **ELLIPSE** command to create the holes.

Fig. 38-3

HINT
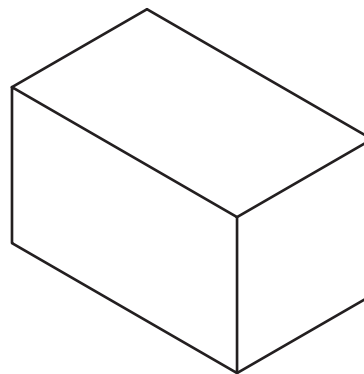
Since the ellipses are to be drawn on the three isometric planes, toggle the crosshairs to the correct plane. Then choose the **ELLIPSE** command's **Isocircle** option. You will need to change the snap spacing to **.25**.
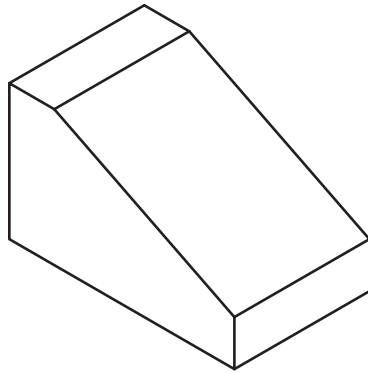
Fig. 38-4
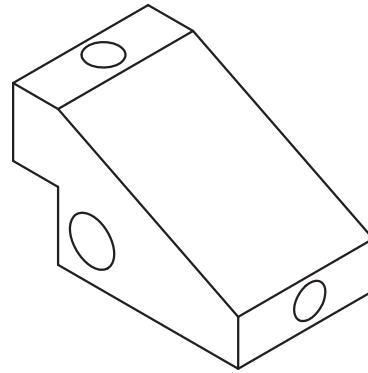
Fig. 38-5

Dimensioning an Isometric Drawing

To dimension an isometric drawing properly, you must rotate the dimensions so that they show more clearly what is being dimensioned. To do this, you can use the **Oblique** option of the **DIMEDIT** command. Compare the two drawings shown in Fig. 38-6. The drawing at the right is the result of applying oblique to the drawing at the left.
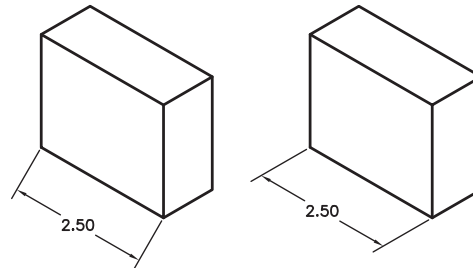
Fig. 38-6

Let's dimension the aluminum block.

1. Create a new layer named **Dimensions**, assign a different color to it, and make it current.

2. In the lower left corner of the block, draw one horizontal and one vertical extension line as shown in Fig. 38-7.

3. Display the **Dimension** toolbar and use the **Aligned** button to create the dimensions as shown in Fig. 38-7.
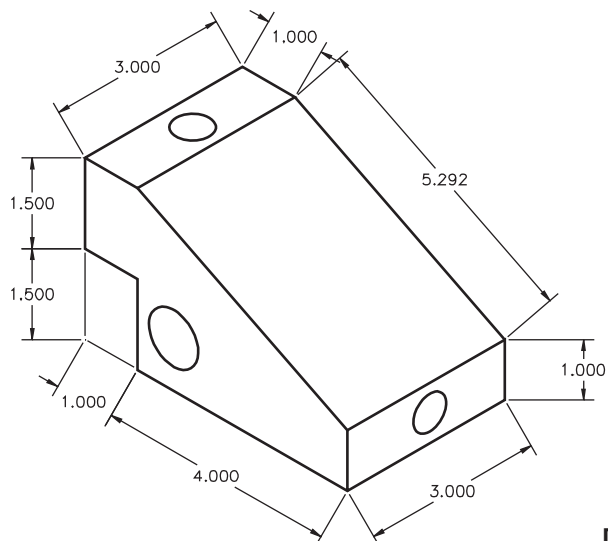
Your drawing should now look similar to the one in Fig. 38-7.

DIMENSION

Fig. 38-7

Chapter 38: Isometric Drawing   503

Notice that dimensions are not well-oriented. This is because you have not yet applied oblique to them.

DIMENSION

4. From the **Dimension** toolbar, pick the **Dimension Edit** button to enter the **DIMEDIT** command.

5. Enter **O** for **Oblique**.

6. Select the **3.000** dimension at the top of the block and press **ENTER**.

7. Enter **–30** for the obliquing angle.
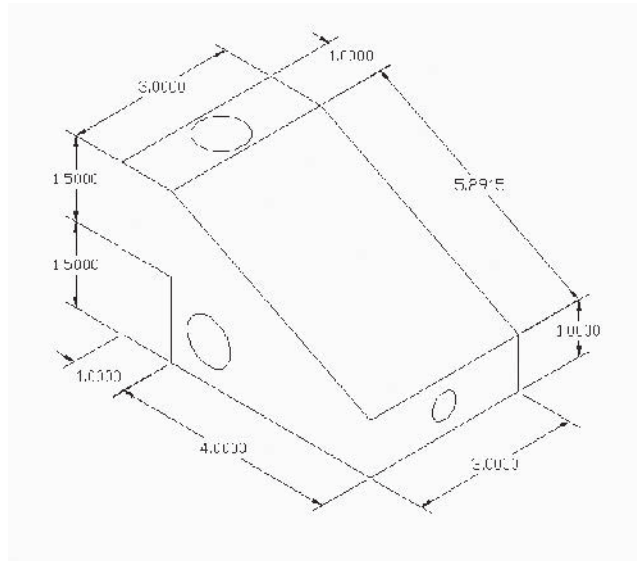
Fig. 38-8

The dimension rotates –30° to take its proper position.

8. Use **DIMEDIT** to correct the appearance of the remaining dimensions so that they match those in Fig. 38-8.

> **HINT**
>
> Enter an obliquing angle of **30** or **–30**, depending on the dimension you are editing. Use grips if necessary to adjust the positions of the dimension lines.

9. Select the **1.000** dimension at the top of the block. Right-click and select **Properties** from the shortcut menu, and change **Arrow 2** to **None** for a cleaner appearance.

10. Use the **Dimension Space** button to align the two sets of dimensions on the left and lower left sides of the drawing.

When you are finished, your drawing should look like the one in Fig. 38-8.

11. Return to AutoCAD's standard drawing format using the **Drafting Settings** dialog box or the **SNAP** command.

12. Save your work and exit AutoCAD.

Chapter 38 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Describe two methods of changing from AutoCAD's standard drawing format to isometric drawing.
2. Describe how to change the isometric crosshairs from one plane to another.
3. Explain how to create accurate isometric circles.
4. How do you correct the appearance of dimensions used on an isometric drawing?
5. How do you change from isometric drawing to AutoCAD's standard drawing format?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Discuss the purposes of isometric drawings. When and why are they used?
2. If you were to dimension the holes in the aluminum block created in this chapter, how would you go about it?

Continued

Chapter 38 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1–3. Create the machined parts shown in Figs. 38-9 through 38-11 using AutoCAD's isometric capability. Estimate their sizes.
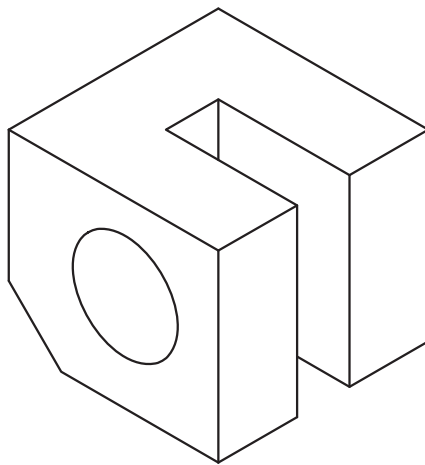
Fig. 38-9
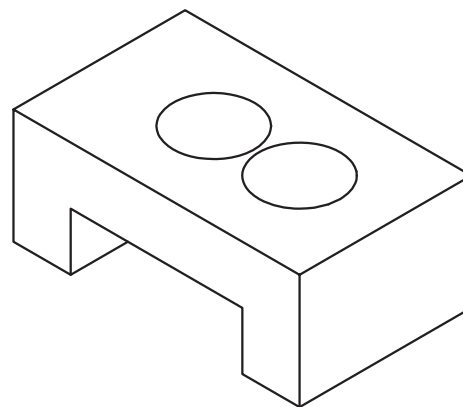
Fig. 38-10
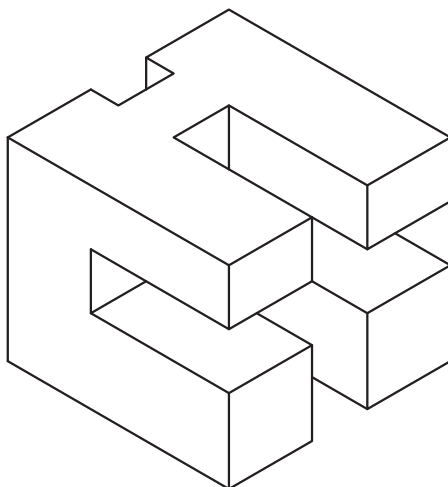
Fig. 38-11

Review & Activities Chapter 38

4. Draw the isometric solid according to the dimensions shown in Fig. 38-12A. Center the hole in the top surface of the part. Specify a diameter of **10.61**. Your finished drawing should look like the one in Fig. 38-12B.

A   B
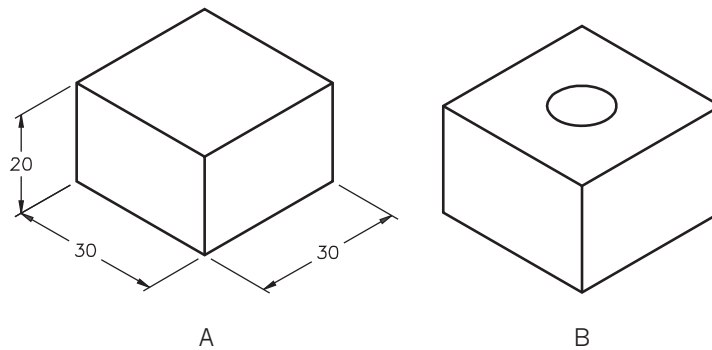
Fig. 38-12

5. Create the cut-away view of the wall shown in Fig. 38-13.
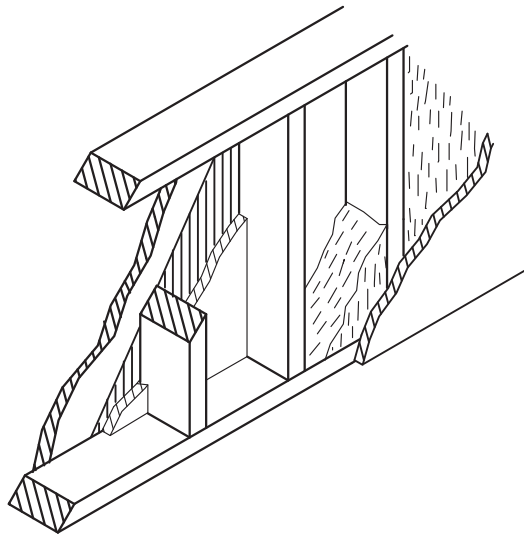
Fig. 38-13

Courtesy of Autodesk, Inc.

Continued

Chapter 38 **Review & Activities**

6. A drawing of a guide block is shown in Fig. 38-14. Draw a full-scale isometric view using the **SNAP** and **ELLIPSE** commands. An appropriately spaced grid and snap will facilitate the drawing.
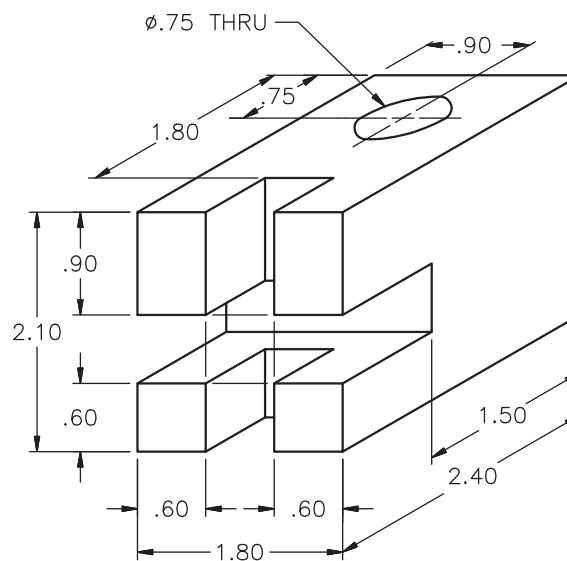
Fig. 38-14

Courtesy of Gary J. Hordemann, Gonzaga University
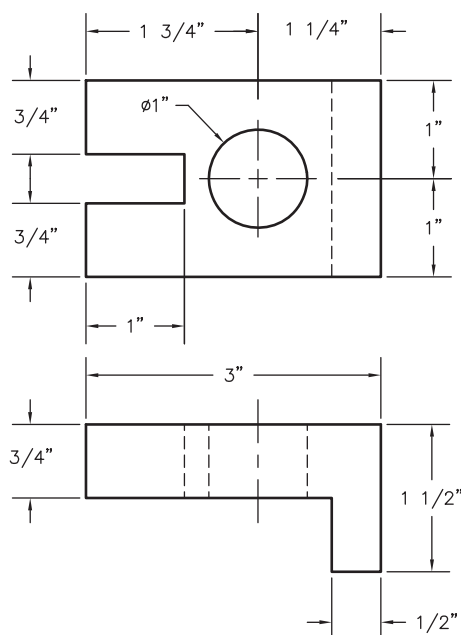
Fig. 38-15

7. Accurately draw an isometric representation of the orthographic views shown in Fig. 38-15. Draw and dimension the isometric according to the dimensions provided.

Review & Activities Chapter 38

8. An exploded, sectioned orthographic assembly is shown in Fig. 38-16. Draw the assembly as a full-scale exploded isometric assembly. Use isometric ellipses to represent the threads, as shown in the isometric pictorial.
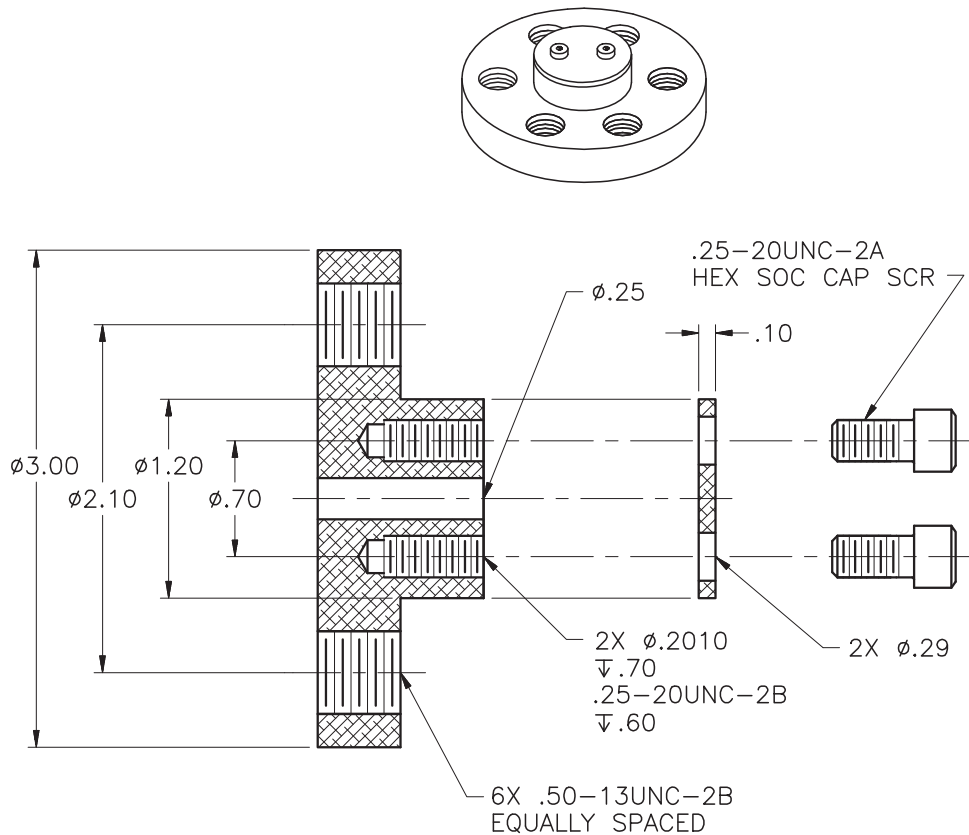
Fig. 38-16

Courtesy of Gary J. Hordemann, Gonzaga University

Continued

Chapter 38 **Review & Activities**

● USING PROBLEM-SOLVING SKILLS

Complete the following problems using problem-solving skills and your knowledge of AutoCAD.

1. The design engineer for the quality assurance division of your company asked you for an isometric drawing of the dovetail block shown in Fig. 38-17, which is used in a milling machine setup. Draw the block, dimension it, and save it as **ch38dove.dwg**.

6.0

3.8

12.0

45°

2.3
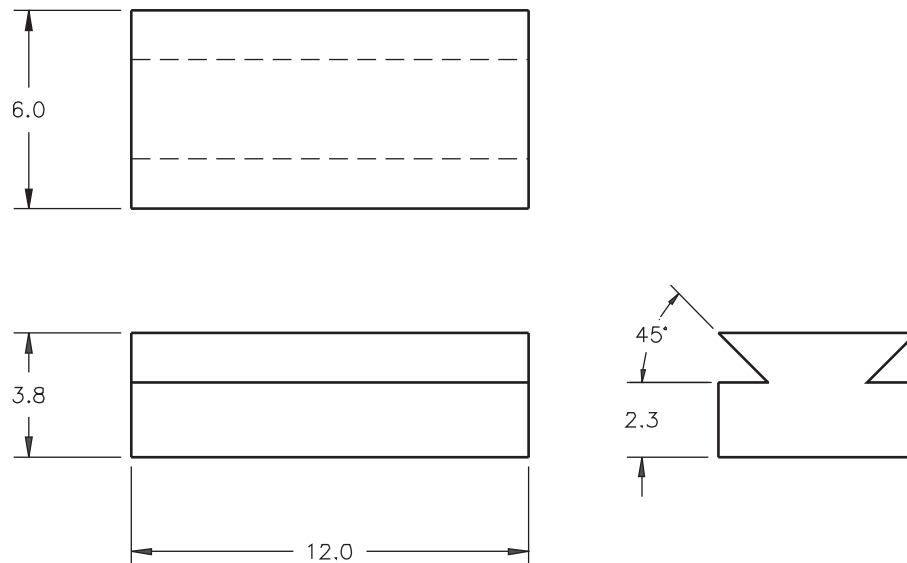
Fig. 38-17

Review & Activities Chapter 38

2. The engineer also needs an isometric drawing of the slide shown in Fig. 38-18 for the same milling machine setup. This one, however, does not need to be dimensioned. Save the drawing as **ch38slide.dwg**.
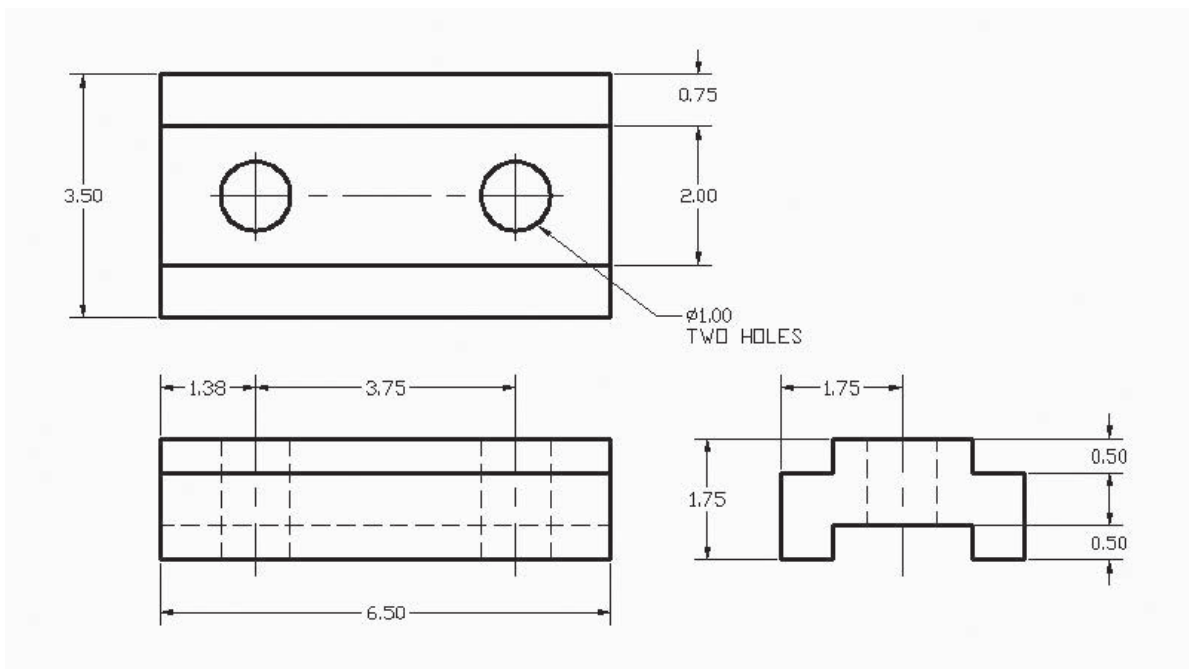
Fig. 38-18

AutoCAD at Work

3D Civil Designer

Civil design and drafting is a highly varied occupation. Civil designers work on highways, harbor facilities, railroads, airports, and even city plans and site developments. The actual creation of documents such as plats, operations maps, and contour maps, however, is often a highly time-consuming and sometimes frustrating endeavor. A change in one view or type of document necessitates changes in multiple other documents.

© PhotoLinkGetty Images/RF

With today's 3D civil software, however, designers can use engineering models directly to produce documents, annotations, and tables that update automatically when the engineering model changes. The software can even enforce drafting standards and company styles. Never again will a designer or drafter have to make change after change manually to civil documents. Model-driven documents are always up-to-date.

Advantages of 3D Civil Software

The most obvious result of using 3D modeling software to keep design and drafting documents up-to-date automatically is a huge time savings. Many companies are finding that their land development projects can be done in 25% less time than projects based on simpler CAD products that do not interact with the engineering model. The use of 3D civil software can literally shave years off projects compared to traditional, manual drafting methods.

Education

A 3D civil designer or drafter needs the same basic education needed by all civil drafters. In addition, a 3D designer needs to be comfortable working with and extracting data from 3D models prepared by engineers. A solid background in science and math is also helpful.

▶ Career Activities

1. If possible, arrange to see the plat for your town or city. How was it created? Explain how it could benefit from 3D civil CAD software.

2. Research civil design and drafting jobs in your area. What is the job outlook?

Chapter **39**

The Third Dimension

Objectives

- Create a basic 3D model
- Apply several 3D viewing options
- View and control a model interactively in 3D space
- Become familiar with the **3D Modeling** workspace
- Learn to use the **SteeringWheel** and **ViewCube** tools

Vocabulary

arcball
camera
clipping plane
compass
extrusion thickness
ground plane
hidden line removal
parallel projection
perspective projection
roll
Z axis

Three-dimensional (3D) drawing and modeling is becoming increasingly popular due, in large part, to improvements in hardware and software. AutoCAD's 3D capabilities allow you to produce realistic views of car instrument panels, snow skis, office buildings, and countless other products and structures. A key benefit of creating 3D drawings is to communicate better a proposed design to people and groups inside and outside a company. Also, the availability of 3D data speeds many downstream processes such as engineering analysis, digital and physical prototyping, machining, and

3D View   Plan View
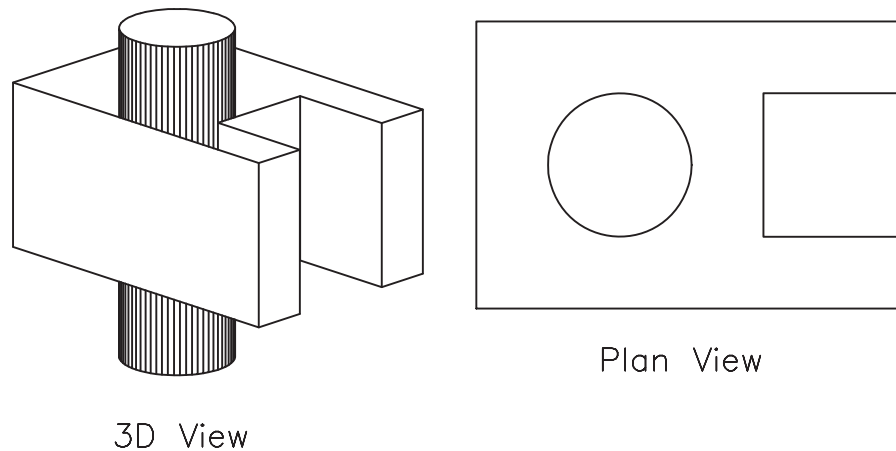
Fig. 39-1

mold-making. In the field of architecture, 3D modeling can help companies persuade customers to move ahead with a proposed construction project such as a shopping mall.

This chapter introduces AutoCAD's 3D capabilities. You will create a simple 3D model, learn to view it from any point in space, use many different viewing options, and learn the different visual styles available for 3D objects in AutoCAD.

An example of a model generated in AutoCAD is shown on the left side of Fig. 39-1. The model on the right is the same object, viewed from the top. The model consists of a section of steel rod and a sheet-metal shroud that fasten into an aircraft fuel control system. The model shows how the shroud protects a wiring harness that runs through the square notch in the shroud next to the rod. The shroud protects the wires from the heat of the rod. This prevents the plastic insulation from melting and causing an electrical short.

Creating a Basic 3D Model

To work in three dimensions in AutoCAD, you will use a third axis on the Cartesian coordinate system. This axis, called the **Z axis**, shows the depth of an object, adding the third dimension. (In this context, the X axis shows the width of the object, and the Y axis shows the length or height.) The Z axis runs through the intersection of the X and Y axes at right angles to both of them.

Let's draw a steel rod and sheet-metal shroud similar to those shown in Fig. 39-1.

1. Start AutoCAD and create a new drawing using the following information.

   - Use the **Quick Setup** wizard.
   - Set the units to **Decimal**.
   - Set the area to **12″ × 9″**.
   - Select the **2D Drafting & Annotation** workspace
   - Display the **Menu Bar** by selecting the flyout arrow on the **Quick Access** toolbar and selecting **Show Menu Bar.**
   - **ZOOM All**.

2. Create a **1″** grid and a **.5″** snap.

3. Create a new layer named **Objects**, assign the color magenta to it, and make it the current layer.

4. Enter the **ELEV** command and set the new default elevation at **1″** and the new default thickness at **3″**.

An elevation of **1″** means the base of the object will be located 1″ above a plane of 0 on the Z axis. The thickness of **3″** means the object will have a thickness of 3″ in the Z direction. This is called the **extrusion thickness**.

5. Draw the top (plan) view of the sheet-metal shroud using the **LINE** command. Construct it as shown on the right in Fig. 39-1. Omit the rod at this time. The width and length of the shroud are **4″ × 6″**, respectively. The square notch measures **2″ × 2″**.

6. Save your work in a file named **3d.dwg**.

Viewing and Hiding

Let's view the object in 3D.

1. Select the **View** tab of the **Ribbon** and focus on the **Views** panel.

2. Pick the flyout arrow on the **Views** panel, and select **SE Isometric** from the flyout menu.

A 3D model of the object appears on the screen.
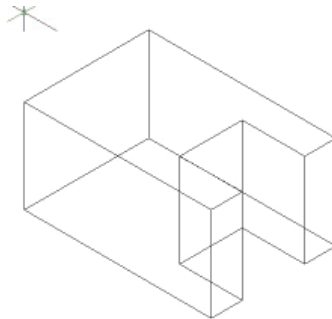
Fig. 39-2

3. Enter **Z** (for **ZOOM**) and **.9x**.

The object should look somewhat like the one in Fig. 39-2.

The **HIDE** command removes edges that would be hidden if you were viewing a real, solid 3D object. Hiding these edges produces a more realistic view of the object. This is called **hidden line removal**.

4. From the **View** menu on the **Menu Bar**, select **Hide**, or enter the **HIDE** command at the keyboard.

The object should now look similar to the one in Fig. 39-3.

5. Return to the plan view of the object by selecting **Top** from the flyout menu on the **Views** panel of the **Ribbon**.

6. **ZOOM All**.
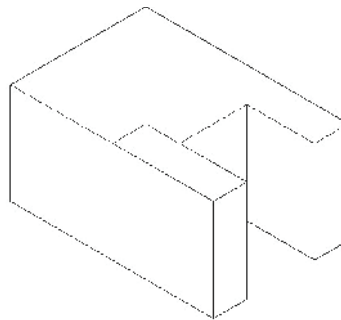
Fig. 39-3
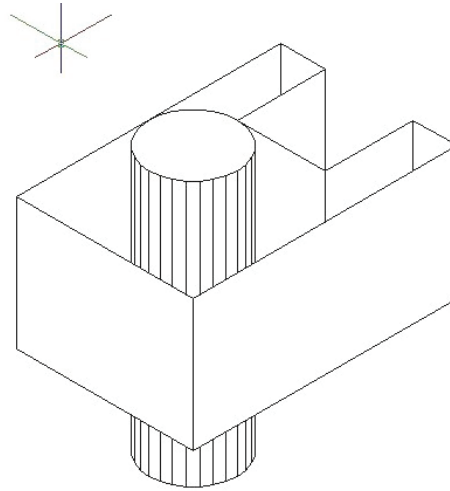
Fig. 39-4

Changing Elevation and Thickness

Let's add the steel rod to the 3D model at a new elevation and thickness.

1. Enter the **ELEV** command and set the elevation at **–1**″ and the thickness at **6**″.

2. Draw a **2**″-diameter circle in the center of the model, as shown in Fig. 39-1 on page 514.

3. On the **Views** panel on the **View** tab of the **Ribbon**, select **SW Isometric** from the flyout menu.

4. Remove the hidden lines by entering the **HIDE** command.

Your drawing should now look similar to the one in Fig. 39-4.

5. Save your work.

Viewing Options

AutoCAD offers several options for viewing models in 3D space. One method you've already used is the **Views** panel on the **Ribbon**. Another is **3D Orbit**, an option that is both visual and easy to use. The two newest viewing tools are the **SteeringWheel** and the **ViewCube**. The method you choose should be based on your personal preference. One method is only better than another if it's faster for you and if it accomplishes what you are trying to achieve.

3D Orbit

3D Orbit is a tool for manipulating the view of 3D models by clicking and dragging. It's especially useful because you can use it to manipulate 3D models while they are shaded or when hidden lines are removed. **3D Orbit**'s right-click shortcut menu offers options for removing hidden lines, rotating objects, panning, zooming, shading, perspective viewing, and clipping planes. You will explore these options in the following steps.

1. In the **Navigate** panel on the **View** tab of the **Ribbon**, pick the arrow next to **Orbit** and select **Free Orbit** from the flyout menu.

Free Orbit

NAVIGATE

This enters the **3DFORBIT** command and activates **3D Free Orbit**. When you enter **3D Free Orbit**, AutoCAD displays an arcball and the shaded 3D user coordinate system. The **arcball** is the green circle divided into four quadrants by smaller circles. The UCS icon is the red, green, and blue icon made up of the X, Y, and Z axes. The icon serves as a visual aid to help you understand the direction from which you are viewing the model. You will learn more about user coordinate systems later in this chapter.

The XY plane—the 2D plane on which all Z values are zero—is called the **ground plane**. The ground plane is the same plane on which you have created drawings throughout this book, but now you can view them in three dimensions.

When the **3DFORBIT** command is active, the target of the view stays stationary and the point of view (also called the **camera**) moves around the target. The target point is the center of the arcball, not the center of the object(s) you are moving.

2. Position the cursor inside the arcball and click and drag upward to rotate the model.

The model rotates freely in three dimensions around the target. The grid and UCS icon provide visual reference.

3. With the **3DFORBIT** command active, right-click to display a shortcut menu.

The second line of the shortcut menu displays the current **Orbit** mode, which is **Free Orbit**.

4. Rest the cursor over **Other Navigation Modes** to view a cascading menu of options.

The **Constrained Orbit** mode limits the rotation of the model with respect to the Z axis and the grid. It allows rotation around the Z axis, but constrains the rotation around the XY plane to the top or bottom view. In other words, **Constrained Orbit** prevents you from flipping the model over.

Continuous Orbit mode allows you to animate the model in a continuous rotation. You can control the direction and speed of the continuous orbit by moving the cursor.

5. Experiment with the **Constrained Orbit** and **Continuous Orbit** modes on your own.

6. Reselect **Free Orbit** from the shortcut menu.

Rotating Objects in 3D Space

When you move the cursor over different parts of the arcball, the cursor icon changes. When you click and drag, the appearance of the cursor indicates the rotation of the view.

1. Study the cursor; then move it outside the arcball and then back inside it.

When the cursor is inside the arcball, it changes to a small sphere encircled by two lines, as shown in Fig. 39-5A. When this cursor is present, you can manipulate the view freely.

2. With the cursor inside the arcball, click and drag up and down, back and forth.

3. Move the cursor outside the arcball.

The cursor changes to a circular arrow around a small sphere, as shown in Fig. 39-5B.

When you click and drag outside the arcball, AutoCAD moves the view around an axis at the center of the arcball, perpendicular to the screen. This action is called a **roll**.

4. With the cursor outside the arcball, click and drag.

5. Move the cursor over one of the smaller circles on the left or right side of the arcball.

A   B   C   D
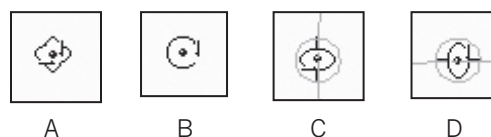
Fig. 39-5

The cursor becomes a horizontal ellipse around a small sphere as shown in Fig. 39-5C. When you click and drag from either of these points, AutoCAD rotates the view around the Y axis that extends through the center of the arcball.

6. With the cursor over one of the smaller circles at the left or right, click and drag to the left and right.

7. Move the cursor over one of the smaller circles on the top or bottom of the arcball.

The cursor becomes a vertical ellipse around a small sphere, as shown in Fig. 39-5D. When you click and drag from either of these two points, AutoCAD rotates the view around the horizontal or X axis that extends through the center of the arcball.

8. With the cursor over one of the smaller circles at the top or bottom, click and drag up and down.

Panning and Zooming

1. With the **3DFORBIT** command active, right-click to display the shortcut menu and select **Other Navigation Modes** and **Pan**.

This causes AutoCAD to change to the pan mode within the **3DFORBIT** command.

2. Click and drag to adjust the view of the model.
3. Right-click and select **Other Navigation Modes** and **Zoom**.
4. Zoom in and out on the model.
5. Display the shortcut menu and select **Reset View**.

Reset View resets the view back to how it was when you first started in a **3D Orbit** mode. The **Preset Views** option displays a list of predefined views such as **Top**, **Front**, and **NW Isometric**.

> **NOTE** Notice that the background turns gray when you select **Reset View**. This background appears by default in the 3D workspace. You will learn more about the 3D workspace later in this chapter.

6. Display the shortcut menu and select **Other Navigation Modes** and **Free Orbit**.

The arcball returns.

7. Press **ESC** or **ENTER** to exit **Free Orbit**.

The arcball disappears and the three-colored UCS icon is replaced with the one-color coordinate system icon. Notice also that the background color in the drawing area is white.

Controlling Visual Aids

AutoCAD provides several visual aids to help orient you in 3D space as you work with objects in **3D Orbit**.

1. Re-enter the **3DFORBIT** command.

2. From the shortcut menu, pick **Visual Aids** and **Compass**.

This displays a 3D sphere within the arcball composed of three ellipses representing the X, Y, and Z axes.

3. Rotate the model and pay attention to the 3D sphere.

4. From the shortcut menu, turn off **Compass**.

5. From the shortcut menu, pick **Visual Aids** and **Grid**.

The grid appears, which represents the 0-elevation ground plane discussed earlier in this chapter. We will leave the grid displayed. The **Visual Aids** shortcut menu also controls the display of the UCS icon.

6. Rotate the model and pay attention to the movement of the ground plane.

7. Press **ESC** or **ENTER** to exit the **Free Orbit** mode.

The SteeringWheel

1. Pick the **SteeringWheels** button on the status bar.

STATUS BAR

This enters the **NAVSWHEEL** command and attaches a large, multi-button icon to the pointer called the **Full Navigation Wheel**, as shown in Fig. 39-6.

2. If the wheel on your screen does not match Fig. 39-6, right-click and select **Full Navigation Wheel** from the shortcut menu.

Study the buttons on the wheel. Several of the buttons should be familiar to you, such as **Zoom, Pan,** and **Orbit.** Notice also the close button and the down-facing arrow button, which displays the shortcut pull-down menu.
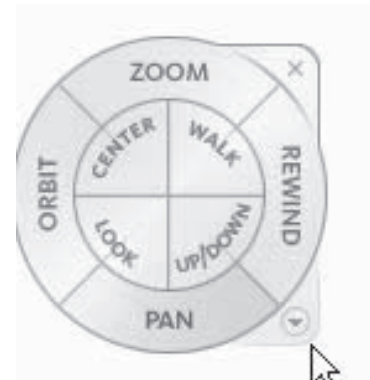
3. Click and drag the **Zoom** button on the **Full Navigation Wheel** to use the **Zoom Tool.**

4. Experiment with the **Pan** and **Orbit** buttons.

Fig. 39-6

Notice the green **Pivot** icon that appears on the screen while you are using the **Zoom Tool** and the **Orbit Tool.** This is the center point of the orbit and zoom actions.

5. Pick the **Center** button on the **Full Navigation Wheel** and drag the wheel to the lower left corner of the sheet-metal shroud. When the green Pivot icon appears, release the mouse button.

6. Pick the **Orbit** button, and rotate the drawing.

Notice that the pivot of the orbit has now changed to the center point you selected.

7. Click once on the **Rewind** button.

AutoCAD takes you to the previous view.

8. Click on the **Rewind** button and hold the mouse button down.

The **Rewind Tool** appears, along with a row of thumbnails that represent all the previous views.

9. Drag the cursor across the row of thumbnails and watch the view on the screen change. Release the mouse button on a previous view of your choice.

10. Pick the **Wheel menu** button in the lower right corner of the **Full Navigation Wheel** to display the wheel's shortcut menu.

Notice the different options for mini wheels. The SteeringWheel contains commands for building a tour, which allows you to "walk" or "fly" through a model. You will learn more about the **Walkthrough** and **Flythrough** commands in Chapter 47.

11. Experiment on your own with the shortcut menu. Select each of the mini wheels, view the **SteeringWheel Settings...** dialog box, select **Go Home,** then close the wheel**.**

Using the 3D Modeling Workspace

Now that you are working in three dimensions, it is time to revisit the **3D Modeling** workspace.

1. Pick the **Workspace Switching** button on the status bar, select the **3D Modeling** workspace, and close any open palettes and floating toolbars.

Focus your attention on the **View** panel of the **Home** tab on the **Ribbon.**

2. Pick the down arrow to the right of the text box to display the Visual Style options.

AutoCAD displays graphic representations of the five default visual styles, including the familiar **2D Wireframe** style, as shown in Fig. 39-7.

3. Pick the **3D Wireframe** style and observe the changes in the drawing area.

4. View the model in the **3D Hidden, Realistic,** and **Conceptual** visual styles.

5. Pick the down arrow to the right of the **3D Navigation** text box and pick the **Top** view.
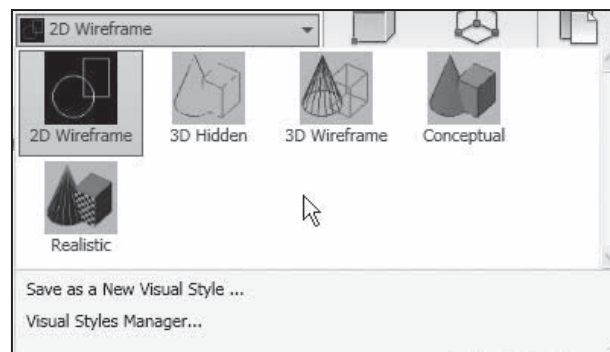
Fig. 39-7

AutoCAD displays the top view of the model.

6. Pick the down arrow again, and pick the **NW Isometric**, **Back**, and **Northeast Isometric** views.

7. Select the **View** tab of the **Ribbon**.

Notice the **Navigate** panel in the **3D Modeling** workspace includes the **Steering Wheels** button. Now focus your attention on the **View** panel.

8. Pick the **Previous View** button on the **View** panel twice.

VIEW

The active view is now **NW Isometric**. As you can see, you can pick the **Previous View** button repeatedly to step back through views of the object.

9. Pick the large **Views** button. As you can see, the same list of predefined views is available on this panel as well.

VIEW

Clipping Planes

A **clipping plane** is a plane that slices through one or more 3D objects, causing part of the object on one side of the plane to be omitted. It is useful in viewing the interior of a part or assembly. You can use clipping planes to achieve views of an object that are similar to sectional views.

1. Enter the **3DCLIP** command at the keyboard.

The **Adjust Clipping Planes** window appears.

2. In the **Adjust Clipping Planes** window, depress the **Front Clipping On/Off** button. (It may already be depressed.)

3. With the cursor inside this window, click and drag up and down.

This moves the clipping plane up and down and provides a corresponding view of the model in the drawing area.

4. In the **Adjust Clipping Planes** window, depress the **Back Clipping On/Off** button, as well as the **Adjust Back Clipping** button.

5. With the cursor inside this window, click and slowly drag up and down to adjust the location of the back clipping plane.

6. Pick the **Create Slice** button, click, and move the cursor up and down.

With the front and back clipping planes on, you can adjust their location at the same time.

7. Depress the buttons to turn **Front Clipping** and **Back Clipping** off.

8. Close the **Adjust Clipping Planes** window.

The ViewCube

1. Pick the **2D Wireframe** visual style and **SW Isometric** view from the **View** panel of the **Home** tab of the **Ribbon.**

The visual style you have used thus far in this textbook is **2D Wireframe.** The other four visual styles are 3D visual styles.

2. Pick the **3D Wireframe** visual style.

Notice the changes in the drawing area. The background color is now gray. The three-colored icon is present, and the ground plane grid is displayed. These are the visual styles for all 3D visual styles.

Focus your attention on the **ViewCube** in the upper right corner of the drawing area, as shown in Fig. 39-8. The **ViewCube** appears in all 3D visual styles, although you can toggle it off with the **ViewCube** toggle button on the **Views** panel.

3. Pick the **FRONT** face of the **ViewCube** and observe the view change in the drawing area.

4. Pick the arrow above the cube to change to the **TOP** view.

5. Pick **W** (for west) on the **ViewCube** compass to change to the **LEFT** view.

6. Pick the rotation arrows in the upper right corner of the **ViewCube** and observe the result in the drawing area.

7. Pick the corners and edges of the **ViewCube** and observe the results in the drawing area.

8. Pick the **Home** button in the upper left corner of the **ViewCube.**

As you can see, the ViewCube is a fast, powerful option for viewing your model in three dimensions.
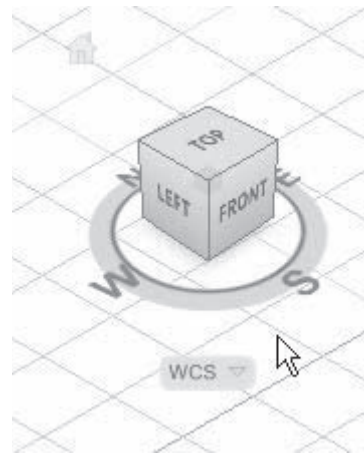
Fig. 39-8

Parallel and Perspective Views

Consider the buildings shown in Fig. 39-9. Both are shown from the same point in space. However, the one on the left is shown in **parallel projection.** The lines are parallel and do not converge toward one or more vanishing points. The building on the right is a **perspective projection.** Parts of the building that are farther away appear smaller, the same as in a real-life photograph.

1. Right-click on the **ViewCube** and pick **Parallel** from the shortcut menu.

Notice the lines in the model are parallel.

Author: Button change correct?

2. Right-click on the **ViewCube** and pick **Perspective** from the shortcut menu.

Notice the model looks more realistic, with the lines in the model converging toward a vanishing point.

⟳ Free Orbit ▾

VIEW

3. Pick the black triangle next to the **Orbit** button on the **Navigate** panel to display the **Free Orbit** button, and rotate the model.

As you rotate the model, notice the subtle color difference between the "sky" and the "ground." When perspective projection is on, AutoCAD provides one shade for the ground plane and another for the space above the ground plane. The boundary between the two shades represents the horizon line.

4. Press **ESC** to exit **3D Orbit.**

5. Save your work and exit AutoCAD.

Fig. 39-9

Chapter 39 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. The extrusion thickness of an object is specified with which command?
2. Briefly explain the process of creating objects (within the same model) at different elevations and thicknesses.
3. With what command are hidden lines removed from 3D objects?
4. Why is the **3D Free Orbit** feature especially useful, compared to other 3D viewing options?
5. Briefly explain how the **3D Modeling** workspace and the **Ribbon** can be useful for 3D modeling.
6. What is the difference between parallel projection and perspective projection?
7. What is a clipping plane?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Of what benefit are front and back clipping planes? Describe at least one situation in which clipping planes could save drawing time or reveal important information about a design.

Continued

Chapter 39 **Review & Activities**

2. Enter the *x*, *y*, and *z* coordinate values for the alphabetically identified points in Fig. 39-10. To get you started, point **A** has coordinate values of **0,0,0**.
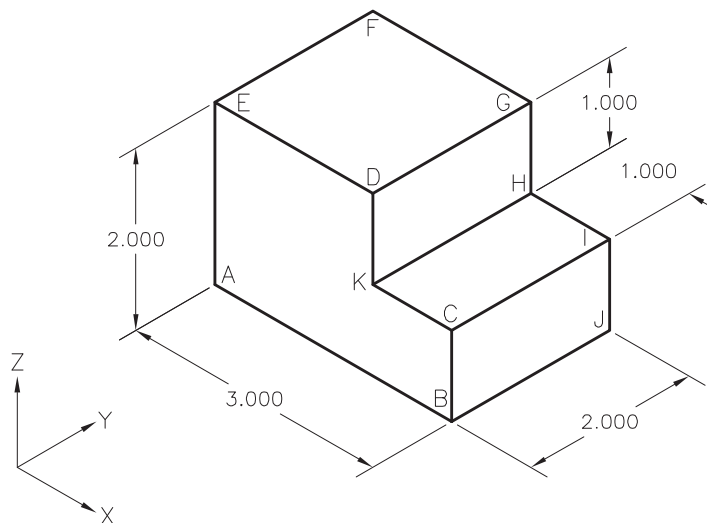
Fig. 39-10

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter. Draw the following objects and generate a 3D model of each.

1. Set the elevation at **1″** to create a 3D model of the guide shown in Fig. 39-11.
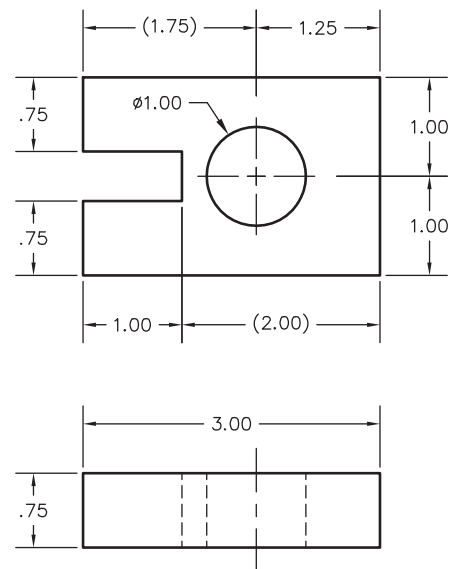
Fig. 39-11

2. To draw the shaft shown in Fig. 39-12, set the elevation for the inner cylinder at **0**. Set the snap resolution before picking the center point of the first circle.

3. Draw the gear blank and hub shown in Fig. 39-13 as a basic 3D model. In the **AutoCAD Classic** workspace, view the model using each of the buttons on the **View** toolbar. Are all the views practical for this model? Which ones are not?
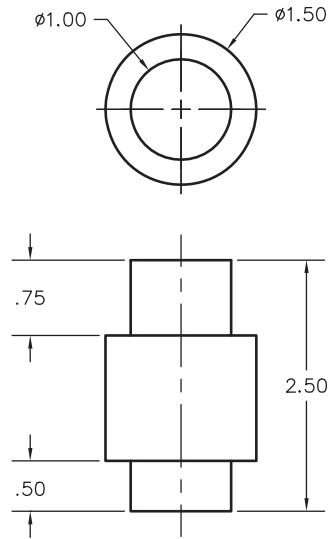
Fig. 39-12
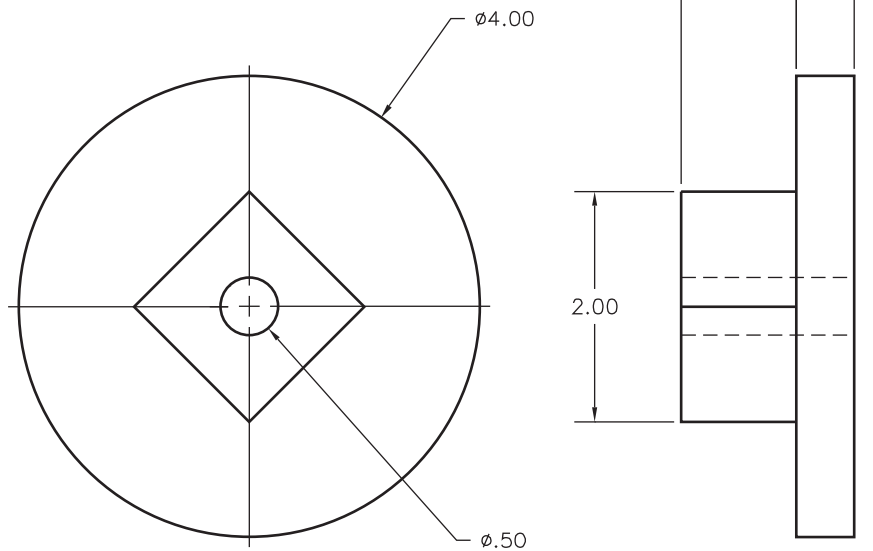
Fig. 39-13

Continued

Chapter 39 **Review & Activities**

• USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. The gear manufacturer for whom you supply the gear blank (see "Applying AutoCAD Skills" problem 3) wants to see the blank in a realistic form. Provide a 3D view of the gear blank in the **Realistic** visual style. Save and plot the drawing. Did it plot the way you expected?

2. Use your knowledge of AutoCAD to create the shaft, gear, and cam shown in Fig. 39-14 as closely as possible.
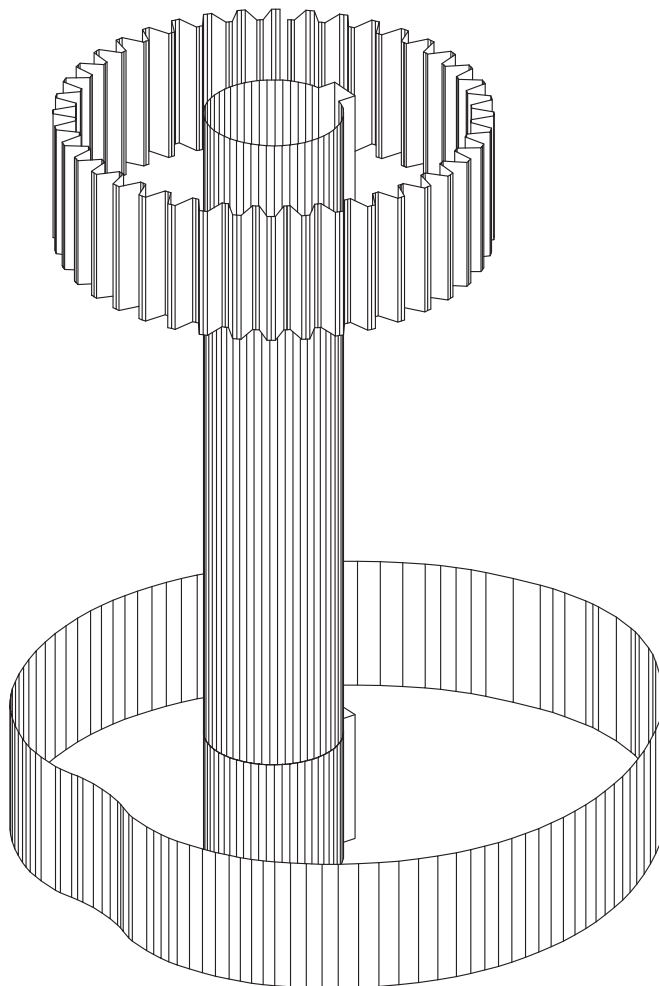
Fig. 39-14

Chapter 40

Drawing Objects in 3D

Objectives

- Draw objects at different Z elevations
- Draw objects using coordinate entry
- Mirror objects in 3D
- Create 3D faces

Vocabulary

elevations
3D faces

In Chapter 39, you were introduced to the third axis of the Cartesian coordinate system, the Z axis. AutoCAD offers a variety of ways to create objects in 3D space. In this chapter, you will use familiar tools and new commands to create objects in three dimensions.

Drawing 2D Objects in 3D Space

The **ELEVATION** command allows you to create objects at different **elevations**, or Z heights.

1. Start AutoCAD and create a new drawing as follows.

- Pick the **Use a Template** option.
- Select the **acad3d.dwt** template.
- Select the **3D Modeling** workspace.
- Enter the **UNITS** command and pick **Feet** as the **Insertion Scale**.
- Right-click on the **ViewCube** and pick **Parallel** projection from the shortcut menu.
- Select the **Top** view and **ZOOM All**.

2. Set snap spacing to **.5** and grid spacing to **1.0**.

3. Create a new layer named **Objects**, assign the color red to it, and make it the current layer. (Enter **LAYER** at the keyboard to open the **Layer Properties Manager** palette.)

4. Draw the base of the storage shed as shown in Fig. 40-1 using the **LINE** command. Place the lower left corner at $x = 1$, $y = 1$. The shed's base is **8′ × 10′**.

As you draw the lines, look at the coordinate display on the status bar. Notice that the Z value of the lines is zero because you are drawing on the ground plane, where $z = 0$.

5. Enter the **ELEV** command and set the new default elevation to **8**. This is the elevation at the top of the shed's walls. For the new default thickness, enter **0**.

6. Move the cursor around in the drawing area and notice the Z coordinate value on the status bar.

As you can see, you are now on a work plane of $z = 8$. The objects you draw now will be 8 feet above the shed's base.

7. Use the **LINE** command to draw the outline of the shed at the top of the walls. The walls are vertical, so the outline will be identical to the outline of the base, **8′ × 10′**. Note that object snap must be off during this operation to avoid snapping to points at the 0 elevation.
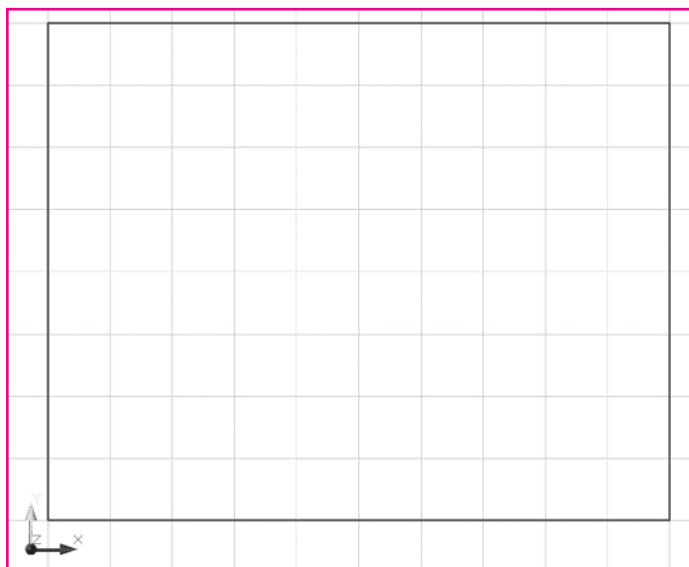
Fig. 40-1
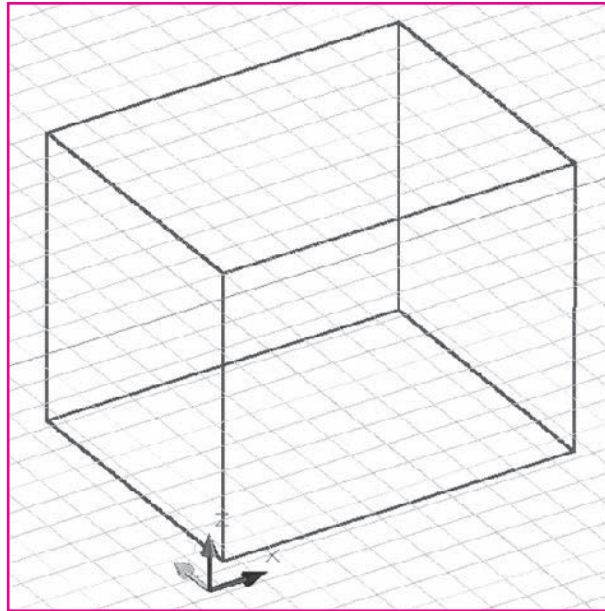
Fig. 40-2

8. Select the **SW Isometric** view from the **Views** panel on the **View** tab. (You may need to turn the grid off to see the base outline clearly.)

9. Use the **LINE** command to draw the four vertical corners of the shed, as shown in Fig. 40-2.

> **HINT** Turn snap off and object snap on. Use the **Orbit** command to rotate the model slightly to make endpoint selection easier.

10. Save your work as **workshed.dwg**.

3D Drawing by Coordinate Entry

Let's add a roof to our storage shed.

1. In the status bar, pick the **Dynamic Input** button to deselect it.

STATUS BAR

This deactivates AutoCAD's dynamic input. Coordinate entry with dynamic input is preset in the **Pointer Input Settings** dialog box and can yield unpredictable results. Refer to Chapter 7 for more information on coordinate entry with dynamic input.

COORDINATES

2. Display the **View** tab of the **Ribbon** and pick the **World** button from the **Coordinates** panel.

Before entering endpoint coordinates for the new roofline, you must be sure that the dynamic UCS feature is turned off and the coordinate system is set to the standard World coordinate system. You will learn more about user coordinate systems in Chapter 41.

3. Enter the **LINE** command.

4. In response to **Specify first point**, type **0,0,8** and press **ENTER**.

5. In response to **Specify next point**, type **12,0,8** and press **ENTER**.

6. Enter **12,5,12** for the third point and **0,5,12** for the fourth point.

7. Type **C** to close the line segments and terminate the **LINE** command.

VIEWS

8. View the shed from the **Front** and **Left**; then use **Previous View** to return to the **SW Isometric** view.

Mirroring Objects in 3D

The **MIRROR** command, like most AutoCAD commands, is not limited to two dimensions. Let's use the **MIRROR** command to create the other half of the roof.

COORDINATES

1. Pick the **World** button on the **Coordinates** panel.

2. Enter the **MIRROR** command, select lines **1**, **2**, and **3**, as shown in Fig. 40-3, and press **ENTER**.

3. For the first and second points of the mirror line, pick the endpoints of line **4**. Do not delete the source object.

VIEWS

4. View the shed from the **Front** and **Left**; then use **Previous View** to return to the **SW Isometric** view.
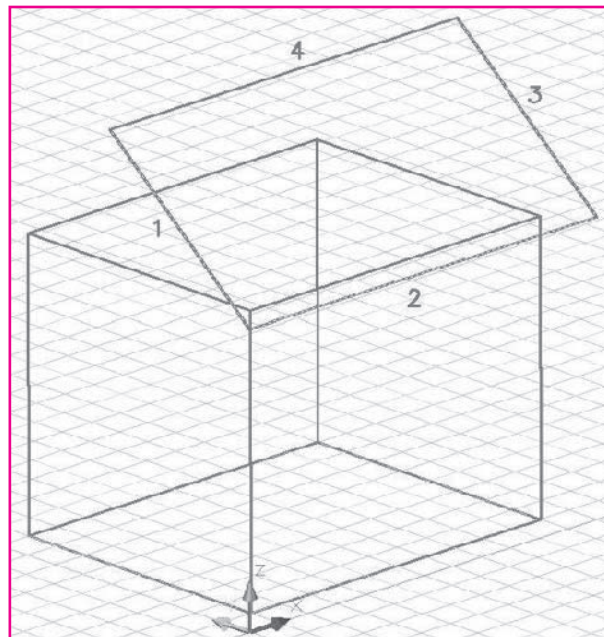
As you can see, you have now created the back half of the roof.

Fig. 40-3

Fig. 40-4

Creating 3D Faces

Thus far, you have created only wireframes in three dimensions. In this context, a wireframe is a representation of an object that uses lines and curves to define its boundaries. The shed you have created should have solid walls and a solid roof, of course. The **3DFACE** command creates **3D faces**, which are three- or four-sided planar meshes, to make the roof and walls appear solid.

1. On the **View** panel of the **Home** tab, select the **Conceptual** visual style.

2. Enter **3DFACE** at the keyboard.

3. Pick the four corner points of one side of the roof in a clockwise or counterclockwise order and press **ENTER**.

You have placed a 3D face on the front section of the roof so that it now appears solid.

4. Enter the **3DFACE** command, select the three points on the left end of the roof, and press **ENTER**.

This creates a face to define the triangular gable end of the roof on the left side of the shed. Your model should look similar to the one in Fig. 40-4.

5. Repeat this process to create 3D faces for the other two roof surfaces.

6. Save your work and exit AutoCAD.

Chapter 40 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Can the **ELEVATION** command be used to create 2D objects on XZ and YZ planes? Explain.
2. In practical terms, what happens when you change the elevation from 0 to 5?
3. What UCS must be active before you create a 3D model by entering 3D coordinates?
4. In your own words, define *wireframe*.
5. Describe a 3D face object.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Briefly explain the difference between the **ELEVATION** command and the **THICKNESS** system variable.
2. As you can tell, entering coordinates for object creation in 3D can be tedious and complex. Without looking ahead in the book, can you think of easier ways to create geometry in three dimensions?

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in the chapter.

1. Open **workshed.dwg** and save it as **prb40-1.dwg**. Create 3D faces for the four walls of the shed. View the shed from several views and with several different visual styles.

Review & Activities Chapter 40

2. Open **prb40-1.dwg** and save it as **prb40-2.dwg**. Create a 3D face for a concrete pad under the shed. The pad measures **18′ × 20′** and extends beyond the shed walls evenly (**5′**) on all sides. When you are finished, the model should look similar to the one in Fig. 40-5.
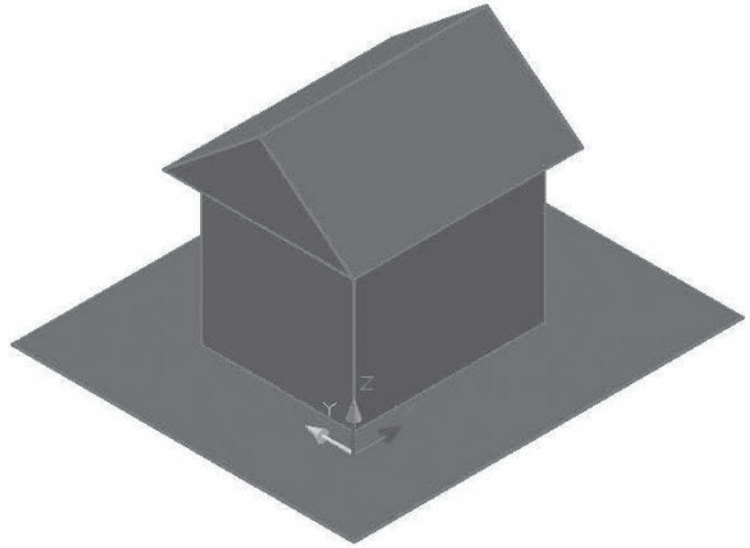
Fig. 40-5

• USING PROBLEM-SOLVING SKILLS

Complete the following problems using problem-solving skills and your knowledge of AutoCAD.

1. Your proposed lamp shade design needs to be drawn in three dimensions before your supervisor will approve the design. Draw a 3D model of the lamp shade shown in Fig. 40-6. Omit the hanger. Create 3D faces where appropriate. Save the drawing as **ch40-lampshade.dwg**.
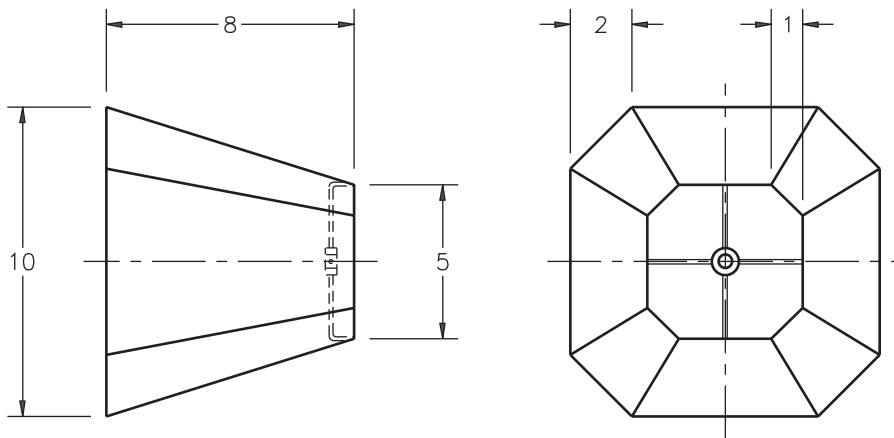
Fig. 40-6

Continued

Chapter 40 **Review & Activities**

2. Create the 3D representation on a globe shown in Fig. 40-7. The center point of the globe is the origin (**0,0,0**). Use the **ELEVATION** command and the data in Table 40-1 to create the series of 2D circles. The center of the circle at each elevation is located at **0,0**. Save the drawing as **ch40-globe.dwg**.
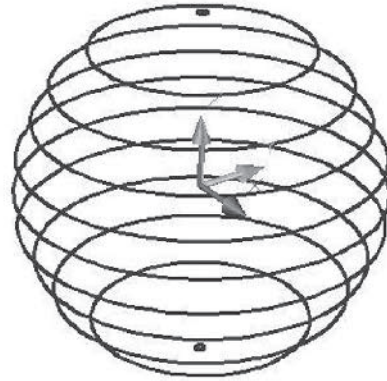
Fig. 40-7

Table 40-1. 2D Circle Data

Elevation (Z Height)	Center of Circle	Diameter of Circle
5.0	0,0	.25
4.0	0,0	6.00
3.0	0,0	8.00
2.0	0,0	9.00
1.0	0,0	9.85
0.0	0,0	10.00
−1.0	0,0	9.85
−2.0	0,0	9.00
−3.0	0,0	8.00
−4.0	0,0	6.00
−5.0	0,0	.25

AutoCAD at Work

Engineer

Most jobs provide opportunities for workers to fine-tune their skills. Young employees often start out in entry-level positions and, by showing their employers that they are hardworking, loyal, and motivated individuals, they are able to advance their status. Some workers move around within one company while others may work for several employers, holding jobs at different levels. Both of these career paths provide a learning platform for employees interested in advancing their status and earnings.

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Growing on the Job

Drafters have a variety of opportunities to move upward in the workplace. Those who develop a particular area of interest, such as mechanical, electrical, architectural, or aeronautical, may eventually wish to work as an engineer. Enrolling in college programs and receiving a degree in a concentrated area is a requirement for any drafter interested in advancing to an engineering position.

Thinking About It

Engineers use their expertise to assist drafters, builders, and maintenance crews. They provide detailed working instructions and drawings for others involved in the production process, and they work closely with drafters to design systems and products. Engineers are problem-solvers; they are able to think clearly and logically in order to derive possible solutions to both existing and potential problems. The ability to work with others is also an important quality. Engineers often find themselves acting as project managers. Being able to prioritize and organize tasks helps them maintain an efficient working team.

▶ Career Activities

1. Write a report about the various types of engineers and their typical duties.

2. Are employment forecasts encouraging for engineers?

Chapter 41

User Coordinate Systems

Objectives

- Create user coordinate systems
- Move among UCSs as necessary to work on a 3D Model
- Create, name, and save UCSs

Vocabulary

UCS icon
user coordinate systems (UCSs)

AutoCAD's default coordinate system is the world coordinate system (WCS). In this system, the X axis is horizontal on the screen, the Y axis is vertical, and the Z axis is perpendicular to the XY plane (the plane defined by the X and Y axes).

You may create drawings in the world coordinate system, or you may define your own coordinate systems, called **user coordinate systems (UCSs)**. The advantage of a UCS is that its origin is not fixed. You can place it anywhere within the world coordinate system. You can also rotate or tilt the axes of a UCS in relation to the axes of the WCS. Many drawing aids (such as the grid, snap, coordinate display, and polar tracking features) move, rotate, or tilt to match the current UCS. These are useful features when you are creating a 3D model.

The **UCS icon** shows the orientation of the axes of the current coordinate system. Its default location is in the lower left corner of the drawing area. When you are viewing the plan view of the WCS in the 2D Wireframe visual style, the Z axis of the UCS icon is not visible, as shown in Fig. 41-1.
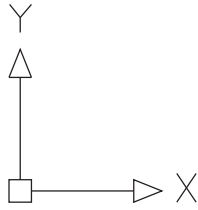
Fig. 41-1

Fig. 41-2

In the 3D visual styles, the UCS icon consists of three arrows with three-dimensional, cone-shaped arrowheads. The X axis is red, the Y axis is green, and the Z axis is blue. The 3D UCS icon is shown in Fig. 41-2.

Creating UCSs

The UCS command enables you to create a new UCS.

1. Start AutoCAD and open **workshed.dwg**. Select the **3D Modeling** workspace.
2. Display the **View** tab on the **Ribbon**.
3. On the **Coordinates** panel, select the **World** button and then the **Origin** button.
4. Use the endpoint object snap to pick the lower left corner of the shed, as shown in Fig. 41-3.

COORDINATES

Notice that the UCS icon has moved to a new location, but it retains the same orientation as the WCS.

5. With snap on, draw a **3′**-diameter circle on the floor of the shed. Place the center of the circle **4′** from the front of the shed and **7′** from the left side of the shed, as shown in Fig. 41-3.

Notice how easy it is to position the circle accurately when the coordinate display and snap are oriented to the new UCS.
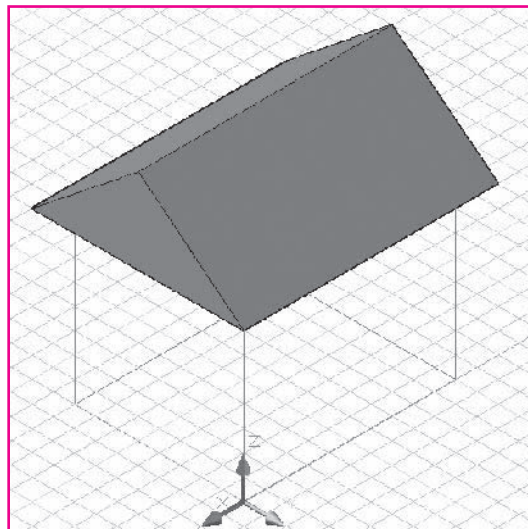
Fig. 41-3

Using Three Points

Next, let's create a door on the front of the shed. To do this, we will create a new UCS on the plane where the door should be. One of the most useful ways to define a new UCS is to select three points that lie on the axes. The first point becomes the origin and the other two lie on the X and Y axes.

COORDINATES

COORDINATES

1. Erase the circle you just created.

2. On the **View** panel of the **Home** tab, pick the **3D Wireframe** visual style.

3. Pick the **World** button on the **Coordinates** panel to reset the current UCS to the world coordinate system and check to be sure dynamic input is off.

4. Pick the **3-Point** button on the **Coordinates** panel.

5. Once again, snap to the lower left corner of the shed for the new origin point.

6. In response to the next prompt, snap to the lower right corner of the shed.

This defines the positive orientation of the X axis.

7. In response to the next prompt, snap to the top of the wall at the front left corner of the shed, directly above the new origin.

Notice that the drawing does not change. However, the crosshairs, grid, and UCS icon shift to reflect the new UCS.

8. Use the **RECTANG** command to create a door. Type **3.5,0** to specify the first corner and **6.5,6.5** to specify the other corner.

As you can see, when you create a UCS oriented to the plane of the front wall of the shed, it greatly simplifies the process of drawing objects there. Now let's add some skylights to the roof.

9. From the **View** panel, switch to the **3D Hidden** visual style.

This will give you a clearer view of the roof surface.

COORDINATES

10. Pick the **3-Point** button on the **Coordinates** panel.

11. For the origin, snap to the front left corner of the roof.

12. Snap to the front right corner of the roof to define the orientation of the X axis.

13. Snap to the roof peak on the left side of the shed to define the orientation of the Y axis.

The new UCS is oriented to the front side of the roof. Notice also that the **ViewCube** is oriented to the new UCS.

14. Draw two **2′** × **3′** skylights on the front roof. Locate the skylights **2′** from the downhill corners of the front roof, as shown in Fig. 41-4.
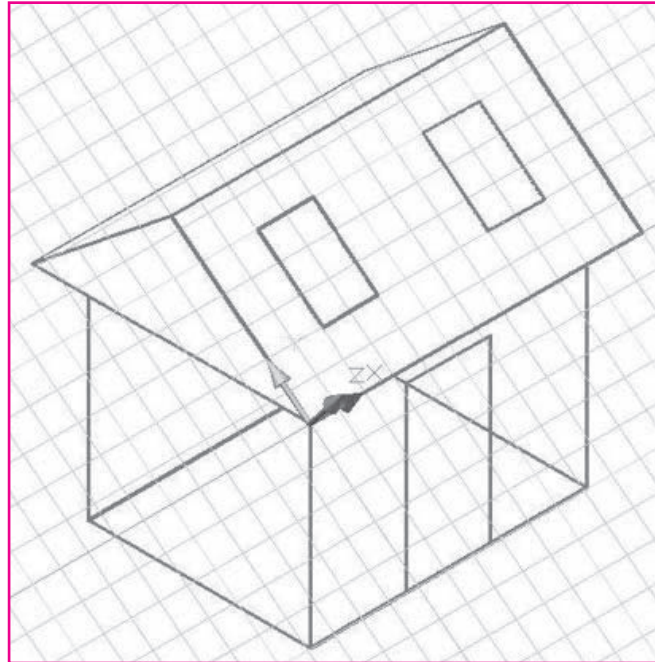
Fig. 41-4

Managing UCSs

The purpose of creating user coordinate systems is to make it easier to work with 3D models. When you have created several UCSs, you need to be able to move from one to another quickly. It is helpful to save UCSs so you can do this without re-creating each UCS.

1. Pick the **Named** button on the **Coordinates** panel.

COORDINATES

The **UCS** dialog box is displayed, as shown in Fig. 41-5 on page 544. In the **Named UCS** tab, the **Current UCS** is listed as **Unnamed**. This is our angled front roof UCS.

2. Select **Unnamed** from the list of named UCSs, type **frontroof**, and press **ENTER**.

3. Select the **World** UCS, pick the **Set Current** button, and pick **OK**.

The world coordinate system becomes the current UCS.

4. Pick the **Named** button on the **Coordinates** panel.

5. Select the **frontroof** UCS, pick the **Set Current** button, and pick **OK.**

COORDINATES

Chapter 41: User Coordinate Systems   **543**
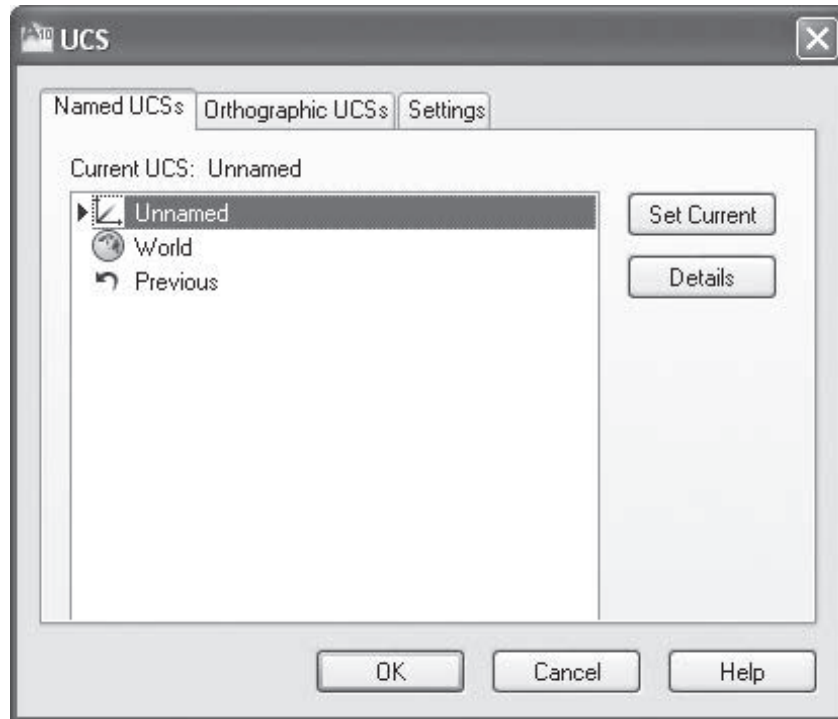
Fig. 41-5

The saved **frontroof** UCS becomes the current UCS again.

6. Pick the **Named** button on the **Coordinates** panel to display the **UCS** dialog box.

Notice the choices of UCSs. In addition to the **World** UCS and the newly created **frontroof** UCS, AutoCAD provides orthographic UCSs oriented to the six viewpoints in three-dimensional space (**Top**, **Bottom**, **Front, Back, Left**, and **Right**). **Previous** makes the previous UCS the current UCS. AutoCAD saves the last ten UCSs, so you can back up with repeated **Previous** selections.

7. Pick **Cancel** to close the dialog box.

Now focus your attention on the other tools on the **Coordinates** panel (Fig. 41-6). The first button, **UCS**, allows you to specify any UCS with keyboard entry. You have already used the **Named**, **World**, and **Origin** buttons. The **X, Y,** and **Z** buttons allow you to rotate the UCS a specified distance around the respective axes. These options are useful when you have selected an origin but wish to change the orientation of the UCS. **Z Axis Vector** allows you to specify the positive direction of the Z axis. The **Show UCS** icon allows you to show or hide the UCS icon.
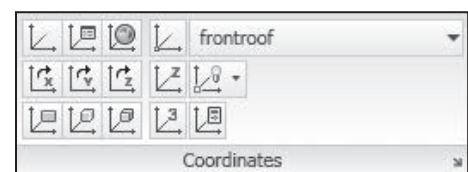
Fig. 41-6

View defines a UCS oriented to the current drawing screen, with the Z axis oriented toward the viewer. This UCS is useful when you want to add text to 3D drawings. **Object** defines a new UCS from a selected planar object, such as a 3D face or a rectangle. The **Face** button aligns the UCS to the selected face of a solid. (You will learn about solids in the next chapter.)

Finally, the **UCS Icon** button allows you to control the size, color, and look of the icon itself.

8. Create and name several new UCSs, then delete them in the **UCS** dialog box.

9. From the **View** panel of the **Home** tab, change the current view to **Left**.

Notice that the UCS icon has changed location and orientation, and that the current UCS is now the **Left** orthographic UCS. When you change views, AutoCAD automatically changes the current UCS to match the view. If you want **frontroof** to remain the current UCS after selecting a new view, you must reselect it with the **Named UCS** button.

> Dynamic UCS is a feature of AutoCAD that moves the UCS automatically. However, dynamic UCS only works on the faces of solid models, 3D objects you have yet to create. You will learn about dynamic UCS in Chapter 45 when you modify solid models.

10. Save your work, and exit AutoCAD.

Chapter 41 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. What is the primary benefit of AutoCAD's user coordinate system?
2. What purpose does the UCS icon serve?
3. Describe the **3 Point** option of the **UCS** command.
4. Briefly describe the **UCS** command's **View** option.
5. In this chapter, you created a UCS for the front roof using the **3-Point** button. Which other option could you have used to create this UCS?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Explain the difference between a view and a UCS and describe how the two can be used together to create complex 3D models.
2. Look back at Chapter 40, in which you created the shed drawing. Now that you know how to create and change UCSs, would you have created the shed differently? Explain.

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Open **workshed.dwg**. Use the **3–Point** option of the **UCS** command to create a UCS on the back wall of the shed. Save the UCS using a name of your choice. With this as the current UCS, add two windows to the back wall. Choose the shape, size, and location of the windows. Save the drawing as **prb41-1.dwg**.
2. Open the **workshed.dwg** and view it in an isometric view of your choice. Create a UCS with the **View** button. Then make a border and title block for the 3D model. Save your work as **prb41-2.dwg**.

Review & Activities Chapter 41

● USING PROBLEM-SOLVING SKILLS

Complete the following problems using problem-solving skills and your knowledge of AutoCAD.

1. Make a 3D drawing of the track shown in Fig. 41-7. Your supervisor wants further design work on the 60° angular face. Create a UCS on the face and save it in preparation for the redesign. Save the drawing as **prb41-track.dwg**.

2. Open **workshed.dwg** and create the gable-end roof vent shown in Fig. 41-8A. First, view the shed from the left. Create a new UCS as shown by snapping to the midpoint of the horizontal line. The vent is six-sided and is located **1.5′** above the bottom of the gable. The size of the vent is your choice. Add an identical vent to the opposite side of the shed. When you are finished, your drawing in the **3D Wireframe** style should look similar to the one in Fig. 41-8B. Save your drawing as **prb41-shed.dwg**.

Fig. 41-7

A   B

Fig. 41-8

Chapter 41: User Coordinate Systems   547
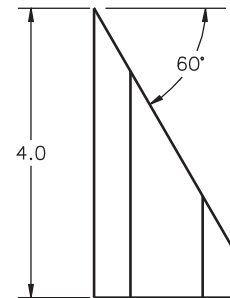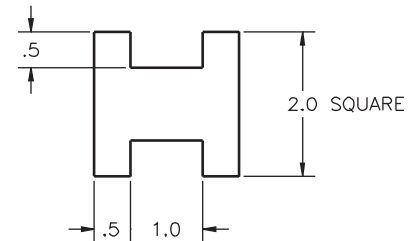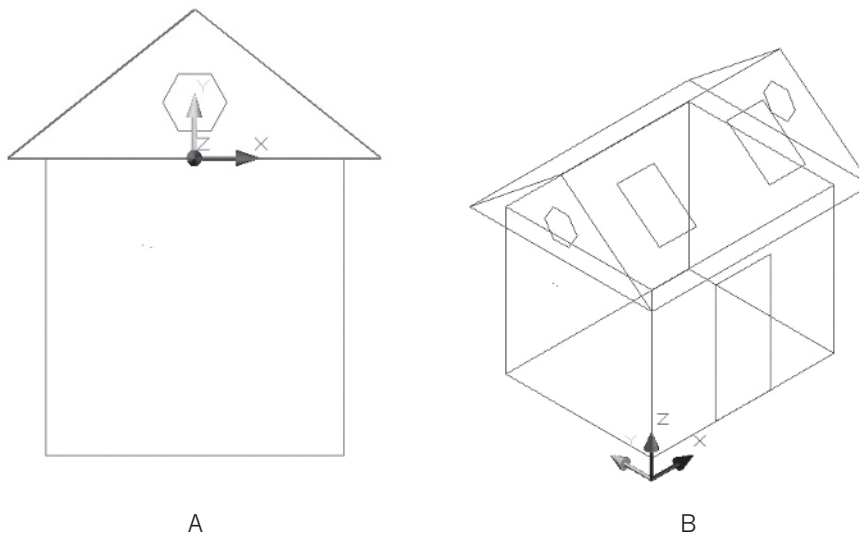
Part 8 Project Applying Chapters 38–41

3D Modeling

Sometimes, to represent an object accurately, you need an alternative to orthographic projections. Two such alternatives are isometric views and 3D models. AutoCAD provides special options to facilitate such drawings.

▶ Description

This project includes two problems:

1. Draw an isometric version of a die (one of two dice), as shown in Fig. P8-1. Include the spots, indicating values of one, two, and three on the three faces showing. Center the spots accurately, and make the length of each side **2″** long. Dimension the length, height, and depth of the die.

2. Create a 3D model of a die the size of a **3″** cube. Draw the spots (numbered one through six) on each of the six faces, symmetrically arranged upon each surface, as shown in Fig. P8-2A and B. Looking at an actual die may help you create an accurate model.

Fig. P8-1

2.00 2.00 2.00

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VIII. 3D Drawing and
Modeling

Part 8 Project: Applying
Chapters 38–41

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555
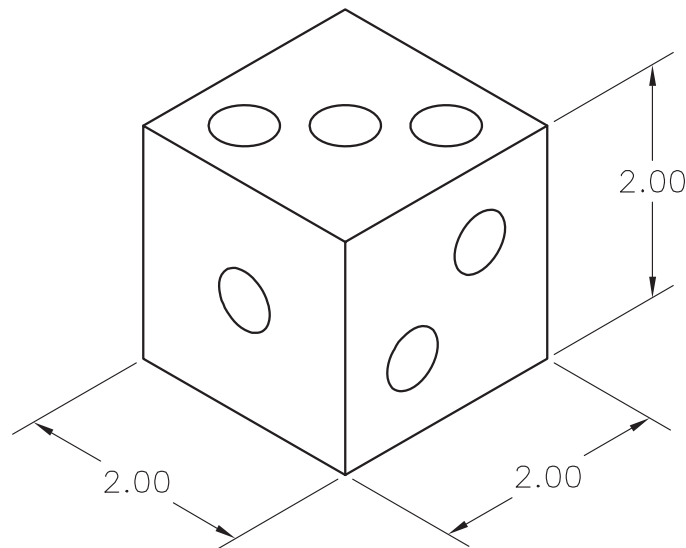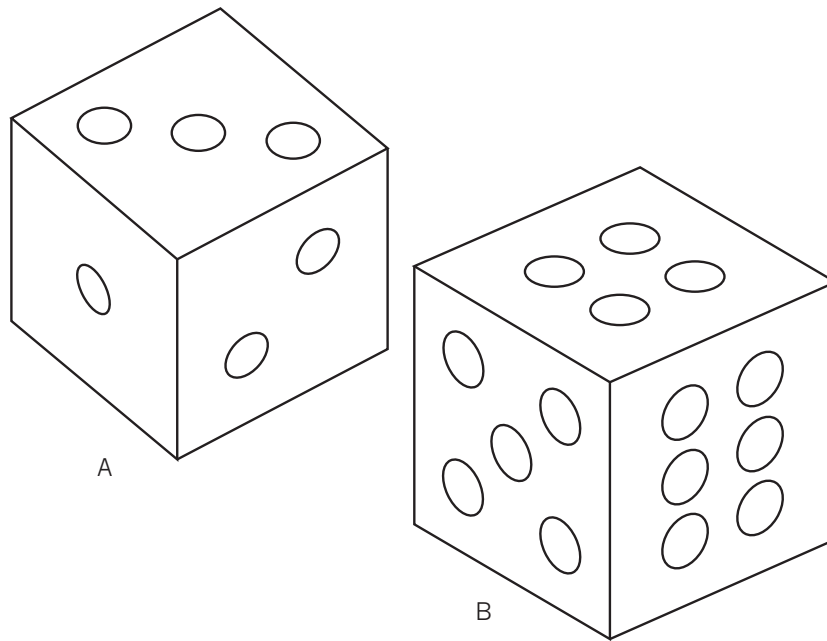
Fig. P8-2

▶ Hints and Suggestions

1. Remember that circles appear elliptical in isometric drawings. Use the **Isocircle** option with the correct plane orientation.
2. Use the **DIVIDE** command to place the spots. Remember to set **PDMODE** to a value that allows you to see the divisions, and use **REGEN** to make the divisions appear.
3. Create two viewports of the 3D model to show all six surfaces. You should only need to create one model.

▶ Summary Questions

Your instructor may direct you to answer these questions orally or in writing. You may wish to compare and exchange ideas with other students to help increase efficiency and productivity.

1. What effect does toggling the isometric plane have upon the use of the ortho feature?
2. Distinguish between thickness and elevation.
3. Of what value is the 3D globe (compass) when you work on 3D models?
4. Describe one possible advantage of reorienting the UCS icon.

Chapter **42**

Primitives

Objectives

- Define solid modeling
- Create solid primitives
- Create mesh primitives

Vocabulary

solid modeling
mesh
polysolid
primitives
tesselations

Computer modeling of three-dimensional volumes of solid parts is called **solid modeling**. This type of modeling has become increasingly important because designs created with solid models more accurately reflect the actual physical properties of the part. The appearance of a solid model is somewhat similar to that of a 3D wireframe with faces, but the two are quite different. Solid models are closed volumes, like a real physical object, and they contain physical and material properties.

Solid **primitives** are the basic building blocks of 3D modeling. Examples of primitives include boxes, spheres, cones, and cylinders. Another type of primitive is the **mesh** primitive. Unlike solid primitives, mesh models are not closed volumes. However, mesh primitives can be modified more easily and more extensively than solid primitives. Mesh models can also be converted to solids at any time.

In this chapter, you will learn how to create solid and mesh primitives by specifying the necessary points and dimensions. In later chapters, you will learn to modify primitives and combine them with other three-dimensional objects to create more complicated shapes.
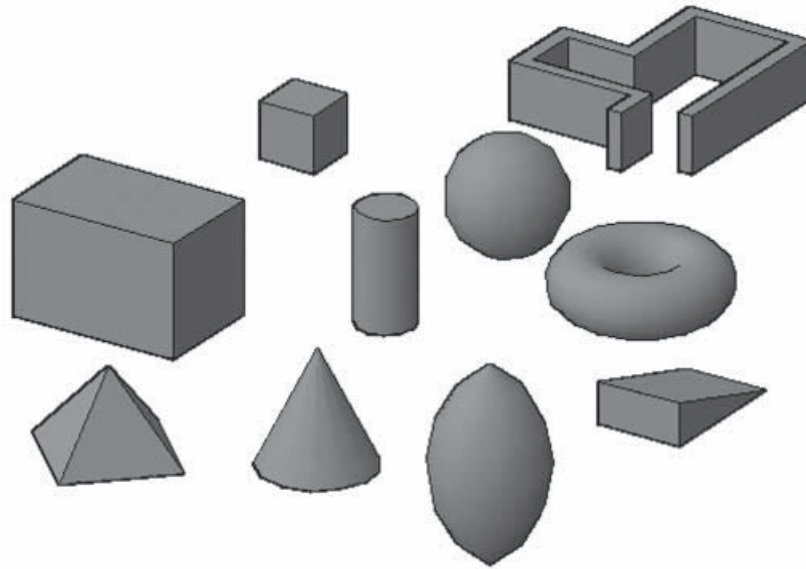
Fig. 42-1

Creating Solid Primitives

The **CYLINDER** command allows you to create a solid cylinder primitive of any size.

1. Start AutoCAD and use the **Acad3d.dwt** template file to create a new drawing.

2. Select the **3D Modeling** workspace and close all open palettes and floating toolbars.

3. Enter **LAYER** at the keyboard and create a new layer named **Objects**, assign the color red to it, and make it the current layer.

4. Set grid to **1** and snap to **.25** and pick **Parallel** projection from the shortcut menu on the **ViewCube.**

5. Save your work in a template file named **solid.dwt**. For the template description, enter **For 3D models,** and pick **OK**.

6. Close **solid.dwt** and use it to create a new drawing file named **primit.dwg**.

7. Select the **3D Hidden** visual style from the **View** panel.

8. Pick the **Cylinder** button from the flyout menu on the **Modeling** panel of the **Ribbon.**

MODELING

This activates the **CYLINDER** command.

9. Pick a point at any location on the drawing area. (Notice that you are working on the ground plane, where $z = 0$.)

10. Enter **.5** for the base radius and **2** for the height of the cylinder.

11. Select the **SW Isometric** viewpoint.

12. Select the **Conceptual** visual style.

The cylinder should appear similar to the one in Fig. 42-2.

The **TORUS** command lets you create donut-shaped solid primitives.

Fig. 42-2

MODELING

13. Zoom out and pan if necessary to make space for the torus.

14. Pick the **Torus** button from the flyout menu on the **Modeling panel.**

This activates the **TORUS** command.

15. Pick a center point for the torus (anywhere) and enter **1** for the radius of the torus.

The radius of the torus specifies the distance from the center of the torus to the center of the tube.

16. Enter **.5** for the radius of the tube.

A torus appears, as shown in Fig. 42-3.
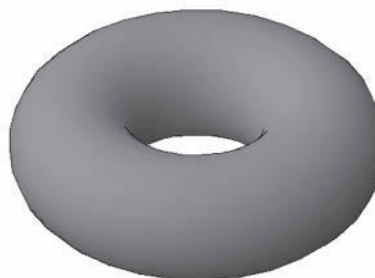
Fig. 42-3

MODELING

If you enter a negative number for the radius of the torus, a football-shaped solid primitive appears, as shown in Fig. 42-4.

17. Enter the **TORUS** command and pick a center point. Enter a radius value of **–1** and a tube radius value of **2**.

A **polysolid** is a solid primitive that is similar to a two-dimensional polyline, but is three-dimensional.

MODELING

18. Pick the **Polysolid** button from the flyout menu **Modeling** panel.

Fig. 42-4

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IX. Solid Modeling

42. Primitives

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559

19. Before picking a starting point, select the **Width** option and type **.25** for the width of the polysolid.

20. Select the **Height** option and type **1**.

21. Pick a point in an open area of the ground plane and move the crosshairs.

The new polysolid begins to take shape.

22. Pick a series of points to create the polysolid. When you are finished, press **ENTER**.

An example of a polysolid is shown in Fig. 42-5. The wall width and height of your polysolid should be similar, but the shape will probably be quite different.
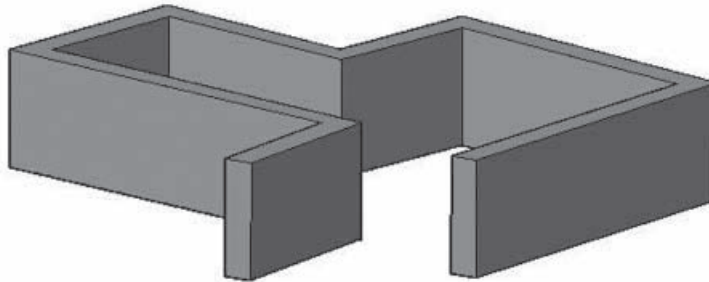
Fig. 42-5

23. On your own, create a cone, a wedge, a box, a sphere, and a pyramid using the solid primitive buttons on the **Modeling** panel.

24. Save your work.

Creating Mesh Primitives

Now we will create several mesh primitives. The **CONE** command allows you to create a cone-shaped mesh primitive.

PRIMITIVES

1. Pick the **Realistic** visual style from the **View** panel of the **Home** tab.

2. Pick the **Mesh Modeling** tab, then pick the **Mesh Cone** button from the flyout menu on the **Primitives** panel.

This activates the **MESH CONE** command.

3. Pick a center point at any location on the ground plane.

4. Enter **1** for the base radius and **2** for the height of the cone.

A cone appears, as shown in Fig. 42-6.

Fig. 42-6

Chapter 42: Primitives   **553**
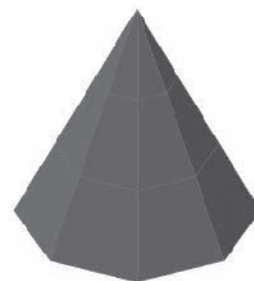
PRIMITIVES

It is possible to create a wedge using the **WEDGE** command.

5. Pick the **Wedge** button on the **Modeling** panel.

6. Pick two corner points to define the size of the wedge's base.

7. Enter **.75** for the height.

A wedge-shaped solid primitive appears, as shown in Fig. 42-7.
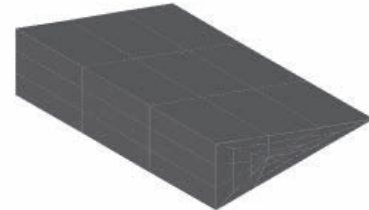
The **BOX** command enables you to create a solid box.

Fig. 42-7

PRIMITIVES

8. From the **Modeling** panel, pick the **Box** button.

NOTE The **Center** option permits you to create a box by specifying a center point.

This activates the **BOX** command.

9. Pick a point to represent one of the corners of the box.

10. Pick the diagonally opposite corner of the base rectangle.

11. Enter **2** for the height.

A box appears, as shown in Fig. 42-8.
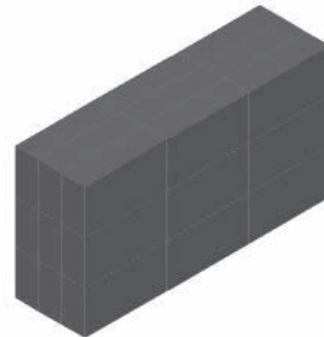
Fig. 42-8

NOTE To create a solid cube, you can select the **BOX** command's **Cube** option after selecting the first point.

PRIMITIVES

The **SPHERE** command creates a solid sphere primitive.

12. Pick the **Sphere** button on the **Modeling** panel.

This activates the **SPHERE** command.

13. Pick a point to define the center of the sphere.

14. Enter **1** for the radius of the sphere.

The sphere appears, as shown in Fig. 42-9.

Fig. 42-9

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IX. Solid Modeling

42. Primitives

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561
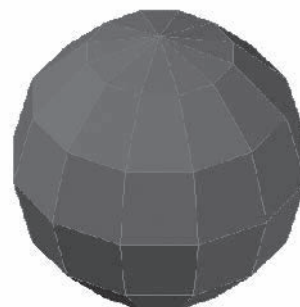
The **PYRAMID** command creates a solid pyramid primitive.

15. Pick the **Pyramid** button on the **Modeling** panel.

16. Pick a point in an open area to define the center of the pyramid's base.

17. Enter **1** for the base radius and **1.5** for the height of the pyramid.
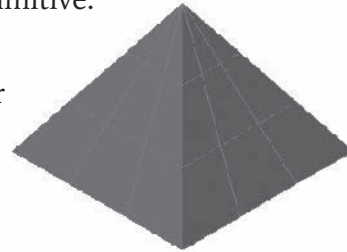
The pyramid appears, as shown in Fig. 42-10.

Mesh Pyramid

PRIMITIVES

Fig. 42-10

> **NOTE**
>
> The **Sides** option specifies the number of sides on the pyramid. The **Edge** option allows you to define the base by specifying an edge instead of the center point.

Controlling Mesh Divisions

Notice the gray lines that subdivide the surfaces on the mesh primitives. These planar subdivisions are called **tesselations**. Tesselations on curved surfaces are a collection of planar facets that approximate the actual curvature. You can control the amount of tesselation in meshes.

1. Pick the arrow in the lower right corner of the **Primitives** panel.

The **Mesh Primitives Options** dialog box appears, as shown in Fig. 42-11.

2. Select each of the mesh primitives, adjust the tesselation divisions, and notice how the model in the **Preview** pane changes. (Make sure the **Auto-update** box is checked.)

Note that increasing tesselation produces smoother models, but also requires more computing power and may affect the speed of your system.

3. Pick **Cancel** to close the dialog box, save your work, and exit AutoCAD.
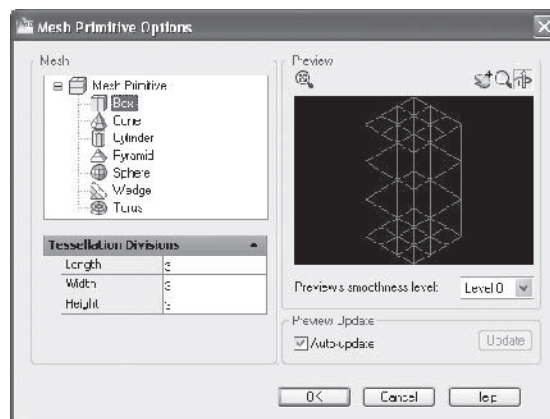
Fig. 42-11

Chapter 42 **Review & Activities**

• REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Describe the function of each of the following commands.
    a. **CYLINDER**
    b. **TORUS**
    c. **CONE**
    d. **WEDGE**
    e. **BOX**
    f. **SPHERE**
    g. **PYRAMID**
    h. **POLYSOLID**
2. With what command can you create a football-shaped primitive?
3. With what command can you create a mesh cube?
4. With what command can you create solid walls?

• CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. If you could slice open a solid primitive, what would you find in the interior? What would you find in the interior of a mesh primitive?
2. Experiment with the **CYLINDER** and **CONE** commands. Is it possible to create a solid primitive cylinder or cone at other than a 90° (right) angle to its base? Explain.

Review & Activities Chapter 42

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create a solid shaft that measures .**5125**″ in diameter by **10**″ in length. Rotate the shaft so that its length is parallel to the WCS.

2. Create a mesh ball bearing that measures .**3625**″ in diameter. Use a polar array to create a ring of 16 balls using a **1**″ radius. View the bearings in an isometric view, and apply the **Realistic** visual style, as shown in Fig. 42-12. Save the drawing as **3dbearing.dwg**.

Fig. 42-12

3. Create an inflated bicycle inner-tube that measures **24**″ in diameter. Make the tube diameter **2**″.

4. Create a football-shaped primitive of any size on top of a **6**″ cube.
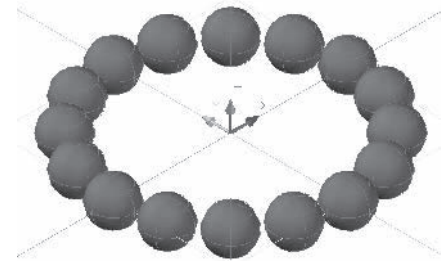
● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Your company's graphics department is designing ornaments for various holidays. They need forms for the art decals. Create the solid primitives for them using the sizes shown in Fig. 42-13. Shade the primitives with the **Conceptual** visual style. Save the drawing as **ornament.dwg**.

2. You need forms around which to design a laptop computer carrying case. Do research to find the most common dimensions for laptops today. Then create two or three mesh boxes to represent the most common sizes. Create a UCS that is parallel to the screen and add text to give the basic computer sizes. Shade the boxes in the **Realistic** style. Save the drawing as **case.dwg**.
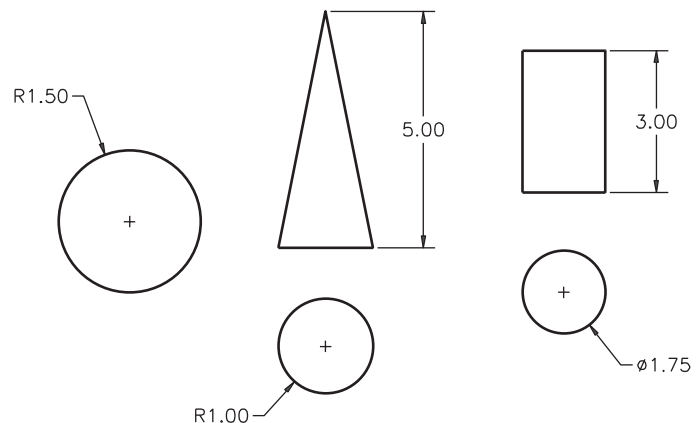
Fig. 42-13

AutoCAD at Work

HVAC Designer

Have you read stories about the days before electricity, when a single fireplace was used to warm an entire household during the coldest months of winter? Fortunately, we live during a time when heating and air-conditioning systems are fairly common-place. Certainly you have a furnace in your home, and almost any store you visit in summertime will have its air conditioning going full blast. HVAC (heating, ventilation, and air conditioning) designers are the people we have to thank for designing the heating and air-conditioning units that make our lives more comfortable.
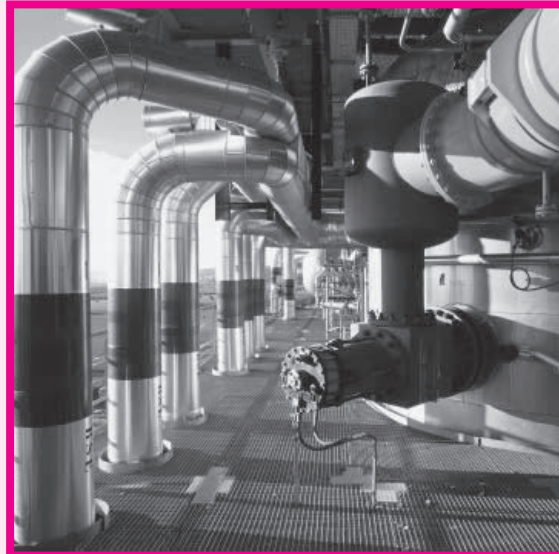
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Combining Skills

People who work with HVAC systems should be familiar with standard codes for heating and cooling systems. Math skills are important to determine heat loss and air flow calculations. HVAC designers may move into positions focusing on heating and drafting analysis and procedures. Computer skills are helpful for all individuals involved in this field, including knowledge of CAD-related programs and word processing programs.

Understanding Efficiency

HVAC designers create plans for heating and cooling systems for residential, commercial, industrial, and institutional uses. The most efficiently designed systems use techniques to improve the quality of air indoors. They can also reduce the amount of energy used to heat or cool a building. This can help businesses and homeowners save money. Energy-efficient systems are also beneficial to the environment.

▶ Career Activities

1. Interview someone who can tell you about employment of HVAC designers in your area. What education is required? Are apprenticeships available?

2. Has employment in this field been steady over the last few years? What are forecasts for future employment?

Basic Modeling

Objectives

- Explain the difference between a solid model and a surface model
- Create models from 2D objects with the **REVOLVE**, **EXTRUDE**, **SWEEP**, and **LOFT** commands
- Create trimmed and planar surfaces with the **PLANESURF** command
- Thicken surfaces into solids

Vocabulary

extrude
helix
lofting
mesh model
revolve
solid model
surface model
sweeping

Many models are produced by first creating a 2D object or objects. For example, it is possible to **revolve** objects (turn them about an axis) to create a 3D model. In this chapter, we will do this. We will also create models by extruding, sweeping, and lofting 2D objects.

The modeling techniques in this chapter produce either a solid model, a mesh model, or a surface model, depending on the defining geometry and the modeling technique used. A **mesh model** is produced from a mesh modeling operation. A **solid model** is produced when a solid modeling operation results in a closed (also called *watertight*) volume. A **surface model** is produced when the operation results in an open 3D surface. By definition, a surface has no thickness and no internal volume.

For example, when a circle is revolved around an axis to form a torus (donut), a solid model or mesh model is created. When a half circle is revolved around an axis, the result is a surface model. Figure 43-1 demonstrates this example.

Fig. 43-1

Revolution

We will use the **REVSURF** command to create a mesh model of the shaft shown in Fig. 43-2.

1. Start AutoCAD and use the **solid.dwt** template file to create a new drawing named **shaft.dwg**.
2. Use a polyline to create the profile of the shaft, as shown in Fig. 43-3. Position point 1 on the ground plane at $x = 1$, $y = 1$. Make sure the polyline is closed. Omit the dimensions.

Fig. 43-2

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IX. Solid Modeling

43. Basic Modeling

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567
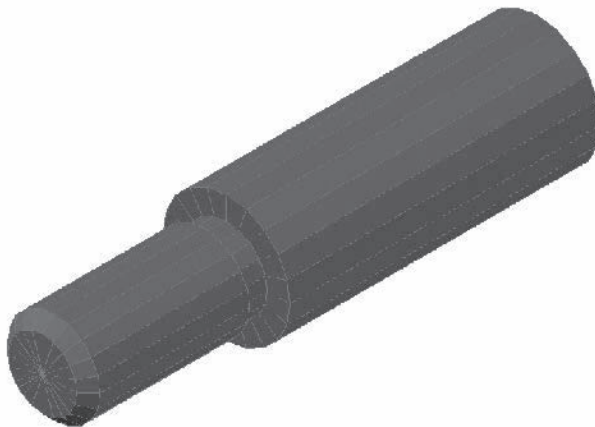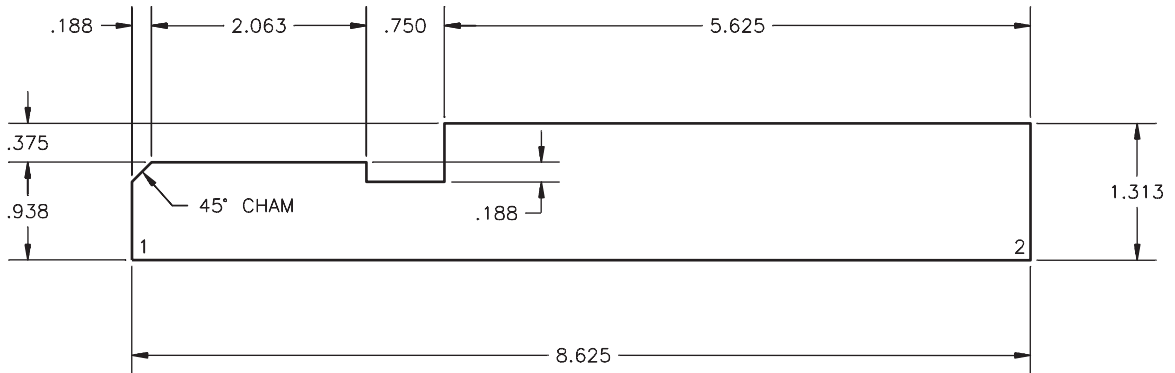
Fig. 43-3

HINT Begin at point 1, work counterclockwise, and take advantage of polar tracking. Set the polar tracking angle to 45°, which will help you create the 45° chamfer.

3. Create a line that passes through points 1 and 2.

4. View the profile from the **SW Isometric** viewpoint and zoom out slightly.

5. Enter **ZOOM** and **.7x**. (You can also use the wheel button on the mouse to zoom out.)

You should now see a view similar to the one in Fig. 43-4.
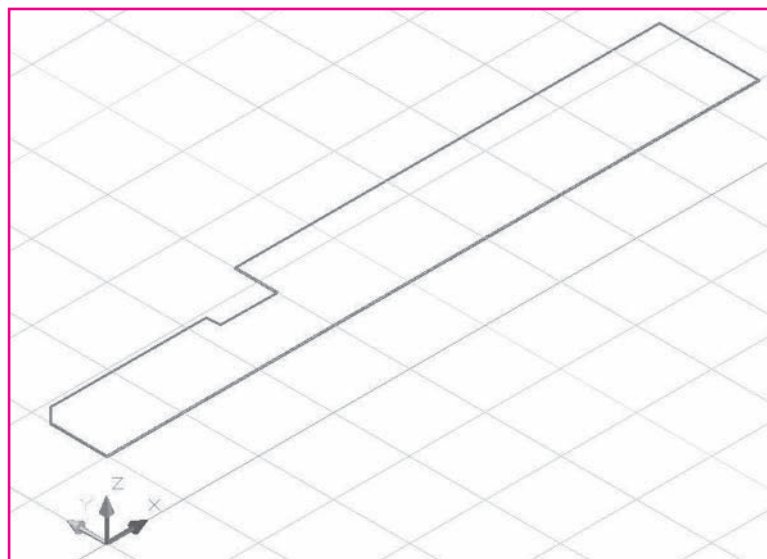
Fig. 43-4

6. Enter **DELOBJ** and **0**.

DELOBJ is the **Delete Object** system variable. With the default value of **1**, the 2D profiles used to create solids are deleted. The **DELOBJ** value of **0** retains the profile, which is what we wish to do.

7. Enter **SURFTAB1** and **20**.

SURFTAB1 is a system variable that controls the number of tesselations on a mesh model in the direction of revolution. The default value is **6**, which would produce a revolved mesh with a hexagonal cross section.

8. Pick the **Revolved Surface Mesh** button on the **Primitives** panel.

PRIMITIVES

This activates the **REVSURF** command.

9. Select the polyline and press **ENTER**.

10. Pick the line as the axis of rotation.

11. Press **ENTER** twice to select the default 0- to 360-degree angle of revolution.

The mesh model of the shaft generates on the screen, as shown in Fig. 43-2 on page 560. Note that you could have used the **REVOLVE** command to produce a solid model, instead of a mesh model, from the polyline.

12. Save your work and close the drawing.

Extrusion

The solid model of an aluminum casting shown in Fig. 43-5 was created using the **EXTRUDE** command. This command allows you to **extrude**, or add depth to, a closed area. The **EXTRUDE** command works on closed 2D objects such as planar 3D faces, closed polylines, polygons, circles, ellipses, closed splines, and donuts. It is not possible to extrude objects contained within a block or a polyline that has a crossing or self-intersecting segment.

1. Begin a new drawing using the **solid.dwt** template file.

2. Using the **PLINE** command's **Arc** and **Line** options, approximate the size and
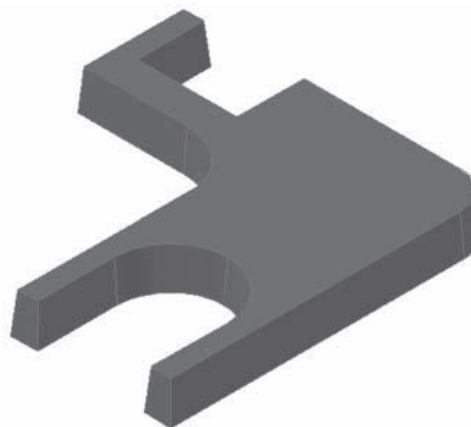
Fig. 43-5
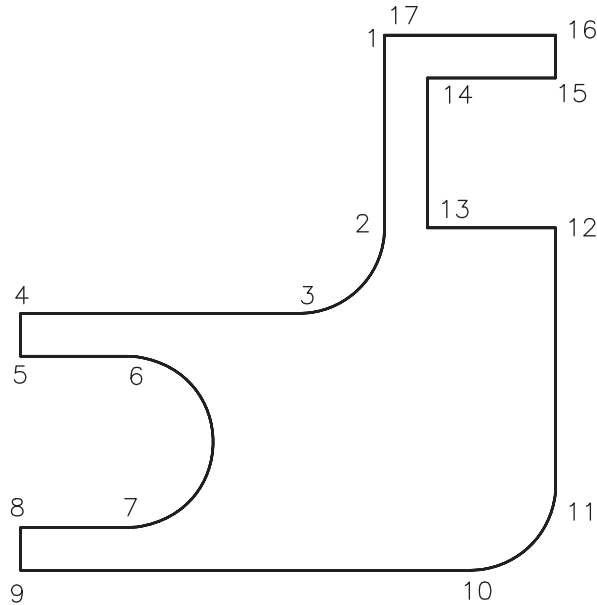
Fig. 43-6

shape of the casting shown in Fig. 43-6. The casting's overall size is **6.25″** × **6.25″**. Pick the points in the order shown. Do not place the numbers on the drawing.

3. Save your work in a file named **extrude.dwg**.

4. Select the **SW Isometric** viewpoint.

5. Pick the **Extrude** button on the **Modeling** panel.

6. Select the polyline and press **ENTER**.

7. Move the crosshairs up and down in the drawing area and notice that AutoCAD displays the height of the extrusion dynamically.

8. Type **T** to activate the **Taper angle** option and enter **5** (for 5 degrees of taper).

9. Enter **.75** for the height of the extrusion.

An extruded solid model appears, with a height of .75 and sides tapered at 5°, as shown in Fig. 43-5.

10. In an open area of the screen, draw an arc and a circle of arbitrary size on the ground plane.

11. Use the **EXTRUDE** command to extrude both objects **1.5** inches. Do not taper the extrusions.

Your drawing should look similar to the one in Fig. 43-7 on page 564.

MODELING

MODELING
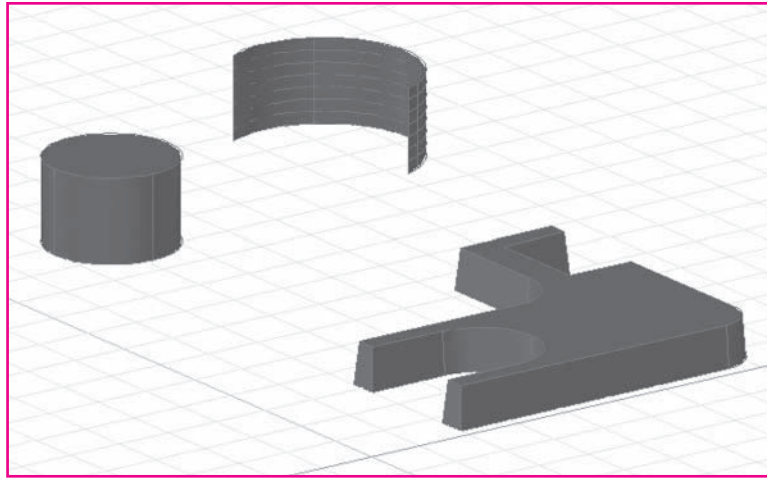
Fig. 43-7

12. Open the **Properties** palette and select each of the two new extrusions individually.

Notice that the extrusion created from the circle is a 3D solid, while the extrusion created from the arc is a surface. This is because the circle is a closed profile, like a closed polyline, while the arc is an open profile.

If we had used a series of lines instead of a polyline to create the profile of the aluminum casting, the **EXTRUDE** command would have produced a series of surfaces. All of the tapered wall areas would have appeared, but the model would have no top or bottom—and no closed volume.

13. Save your work and close the drawing.

Sweeping

The **SWEEP** command creates a new solid or surface by **sweeping**, or moving a 2D profile along a path curve. The sweep curve can be open or closed, and 2D or 3D. We will demonstrate the **SWEEP** command by sweeping a circle along the path of a helix.

DRAW

1. Begin a new drawing using the **solid.dwt** template file.

2. Draw a **1″** line and a **1″**-diameter circle anywhere on the ground plane.

3. Pick the **Helix** button on the expanded **Draw** panel to enter the **HELIX** command.

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IX. Solid Modeling

43. Basic Modeling

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571

This command creates a **helix**, or open 3D spiral, given the radius of the spiral at the top and bottom and the height of the spiral. It is often used to create springs and coils. You can also create a 2D spiral by specifying a height of 0.

4. Pick a point on the ground plane and enter **1.5** for the base radius.

5. For the top radius, enter **3.5**.

6. For the height, enter **10**.

The helix forms. Note that we accepted the default number of turns (3) and twist direction (counterclockwise).

7. Pick the **Sweep** button on the **Modeling** panel.

8. Pick the **1″** line as the object to sweep and press **ENTER**.

9. Select the helix as the sweep path.

MODELING

The 3D surface model—a helical ribbon—appears on the screen, as shown in Fig. 43-8.

10. Undo the sweep and pick the **Sweep** button once again.

11. This time, pick the circle as the object to sweep and press **ENTER**.

12. Again select the helix as the sweep path.

MODELING

A 3D solid model of the tapered spring appears, as shown in Fig. 43-9.

13. Save your work in a file named **sweep.dwg**.
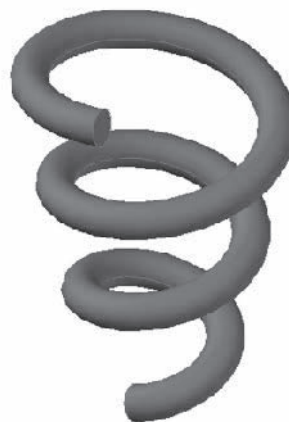
Fig. 43-8

Fig. 43-9

Lofting

Creating a new solid model or surface by specifying a series of 2D cross sections is known as **lofting**.

1. Begin a new drawing using the **solid.dwt** template file.

2. Draw a **6″**-diameter circle on the ground plane. Center the circle at $x = 5$, $y = 5$.

3. Enter the **ELEV** command, specify a new elevation of **5** ($z = 5$), and a thickness of **0**.

4. Draw a **2″**-diameter circle at the $z = 5$ elevation. Center the circle at $x = 5$, $y = 5$.

5. Enter the **ELEV** command and specify a new elevation of **10** ($z = 10$) and a thickness of **0**.

6. Use the **RECTANG** command to draw a square at the $z = 10$ elevation. Locate the first corner at $x = 3$, $y = 3$. Locate the other corner at $x = 7$, $y = 7$. (If you are entering coordinates at the keyboard, you may need to turn dynamic input off on the status bar to get accurate results.)

7. View the drawing area from the **SE Isometric** viewpoint.

8. Pick the **Loft** button on the **Modeling** panel.

9. Select, in order, the large circle, the small circle, and the square; then press **ENTER**.

10. Press **ENTER** again to accept the **Cross sections only** default.

A 3D solid model is created, with a lofted contour that passes through the three cross sections. The **Loft Settings** dialog box is displayed as well. The lofted solid and the **Loft Settings** dialog box are shown in Fig. 43-10.

AutoCAD offers several options to control the contour of the lofted solid. Initially, the **Smooth Fit** option is selected.

11. Select the **Ruled** option and notice that the lofting between the cross sections changes to straight lines. (Make sure the **Preview changes** box in the lower left corner is checked.)

12. Select the **Normal to** option and explore the different options in the drop-down menu.

The **Draft angles** option allows you to specify the angle of the lofted contour at the first and last cross section. By default, they are set to 90°.

13. Reselect **Smooth Fit** and pick the **Close surface or solid** box.

MODELING
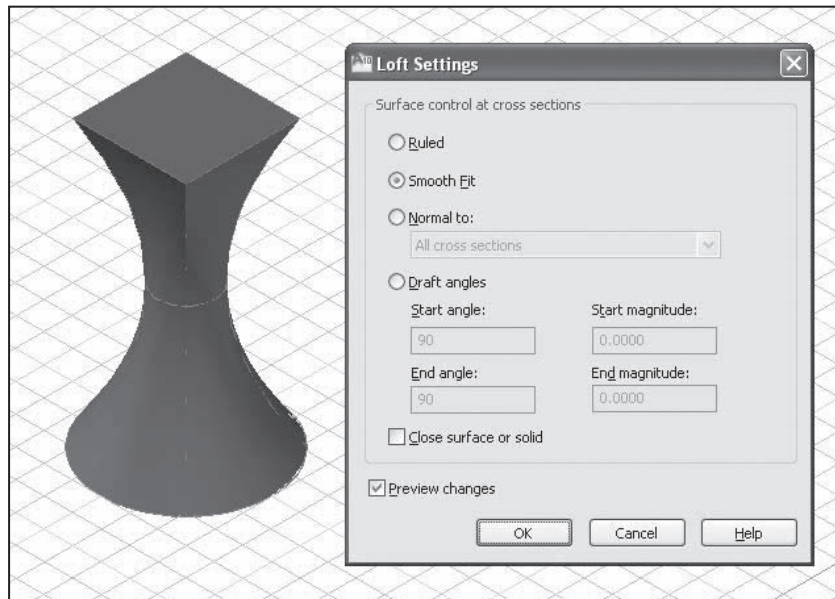
Fig. 43-10

This option creates a torus-like tube, with the lofted cross sections defining the inside surface of the tube.

14. Uncheck the **Close surface or solid** box and pick **OK**.

15. Save your work in a file named **loft.dwg**.

Working with Surfaces

The **PLANESURF** command allows you to create a planar surface either by specifying a rectangular area or by selecting objects that form an enclosed area.

Creating Surfaces

1. Begin a new drawing using the **solid.dwt** template file.

2. Draw a circle, a six-sided polygon, and a closed polyline of any shape and size anywhere on the ground plane.

3. Pick the **Planar Surface** button on the **Modeling** panel to enter the **PLANESURF** command.

Planar Surface

MODELING

4. In response to **Specify first corner or [Object]**, press **ENTER** to select the **Object** mode.

5. Select the circle, polygon, and polyline and press **ENTER**.

Three trimmed surfaces are created, defined by the three enclosed areas. The **PLANESURF** command works for any enclosed 2D area, including a series of line segments.
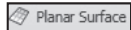
Planar Surface

MODELING

6. Pick the **Planar Surface** button on the **Modeling** panel.

7. In response to **Specify first corner or [Object]**, pick a point in an open area of the ground plane.

8. Move the crosshairs and preview the planar surface in the drawing area.

9. Pick a second point to form the planar surface.

The **PLANESURF** command is not limited to the 2D ground plane. Let's create a surface on an angled plane.

10. Display the **Coordinates** panel and select the **X** button.

11. Enter **45** to rotate the current UCS 45° about the X axis.

Planar Surface

MODELING

12. Pick the **Planar Surface** button again.

13. Pick two points to define the corners of the planar surface.

14. Pick the **World** button on the **Coordinates** panel.
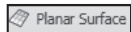
Converting Surfaces to Solids

SOLID
EDITING

The **THICKEN** command allows you to create a solid from a surface.

1. Pick the **Thicken surface** button on the **Solid Editing** panel.

2. Select the hex-shaped surface and the angled planar surface and press **ENTER**.

3. Specify a thickness of **4**.

Two solids are created. Your drawing should look similar to the one in Fig. 43-11. (Your polygonal surface will be different.)

4. Save your work in a file named **planesurf.dwg** and exit AutoCAD.
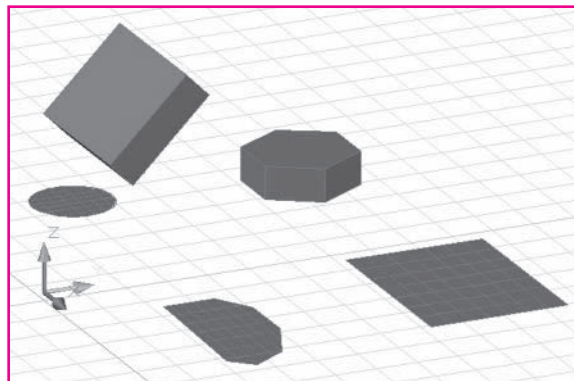
Fig. 43-11

Chapter 43 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Describe each of the following commands.

    a. **REVOLVE**

    b. **EXTRUDE**

    c. **SWEEP**

    d. **LOFT**

    e. **PLANESURF**

    f. **THICKEN**

2. What command(s) would you use to create a 3D model of a quart oil container?

3. What effect does the **Ruled** surface control have on a lofted model?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. What command should be used to create a model of a toothpaste tube? …a model of a Frisbee®? Explain your answers.

2. Is it possible to extrude a wide polyline (one that has a defined width)? Explain.

Continued

Chapter 43 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Identify a simple object in the room. Create a profile of the object and then extrude it to create a solid model. Note that only certain objects—those that can be extruded—can be used for this problem.

2. Open **shaft.dwg** and save it as **ch43-revolve.dwg**. Erase the revolved mesh. Create two new revolved models using the **X axis** and **Y axis** options. Make one a solid model and the second a mesh model. Move one of the models so they do not overlap (use the **Move** button on the **Modify** panel of the **Ribbon**). Your completed drawing should look similar to Fig. 43-12. Save your work.

Fig. 43-12

3. Use the following four methods to create a solid cylinder with a radius of **.4** and a height of **2**.

   a. Draw a rectangle measuring **.4** × **2** and revolve it **360°**.

   b. Draw a circle with a radius of **.4** and extrude it to a height of **2**.

   c. Use the **CYLINDER** command to create a solid cylinder with a radius of **.4** and a height of **2**.

   d. Use **Mesh Cylinder** to create a mesh model with a radius of **.4** and a height of **2**.

4. Open **sweep.dwg** and save it with a new name of **ch43-sweepprob.dwg**. Erase the swept solid. Create a new profile by drawing a **.5″** × **1.5″** rectangle on the ground plane. Create a new swept solid using the new rectangle as the profile, and use the **Twist** option to twist the swept profile **270°**. Your drawing should look similar to the one in Fig. 43-13. Save your work.

Fig. 43-13

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IX. Solid Modeling

43. Basic Modeling

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577
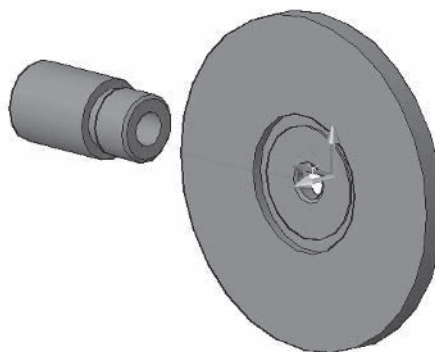
Review & Activities Chapter 43

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Create a solid model of the boat trailer roller. Define the shape of the roller with a polyline, as shown in Fig. 43-14.
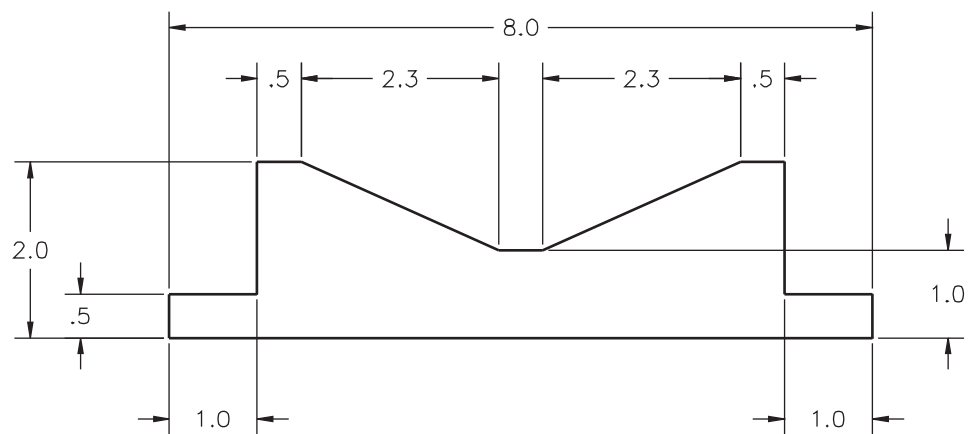
Fig. 43-14

Revolve the profile and view it from the **SW Isometric** viewpoint. Save the drawing as **ch43roller.dwg**. Your drawing should look similar to the one in Fig. 43-15.

Fig. 43-15

Continued

Chapter 43: Basic Modeling   571

Chapter 43 **Review & Activities**

2. Create the compression spring shown in Fig. 43-16 using the appropriate 3D modeling commands. Make the spring **7″** long, with an overall diameter of **2.5″** and a wire diameter of **.25″**. Save your work as **compression spring.dwg**.

3. Use the **LOFT** command to create a lamp similar to the one shown in Fig. 43-17. The base has a diameter of **7″**, and the lamp is **13.7″** tall. Determine how many cross-sectional profiles you need and what settings to use in the **Loft Settings** dialog box. Save your work as **lamp base.dwg**.

4. The bumper assembly for a bumper pool table is shown in Fig. 43-18. Use the dimensions of the three parts shown in Fig. 43-19 and use the **REVSURF** command to create three mesh models. Note that the bumper support has a square hole. Assume the hole to be circular. After you have finished modeling the three parts, place them in an isometric view. View the models in every 3D visual style. Save your work as **bumper.dwg**.

Fig. 43-16

Fig. 43-17
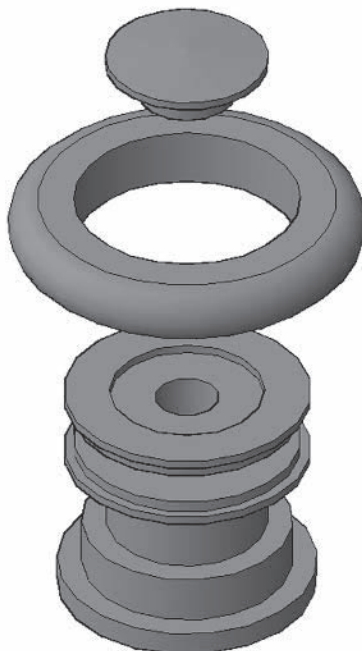
Fig. 43-18
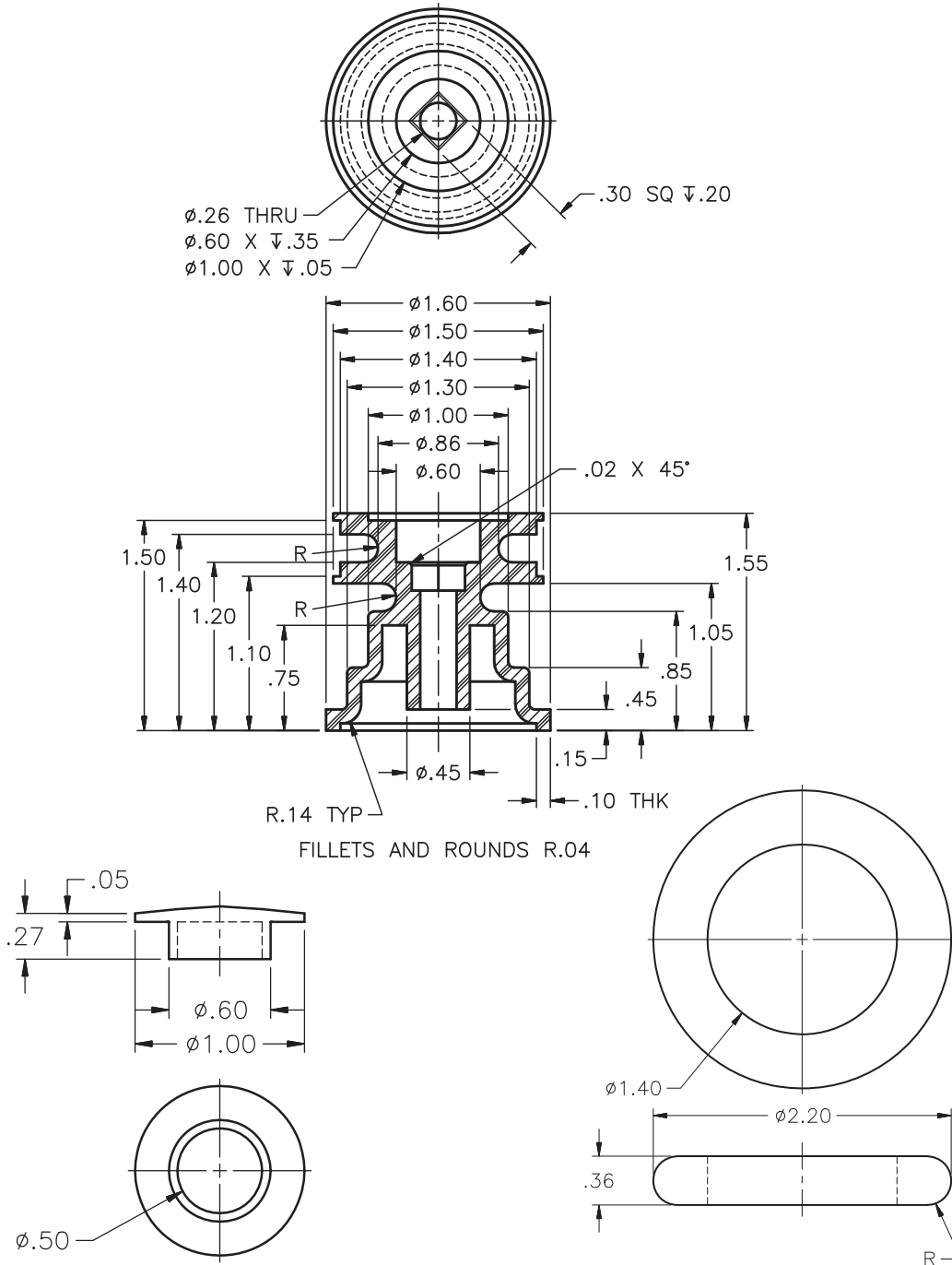
Fig. 43-19

Problem 4 courtesy of Gary J. Hordemann, Gonzaga University

Boolean Operations

Objectives

- Prepare solid primitives for Boolean operations
- Use the Boolean subtraction operation to remove portions of a solid model
- Use the Boolean union operation to combine composite models

Vocabulary

Boolean logic
Boolean mathematics
Boolean operations
composite solids

Much of AutoCAD solid modeling is based on the principles of **Boolean mathematics**. Boolean math, also called **Boolean logic**, is a system created by mathematician George Boole for use with logic formulas and operations. AutoCAD **Boolean operations**, such as union, subtraction, and intersection, are used to create composite solids. **Composite solids** are composed of two or more solid objects. Using Boolean operations, let's create a composite solid of the steel support shown in Fig. 44-1.
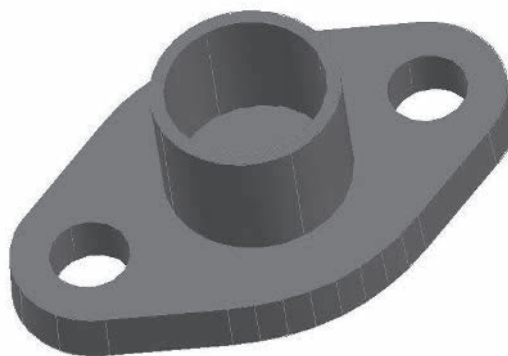
Fig. 44-1

Preparing the Base Solid

The first step in creating a composite solid is to create the base solid—the solid to which other solids will be joined.

1. Start AutoCAD and begin a new drawing using the **solid.dwt** template file.

2. Using the **PLINE Arc** and **Line** options, create the shape shown in Fig. 44-2 using the information below.

   - Use snap and grid settings of **.25** to produce the polyline accurately.

   - Pick the four points in the order shown. Each point falls on the snap grid.

   - In the **PLINE** command, use the **Arc**'s **Radius** option.

Fig. 44-2

You have just completed one quarter of the object shown in Fig. 44-3.
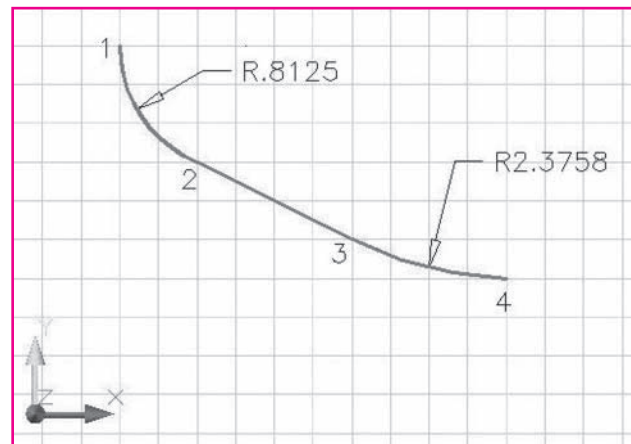
3. Mirror the shape to create one half of the object, and mirror that to create the entire object.

4. Use the **PEDIT Join** option to join the four polylines into one.

Fig. 44-3

Chapter 44: Boolean Operations   575
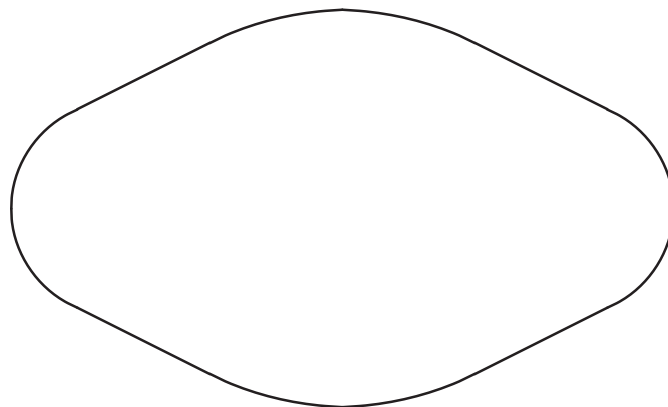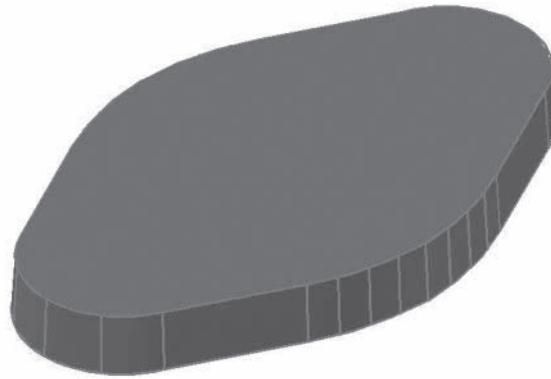
AU: Is SW correct here?

Fig. 44-4

MODELING

5. Select the **SW Isometric** viewpoint.

6. Save your work in a file named **composite.dwg**.

7. Pick the **Extrude** button on the **Modeling** panel.

8. Select the polyline and press **ENTER**.

9. Specify a taper angle of **3** degrees, and enter **.5** for the height.

A view similar to the one in Fig. 44-4 appears.

10. Pick the **Top** viewpoint, switch to the **3D Hidden** visual style, and **ZOOM All**.

The top view allows you to see the taper, as shown in Fig. 44-5, but without the two holes and the dimensions.
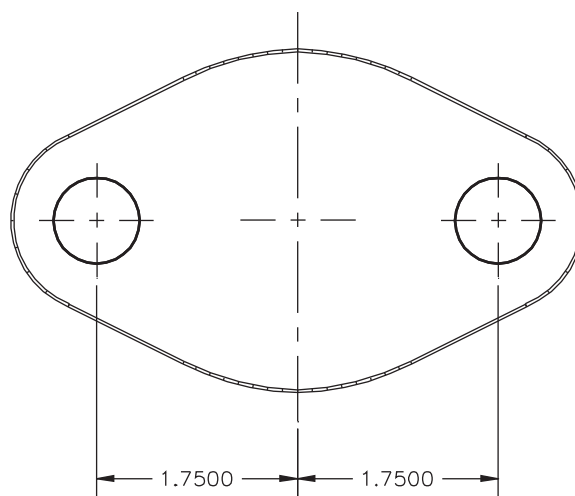
Fig. 44-5

11. Display the **Coordinates** panel and pick the **World** button. (You can also select **WCS** at the flyout menu below the **ViewCube.**)

12. On the status bar, toggle **Dynamic UCS** off.

13. Pick the **Cylinder** button on the **Modeling** panel and place two cylinders as shown in Fig. 44-5 to represent the holes. Make their radius **.375** and their height **.5**.

COORDINATES

HINT Create one cylinder and use the **COPY** or **MIRROR** command to create the second.

You have just created two solid cylinders within the original extruded solid.

14. Select the **SE Isometric** viewpoint and the **Conceptual** display.

Subtracting the Holes

The **SUBTRACT** command performs a Boolean operation that creates a composite solid by subtracting one solid object from another.

1. Pick the **Subtract** button from the **Solid Editing** panel to enter the **SUBTRACT** command.

2. Select the extruded solid (but not the cylinders) as the solid to subtract from, and press **ENTER**.

3. Select the two cylinders as the solids to subtract, and press **ENTER**.

SOLID EDITING

AutoCAD subtracts the volume of the cylinders from the volume of the extruded solid.

4. Rotate the model to see that the cylinders have been removed, creating two holes in the solid.

The composite solid should look similar to the one in Fig. 44-6.
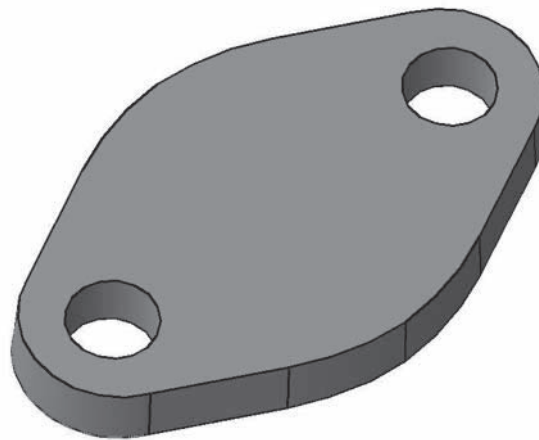
Fig. 44-6

Adding the Support Cylinder

The **UNION** command permits you to join two solid objects to form a new composite solid.

1. Select the **Top** view and the **World** UCS.

2. Place a solid cylinder at the center of the model. Make the diameter **1.75″**, and make it **1.75″** in height.

> **HINT** Use the snap grid to snap to the center of the model.

3. Using the same center point, place a second cylinder inside the first. Specify a diameter of **1.5″** and a height of **1.75″**.

4. Select the **SE Isometric** viewpoint.

5. Subtract the smaller cylinder from the larger cylinder.

SOLID EDITING

The model is currently made up of two separate solid objects. Let's use the **UNION** command to join them to form a single composite solid.

6. Pick the **Union** button on the **Solid Editing** panel.

SOLID EDITING

This enters the **UNION** command.

7. Select the two solid objects and press **ENTER**.

The two objects become a single composite solid.

8. Pick the **Conceptual** visual style.

Your model should look similar to the one in Fig. 44-1 (page 574).

You have learned to create composite solids by adding and subtracting solid objects. You can also create them by overlapping two or more solid objects and calculating the solid volume that is common to each of the objects—the intersecting volume. The **INTERSECT** command performs this Boolean operation. Experiment on your own with the **INTERSECT** command.

9. Save your work and exit AutoCAD.

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IX. Solid Modeling

44. Boolean Operations

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585

Chapter 44 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Describe the procedure used to create a hole in a solid object.
2. What is a composite solid?
3. Explain the purpose of the following solid modeling commands.
   a. **SUBTRACT**
   b. **UNION**
   c. **INTERSECT**

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore further the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1–2. Describe how you would create the solid models shown in Figs. 44-7 and 44-8. Be specific.
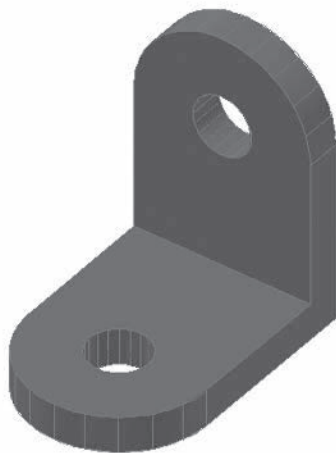
Fig. 44-7
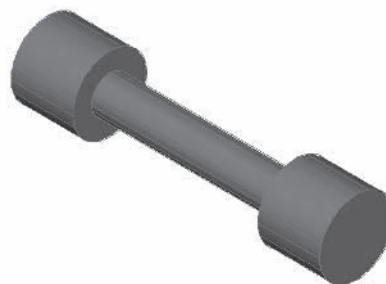
Fig. 44-8

Continued

Chapter 44 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create the composite solid shown in Fig. 44-9.
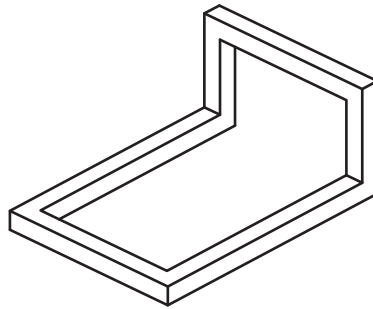
Fig. 44-9

2. Create the composite solid shown in Fig. 44-10.
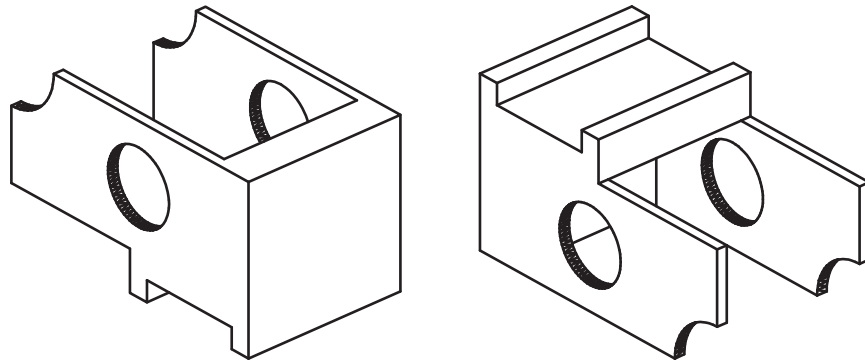
Fig. 44-10

3. Identify a machine part, such as a gear on a shaft or a bracket. Sketch the part in your mind or on paper. Using AutoCAD's solid primitives and Boolean operations, shape the part by adding and subtracting material until it is complete.

Review & Activities Chapter 44

4. Draw a 3D model of the height gage shown in Fig. 44-11 using the **EXTRUDE** command and Boolean operations. Can you use a box or a wedge to create the V-groove? Why or why not? Save the drawing as **ch44htgage.dwg**.
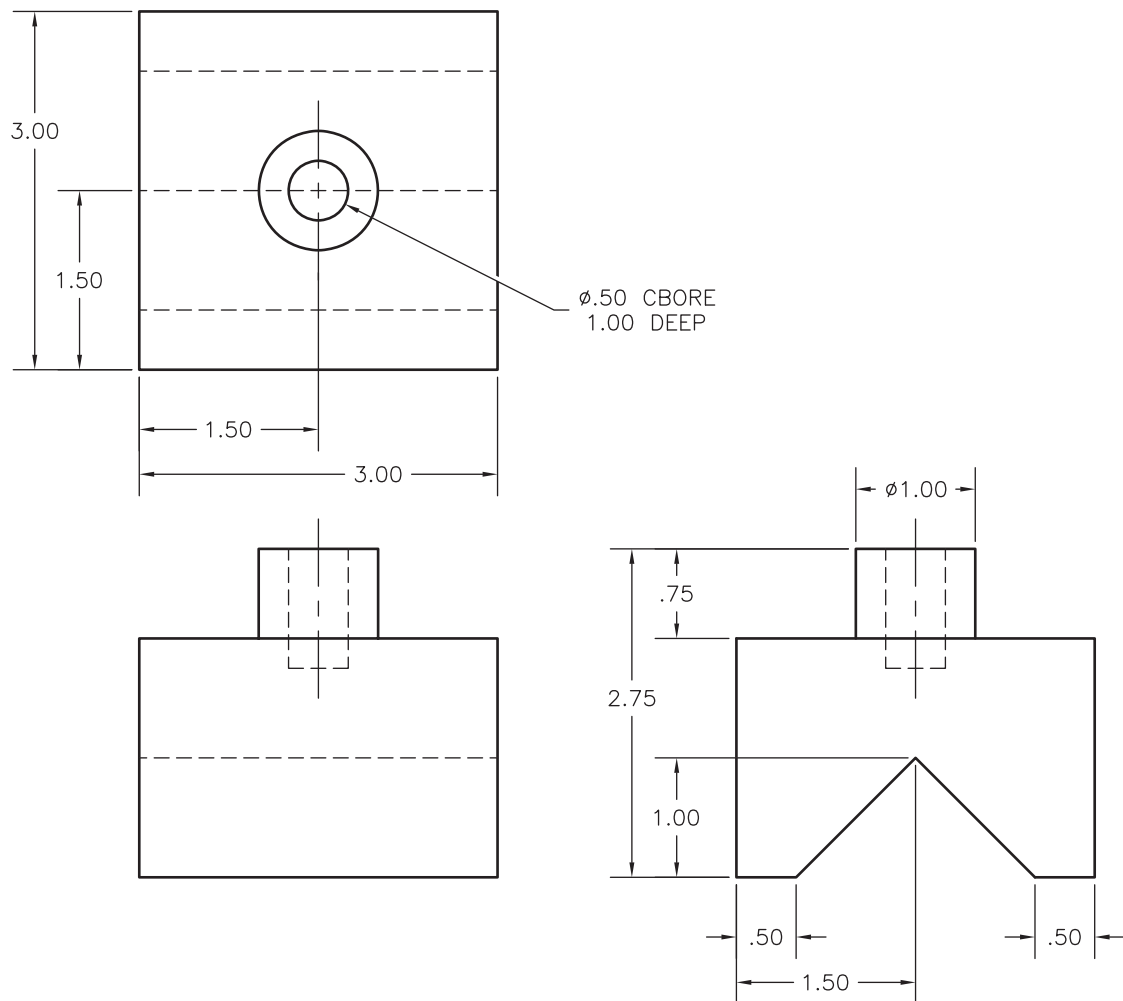
Fig. 44-11

Continued

Chapter 44 **Review & Activities**

5. Your customer wants to see a 3D view of the base plate shown in Fig. 44-12. Create the base, extrude it, and then use Boolean operations to create the 3D model. View the base plate in the **Conceptual** visual style, as shown in Fig. 44-13. Save the drawing as **ch44baseplate.dwg**.
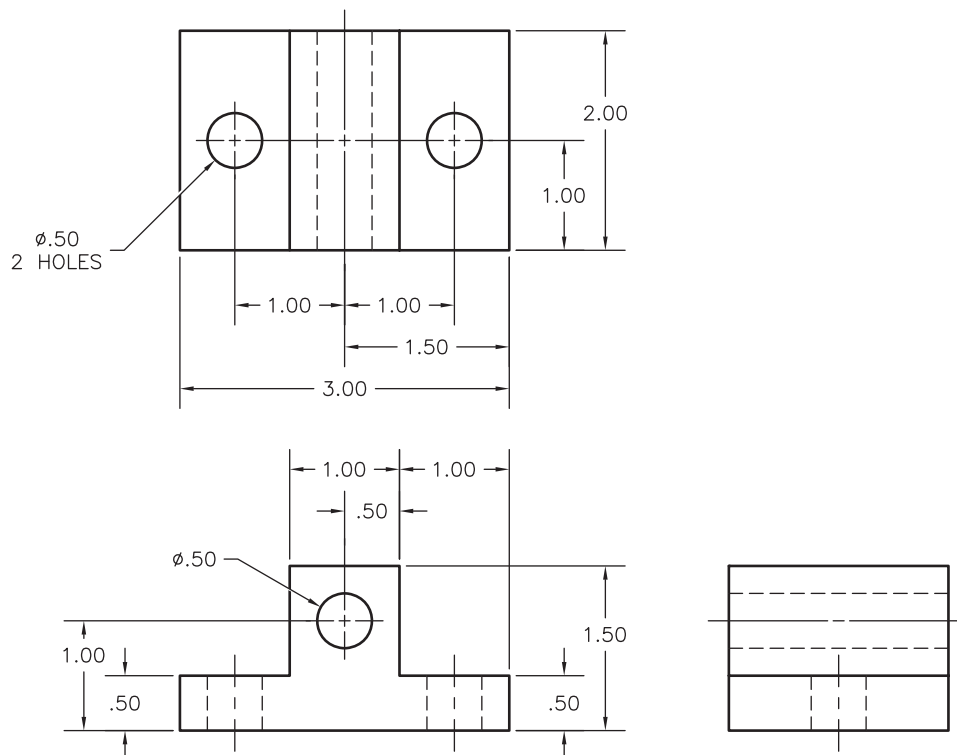
Fig. 44-12

Figure 44-12 prepared by Gary J. Hordemann, Gonzaga University
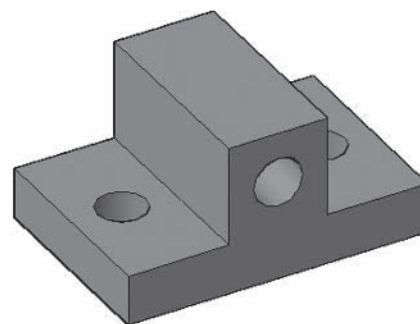
Fig. 44-13

Review & Activities Chapter 44

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Create the fluid coupling end cap shown in Fig. 44-14 using the **REVOLVE** command. Begin by creating a layer named **Endcap**, and set the color to cyan. The object's profile is shown in Fig. 44-15.

   Create the revolved model of the endcap on layer **Objects** and freeze the **Endcap** layer. Enter **ISOLINES** and enter a value of **25**. (The **ISOLINES** system variable controls the number of tessellation lines used in the **3D Wireframe** and **Realistic** visual styles.) View the endcap in all four of the 3D visual styles at various isometric viewpoints. Save your work as **endcap.dwg**.
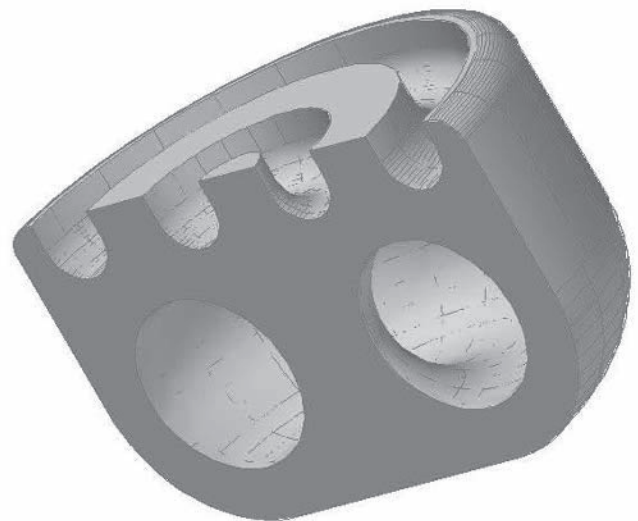
Fig. 44-14

Problem 1 courtesy of Gary J. Hordemann, Gonzaga University
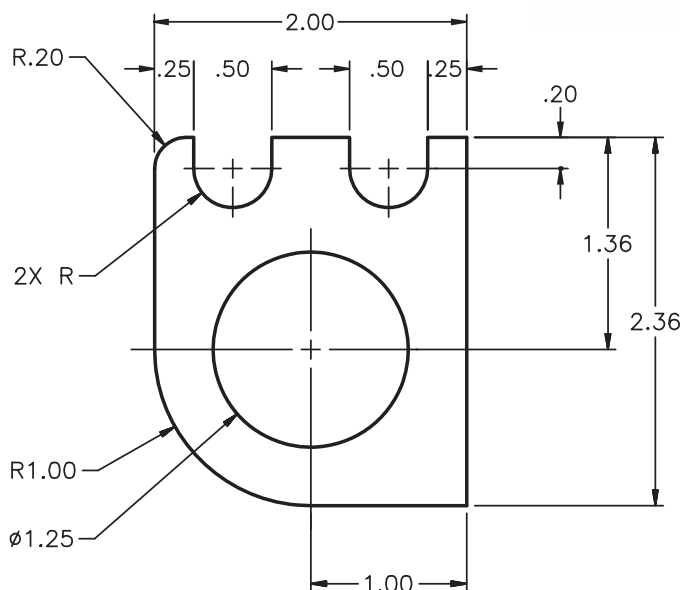
Fig. 44-15

Continued

Chapter 44 **Review & Activities**

2. Create the shaft clamp shown in Fig. 44-16 using the **SUBTRACT** and **EXTRUDE** commands. Create the **.40**-diameter hole by inserting and subtracting a cylinder. Display the model in the **Conceptual** visual style.
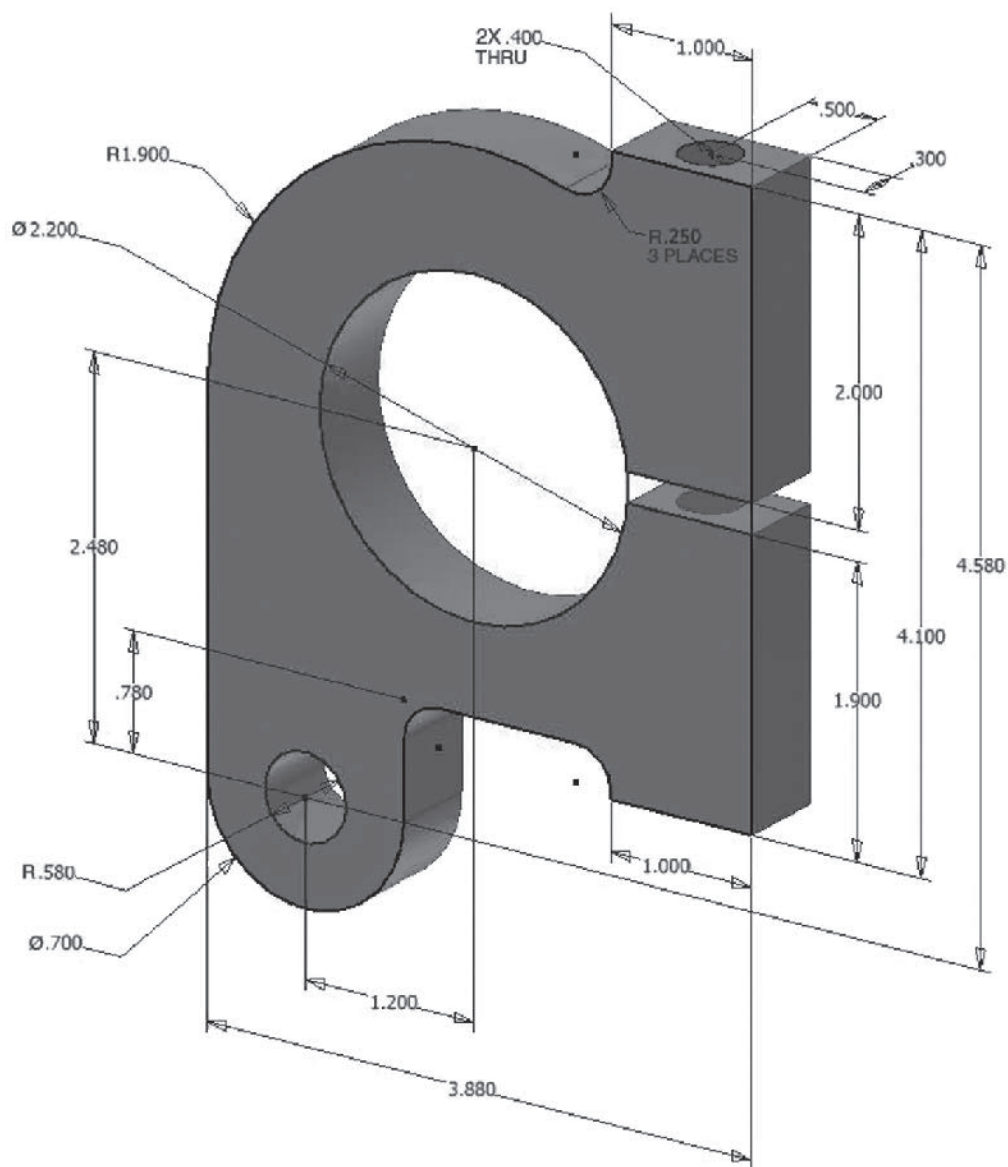
Fig. 44-16

Courtesy of Gary J. Hordemann, Gonzaga University

3. Create a solid model of the adjustable pulley half shown in Fig. 44-17. If you allow for the counterbored hole when drawing the profile, you will not have to bring in and subtract cylinders. After revolving the profile, go to the front view and insert and subtract the cylinders for the two smaller holes.
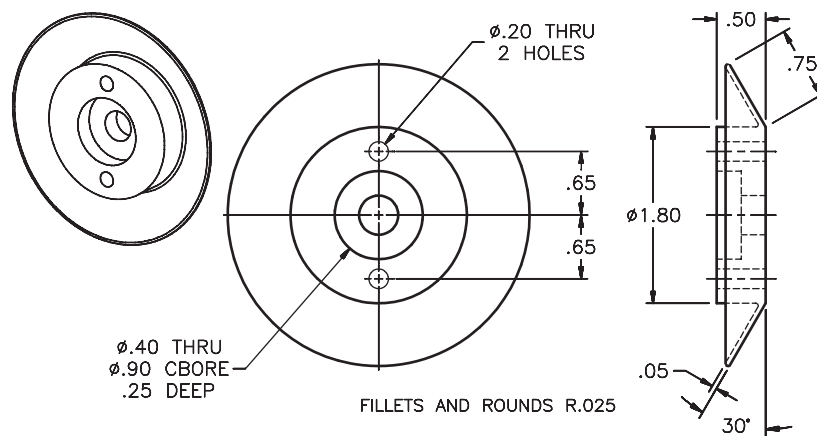
Fig. 44-17

Chapter 45

Adjusting 3D Models

Objectives

- Move, stretch, and rotate objects in 3D
- Use grips and subobject selection to adjust solids
- Modify solids with the **PRESSPULL** command
- Add chamfers and fillets to solids
- Adjust solids using the **SLICE** and **SHELL** commands
- Modify 3D mesh objects and convert them to solids

Vocabulary

dynamic UCS
gizmo
interference
subobject
vertex grip

In the last three chapters, you have been introduced to many 3D–modeling methods. When these methods are used in combination, AutoCAD becomes a powerful tool for modeling solids and surfaces. In this chapter, you will be introduced to several techniques to adjust and modify existing solids.

AutoCAD allows you to use grips to move, stretch, and rotate solids in 3D. You can also select and modify individual subobjects. A **subobject** is a face, edge, or corner of a solid. Additionally, many of the tools you have used to modify 2D objects can also be used to modify solids. You will use familiar tools and learn several new commands to adjust solids.

Many of today's products, from cell phone housings and handheld remote controls to the exteriors of automobiles, are designed with curved, flowing styles. 3D mesh modeling allows you to create these smooth, organic shapes that are difficult to model with the solid modeling tools.

Modifying Solid Primitives

Let's start by modifying solid primitives using 3D commands, grips, and subobject selection.

1. Start AutoCAD and use the **solid.dwt** template file to create a new drawing named **3Dgrips.dwg**.
2. Select the **3D Modeling** workspace and close all palettes and floating toolbars.
3. Select the **NE Isometric** viewpoint and the **3D Wireframe** visual style.
4. Use the **BOX** command to create a box primitive of any size and shape on the ground plane.

MODELING

Moving Objects in 3D

The **3DMOVE** command allows you to move solids in 3D space.

1. Pick the **3D Move** button on the **Modify** panel of the **Ribbon.**
2. Select the box primitive and press **ENTER**.

MODIFY

The **Move** gizmo appears at the center of the selection set. A **gizmo** is an icon that simplifies the selection process for 3D commands. The **Move** gizmo consists of three axis handles and a center box.

3. Move the crosshairs to the middle of the gizmo and right click to display the shortcut menu.
4. Verify that the **Move** gizmo is checked, then select **Relocate Gizmo** from the shortcut menu.
5. Move the center box of the gizmo to the front lower corner of the box, as shown in Fig. 45-1.
6. Pick the center box of the gizmo as the base point.
7. Pick a second point in the drawing area to move the solid.
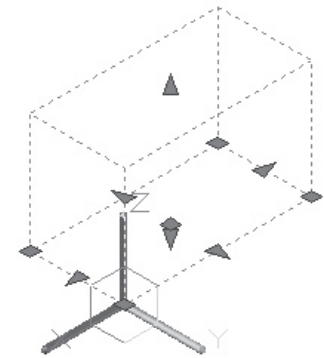
Fig. 45-1

You can use the grip tool to constrain the direction of a move.

8. Undo the 3D move.
9. Repeat steps 1 through 5.
10. Move the crosshairs over the green axis handle until it is highlighted and a green vector appears, as shown in Fig. 45-2, and pick the handle.

Fig. 45-2

Chapter 45: Adjusting 3D Models   **587**
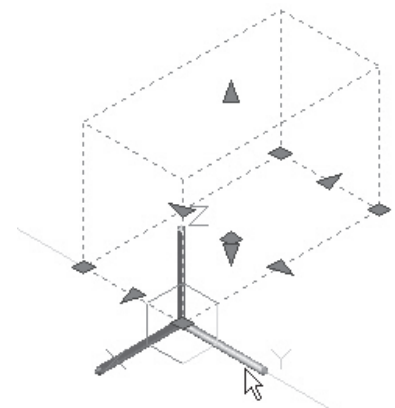
MODIFY

11. Drag the crosshairs in the drawing area to move the box and notice that its movement is now constrained along the selected axis vector.

12. Pick a point to complete the 3D move.

Rotating Objects in 3D

The **3DROTATE** command allows you to rotate objects in 3D space.

1. Pick the **3D Rotate** button on the **Modify** panel.

2. Select the box primitive and press **ENTER**.

The **Rotate** gizmo appears. Like the **Move** grip tool, the **Rotate** grip tool consists of three axis handles and a center box.

3. Pick the front lower corner of the box as the base point.

4. In reply to **Pick a rotation axis**, move the crosshairs over the circular blue axis handle to highlight it. (It becomes yellow when it is highlighted.)
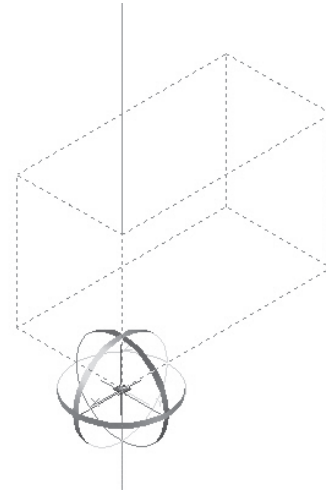
Fig. 45-3

A blue vector appears, as shown in Fig. 45-3.

5. Pick the axis while the blue vector is present.

6. Pick an endpoint on the box primitive as the angle start point; then rotate the box dynamically by moving the crosshairs in the drawing area.

7. Pick anywhere in the drawing area to complete the 3D rotation.

8. Experiment with the **3DSCALE** command and the **Scale** gizmo on your own.

9. Erase the box primitive.

Using Object Grips

In the previous exercise, you first entered the **3DMOVE** and **3DROTATE** commands and then selected the object. In this section, you will select the object first, before applying a command.

1. Create a new box primitive on the ground plane.

2. Pick the box with a single click and observe the grips and the **Move** gizmo that appear. (The **Move** gizmo should be the default gizmo, as shown in the **Subobject** panel.)

3. Rest the crosshairs on the rectangular grip on the bottom (down-facing) face of the box.

The **Move** gizmo appears on the grip. You can move the box primitive without selecting the **3D Move** button or typing the **3DMOVE** command.
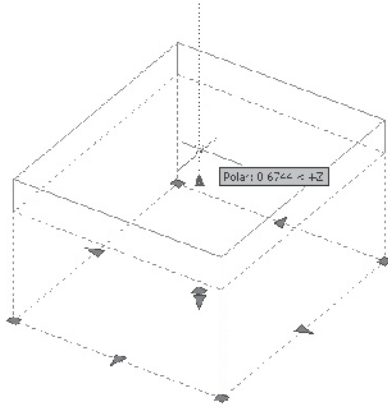
Fig. 45-4
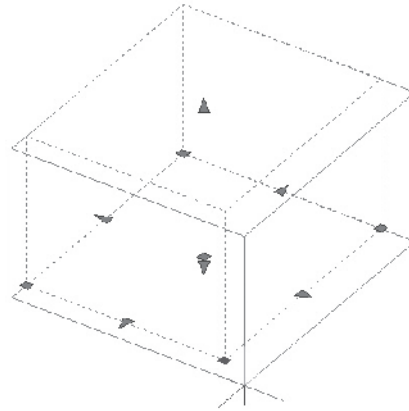
Fig. 45-5

4. Press **ESC** to deselect the grip tool location and the box primitive.

5. Select the box again.

6. Select the triangular grip on the top face of the box, as shown in Fig. 45-4, move the crosshairs upward, and pick a point to stretch the box (you may need to rotate the model using Orbit to see the grip clearly).

7. Select the square grip at the front lower corner of the box and stretch the box outward, as shown in Fig. 45-5.

The corner grip allows you to stretch the solid in two directions at the same time.

Using Subobjects

The moving, rotating, and stretching operations you have just learned can be applied to a subobject—a face, edge, or corner of a solid.

1. Select the **3D Hidden** visual style.

2. Select the **Face** subobject filter button from the flyout menu on the **Subobject** panel.

3. Press and hold the **CTRL** key and notice that the crosshairs changes to a pick box with a **Face** filter icon attached.

4. While pressing **CTRL**, move the pick box over the top face of the box primitive.

The top face of the box becomes highlighted. (If the top face is not highlighted, press the spacebar to toggle through the available selections.)

5. Pick the top face.

SUBOBJECT

Chapter 45: Adjusting 3D Models   **589**

Edge

SUBOBJECT

A face grip appears in the center of the top face, as shown in Fig. 45-6.

6. Select the **Edge** subobject filter button from the flyout menu on the **Subobject** panel.

7. While pressing **CTRL**, pick the front right edge of the top face.

An edge grip appears, as shown in Fig. 45-6.

8. Select the **Vertex** subobject filter button, press **CTRL**, and pick the upper right corner of the box.

Vertex

SUBOBJECT

A **vertex grip** (corner grip) appears, as shown in Fig. 45-6.

Notice that the grip for each type of subobject has a distinct shape. As you can see, you can select one subobject or a set of subobjects.

We will now use subobjects to modify the shape of the solid.

9. Press **ESC** to clear the selection set.

10. Select the **Edge** subobject filter button, press **CTRL**, and pick the front right edge of the top face.

11. Pick the edge grip and move the crosshairs in the drawing area.

Notice the change in the solid.

12. Press the **CTRL** key once again to toggle to the next modification option. Again, notice the change in the solid as you move the crosshairs.

13. Press the **CTRL** key again, observing the next option for modification.

As you can see, AutoCAD offers three different options for modifying the solid as you drag the edge.

14. Experiment on your own with subobject selection, selection sets, and modification options. Apply what you have learned to different primitives, and experiment on other types of solids, too.

15. Save your work and close the drawing file.
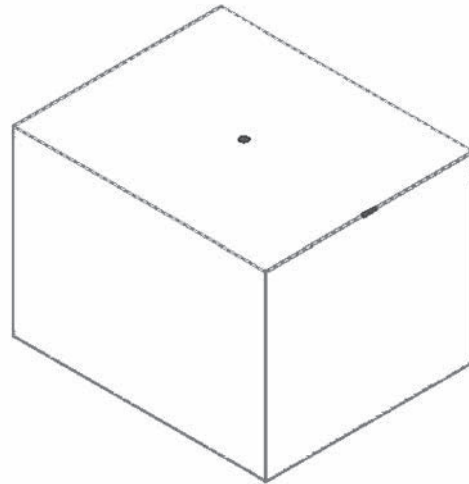
Fig. 45-6

Pressing and Pulling Solids

The **PRESSPULL** command allows you to "press" or "pull" any closed 2D pro-file on the face of a solid. When you press, the effect is a combination of the **EXTRUDE** and **SUBTRACT** commands. When you pull, the effect is a combination of the **EXTRUDE** and **UNION** commands.

1. Use the **solid.dwt** template to create a new drawing named **presspull.dwg**.

2. Select the **NE Isometric** viewpoint and the **3D Hidden** visual style.

3. Set snap and grid to **10**.

4. Make sure the **Dynamic UCS** button on the status bar is toggled on so that dynamic UCS is active.

5. Use the **BOX** command to create a box primitive with a base measuring **40 × 20** units and a height of **40** units.

6. Enter the **CIRCLE** command and move the crosshairs over the large side face of the box solid.

Notice that the face becomes highlighted.

7. With snap off, select a point near the center of the face as the center of the circle.

MODELING

The **dynamic UCS** icon appears, as shown in Fig. 45-7. The dynamic UCS works only on solid faces, and only from within active commands. It automatically and temporarily moves the current UCS to the selected face. This saves you from constantly managing and changing UCSs as you work on a model.

8. Enter a radius of **10**.

Fig. 45-7

Chapter 45: Adjusting 3D Models   **591**
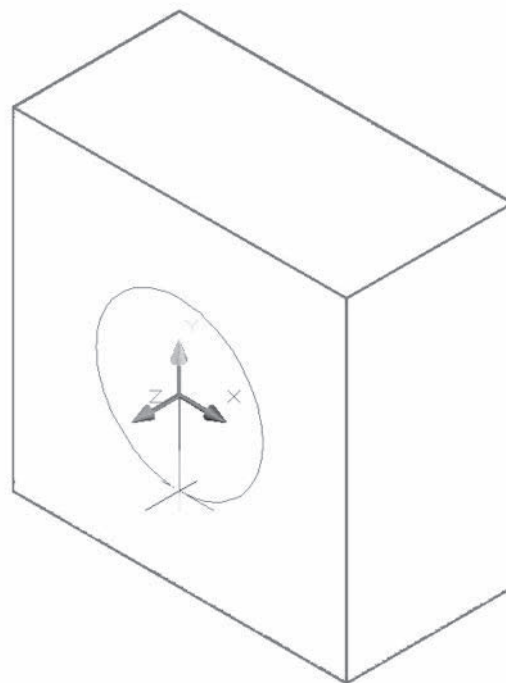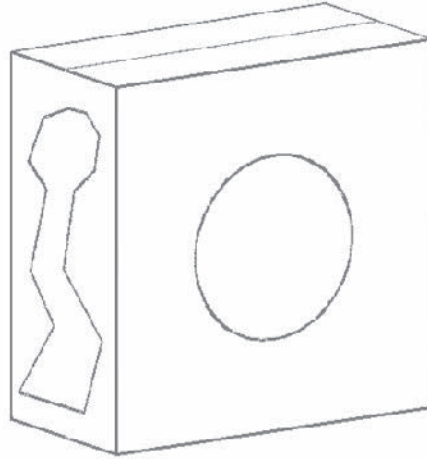
Fig. 45-8

9. On the top face, use the **LINE** command and the **Midpoint** object snap to split the face into two long, equal rectangles, as shown in Fig. 45-8.

10. Rotate the model as shown in Fig. 45-8.

11. On the front face, use a polyline to create a closed 2D profile of arbitrary shape and size, as shown in Fig. 45-8.

Press/Pull

MODELING

12. Select the **Press/Pull** button on the **Modeling** panel.

13. Pick inside the area bounded by the circle.

14. Press the circle through the box solid until you see the extruded cylinder on the far side of the solid, as shown in Fig. 45-9, and click the mouse button.

A hole is created in the solid.

Press/Pull

MODELING

15. Pick the **Press/Pull** button and pick inside the back rectangle on the top face of the solid.

16. Type **10** and press **ENTER**.

A raised step is created on top of the solid. As you can see, the **PRESSPULL** command works with keyboard entry.

17. Switch to the **3D Wireframe** visual style.
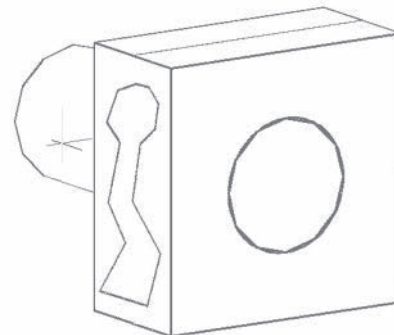
Fig. 45-9

18. Pick the **Press/Pull** button, pick the area bounded by the closed polyline, and press inward a short distance.

19. View the model in the **Conceptual** visual style.

Your model should look similar to the one in Fig. 45-10. **PRESSPULL** is a powerful command for quick modifications to the faces of solids. Keep in mind that you can press or pull only in the direction that is perpendicular to the face, and that you cannot taper the bounded area as it is pressed inward or pulled outward.

20. Save your work and close the **presspull.dwg** file.
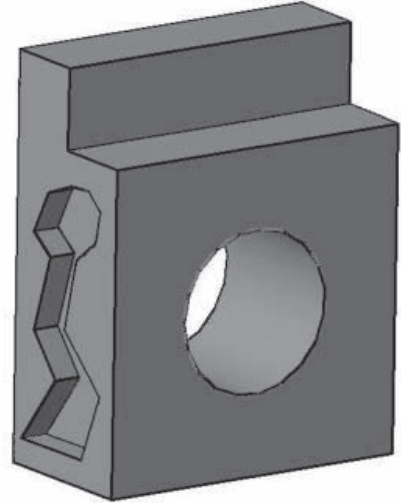
Fig. 45-10

Beveling and Rounding Solid Edges

Let's create the table shown in Fig. 45-11 using the methods we've used in previous chapters. We will apply the **CHAMFER** and **FILLET** commands to make the filleted and beveled edges.
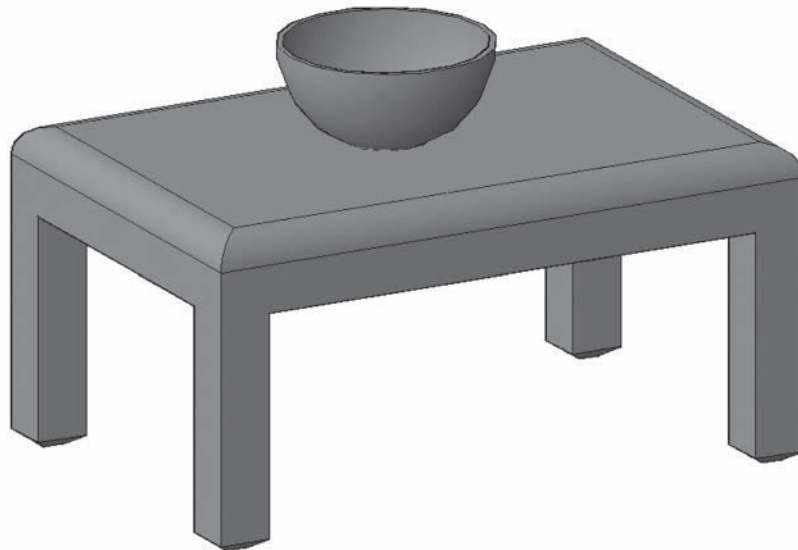
Fig. 45-11
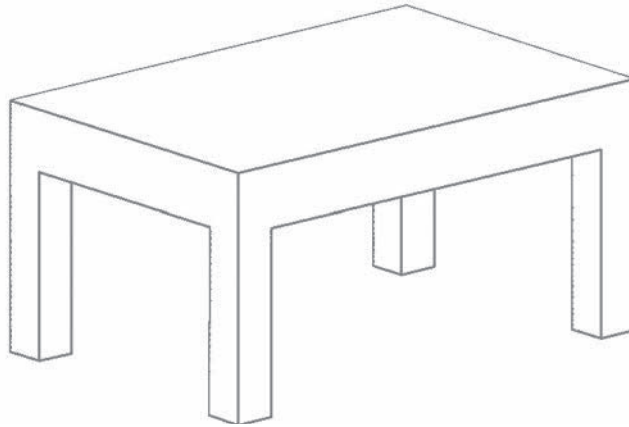
Fig. 45-12

MODELING

1. Use the **solid.dwt** template file to create a new file named **table.dwg**.

2. Pick the **Box** button on the **Modeling** panel. Start at X = 2, Y = 2 and create a solid box **6** units in the X direction and **4** units in the Y direction. Make it **3** units in height and place it in the center of the screen.

3. Select the **SW Isometric** viewpoint.

MODELING

4. Use the **PRESSPULL** command to remove two rectangular boxes from the first one to form the basic table shown in Fig. 45-12. The legs are **.5** unit wide and the table top is **.75** thick.

> **HINT**   Use the **LINE** command to create 2D profiles on the front and left faces of the box primitive.

MODIFY

5. Rotate the table so that you can see the bottom edge of all four legs clearly.

6. Enter the **CHAMFER** command and select the bottom edge of one of the four table legs. If AutoCAD selects the side of the table rather than the bottom of the leg, type **N** for **Next** and press **ENTER**. When AutoCAD selects the bottom, press **ENTER**.

7. Enter a base surface chamfer distance of **.15**.

8. At the **Specify other surface chamfer distance** prompt, enter **.1**.

9. At the next prompt, pick all four edges that make up the bottom of the leg and press **ENTER**.

Wohlers: Applying
AutoCAD® 2010

IX. Solid Modeling

45. Adjusting 3D Models

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601

AutoCAD chamfers the edges.

10. Repeat Steps 6 through 9 to chamfer the remaining three legs.

Notice that the lines you used to press the boxes to create the table legs now extend beyond the chamfered legs. It is safe to erase these lines now—doing so will have no effect on the model.

11. Erase the lines that extend beyond the chamfered legs.

The bottom of the legs should now look similar to those in Fig. 45-11 on page 593.

12. Enter the **FILLET** command, select any edge of the table top, and enter **.3** for the radius.

13. Enter **C** for **Chain**, select the remaining three edges of the table top, and press **ENTER**.

14. Display the table in the **Conceptual** visual style.

15. Save your work.

The table should look very similar to the one in Fig. 45-11.

MODIFY

Slicing and Shelling a Solid Object

We will make a sphere and use the **SLICE** and **SHELL** commands to create the bowl shown on the table in Fig. 45-11.

1. Enter the **UCS** command and type **W** to make sure you are in the world coordinate system. (The current UCS is also displayed at the bottom of the **ViewCube.**)

2. Select the **Sphere** button on the **Modeling** panel.

3. On the ground plane, create a sphere with a radius of **1**.

4. Select the **Top** viewpoint.

5. Pick the **Planar Surface** button on the **3D Modeling** panel.

6. Create a planar surface on the ground plane that overlaps the sphere on all sides.

7. Return to the **SW Isometric** view.

8. Select the **Slice** button on the **Solid Editing** panel.

9. Pick the sphere as the object to slice and press **ENTER**.

10. Type **S** to specify **Surface** as the slicing option.

11. Pick the planar surface you just created.

MODELING

SOLID EDITING

12. Pick the lower half of the sphere as the solid to keep.

13. Erase the planar surface.

We want to make the hemisphere into a bowl. The fastest way to do this is to shell it out.

14. Pick the **Shell** button from the last flyout menu on the **Solid Editing** panel.

SOLID EDITING

This enters the **Shell** option of the **SOLIDEDIT** command.

15. Select the hemisphere.

Next, we need to remove the top face from the shelling. If we don't, AutoCAD will hollow out the hemisphere, but it will not have an opening.

16. In response to **Remove faces**, pick the edge of the hemisphere.

The top face and the main body face share the same edge, so AutoCAD displays **2 faces found, 2 removed**. We don't want to remove the main body face, so we need to add it back.

17. Enter **A** for **Add**, select the rounded part of the hemisphere (do not select the top edge), and press **ENTER**.

18. For the shell offset distance, enter **.05** and press **ENTER** twice to terminate the command.

AutoCAD produces a shell, or bowl, as shown in Fig. 45-13.

Fig. 45-13

19. Move the bowl to rest on the tabletop, as shown in Fig. 45-11 on page 593. You will need to view the model from several views to be sure you have moved the bowl correctly.

20. Save your work.

Modifying 3D Mesh Objects

3D meshes can be modified in a variety of ways that are impossible with solids. You can also convert a primitive solid to a mesh and convert a mesh to a solid.

1. Use the **solid.dwt** template file to create a new file named **meshmod.dwg.**

2. Use the **BOX** command to create a solid box primitive of any size and shape on the ground plane.

3. Select the **Mesh Modeling** tab of the Ribbon, if it is not already selected.

4. Select the **Smooth Object** button from the **Mesh** panel, select the box, and press **ENTER**.

The solid primitive has been converted to a mesh.

MESH

5. Select the **Smooth More** button, select the mesh object, and press **ENTER**.

The edges and corners of the mesh are now smoother.

MESH

Faces on meshes can be extruded, much like the **PRESSPULL** command extrudes closed areas of a solid.

6. Select the **Extrude Face** button from the **Mesh** panel, press and hold the **CTRL** key while selecting the center face on the top of the mesh, and press **ENTER**. (Make sure the default subobject filter is set to **Face**.)

MESH EDIT

7. Move the crosshairs in the drawing area and watch the dynamic changes to the mesh, then select a point above the mesh to create a protrusion above the object.

8. Select the **Add Crease** button from the **Mesh** panel, press and hold the **CTRL** key while selecting the three faces on the top of the mesh as shown in Fig. 45-14, and press **ENTER** twice. (Make sure the default subobject filter is set to **Face**.)

MESH

As you can see, the edges of the three faces have changed.

9. Select the **Refine Mesh** button, select the mesh object, and press **ENTER**.

MESH

The tesselation of the mesh object has increased. You can also refine individual faces on a mesh, rather than the entire object.

10. Select the **Convert to Solid** button on the **Convert Mesh** panel, select the mesh object, and press **ENTER**.

CONVERT MESH

Fig. 45-14

The modified mesh object is now a solid. As this exercise demonstrates, mesh modeling allows you to create complicated, smooth shapes that would be difficult to model with AutoCAD's solid modeling tools.
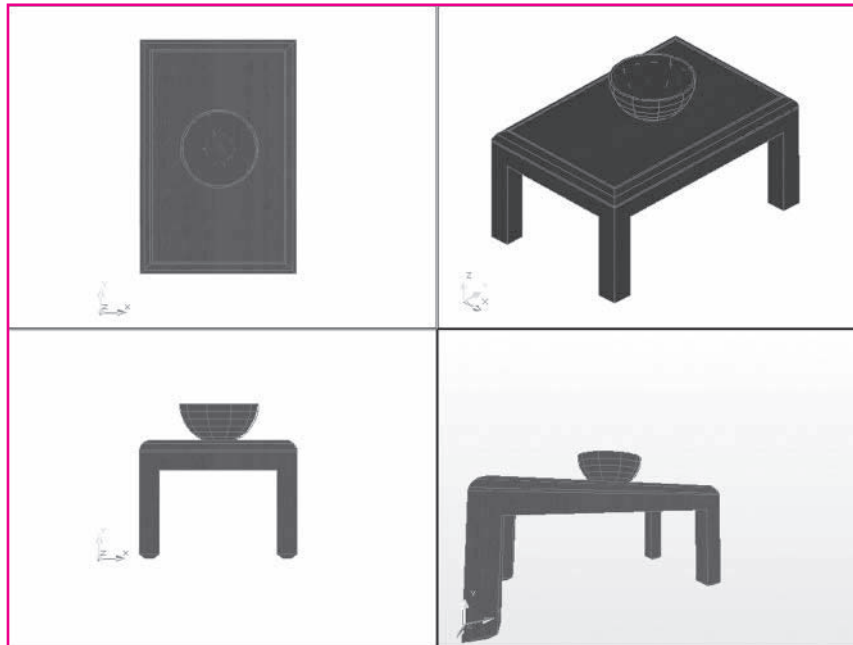
11. Save your work and exit AutoCAD.
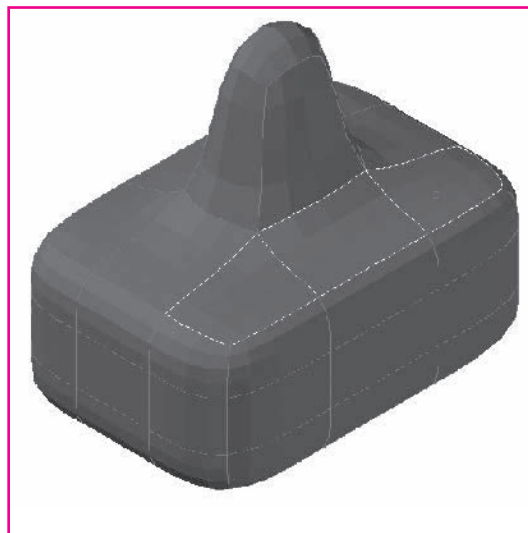
Chapter 45 **Review & Activities**

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Describe the components of the **Move** and **Rotate** grip tools and explain the function of each tool.

2. Describe how the dynamic UCS works and explain why it is useful.

3. Explain why shelling is useful.

4. What are the major differences between the **EXTRUDE** and **PRESSPULL** commands?

5. Explain the purpose of the following mesh modeling commands.

   a. Smooth Object

   b. Smooth More

   c. Refine Mesh

   d. Add Crease

   e. Extrude Face

   f. Remove Crease

   g. Convert to Solid

   h. Convert to Surface

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. How do the **CHAMFER** and **FILLET** commands affect the appearance of solid objects containing outside corners?

2. When a primitive solid is modified using a subobject selection set, the subsequent selection of the solid reveals fewer grip options. Why do you think this occurs?

Review & Activities Chapter 45

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Open **loft.dwg**, a model you created in Chapter 43. We will use it to create a model of a candlestick. Rotate the model 180° to position the square end at the bottom. Draw two circles on the round top face using dynamic UCS and object snap. Make the larger circle the same size as the outer edge of the top face. Then use the **PRESSPULL** command to create a circular rim. Your candlestick should look similar to the one in Fig. 45-15. Save your model as **candlestick.dwg**.

2. Open **ch44baseplate.dwg**. Chamfer the two edges where the upper horizontal and vertical surfaces meet. Use a **.25″** chamfer at **45°**. Create fillets with a radius of **.15** where the upper vertical surfaces meet the plate's horizontal surfaces. Create fillets with a radius of **.15** at the two outer corners. Refer to Fig. 45-16 for specific locations.

Fig. 45-15

CHAMFER .25X45°
2 PLACES

FILLET R.15
2 PLACES

FILLET R.15
2 PLACES
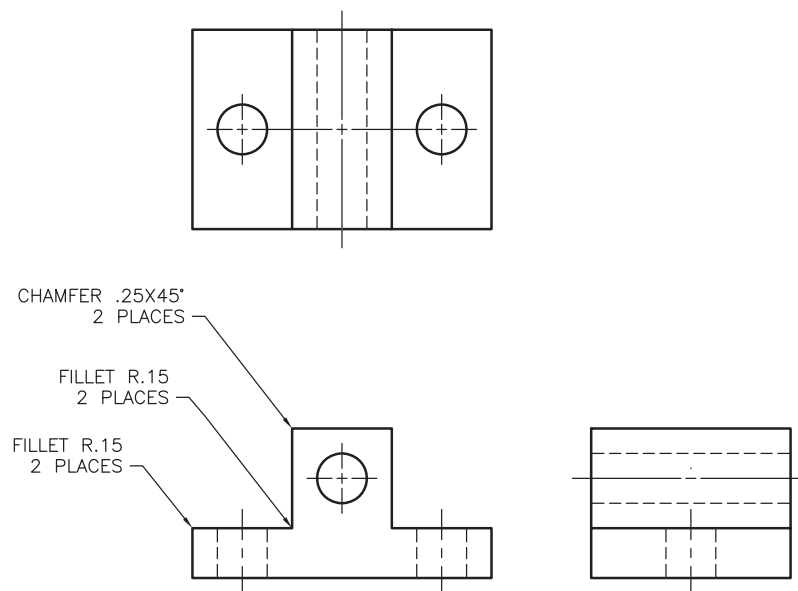
Fig. 45-16

Figure 45-16 prepared by Gary J. Hordemann, Gonzaga University
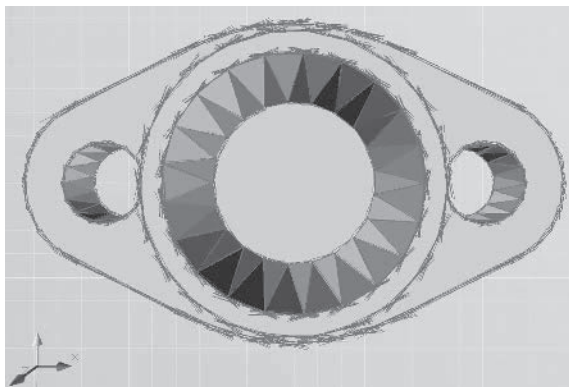
Continued

Chapter 45: Adjusting Solid Models   **599**

Chapter 45 **Review & Activities**

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Use the **PRESSPULL** command to model the plate shown in Fig. 45-17. The dimensions are given in the orthographic projections in Fig. 45-18.

Use the **FILLET** command to create a fillet where the boss joins the main part of the plate. After you have finished the model, produce an isometric view in the **Conceptual** visual style. The drawing should look like the one in Fig. 45-17.
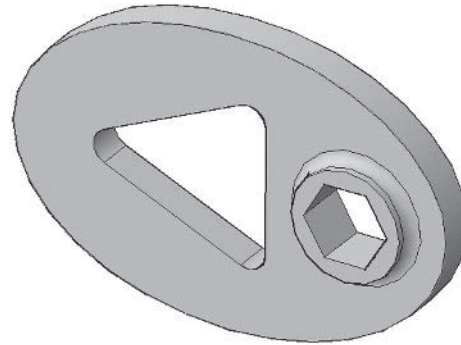
Fig. 45-17
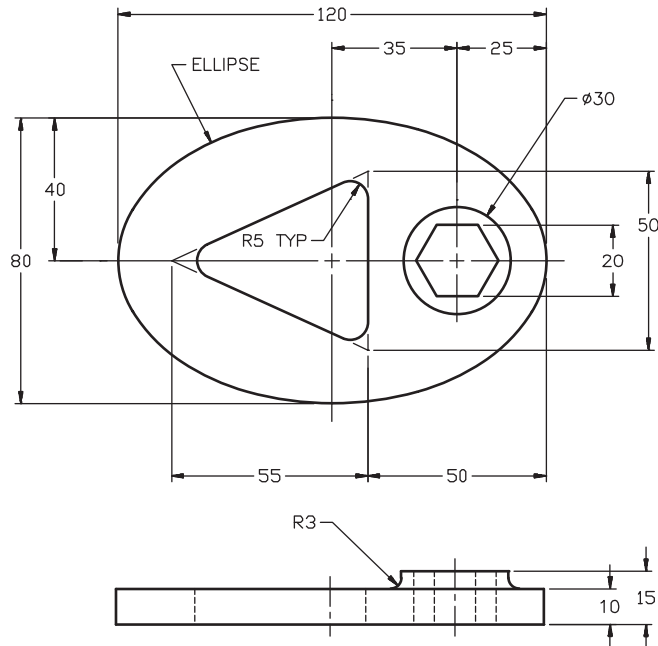
Fig. 45-18

Problem 1 courtesy of Gary J. Hordemann, Gonzaga University

2. Using the **SUBTRACT** and **EXTRUDE** commands, model the tube bundle support shown in Fig. 45-19. The dimensions are given in the orthographic projections shown in Fig. 45-20.

The rounded top can be produced in several ways. One way is to use the **REVOLVE** command to create a piece to be removed from the original extrusion using the **SUBTRACT** command. A second way is to use the **REVOLVE** command to create a rounded piece without holes; then use the **INTERSECT** command to obtain the common geometry. Try both ways. Display the model using an isometric viewpoint and the **Conceptual** visual style.

Fig. 45-19
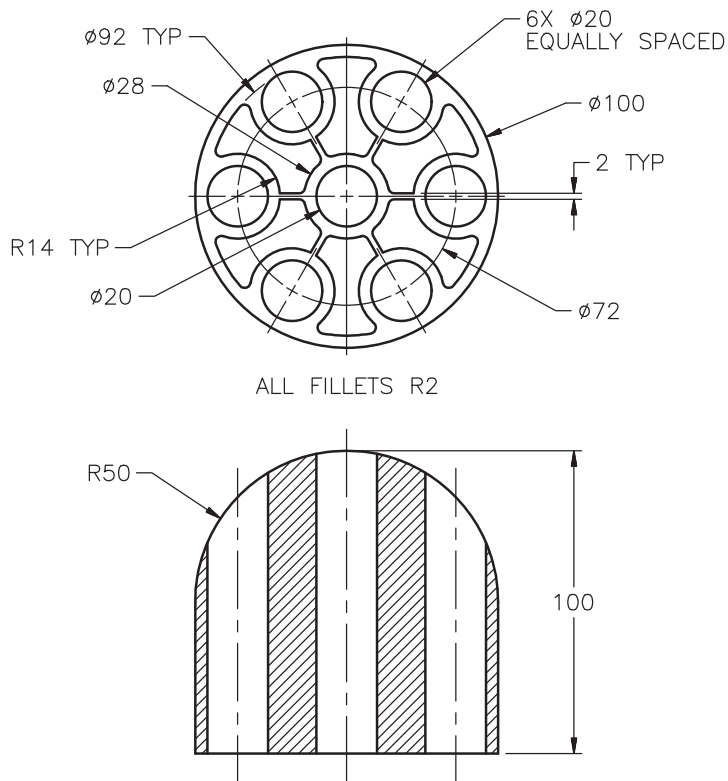
Fig. 45-20

Problem 2 courtesy of Gary J. Hordemann, Gonzaga University

Continued

Chapter 45: Adjusting Solid Models   601

Chapter 45 **Review & Activities**

3. Three orthographic views of a T-swivel support are shown in Fig. 45-21. Model the piece as a solid. Use the **FILLET** command to fillet all of the indicated edges.
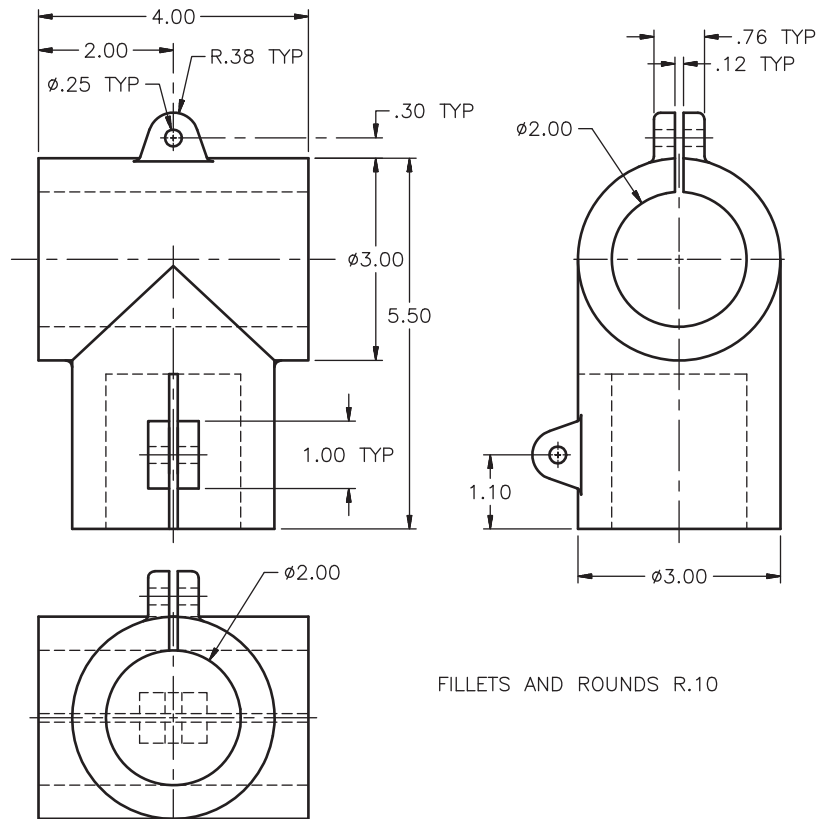
Fig. 45-21

Courtesy of Gary J. Hordemann, Gonzaga University

CAD Product Manager

Not too long ago, CAD drawings and models were created and used only by CAD departments. However, companies today share CAD files among multiple departments and locations. Improved drawing collaboration tools allow many people to work on these files. With CAD files being used for everything from design to marketing to computerized manufacturing, controlling these digital assets can quickly become a nightmare. This is especially true in large companies, which may need to track hundreds of thousands of CAD files.

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Software Solutions

So how do they do it? The CAD product manager's job is to keep track of everything from who needs which files to revision records. But how can one person—or even an entire department—track thousands of digital files accurately? Many managers are turning to product lifecycle management (PLM) software. This software consists of a huge relational database that can track digital models, assemblies, development costs, rights, creation dates, usage, associated stock numbers, and just about any other information a company needs to track.

A Different Skill Set

In addition to being familiar with the CAD software being used in the company, a CAD product manager must be familiar with the needs of every department that uses the software and all of the products the company produces. To use PLM software, the manager should also have a working knowledge of databases, including relational databases. A logical mind and an excellent sense of humor may also come in handy!

▶ Career Activities

1. Make a list of the items you think would need to be tracked for each CAD file created by the company.

2. Research PLM software on the Internet. What options are available for companies of various sizes?

Chapter **46**

Documenting Solid Models

Objectives

- Create a full section from a solid model
- Create 2D flatshot views of a solid model
- Create an orthographic projection drawing sheet of a solid model

Vocabulary

live sectioning
section plane indicator

In this chapter, you will learn to create orthogonal, isometric, and section views from a solid model of a pulley. The **SECTIONPLANE** and **FLATSHOT** commands make this task quite simple. You will then create a multiview drawing of the pulley.

Creating the Solid Model

1. Start AutoCAD and use the **solid.dwt** template file to create a new drawing named **pulley.dwg**.

2. Select the **Top** viewpoint.

3. Set snap to **.25** and grid to **.25**.

4. Set the **ISOLINES** system variable to **30**, and select the **Realistic** visual style.

5. Create the polyline shown in Fig. 46-1 using the following information.

   - The lines on the grid are spaced **.25** apart. All polyline endpoints lie on the grid, so use it to produce the object accurately.

Wohlers: Applying
AutoCAD® 2010

IX. Solid Modeling

46. Documenting Solid
Models

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611
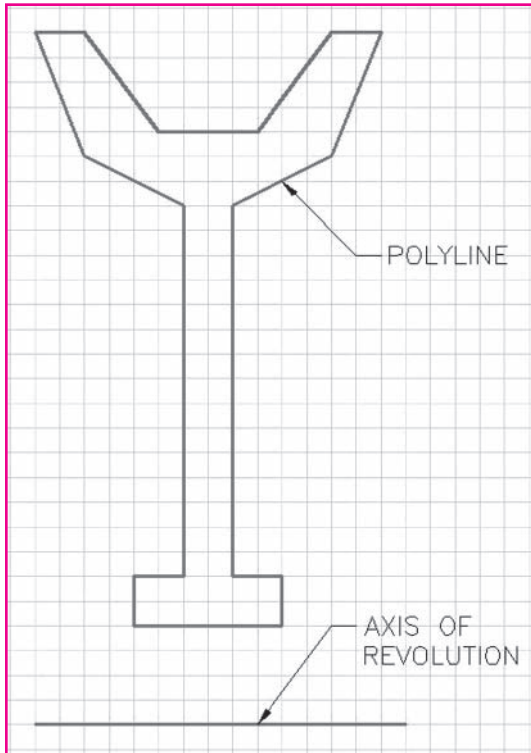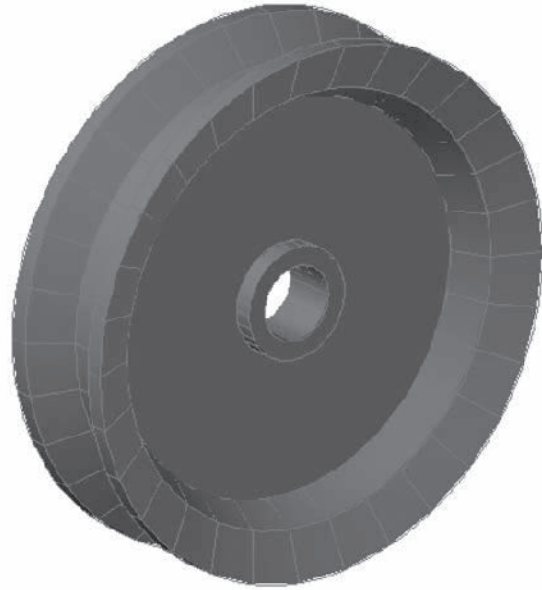
Fig. 46-1

Fig. 46-2

- Produce the polyline in the top right quadrant of the drawing area.
- The axis of revolution is a line of arbitrary length.

6. Using the **REVOLVE** command, revolve the polyline to produce the pulley.

7. Select an isometric viewpoint. Your view should be similar to the one in Fig. 46-2.

8. Erase the line used for the axis of revolution and save your work.

Creating a Full Section

A full section of a part is a view that slices all the way through the part, showing the interior of the part as it appears at that slice, or cross section. The **SECTIONPLANE** command creates a full cross section of a solid model.

1. Create a new layer named **Views**, assign the color blue to it, and make it the current layer.

2. Pick the **Section Plane** button on the **Section** panel.

3. Enter **O** to specify the **Orthographic** option.

SECTION

4. In response to **Align section to**, enter **F** for **Front**.

A transparent plane, called a **section plane indicator**, appears and a full section of the pulley is created, as shown in Fig. 46-3. The **SECTIONPLANE** command automatically places the section plane at the center of all the 3D objects in the drawing area. This is why the section plane cuts through the exact center of the pulley.

5. Pick the section plane indicator.

Notice that the section plane indicator has grips, as shown in Fig. 46-3. Using the grips, you can move, rotate, and stretch the section plane, as well as specify which half of the pulley is hidden.

6. Experiment with moving and selecting the section plane grips; then undo all the changes you have made until the section plane returns to the original plane, as shown in Fig. 46-3.

7. With the section plane indicator selected, right-cick to view the shortcut menu.

8. Pick the **Live section settings...** option to view the **Section Settings** dialog box.

Fig. 46-3

The **Section Settings** dialog box allows you to specify many settings for sectioning, including **live sectioning** (which provides realtime views while manipulating the grips) and the settings for the 2D or 3D sections you create.

9. Explore the settings in the **Section Settings** dialog box. Note that you can change the hatch pattern in a 2D or 3D section.

10. Pick **Cancel** to close the **Section Settings** dialog box.

11. Right-click to view the shortcut menu and select the **Generate 2D/3D section...** option.

12. In the **Generate Section/Elevation** dialog box, select **2D Section/Elevation** and pick the **Create** button.

13. Pick an insertion point to the left of the pulley and accept the default scale and rotation angle.
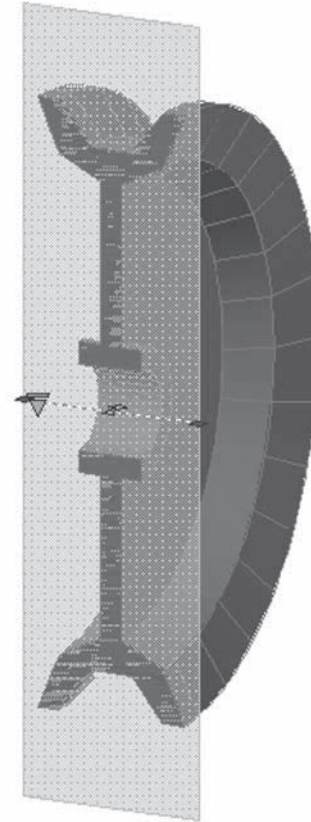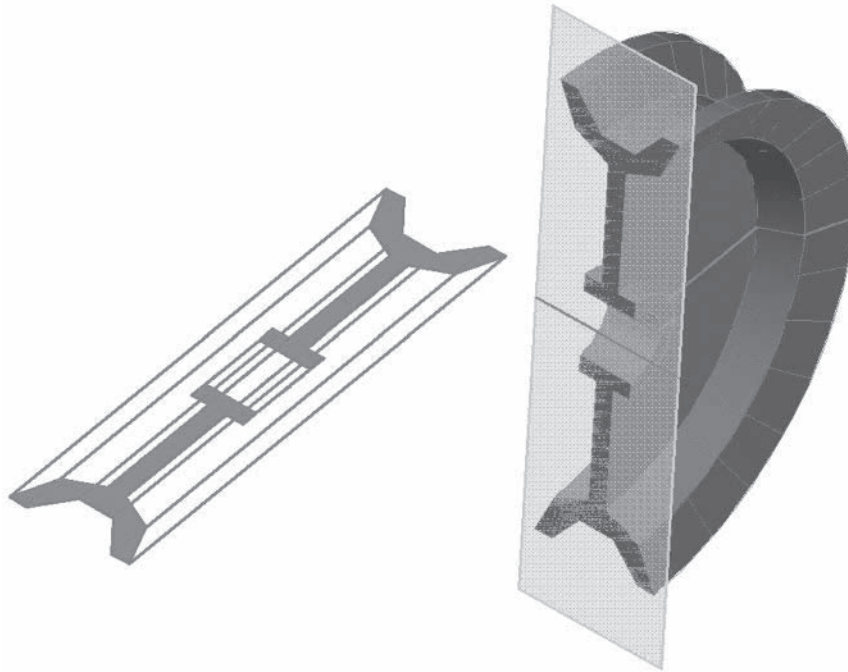
Fig. 46-4

A 2D section of the pulley at the section plane is created on the ground plane, as shown in Fig. 46-4.

14. Erase the section plane.

15. Save your work.

Creating a 2D Representation

The **FLATSHOT** command creates a 2D, or "flattened," representation of all the 3D objects in the current view. The view can be an orthogonal view, predefined isometric view, or any other view that you create using 3D orbit. The 2D representations are created as blocks (as was the 2D section you have already created). The 2D objects are placed on the XY plane of the current UCS.

1. Switch to the **Top** view.

2. Select **WCS** in the menu below the **ViewCube** to make WCS the current UCS.

3. Pick the **Flatshot** button on the expanded **Section** panel.

SECTION

The **Flatshot** dialog box appears. It provides options for the destination and the representation of the lines in the 2D flatshot.

4. In the **Flatshot** dialog box, make sure the **Show** box in the **Obscured lines** area is unchecked and pick **Create**.

A 2D view of the top of the pulley appears, just as it would look when viewed from above, with hidden lines removed.

Author is this button correct?
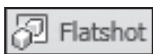
SOLID EDITING

5. Pick a location to the left of the full section view and accept the default values for scale and rotation angle.

6. Switch to the **Right** view.

7. Make sure WCS is the current UCS.

8. Pick the **Flatshot** button.

9. In the **Flatshot** dialog box, make sure the **Show** box in the **Obscured lines** area is unchecked and pick **Create**.

Because this view is perpendicular to the ground plane, we will need to specify the Z location of the new flatshot.

10. In response to **Specify insertion point**, snap to the center point of the pulley. Accept the default values for scale and rotation angle.

11. Pick the **NE Isometric** view and repeat the steps to create an isometric flatshot positioned on the ground plane.

12. Pick the **Top** view and use the **MOVE** command to arrange the 2D objects so they do not overlap. When you are finished, your drawing should look similar to the one in Fig. 46-5.

13. Save your work.

Creating a Drawing

As you may recall, orthographic projection is a method of creating views of an object that are projected at right angles onto adjacent planes. With the **SECTIONPLANE** and **FLATSHOT** commands, we have already created 2D orthogonal and isometric projections of the pulley, as well as a full section view. Now we will create a drawing of those views. This process is similar to the multiple viewport drawing you created in Chapter 26, but since we have all the views projected onto the ground plane, we only need a single viewport.

1. Freeze layer **Objects** so that only the four 2D objects are visible.

2. Pick the **Layout1** tab at the bottom of the drawing area.
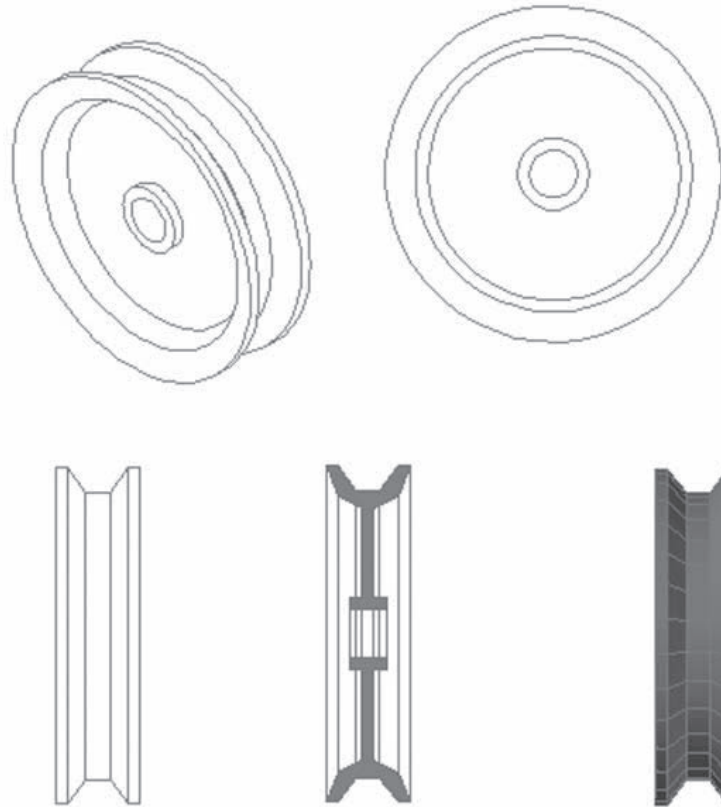
Fig. 46-5

AutoCAD switches to paper space, and the paper space icon replaces the coordinate system icon. The four 2D objects representing the views are present, but they are not positioned or aligned in the conventional style for orthogonal projection.

3. Pick the **Model** tab at the bottom of the drawing area to return to model space.

4. Use the **MOVE** command to arrange the four 2D objects so that the front view is in the lower left, the section view is on the lower right, and the isometric view is in the upper right.

5. Rotate the 2D top view 90° and place it above the front view.

6. Use object snap and ortho to align the top view and the section view to the front view precisely.

When you are finished, your drawing area should look similar to the one in Fig. 46-6 on page 610.
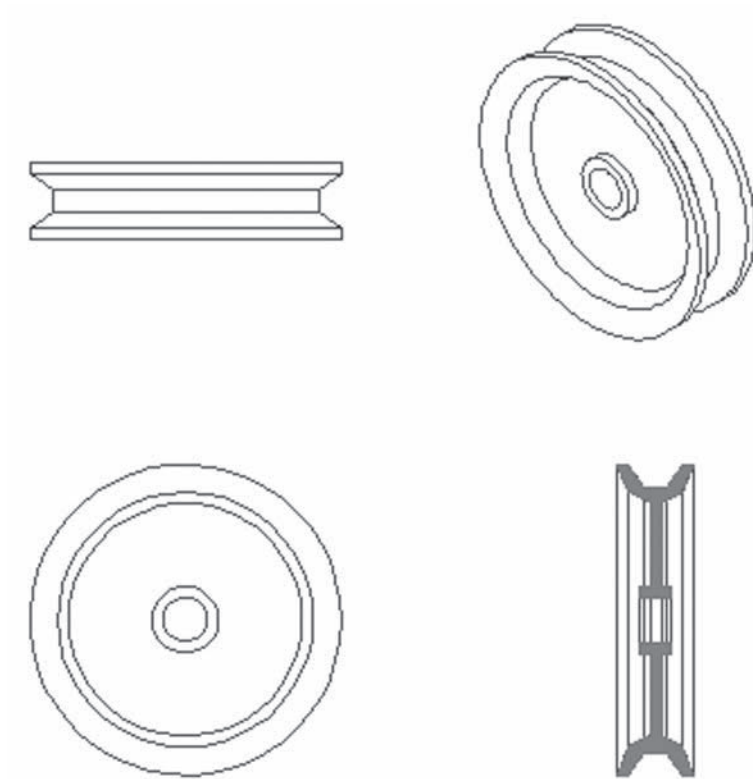
Fig. 46-6

7. Pick the **Layout1** tab to switch to paper space.

The drawing sheet now looks correct, with views properly positioned, oriented, and aligned. As you can see, the **SECTIONPLANE** and **FLATSHOT** commands make the production of multiview drawings fast and simple.

8. If necessary, resize and move the viewport to center the objects on the drawing sheet.

9. Plot the drawing.

10. Save your work and exit AutoCAD.

Chapter 46 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Explain the purpose of the following solid modeling commands.

    a. **SECTIONPLANE**

    b. **FLATSHOT**

2. What is a section plane indicator?

3. Describe each of the grips on the section plane indicator and explain the function of each.

4. What type of object is the 2D representation that results from the **FLATSHOT** command?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Investigate the other options in the **SECTIONPLANE** command. What option would you use to create a jogged (nonplanar) section of an object?

2. Consider the advantages and disadvantages of creating all views in model space and using one viewport in paper space. Can you think of a situation in which it would be preferable to use multiple viewports?

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Select any of the solid models you created in the previous chapters, or create a new one of your own. Then create front, top, right, isometric, and full section views of the model.

Continued

C h a p t e r 46 **Review & Activities**

2. Draw a solid model of the bushing and insert shown in Fig. 46-7. Then create a full sectional view of each to describe them completely. Change the hatch pattern for the insert to ANSI31. Save the drawing as **ch46bushing.dwg**.
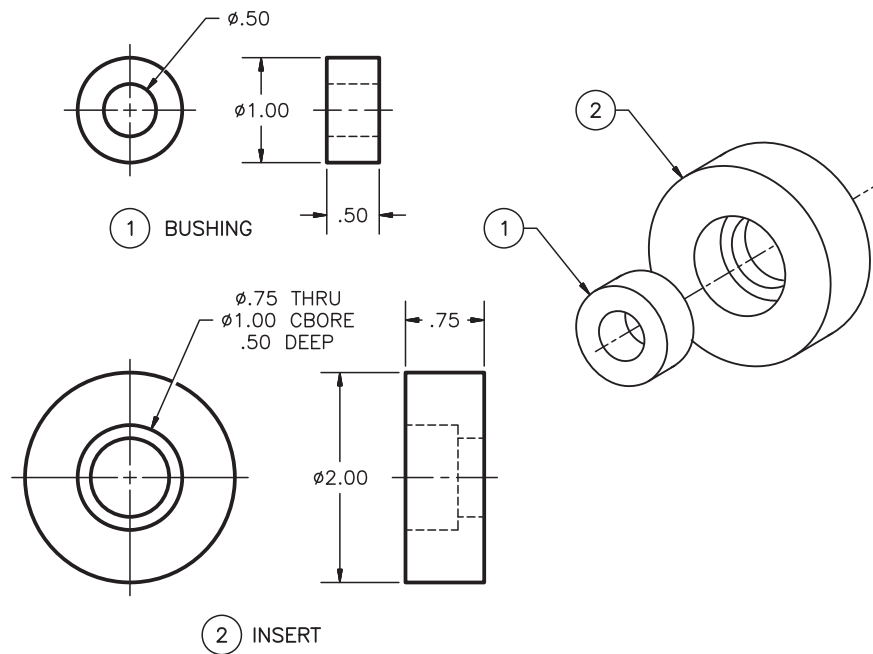
Fig. 46-7

Adapted from a drawing by Gary J. Hordemann, Gonzaga University

3. You are given a file containing the solid model and sectional view of the bushing and insert (problem 2 above). The manufacturing department needs a set of 2D working drawings. Create 2D front and side views of the bushing and insert. Save the drawing as **ch46bushing2.dwg**.

Review & Activities Chapter 46

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Open **bumper.dwg**, which you created in Chapter 43. Use the **FLATSHOT** and **SECTIONPLANE** commands to create a drawing in paper space that is similar to Fig. 43-19 on page 573. Omit the dimensions and notes. Save your drawing as **ch46bumper.dwg**.

2. A solid model of the rocker arm shown in Fig. 46-8 is needed, along with a full section and a complete set of 2D drawings, for the design and manufacturing departments. Provide the necessary drawings. Save the drawings using names of your choice.
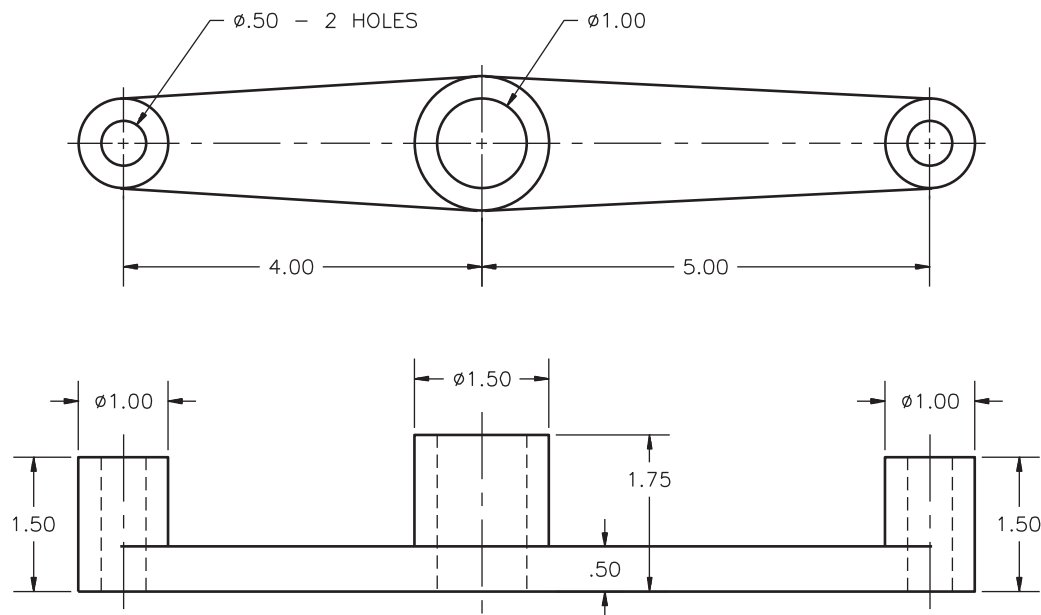
Fig. 46-8

Courtesy of Gary J. Hordemann, Gonzaga University

$$C\,h\,a\,p\,t\,e\,r \quad 47$$

Visualization and Navigation

Objectives

- Apply AutoCAD's visual styles to a solid model
- Render a model with user-defined lighting and materials
- Place a camera in a model and use it to define a viewport
- Navigate a model using AutoCAD's walk and fly modes

Vocabulary

glyph
opacity
rendering

AutoCAD provides many tools to enhance the appearance of a model. These visualization tools help to produce a photorealistic representation, complete with realistic lighting, shading, and shadows. You can also assign a material specification to objects, giving them the patterns and textures of actual materials such as wood, concrete, or plastic. In this chapter, you will learn to use these visualization tools.

You are already familiar with many of AutoCAD's tools for navigating a model. You have moved around the drawing area by zooming, panning, and orbiting. In this chapter, you will learn to use an additional navigation tool: the camera.

Wohlers: Applying
AutoCAD® 2010

IX. Solid Modeling

47. Visualization and
Navigation

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621

Visualization Tools

Let's start by taking a look at the visualization tools AutoCAD provides.

1. Start AutoCAD and open the drawing named **table.dwg**, which you created in Chapter 45. Save the file as **visual.dwg**.

2. Select the **3D Modeling** workspace and close all palettes and floating toolbars.

3. Create two new layers named **Table** and **Bowl**. Make the **Table** layer gray and the **Bowl** layer blue.

4. Move the table to layer **Table** and the bowl to layer **Bowl**. Freeze all other layers so that only the two solids are visible.

5. Select the **SW Isometric** viewpoint and the **Conceptual** visual style.

6. Select the **Render** tab on the **Ribbon,** then pick the **X-Ray** button on the **Visual Styles** panel.

VISUAL STYLES

X-ray mode allows you to view faces on objects transparently. The value of transparency, or **opacity**, is controlled by the **VSFACEOPACITY** system variable. A value of **100** makes objects opaque; a value of **0** makes them transparent. The default value is **60**.

7. Pick the **X-Ray** button again to turn **X-ray** mode off.

8. On the expanded **Visual Styles** pane, click the arrow next to the **Warm-Cool Face Style** button on the **Visual Styles** panel to see the additional choices of face styles.

Warm-Cool Face Style is another name for the **Conceptual** visual style. The other choices are **No Face Style** (the **3D Hidden** visual style) and **Realistic Face Style** (the **Realistic** visual style).

9. View the table and bowl in all face styles and then return to the **Warm-Cool Face Style**.

The appearance of curves on the faces of objects is controlled by the **Facet Lighting, Smooth Lighting,** and **Smoothest Lighting** buttons.

10. Pick the **Smooth Lighting** button, the **Smoothest Lighting** button, and then the **Facets Lighting** button on the expanded **Visual Styles** panel. Observe the appearance of the bowl and the fillets on the rounded table edge.

Modifying Edge Settings

Now focus on the **Edge Effects** panel.

1. Click the arrow below the **Facet Edges** button and select the **No Edges** button from the flyout menu. Observe the effect on the model.

2. Select the **Isolines** button.

In **Isolines** mode, AutoCAD displays edges on curved objects. The system variable **ISOLINES** controls the number of edges. After you change the **ISOLINES** value, you need to regenerate the screen to display the change.

3. Enter the **ISOLINES** system variable at the keyboard and enter a new value of **50**.

4. Type **REGEN** and press **ENTER** to regenerate the screen.

Your model should now look similar to the one in Fig. 47-1.

5. View the model in all combinations of visual styles and edge modes.

6. Return to the **Warm-Cool Face Style** and **Isolines** edge mode.

7. Set the **ISOLINES** system variable to **10** and regenerate the screen.

8. Pin the expanded **Edge Effects** panel open.
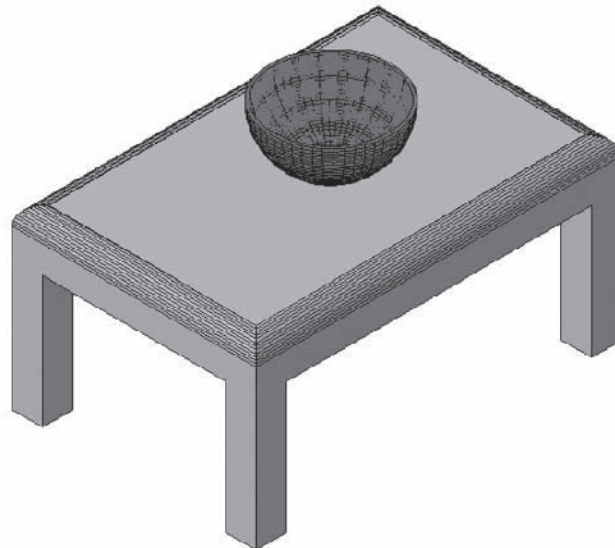
Fig. 47-1

The remainder of the **Edge Effects** panel controls the appearance of edges. **Edge overhang** causes edge lines to overhang their endpoints. **Edge jitter** increases the number of edge lines and makes them irregular. When edges are overhung and jittered, it gives the drawing a hand-sketched, pencil-drawn appearance. **Silhouette edge width** thickens the edges that define the silhouette of the objects. **Obscured edges** and **Intersection edges** are active only in the **Facet edges** mode. As their names imply, they control the display of obscured (hidden) and intersection edges.

9. Experiment on your own with all the **Edge Settings** buttons and sliders. Attempt to duplicate the hand-drawn look of Fig. 47-2.

10. Turn off all edge effects by toggling the **Edge Settings** buttons off.

11. Unpin the expanded **Edge Effects** panel.
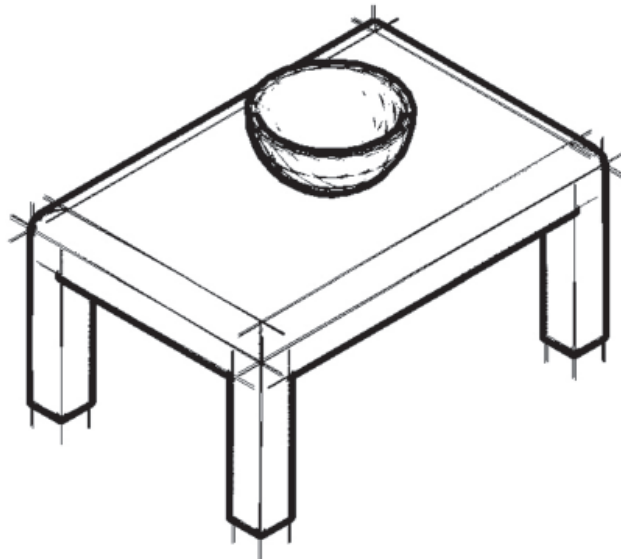
Fig. 47-2

Lighting Options

AutoCAD applies default lighting to models when they are shaded. Default lighting illuminates all faces on objects evenly and follows your viewpoint as you move around in the model. You can turn default lighting off and apply different lighting options to the model. These options include user-defined lights and lighting from the sun.

1. On the expanded **Lights** panel, pick the **Default Lighting** button to toggle default lighting off.

Default Lighting

LIGHTS

2. Click the arrow below the **Shadows Off** button and select the **Ground Shadows** button from the flyout menu.

Shadows appear below the table on the ground plane.

SUN & LOCATION

3. Pick the **Sun Status** button on the **Sun & Location** panel to toggle the sun lighting mode.

4. Move the **Sun date** and **Sun time** sliders left and right and notice the effect on the model.

The properties of sun lighting are controlled in the **Sun Properties** palette. You can open the **Sun Properties** palette by picking the arrow in the lower right corner of the **Sun & Location** panel. The geographic location of the model's sun lighting can be changed from the **Sun Properties** palette or by picking the **Set Location** button on the **Ribbon**.

5. Pick the arrow in the lower right corner of the **Sun & Location** panel to display the **Sun Properties** palette.

Set Location

SUN & LOCATION

6. Change the date to **July 4, 2010**, and change the time to **2:00 PM**.

7. Pick the **Set Location** button, select the **Enter the location values,** and enter a **Latitude** of **37.7950 North** and a **Longitude** of **122.3940 West** (San Francisco).

Your model should look similar to the one in Fig. 47-3.

SUN & LOCATION

8. Pick the **Sun Status** button to toggle the sun lighting mode off.
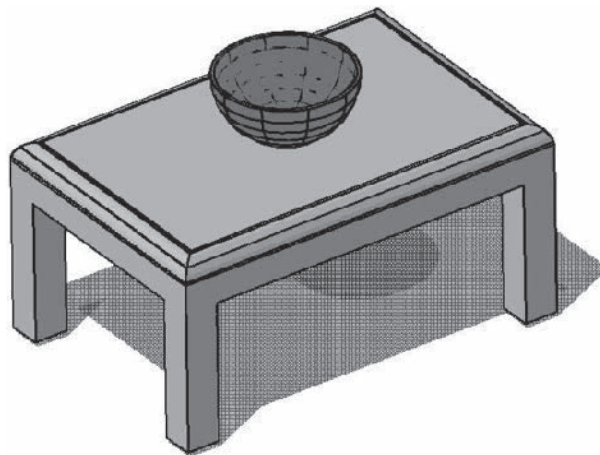
Fig. 47-3

Another option to default lighting and sunlight is user-created lights. We will now create a spotlight.

9. Select the **Realistic** visual style.

10. Make sure dynamic input is off and the UCS is the world coordinate system.

11. Pick the **Spot** button from the flyout menu below the **Create Light** button on the **Lights** panel.

This enters the **SPOTLIGHT** command.

12. For the spotlight's source location, enter **1, 7, 6**.

13. For the target location, pick the front right corner of the table top, as shown in Fig. 47-4.

Endpoint
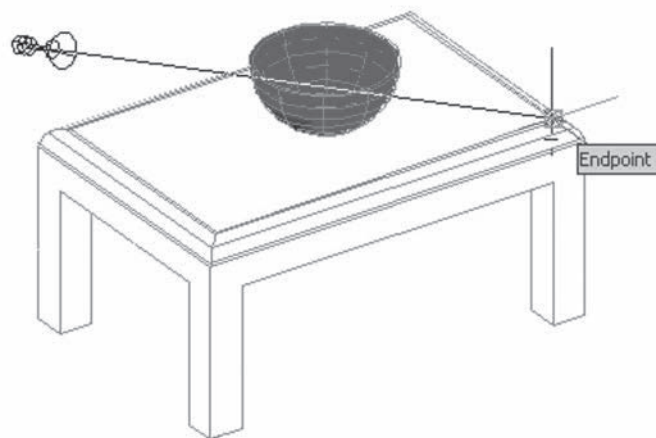
Fig. 47-4

14. Enter **X** to exit the **SPOTLIGHT** command.

A spotlight glyph appears on the screen and the model is now shaded by the spotlight. A **glyph** is an icon that has a specified location in the drawing area and represents one of AutoCAD's tools, such as a spotlight or camera.

15. Pick the light glyph and use the 3D grips to position the light so that the bowl is highlighted on the tabletop.

16. Adjust the **Brightness** and **Contrast** sliders on the expanded **Lights** panel and notice the effect on the model.

17. Save your work.

Selecting Materials and Rendering a Model

Now we will assign materials to the objects in the model. **Rendering** combines visual styles, lighting, materials, and shadows to create a photorealistic image of the model.

1. Select the **Tool Palettes** button from the **Palettes** panel of the **View** tab.

2. Right-click in the title bar of the **Tool Palettes** and select **Materials** from the shortcut menu.

The **Tool Palettes** window changes to a **Materials** palette, with multiple tabs of materials available.

3. Select the **Woods and Plastics – Material Samples** tab.

4. Select the **Woods – Plastics.FinishCarpentry.Wood.Maple** material and move the cursor into the drawing area.

The cursor changes to a pick box with an artist's brush.

5. Pick the table solid.

6. Select the **Metals - Materials Sample** tab.

7. Select the **Metals.Ornamental Metals.Copper** material and pick the bowl solid.

The appearance of the solids has not changed much, but when the model is rendered, you will see a difference.

8. On the **Render** tab, pick the **Render** button on the **Render** panel.

RENDER

A new window, called the **Render** window, appears on the screen in front of the AutoCAD window. The rendered image of the table and bowl are displayed, as shown in Fig. 47-5, complete with your user-defined spotlighting. Notice the realistic shadows and the reflection of light from inside the bowl.

The table looks like a wood table. The **Render** window displays an information pane on the right and a file management pane on the bottom. You can save or delete a rendered image in this pane.

> **NOTE** Your rendering may look different, depending on the location of your spotlight.
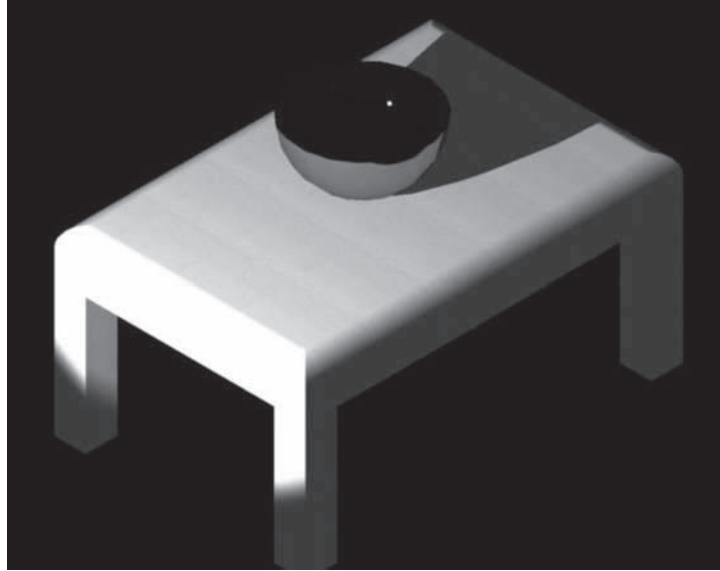
Fig. 47-5

9. Right-click **Output File Name** at the bottom of the **Render** window (the default name for the file will be **visual-Temp 000** or a similar name) and pick **Save...** from the shortcut menu.

10. Save the rendered image in your drawing folder as a **JPEG** file named **Render.jpg**, using the default **JPEG** image options.

11. Experiment on your own with rendering the model.

   - Use different lighting sources, materials, and viewpoints.
   - Select the spotlight glyph and move its grips to change the spotlight's location and cone size.
   - Render the model using the **Presentation** preset, which you can select at the top of the **Render** panel.

12. Close the Tool Palettes window and save **visual.dwg**.

Navigation Tools

We will now take a look at AutoCAD's other navigation tools.

1. If the **Sun Status** is currently off, turn it on.

2. Select the **SteeringWheel.**

3. Use the **SteeringWheel** to zoom, pan, and orbit in the drawing area.

> **HINT**
>
> You can also use shortcuts to navigate. Zoom by rolling the center (wheel) mouse button. Pan by holding the center mouse button down and dragging the cursor in the drawing area. Enable 3D orbit by holding the **SHIFT** key and center mouse button down simultaneously and dragging the cursor in the drawing area.

4. Close the **SteeringWheel** and select the **SE Isometric** viewpoint.

5. Reopen the **SteeringWheel** and click on the **Look** button.

6. Click and hold the left mouse button and move the cursor in the drawing area.

The **Look** navigation mode is different than the **Pan** mode, as you can see. Panning changes the viewpoint on a flat plane, maintaining a constant distance and angle from the objects in the drawing. **Look** swivels the viewpoint around a point in 3D space, similar to moving a camera around on a fixed tripod.

Creating a Camera

We will now create a custom camera, which is a saved viewpoint in a 3D drawing.

1. Select the **SW Isometric** viewpoint.

2. Make sure dynamic input is off and the current UCS is the world coordinate system.

3. Enter **CAMERA** at the keyboard.

This enters the **CAMERA** command. A camera glyph is attached to the crosshairs in the drawing area.

4. For the camera position, enter **3, -3, 2.5**.

5. For the target position, pick the right rear corner of the tabletop, as shown in Fig. 47-6.

6. Enter **N** and name the new camera **MyCamera**.

7. Enter **X** to exit the camera command.

8. Select the camera glyph.

When you select the camera glyph, dotted lines appear that define the camera's lens length and field of view. Grips are also displayed. The grips allow you to change the camera position, target position, and lens length (or field of view).
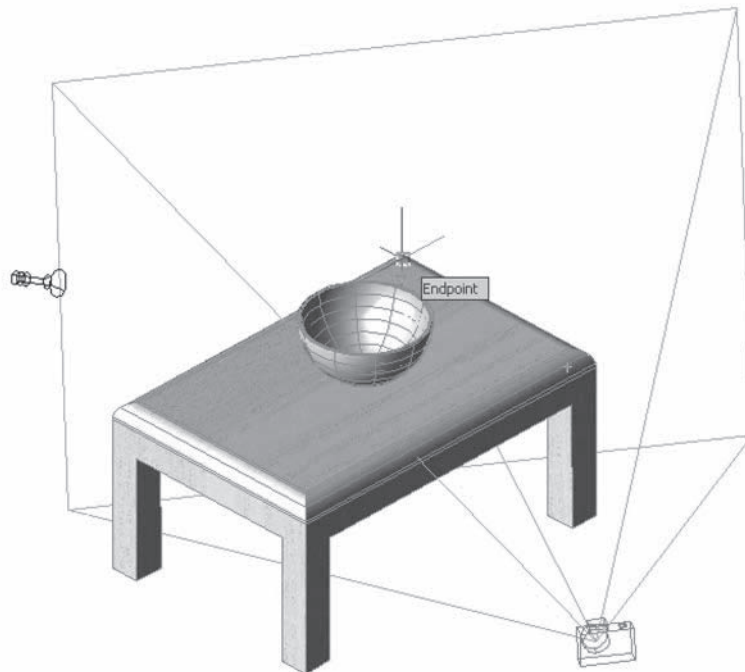
Fig. 47-6
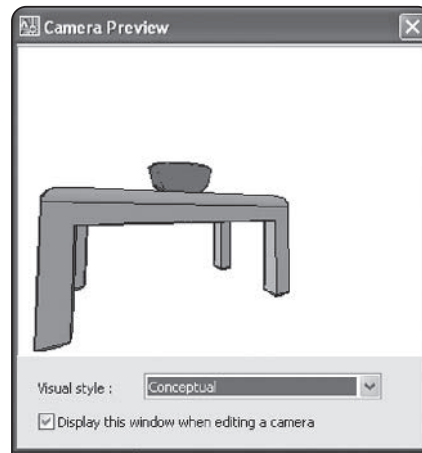
The **Camera Preview** dialog box also appears. The preview pane allows you to see the model through the lens of the camera.

   9. In the **Camera Preview** dialog box, select the **Conceptual** visual style.

 10. Click and drag one of the grips to increase the camera's field of view until the entire table is displayed in the preview pane, as shown in Fig. 47-7.

 11. Pick the **Four: Equal** option from the **Viewport Configurations** drop-down menu in the **Viewports** panel on the **View** tab.

Four viewports are displayed. We will keep an isometric, and make one a front view and another a top view. We will change the fourth to the MyCamera viewpoint.

 12. Click in the lower right viewport to make it active.

 13. Pick the arrow below **Views** in the **Views** panel to select a new view.

Notice that **MyCamera** has now been added to the list of view options.

 14. Pick the **MyCamera** view for the lower right viewport.

The lower right viewport now displays the view from **MyCamera**.

15. Create the other views and improve the appearance of all four views.

- Create a front view and a top view in the two views on the left side.
- Center and zoom the objects in all four viewports.
- Align the top view with the front view.
- Hide the camera glyph, the spotlight glyph, and the **ViewCube.**

Your drawing area should now look similar to the one shown in Fig. 47-8.

16. Render the **MyCamera** viewport.

17. Save and close **visual.dwg** and exit AutoCAD.

Fig. 47-8

Chapter 47 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Describe each of the following AutoCAD visualization modes.
   a. **X-ray**
   b. **Edge overhang**
   c. **Edge jitter**
   d. **Silhouette edges**
   e. **Ground shadows**
   f. **Sun Status**

2. Identify the two glyphs you created in this chapter and explain the function of each.

3. List the nine navigation modes available in AutoCAD. (*Hint:* They are listed in the shortcut menu available during any navigation command.)

4. Describe how to set up ground shadows for January 1 and high noon in New York, NY.

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. What is the difference between the **Orbit** and **Look** modes of the **SteeringWheel?** Can you think of an instance in which **Look** would be more useful than **Orbit** and vice versa?

2. Review all the options available in the various tabs of the **Materials** tool palette. If you were designing the interior of a house, what material would you select for a tile floor? for a leather couch? for a copper sculpture?

Review & Activities Chapter 47

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Open the model **presspull.dwg**, which you created in Chapter 45. Light the model with sunlight for **January 1, 2009,** at **2:00 PM** for **Tulsa, OK**. Assign the material **Ornamental Metals.Brass.Satin** to the solid. With the grid on, your model should look similar to the one in Fig. 47-9.
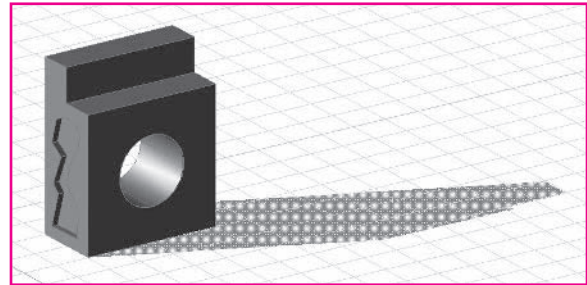
Fig. 47-9

2. Open the model **composite.dwg**, which you created in Chapter 44. Create a camera at the top center of the support cylinder, with a target directly down into the cylinder, as shown in Fig. 47-10. (Snap to circle centers to locate the camera and the target.) Name the camera **CenterCam**.

Use the camera's grips to move the camera up several inches above the solid. (Constrain the movement to the Z axis only with the **Move** grip tool.) Then expand the camera's field of view to encompass all of the model, as shown in Fig. 47-11 on page 630.

Finally, view the solid from camera **CenterCam**. Select the **Realistic** visual style, turn **Facets** on, and apply some edge overhang and jitter. Your drawing area should look similar to that shown in Fig. 47-12 on page 630. Save the model as **CompositeCam.dwg**.

Fig. 47-10

Continued

Chapter 47: Visualization and Navigation **627**

Chapter 47 **Review & Activities**

Fig. 47-11

Fig. 47-12

Review & Activities Chapter 47

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. Start with the model **table.dwg** and create the simple hotel room shown in Fig. 47-13. The following information will help you create each solid in the model.

   - Create the walls using the **POLYSOLID** command. They are **10′** high and **.25′** wide. The room measures **18′ × 30′**.
   - Create the door and windows using the **PRESSPULL** command. The door is **3′ × 7′**. The two matching windows are of arbitrary size.
   - Create the floor using the **PLANESURF** and **THICKEN** commands. The value for thickness is **–1**.
   - The bed is a box measuring **6′ × 7′ × 2.5′**.
   - If necessary, scale, rotate, and move the table and bowl to match their appearance in Fig. 47-13.

   Put each object on its own layer and give each layer a unique color. Save your model as **room.dwg**.
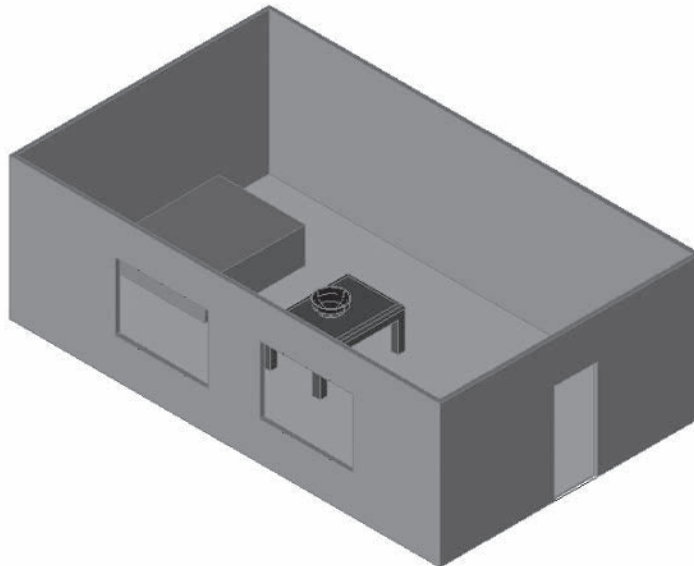
Fig. 47-13

Continued

Chapter 47 **Review & Activities**

2. Apply the following materials to the objects in **room.dwg**:

Walls: **Plaster.Stucco.Troweled.White**

Floor: **Wood.Hardwood1**

Bed: **Fabrics.Velvet.Red**

Table: **Finishes.Wax.Varnish**

Bowl: **Glazing.Glass.Mirrored**

Render the model with late afternoon sunlight, as shown in Fig. 47-14.

Finally, create a spotlight in the top corner of the room, to the right of the door, and locate its target so that the light falls across the table and bowl onto the bed. Set the intensity of the spotlight to a value of **10**. Adjust the spotlight's properties until the rendered view of the room looks similar to the view in Fig. 47-15. Notice that this rendering contains both late afternoon sunlight and light from the spotlight. Save your work.
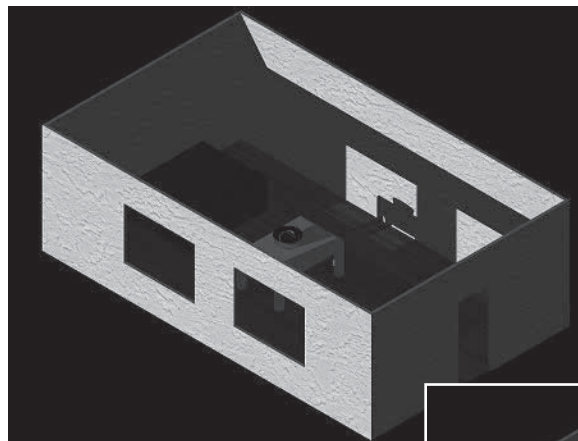
Fig. 47-14
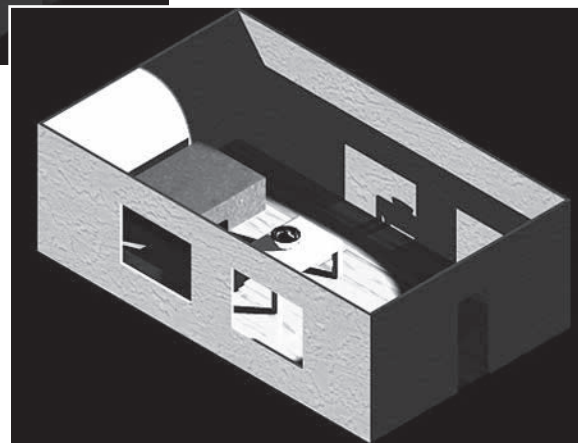
Fig. 47-15

AutoCAD at Work

Pipeline Drafter

Although we don't always realize it, we sometimes take modern-day conveniences for granted. If we want a drink of water, all we have to do is go to the kitchen sink and help ourselves to a glass. If we feel like taking a bath, we can just walk into the bathroom and fill the tub. The water we use for these purposes doesn't just magically appear, of course. Someone had to plan a way to direct water from a main source to our homes. Thanks to people like pipeline drafters, we are able to enjoy one of our most important natural resources by merely turning a nozzle.

A Knack for Design

In addition to finding ways to channel water into homes, pipeline drafters design piping systems to move substances in refineries, oil and gas fields, sanitation systems, and chemical plants. The drawings they create are used for construction and layout of parts. An accurately designed system transports substances efficiently from place to place. It also resists leakages or damage that may be potentially harmful to the environment.

© 2007 Getty Images, Inc./RF

Analyzing the Situation

Pipeline drafting requires knowledge of the materials used to build pipeline systems and the quality and safety standards for each. Analytical skills are necessary to determine which materials are best suited for a particular system in a particular environment. Drafters frequently consult with surveyors, scientists, and engineers to ensure proper design, so communication and "people" skills are an important part of the job, as well.
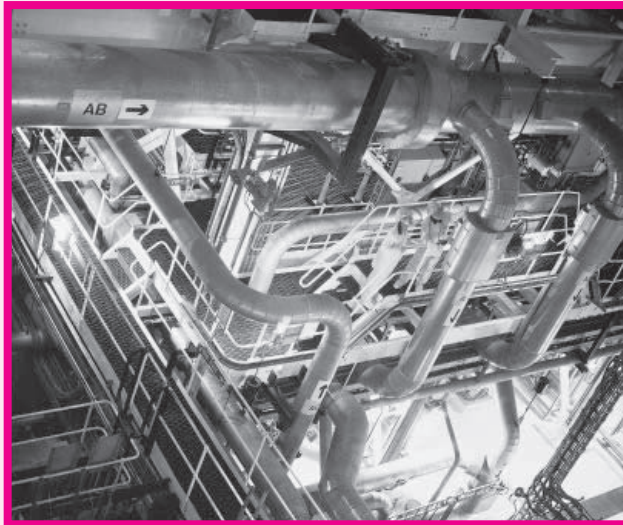
▶ Career Activities

1. Write a report about the work of a pipeline drafter. What challenges are involved in this type of drafting?

2. Why is it important for pipeline drafters to consult with other people when designing piping systems?

Chapter 48

Benefits of Solid Modeling

Objectives

- Split a solid model in half
- Calculate mass properties of a solid model
- Create an **STL** file of a solid model for prototype fabrication

Vocabulary

additive manufacturing
bisect
manufacturing
XYZ octant

Solid modeling provides many benefits that are "downstream" from the solid model. As you saw in Chapter 46, the solid model of the pulley was used to create a production drawing. Other examples of benefits include mass properties generation, detail drafting, finite element analysis, and the fabrication of physical parts. Another benefit is the natural link to additive manufacturing. **Additive manufacturing** is a process of joining liquids, powders, filaments, or sheet materials to form parts from 3D model data. The parts are usually produced layer upon layer, as opposed to a subtractive manufacturing methodology, such as machining. Once you have a solid model, it becomes very easy to output the data needed to drive an additive manufacturing system.

In this chapter, you will split the pulley you created in Chapter 46, as shown in Fig. 48-1. You will use the pulley half to calculate mass properties and create data that can be used to produce a prototype part.
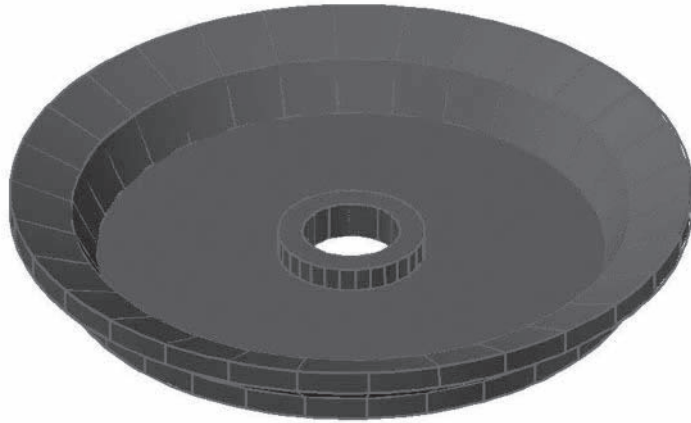
Fig. 48-1

Slicing the Pulley

The **SLICE** command cuts through a solid and retains either or both parts of it. The command is especially useful if you want to create a mold for half of a symmetrical part, such as a pulley.

1. Start AutoCAD and use the **pulley.dwg** drawing file to create a new file named **half.dwg**.
2. Freeze all layers except for layer **Objects**.
3. View the model from the **Top** view and switch to the **3D Wireframe** visual style.
4. Enter the **UCS** command and enter **W** for **World**.
5. Pan and zoom the pulley to the center of the screen.
6. Turn snap on and draw a line that bisects the pulley.

When you **bisect** an object, you divide it into two equal halves. The view of the pulley should be similar to the one in Fig. 48-2.

7. Pick the **Slice** button on the **Solid Editing** panel to enter the **SLICE** command.
8. Select the pulley, press **ENTER**, and enter the **YZ** option.

Fig. 48-2

SOLID EDITING

AutoCAD asks you to pick a point on the YZ plane. Try to visualize the orientation of this plane, which is vertical and perpendicular to the screen. "Vertical" is the Y direction, and "perpendicular to the screen" is the Z direction.

9. Pick the endpoints of the center line to define the slicing plane.

10. Pick a point anywhere to the right of the slicing plane.

This tells AutoCAD that you want to keep the right half of the pulley. AutoCAD cuts the pulley in half and leaves the right half on the screen.

11. Pick the polyline that you used to create the pulley in Chapter 46 and move it to layer **0**.

Because layer **0** is frozen, AutoCAD displays a dialog box to let you know that the object has been moved to a frozen layer.

12. Erase the center line.

Your model should now look like the one in Fig. 48-3.

13. Save your work.

Fig. 48-3

Calculating Mass Properties

Because solid models contain volume information, it is possible to obtain meaningful details about the model's physical properties. For example, you can generate mass properties, including the center of gravity, mass, surface area, and moments of inertia. Properties such as mass and volume permit engineers to analyze the part for structural integrity.
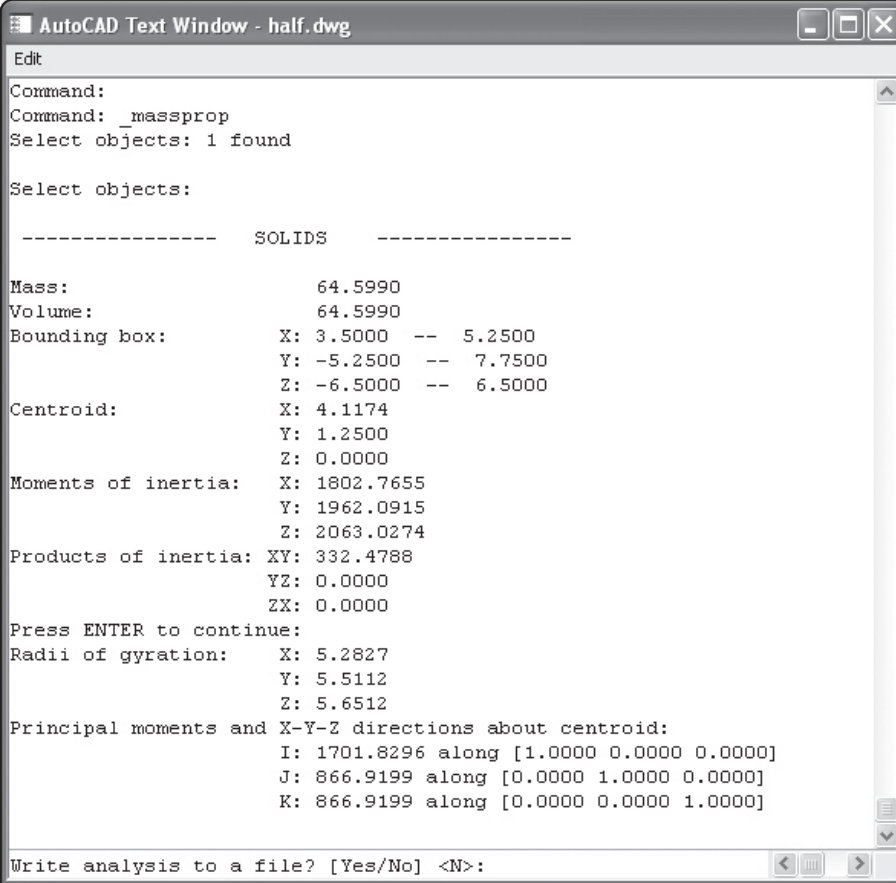
1. Enter the **MASSPROP** command at the keyboard.

2. Select the solid model and press **ENTER**.

AutoCAD generates a report similar to the one shown in Fig. 48-4.

3. Press **ENTER** again to see the entire report.

4. Enter **Yes** in response to **Write analysis to a file?** and press **ENTER** or pick the **Save** button to use the default **half.mpr** file name.

AutoCAD creates an ASCII (text) file, assigning the **MPR** file extension to it automatically. The **MPR** file extension stands for "Mass Properties Report." Let's locate and review the file.

5. Close the **AutoCAD Text** window, if you haven't already, and minimize

Fig. 48-4

AutoCAD.

6. Start Notepad.

7. Select **Open...** from the **File** pull-down menu.

8. Locate and open the file named **half.mpr**.

> **HINT**
>
> Change the file name extension from **txt** to **mpr** to list only those files that have the **MPR** extension.

The contents of the file should be identical to the information displayed by the **MASSPROP** command.

9. Exit Notepad and maximize AutoCAD.

Outputting Data for Additive Manufacturing

The **STLOUT** command creates an **STL** file from a solid model. **STL** is the file type required by most additive manufacturing systems, such as fused deposition modeling and laser sintering. These machines enable you to create a physical part from a solid model. The part can serve many purposes, such as a pattern for prototype tooling.

Positioning the Solid Model

Before creating the **STL** file, you must orient the part in the position that is best for the additive manufacturing system. In most cases, the part should be lying flat on the WCS. You can accomplish this in the current drawing by rotating the part around the Y axis.

1. Enter the **ROTATE3D** command, select the solid, and press **ENTER**.

> **NOTE**
> Be sure to use the **ROTATE3D** command. Do not pick the **3D Rotate** button on the **Dashboard**; doing so enters the **3DROTATE** command, which produces different results.

2. Enter the **Yaxis** option because you want to rotate the part around the Y axis.
3. Pick a point anywhere on the left edge of the part to define the Y axis.
4. Enter **–90** for the rotation angle.

It is important that you rotate the part in the negative direction. A positive rotation positions the part upside-down. Also, if the part lies outside the positive XYZ octant, the **STLOUT** command will not work. The **XYZ octant** is the area in 3D space where the *x, y,* and *z* coordinates are greater than 0.

5. If necessary, move the part so that it lies entirely in positive 3D space.

> **HINT**
> To make sure the part lies entirely in positive 3D space, turn on the grid and view the model from the **Front** viewpoint. Move it up using the **3DMOVE** command, if necessary.

6. Pick the **SW Isometric** view, as shown in Fig. 48-5.

The part should appear to lie on the WCS. Let's check to be sure.
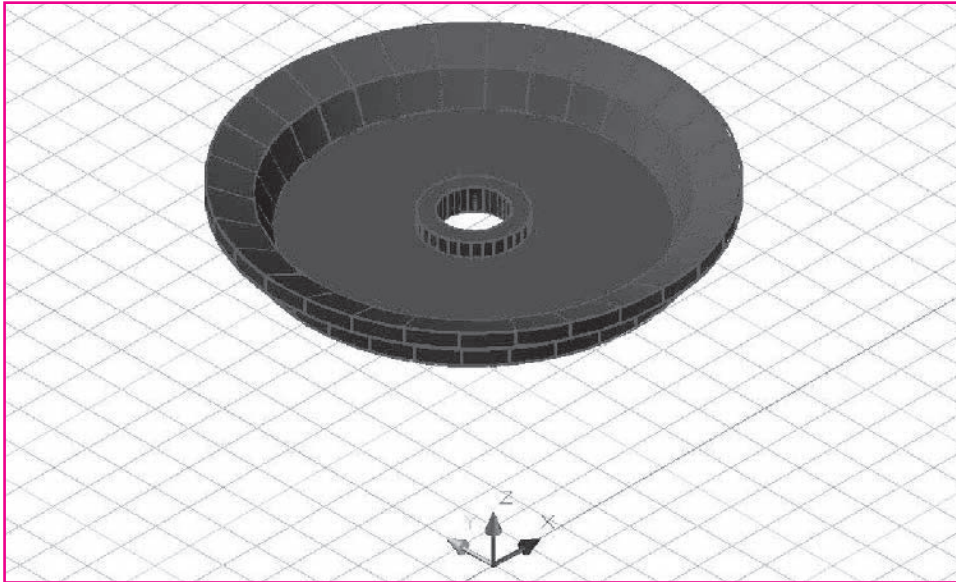
Fig. 48-5

7. Using the **ID** command, pick a point at the bottom of the part.

> **HINT** Use the **Nearest** object snap to snap to a point at the bottom of the part.

The value of the z coordinate should be greater than 0.0000. If it is 0.0000 or a value less than 0, you must move the solid upward into the positive XYZ octant.

8. If necessary, move the solid upward into the positive XYZ octant.

Creating the STL File

Now that the pulley is properly positioned, you can create the **STL** file.

1. Enter the **UCS** command and **W** for **World**.

2. Enter the **STLOUT** command, pick the solid, and press **ENTER**.

AutoCAD wants to know whether you want to create a binary **STL** file. Normally, you would create a binary file because binary files are much smaller than ASCII files. However, you cannot view the contents of a binary **STL** file, and we want to view its contents.

3. Enter **No** to create an ASCII **STL** file.

4. Pick the **Save** button in the **Create STL File** dialog box or press **ENTER** to accept **half.stl** as the name of the file.

Reviewing the STL File Data

Let's review the contents of the **STL** file.

1. Minimize AutoCAD and start WordPad.

2. Select **Open…** from the **File** pull-down menu.

3. In the **File** name box, enter ***.stl**. Be sure to press **ENTER**.

4. Open the file named **half.stl**.

The contents of the file appear. The text listing in Fig. 48-6 shows the first part of the file. **STL** files consist mainly of groups of x, y, and z coordinates. Each group of three defines a triangle.

```
solid AutoCAD
facet normal 0.0000000e+000  0.0000000e+000  –1.0000000e+000
        outer loop
                vertex 8.4399540e+000  4.8596573e+000  1.0000000e–001
                vertex 8.5000000e+000  4.2500000e+000  1.0000000e–001
                vertex 5.8750000e+000  4.2500000e+000  1.0000000e–001
        endloop
endfacet
facet normal 0.0000000e+000  0.0000000e+000  –1.0000000e+000
        outer loop
                vertex 4.8750000e+000  4.2500000e+000  1.0000000e–001
                vertex 2.2500000e+000  4.2500000e+000  1.0000000e–001
                vertex 2.3100460e+000  4.8596573e+000  1.0000000e–001
        endloop
endfacet
facet normal 0.0000000e+000 0.0000000e+000  –1.0000000e+000
        outer loop
                vertex 4.9130602e+000  4.4613417e+000  1.0000000e–001
                vertex 4.8750000e+000  4.2500000e+000  1.0000000e–001
                vertex 2.3100460e+000  4.8596573e+000  1.0000000e–001
        endloop
endfacet
facet normal 0.0000000e+000  0.0000000e+000  –1.0000000e+000
```

Fig. 48-6

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IX. Solid Modeling

48. Benefits of Solid
Modeling

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645
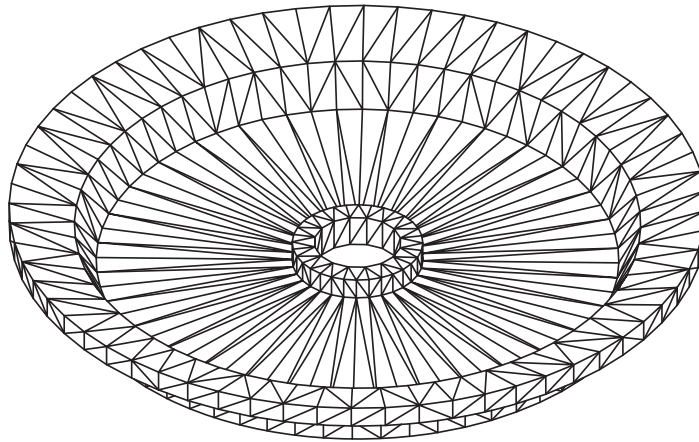
Fig. 48-7

5. After you have reviewed the file, close WordPad and maximize AutoCAD.

6. Save your work and exit AutoCAD.

The view shown in Fig. 48-7 was created using a special utility called **stlview.lsp**. This utility reads and displays the contents of an ASCII **STL** file.

As you can see, an **STL** file is indeed made up of triangular facets. You can adjust the size of the triangles using the **FACETRES** (short for "facet resolution") system variable by changing its value before you create the **STL** file. A high value creates a more accurate part with a smoother surface finish, but the file size and the time required to process the file increase. You should set it as high as necessary to produce an accurate part. Knowing what this value should be for a given part comes with experience and experimentation.

Part Creation

Hundreds of organizations around the world can create parts from **STL** files. If you were to send the **half.stl** file to one of them, that organization could create a part for you, typically for a fee. Autodesk (the maker of AutoCAD) has partnered with service providers to make 3D printing available. The **Send to 3D Print Service** button in the **Output** tab opens a dialog box that allows you to order one or more parts.

Figure 48-8 on page 640 shows pictures of a completed plastic prototype part. It was created from the **half.stl** file using a stereolithography additive manufacturing system.

Notice the flat segments that make up the large diameter of the pulley. They are caused by the triangular facets. You could reduce their size by increasing the value of **FACETRES** before creating the **STL** file. You could also reduce their visibility by sanding the part, but this would change the accuracy of the part. The part pictured in Fig. 48-8 was built from a version of **half.stl** with **FACETRES** set at a relatively low value to show the undesirable effect of large facets.
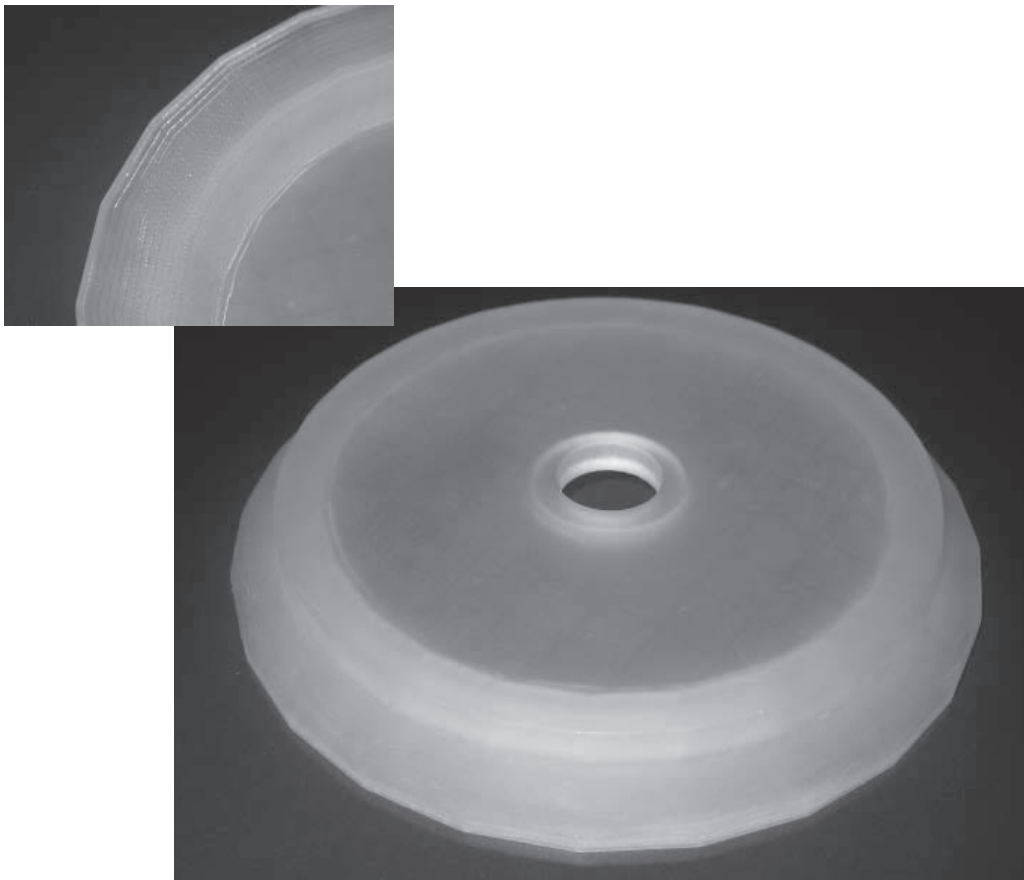
Fig. 48-8

Prototype part courtesy of Laser Prototypes, Inc. of Denville, New Jersey
(Photos courtesy of Terry Wohlers)

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IX. Solid Modeling

48. Benefits of Solid
Modeling

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647

Chapter 48 Review & Activities

● REVIEW QUESTIONS

Answer the following questions on a separate sheet of paper.

1. Describe the **SLICE** command.
2. When would the **SLICE** command be useful?
3. AutoCAD can create files with an **MPR** file extension. How are these files created, and what information do they provide?
4. Name at least three mass properties for which AutoCAD makes information available about a solid model.
5. Explain the purpose of the **STLOUT** command.
6. What is the purpose of creating an **STL** file?
7. What AutoCAD system variable affects the size of the triangles in an **STL** file?
8. What is the primary advantage of creating a binary **STL** file? …an ASCII **STL** file?

● CHALLENGE YOUR THINKING

These questions are designed to further your knowledge of AutoCAD by encouraging you to explore the concepts presented in this chapter. Answer each question on a separate sheet of paper.

1. Describe a situation in which you might need to slice an object using the **3points** option.
2. Explore the different additive manufacturing systems that are commercially available. Write an essay that compares each of them.

Continued

Chapter 48 **Review & Activities**

● APPLYING AUTOCAD SKILLS

Work the following problems to practice the commands and skills you learned in this chapter.

1. Create ASCII and binary **STL** files from the solid model stored in the **composite.dwg** file, which you created in Chapter 44. Compare the sizes of the two files and then view the contents of the ASCII **STL** file.

2. Open the height gage you created in "Applying AutoCAD Skills" problem 4 in Chapter 44. If you have not yet created the model, create it now. Then make two slices. Make the first slice through the V groove and the second slice perpendicular to the first and in the same plane.

3. If you know of an additive manufacturing service provider in your area, request a demonstration of its system(s). Explore the possibility of having the company create a prototype part for you from an **STL** file.

4. Open the file named **shaft.dwg**, which you created in Chapter 43. Use the **SLICE** command to split the model down the middle to create a pattern. Use **STLOUT** to create a binary **STL** file.

5. Produce and save a list of the mass properties for **shaft.dwg**.

● USING PROBLEM-SOLVING SKILLS

Complete the following activities using problem-solving skills and your knowledge of AutoCAD.

1. The bushing and insert shown in Fig. 48-9 are part of an aerospace project. Weight is critical. Slice the bushing and insert. Determine the volume of each part. Write each of the mass property reports to a file and print the files. Then select a material for each part, look up its density, and compute its mass. (To find density values, you may have to go to the reference section of a suitable library or search the Internet.)

Review & Activities Chapter 48
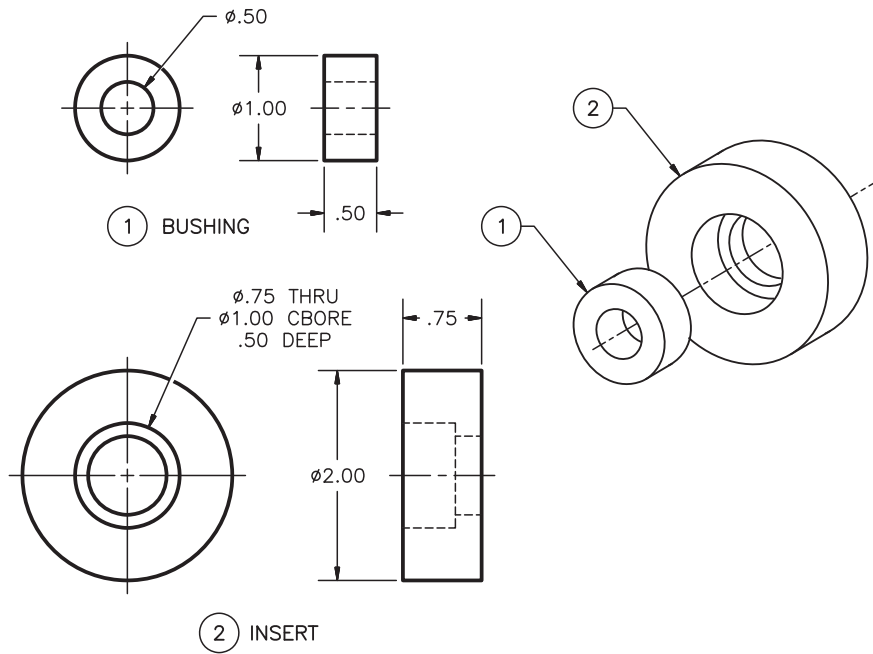
Fig. 48-9

Courtesy of Gary J. Hordemann, Gonzaga University

2. Position the bushing and insert from problem 1. Then make an ASCII **STL** file of the bushing only. View the **STL** file, but do not print it.

650

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IX. Solid Modeling

Part 9 Project: Applying
Chapters 42–48

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Part 9 Project | Applying Chapters 42–48

Designing and Modeling a Desk Set

AutoCAD is capable of creating 2D drawings, 3D surface models, and 3D solid models. The differences are meaningful. Although all three types of drawings yield information, 3D models allow more realistic views. Solid modeling, in addition, allows determination of mass properties and the generation of production data such as that used in automated machine tool processes. Familiarity with solid modeling enhances the ability of the drafter/designer to create solutions; it is well worth the effort to learn.

▶ Description

Your company has sold the desk set shown in Fig. P9-1 for several years. Your supervisor has determined that it is time to modernize and streamline the desk set to meet current market demand.
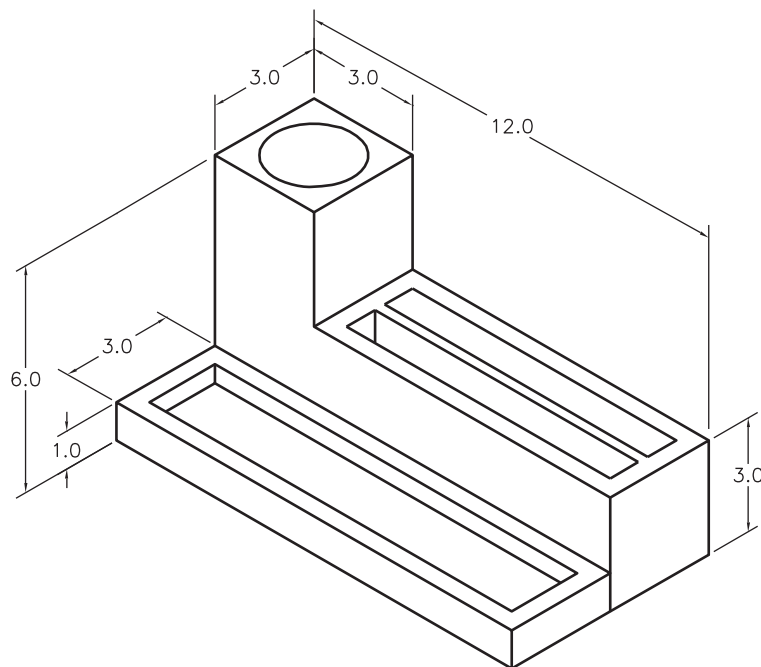
Fig. P9-1

1. Create a solid model of the desk set shown in Fig. P9-1. Include the round space to store pens, pencils, and the like; a large, flat, shallow area for items such as paper clips and rubber bands; and a medium-height section for a note pad, business cards, and similar papers. Wall thickness should range between **.1**″ and **.185**″. Assign a plastic material to the model. Save it as **desk1.dwg**.

2. Save **desk1.dwg** with a new name of **desk2.dwg**. Alter the design to modernize it or to make it more useful. For example, you may decide to provide an inclined surface on the two medium-height holes so that the one in back rises above the one in front of it. You may also want to round or bevel the edges to give it a streamlined look, or even add or subdivide the compartments.

3. Prepare a presentation for your supervisor. In your presentation, compare and contrast the new and old designs and explain why the new design is superior. Prepare rendered images of both desk sets to support your presentation.

▶ Hints and Suggestions

1. There is often more than one way to construct a solid model. If your approach does not seem to be working, try another.

2. Create the recesses for the compartments (such as the pencil holder) using the **SUBTRACT** command or the **PRESSPULL** command.

3. Use the **UNION** command as necessary so that the final product is a single composite solid.

▶ Summary Questions

Your instructor may direct you to answer these questions orally or in writing. You may wish to compare and exchange ideas with other students to help increase efficiency and productivity.

1. In your desk set design, how could the area actually being used to hold items be calculated?

2. Could the entire desk set design be created via extrusion? Explain how it could or why it could not.

3. What are some of the advantages of using predefined solid primitives in the creation of more complex solid models?

4. What is meant by "downstream benefits" of solid modeling?

Additional Problems

Introduction

The following problems provide additional practice with AutoCAD. These problems encompass a variety of disciplines. They will help you expand your knowledge and ability and will offer you new and challenging experiences.

The key to success is to **plan before you begin**. Review the options for setting up a new drawing and consider which commands and features you might use to solve the problem. As you discover new and easier methods of creating drawings, apply these methods to solving the problems. Since there is usually more than one way to complete a drawing, experiment with alternative methods. Discuss these alternatives with other users and create strategies for efficient completion of the problems.

Remember, there is no substitute for practice. The expertise you gain is proportionate to the time you spend on the system. Set aside blocks of time to work with AutoCAD, think through your approach, and enjoy this fascinating technology.

Each problem in this section is preceded by an icon that describes its level of difficulty, as shown below. The level of difficulty assigned to each problem assumes that you have learned the AutoCAD techniques and skills necessary to work the problem successfully. For some of the Level 3 problems, you may be required to expand your experience by combining your knowledge of AutoCAD with knowledge and skills that are not taught in this book.

L1   **Uses basic AutoCAD skills**

L2   **Uses intermediate AutoCAD skills**

L3   **Uses advanced AutoCAD and problem-solving skills**

L1 PROBLEM 1

Create the geometric design using the grid, snap, the **LINE** command, and (optionally) the **MIRROR** command. Set grid to **1** and snap to **.25**.
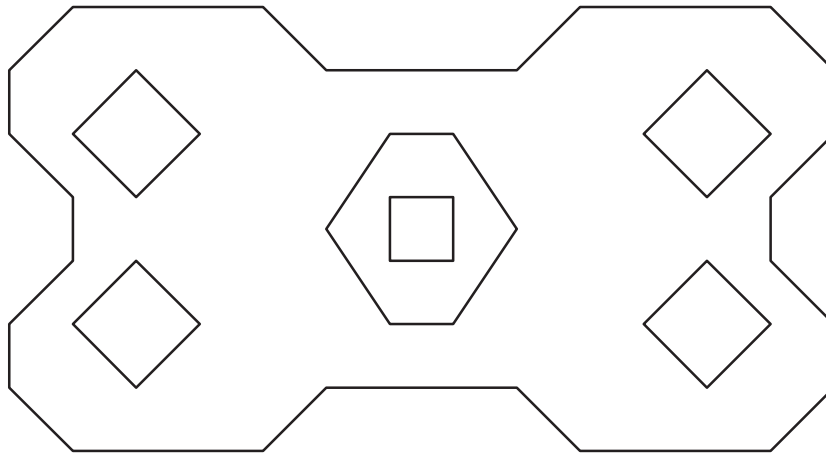
L1 PROBLEM 2

Create the geometric design using the grid, snap, and objects such as lines, arcs, circles, and fillets. Set grid to **1** and snap to **.5**.
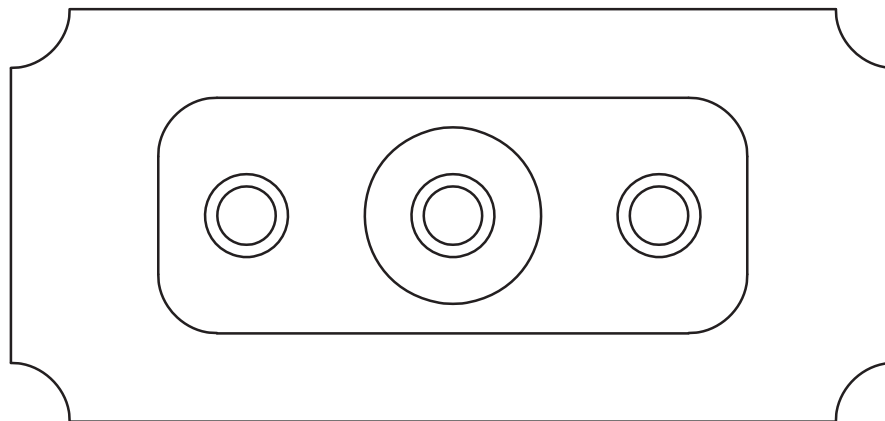
*Problems 1 and 2 courtesy of Carole J. Williams, South Harrison High School*

Additional Problems

L1 PROBLEM 3

Create the first geometric figure using the **POLYGON** and **ARRAY** commands. Inscribe the polygon in a circle with a radius of **2**. Then array the polygon three times to create the figure. Then copy the first geometric figure to a new location in the drawing area. Use the **TRIM** command to create the second figure.
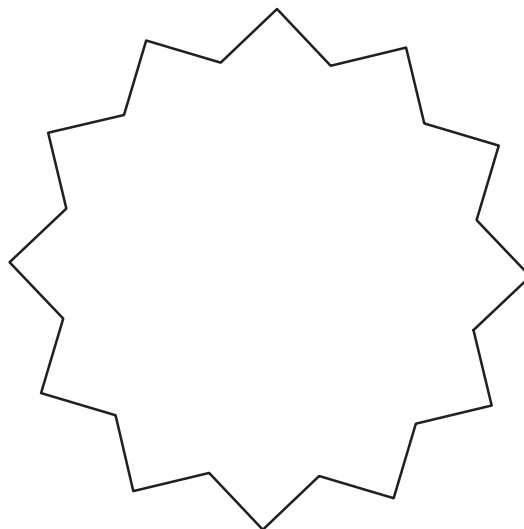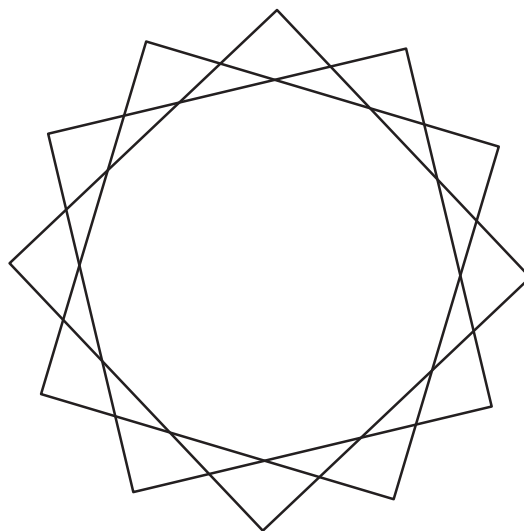
Courtesy of Carole J. Williams, South Harrison High School

L1 PROBLEM 4

Create the set of Olympic Rings using the **CIRCLE**, **COPY**, **BREAK**, and **BHATCH** commands. Create five layers using colors of blue, black, red, yellow, and green and create each ring on a separate layer. The first circle is blue, the second is yellow, the third is black, the fourth is green, and the fifth is red. Set snap to **.25** and grid to **.50**. The circles that make up the outer part of each ring have a radius of **2.75**, and the circles that define the inner part of the ring have a radius of **2.25**. The centers of the rings are **6.00** units apart, and the centers of the two bottom rings are **5.50** units below the centers of the three top rings.
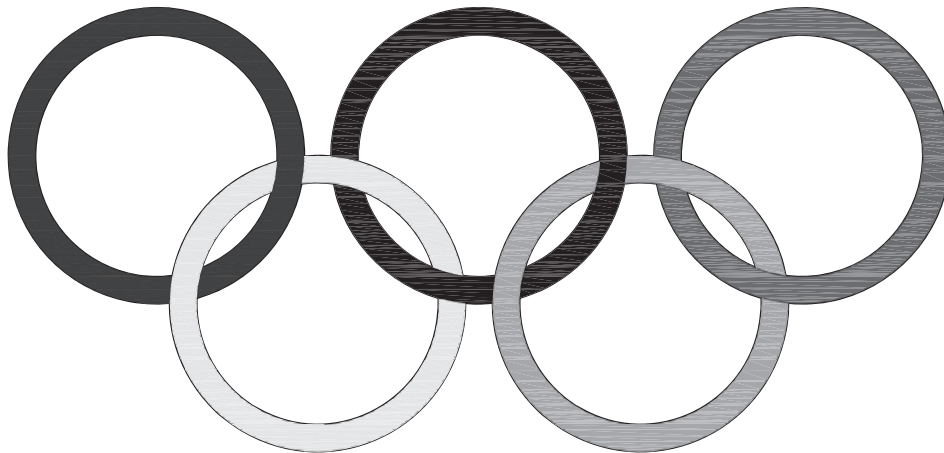
Courtesy of Carole J. Williams, South Harrison High School

Additional Problems

L1 PROBLEM 5

Use the **CIRCLE** and **ARRAY** commands to create each design variation
shown. All of these variations were created by starting with a basic
circle with a radius of **2**.
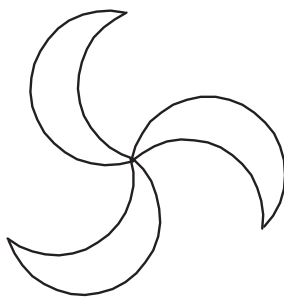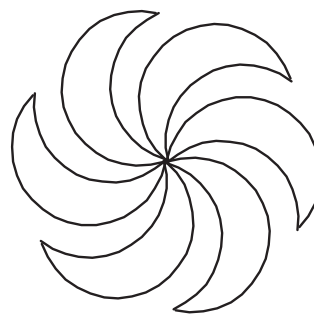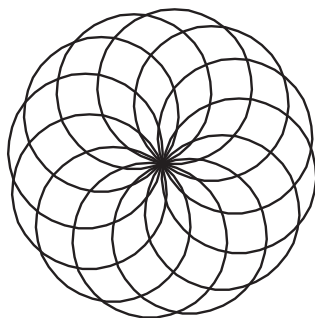
Courtesy of Carole J. Williams, South Harrison High School

L1 PROBLEM 6

Use the **CIRCLE**, **ARC**, **LINE**, **ERASE**, and **BHATCH** commands to create the geometric design. Circumscribe a polygon about a circle with a radius of **3**. Then construct lines from the center of the circle to each of the vertices. Use the **Start**, **End**, and **Center** options of the **ARC** command to create six arcs. Start each arc at one vertex and end it at the next alternate vertex, using the middle vertex as the center. Save the design and copy the entire figure to a new location in the drawing area. Alter the copy to make it look like the six-petal flower.
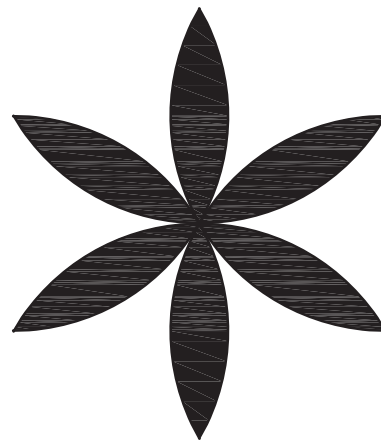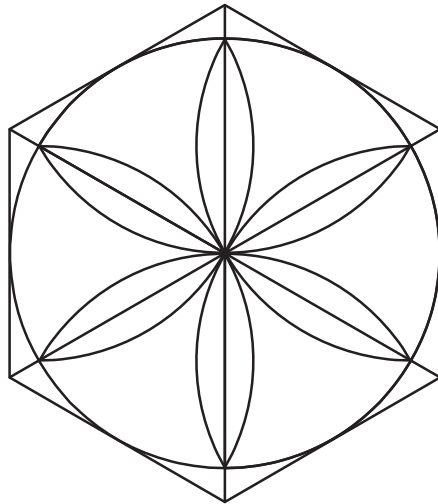
Courtesy of Carole J. Williams, South Harrison High School

Additional Problems

L1 PROBLEM 7

Use the **LINE**, **MIRROR**, and **BHATCH** commands with the necessary object snaps to create the two geometric figures. Set grid at **1** and snap at **.25**.

- Begin with an equilateral triangle with a side length of **3**.
- Construct a second equilateral triangle with a vertex at the midpoint of the base of the first triangle and a side length of **1.5**.
- Using **LINE**, extend the base of the second triangle by **1.25** on each side to create the top of the trapezoid.
- At each endpoint of the line created in the previous step, draw a segment of length **.75** at an angle of **225°** on the left side and an angle of **–45°** on the right side.
- Connect the inner endpoint of the left side to the endpoint of the inner right side.
- Mirror the top half of the design to complete the bottom half.
- Copy the entire figure to a new location in the drawing area.
- Use a solid hatch to fill in the areas shown.
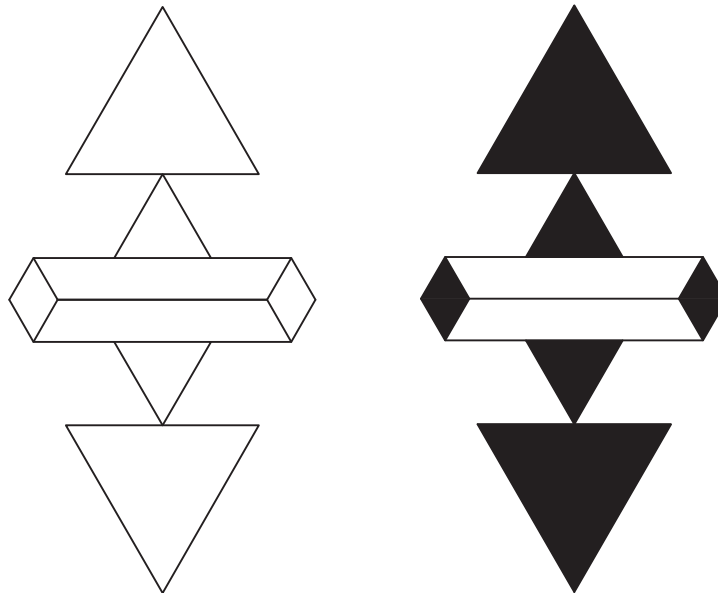
Courtesy of Carole J. Williams, South Harrison High School

L1 PROBLEM 8

Create and fully dimension the drawing of a link.
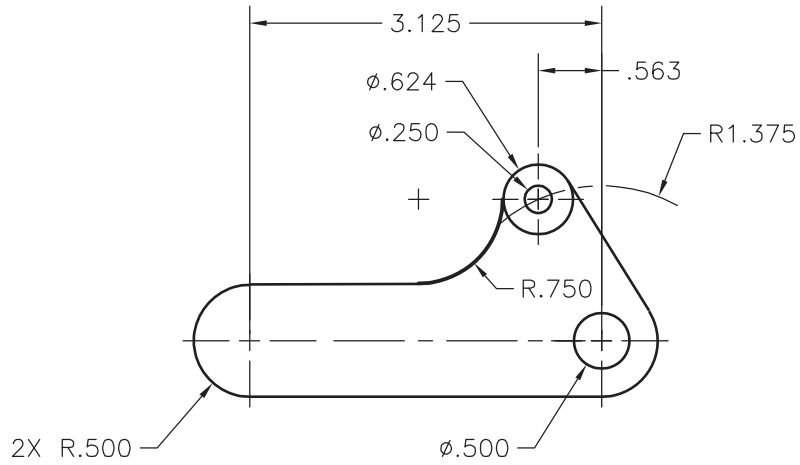
L1 PROBLEM 9

Create and fully dimension the drawing of a filler.
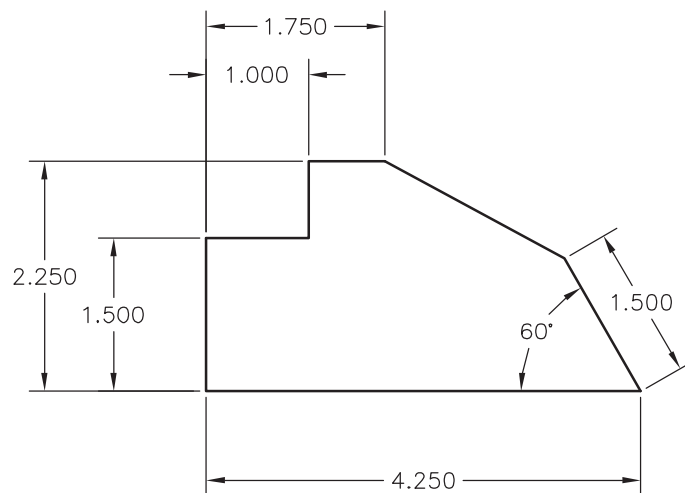
Additional Problems

L1 PROBLEM 10

Create and fully dimension the drawing of a link.
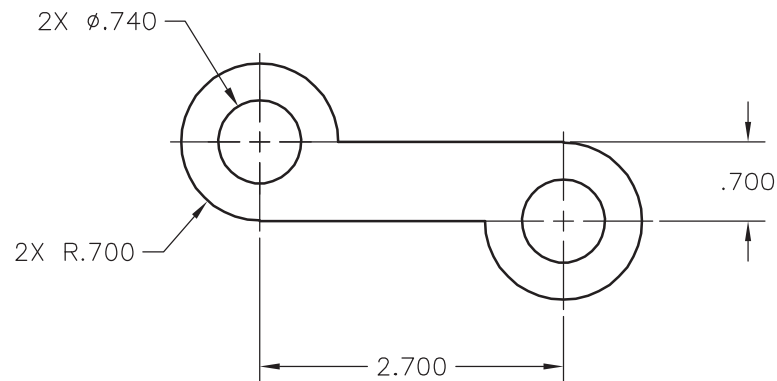
L1 PROBLEM 11

Create and fully dimension the orthographic views of the block.

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Additional Problems

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661
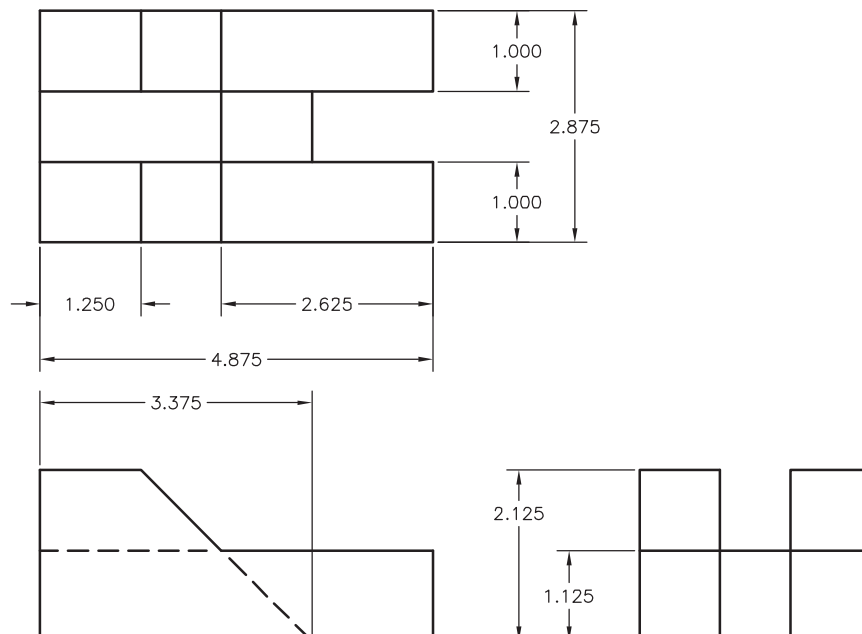
Additional Problems

L1 PROBLEM 12

Create and fully dimension the drawing of a gage.
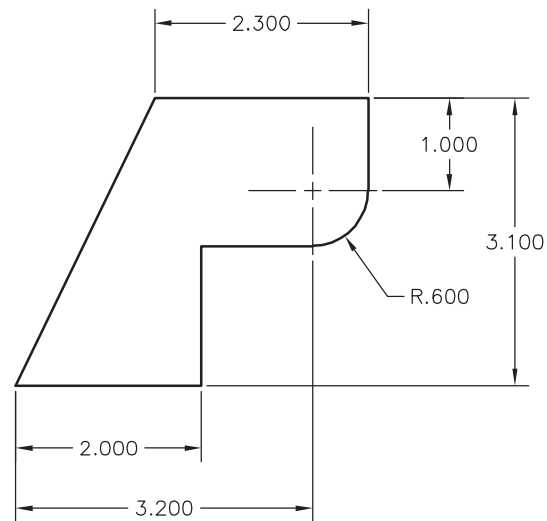
L1 PROBLEM 13

Create and fully dimension the drawing of a shaft bracket.
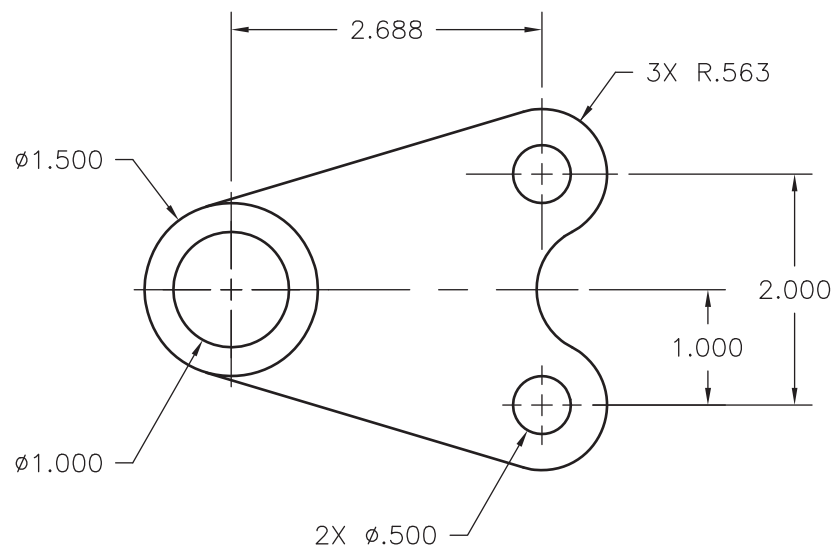
Additional Problems

L1 PROBLEM 14

Create and dimension the drawing of an end block.
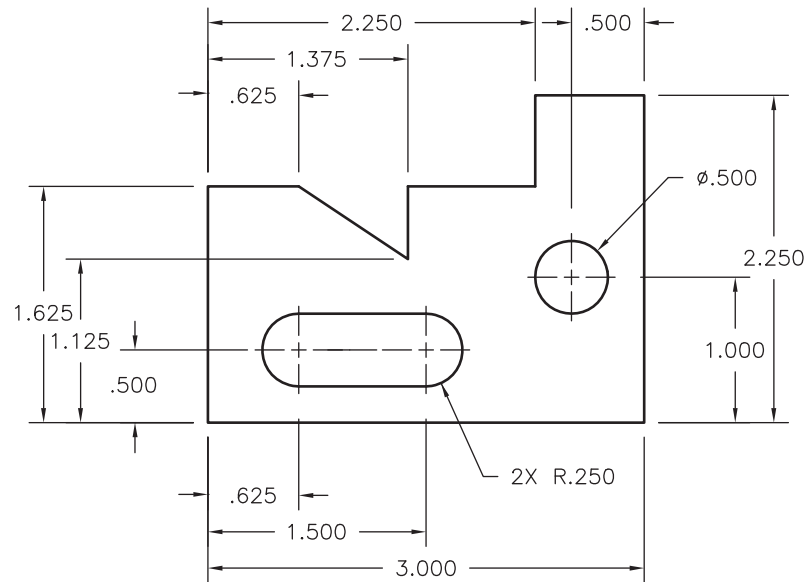
L1 PROBLEM 15

Create and dimension orthographic views of the angle block.
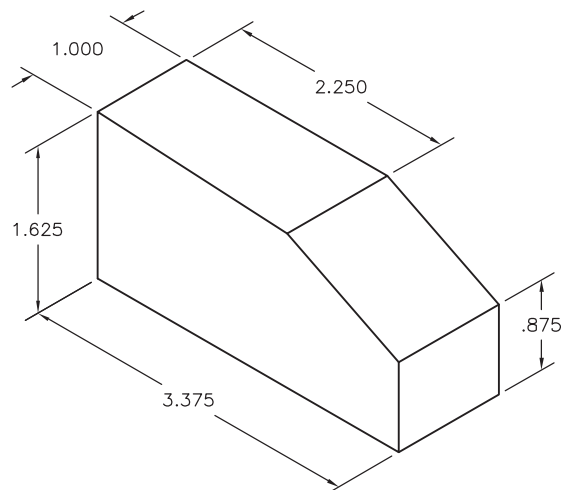
L1 PROBLEM 16

Create and fully dimension orthographic views of the nesting block.
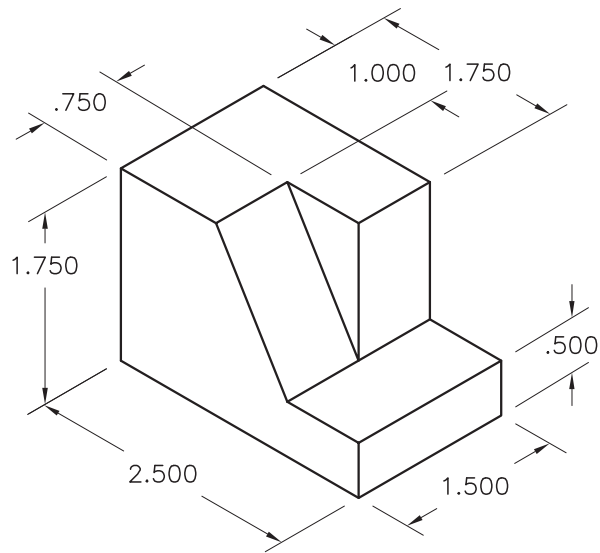
L1 PROBLEM 17

Create and fully dimension orthographic views of the locator.
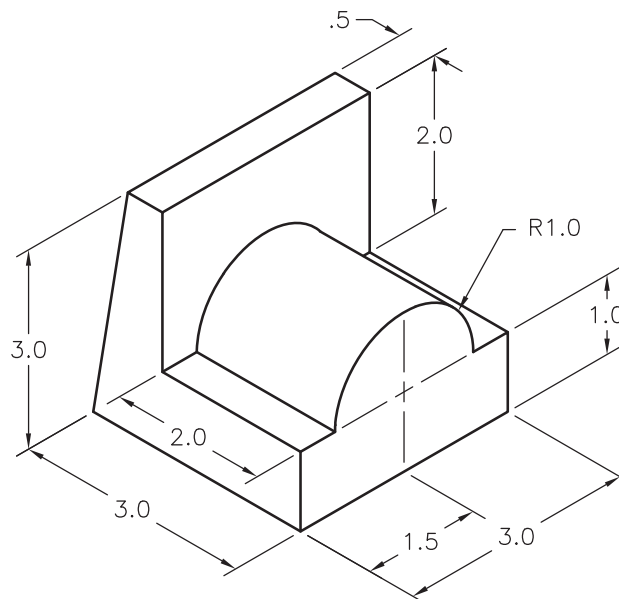
Additional Problems

L1 PROBLEM 18

Create and fully dimension orthographic views of the step block.
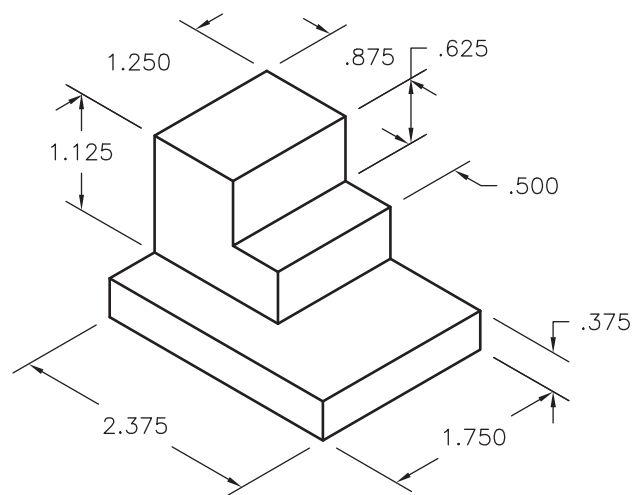
L1 PROBLEM 19

Create and fully dimension orthographic views of the block.
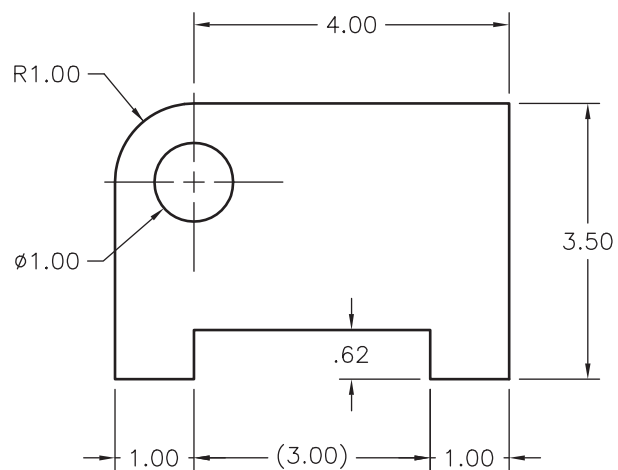
L1 PROBLEM 20

Create the drawing of an angle bracket. Use the dimensions shown, but dimension the drawing using decimal units.
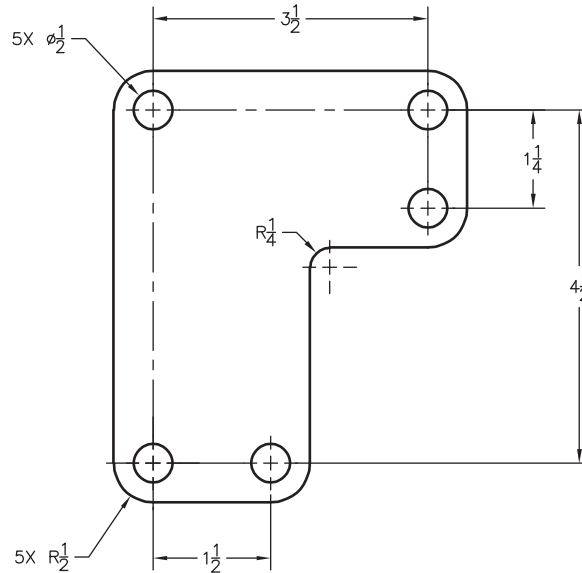
L1 PROBLEM 21

Create and dimension the drawing of a gasket.
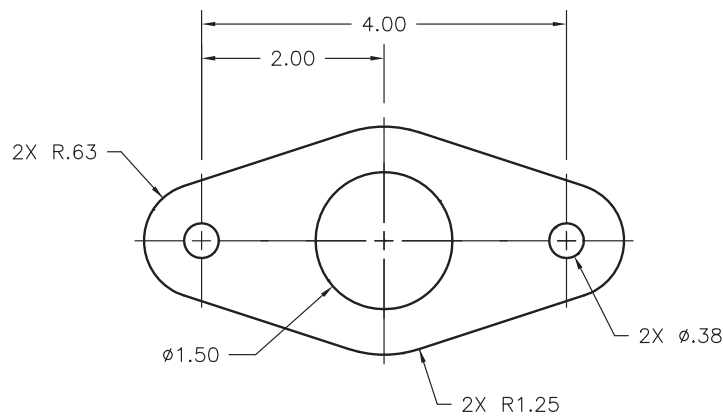
Adapted from the textbook Drafting Fundamentals *by Scott, Foy, and Schwendau*

Additional Problems

L1 PROBLEM 22

Create the electrical symbols as shown. Estimate their sizes. Produce a block of each symbol to create a symbol library.

RESISTOR THERMISTOR VARIABLE RESISTOR

DIODE ZENER CONNECTION

CAPACITOR CRYSTAL TERMINAL

INDUCTOR SWITCH SPEAKER

TRANSISTORS LOGICAL GATES

NPN PNP AND NAND

JFET−N JFET−P OR NOR

THYRISTORS COMMON GROUND CHASSIS GROUND
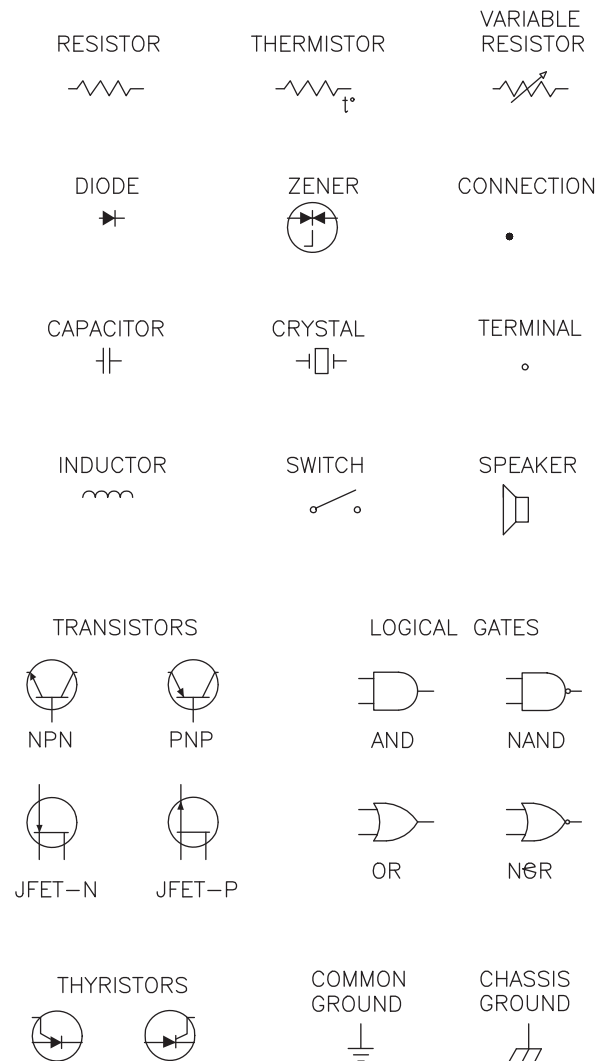
Courtesy of Robert Pruse, Fort Wayne Community Schools

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667

L1 PROBLEM 23

Create the following international symbols.

Courtesy of Joseph K. Yabu, Ph.D., San Jose State University

L1 PROBLEM 24

Create the figure shown below using a pattern of circles of radius **2** and the **ARRAY**, **BREAK**, **TRIM**, and **BHATCH** commands.
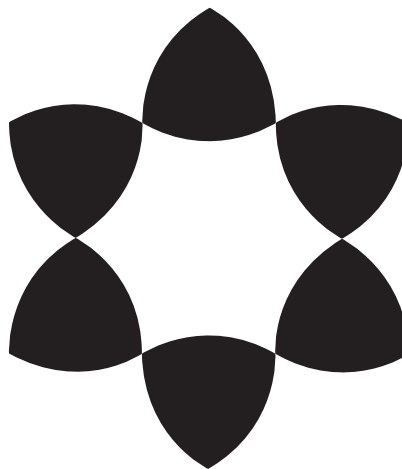
Courtesy of Carole J. Williams, South Harrison High School

Additional Problems

L1 PROBLEM 25

Create the floor plan drawing of a computer lab. Determine the drawing area according to space requirements. Estimate the dimensions that are not given.

ALL WALLS 6" THICK
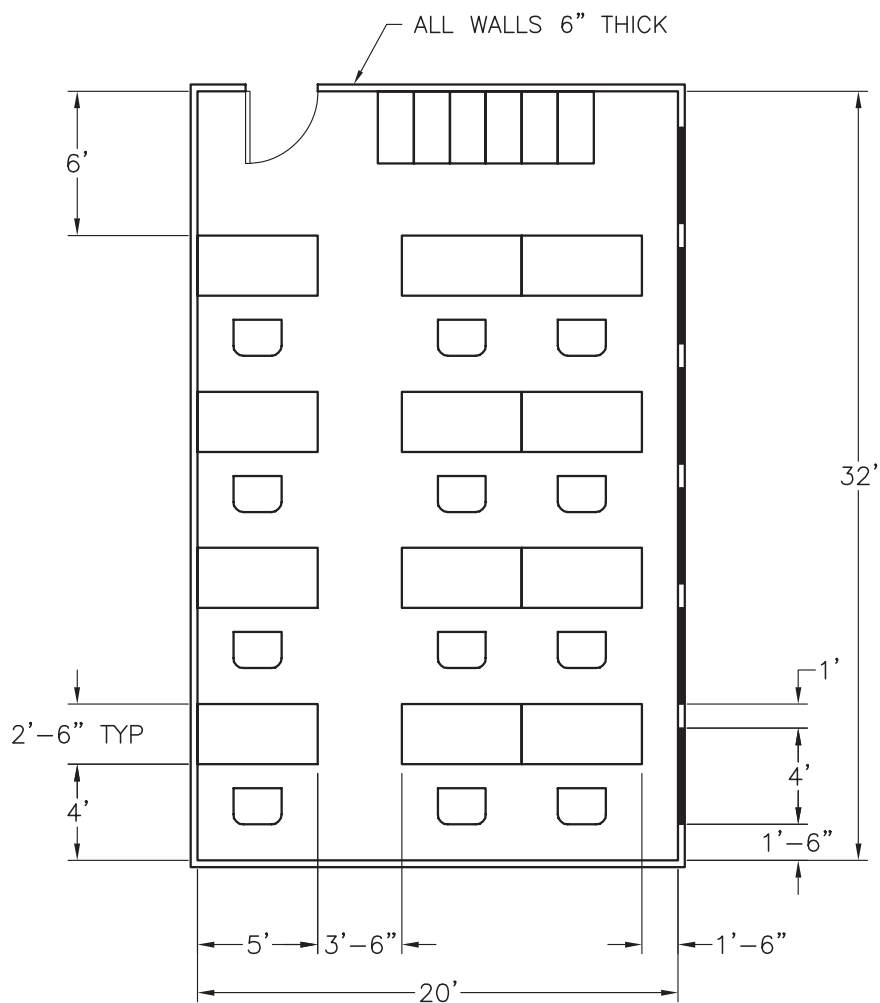
L2 PROBLEM 26

Create and fully dimension the drawing of an adjustable link.
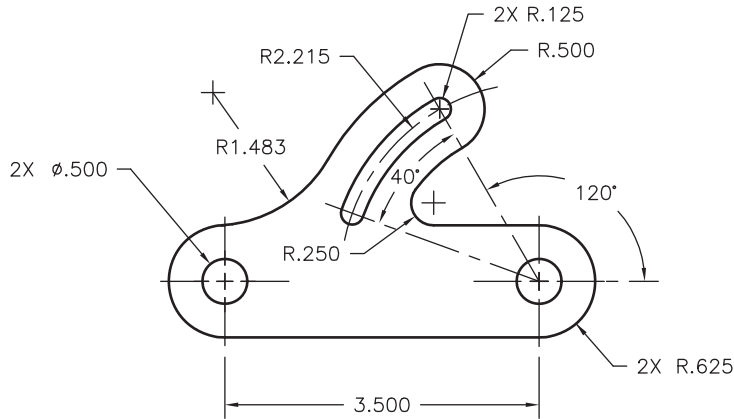
L2 PROBLEM 27

Create and fully dimension the drawing of an idler plate.
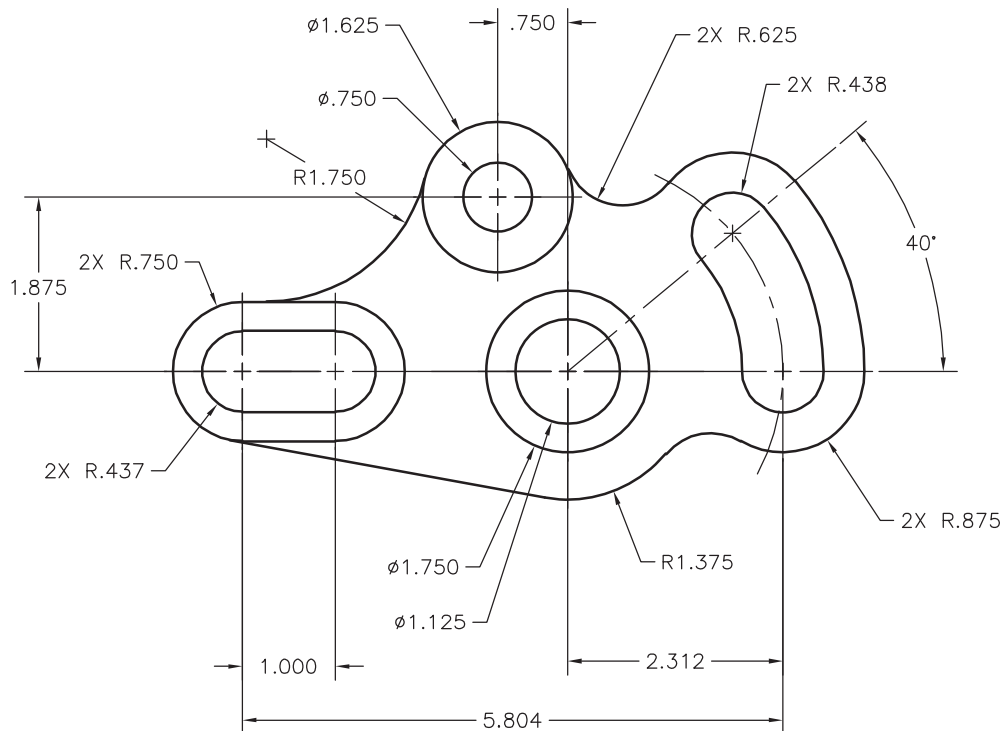
Additional Problems

L2 PROBLEM 28

Create and dimension orthographic views of the support.
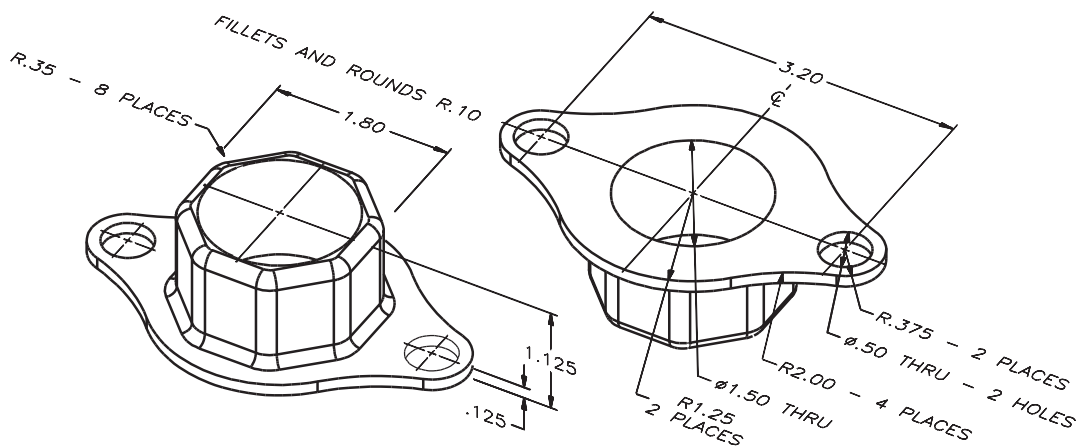
Courtesy of Gary J. Hordemann, Gonzaga University

L2 PROBLEM 29

Create and dimension the drawing of a cover plate.
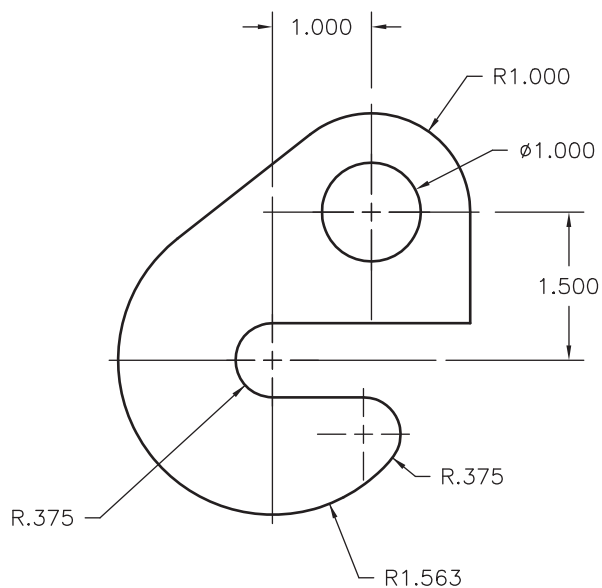
L2 PROBLEM 30

Create and dimension orthographic views of the base. Use a drawing area of 17 × 11.
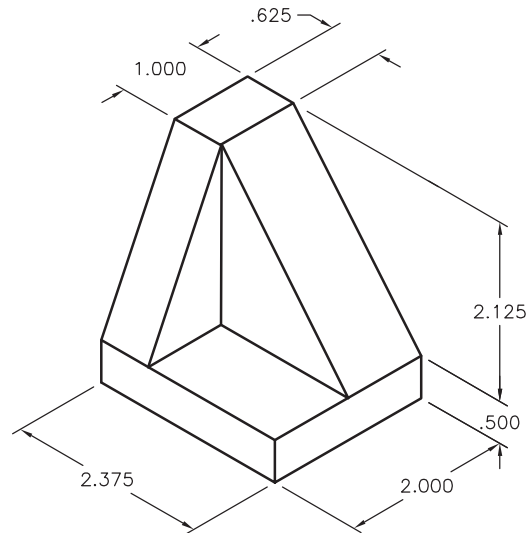
L2 PROBLEM 31

Create and dimension orthographic views of the base. Use a drawing area of 17 × 11.
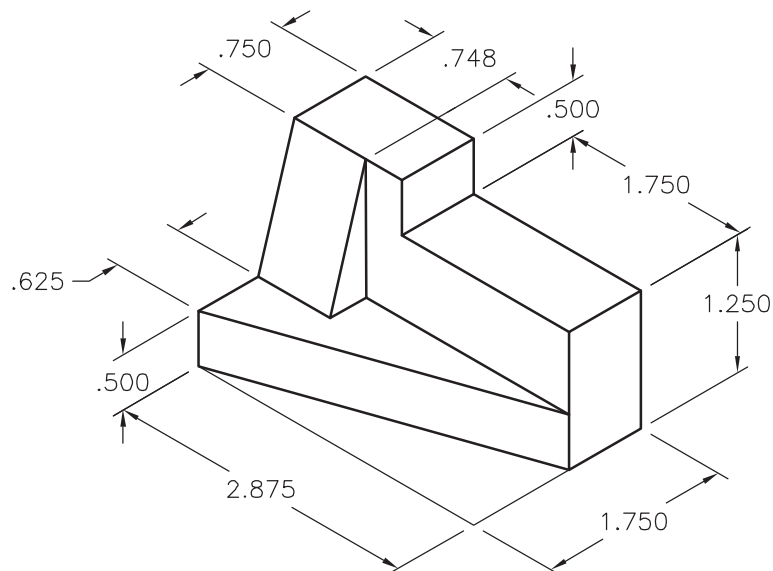
Additional Problems

L2 PROBLEM 32

Create and fully dimension the orthographic views of the angle block.
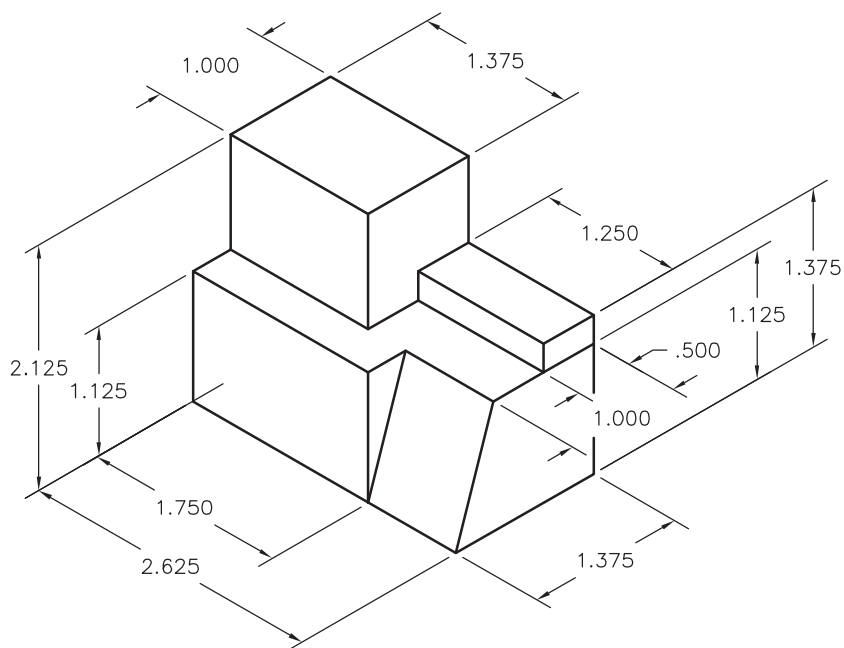
L2 PROBLEM 33

Create and fully dimension the orthographic views of the cradle.
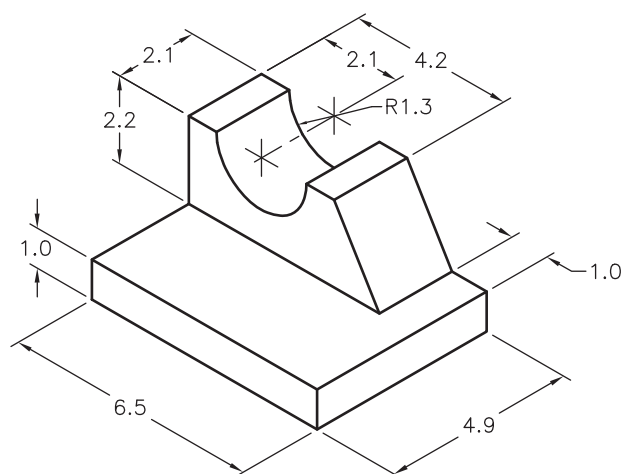
L2 PROBLEM 34

Draw the hex ratchet. Begin by creating the circles for the ends of the ratchet. Then create the lines and polygon. Use the **BREAK** command to remove parts of both circles as indicated to finish the drawing. Use a drawing area of 280 × 215. Dimension the drawing in millimeters.
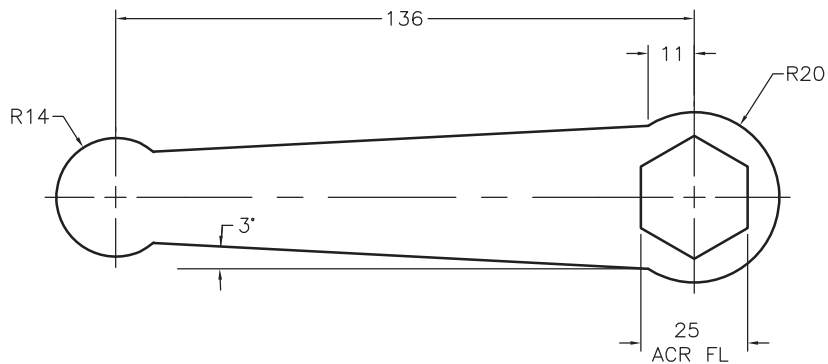
Adapted from the textbook Drafting Fundamentals *by Scott, Foy, and Schwendau*

L2 PROBLEM 35

Create and fully dimension the drawing of a rod support.
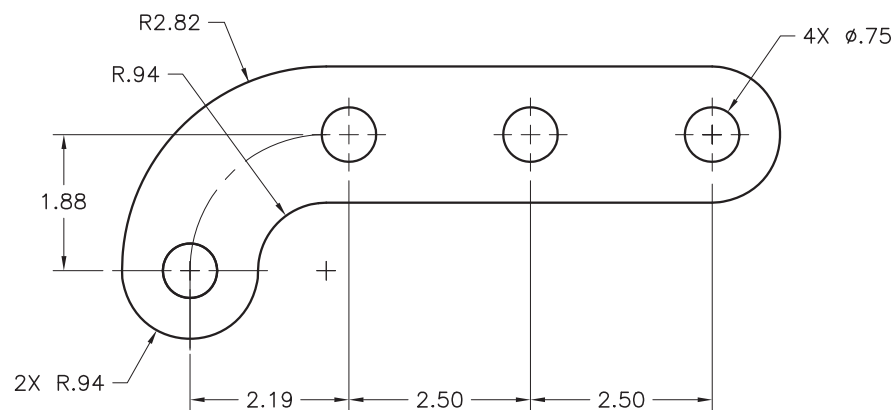
Additional Problems

L2 PROBLEM 36

The mechanism below consists of three parts: a driver, a follower, and a link. Use the dimensions shown and the **CIRCLE**, **OFFSET**, and **FILLET** commands to draw the front view.
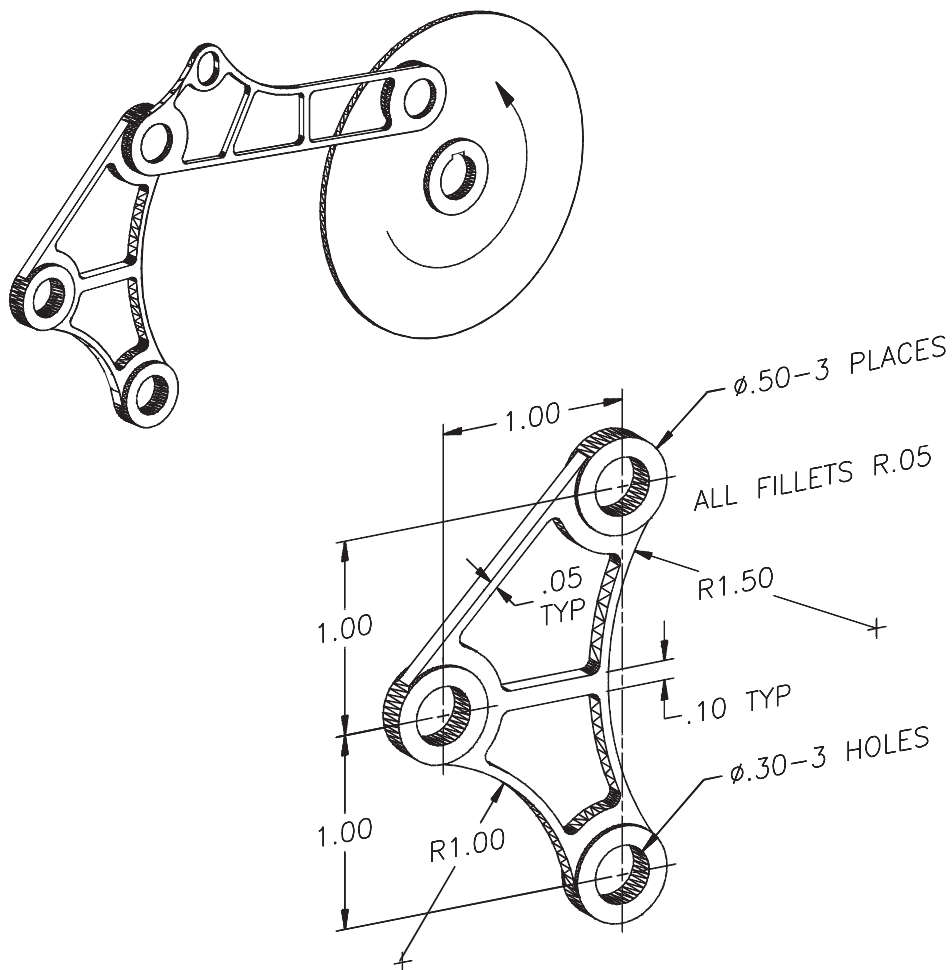
Courtesy of Gary J. Hordemann, Gonzaga University

L2 PROBLEM 37

Create and fully dimension the drawing of a master template. Use a drawing area of 22 × 17.
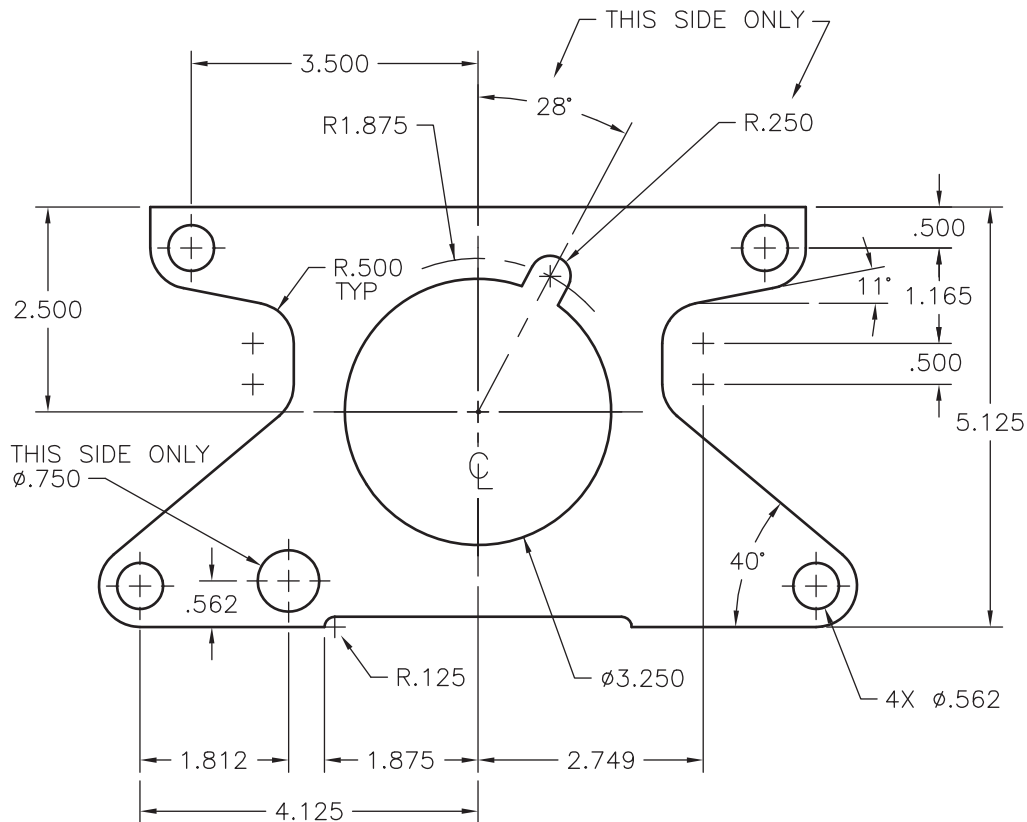
Courtesy of Steve Huycke, Lake Michigan College

Additional Problems

L2 PROBLEM 38

Create and dimension the drawing of a steel plate.

ø4.50±0.03 BC

6X ø.90±.03
EQUALLY SPACED

ø2.75±.03

ø6.25±.03
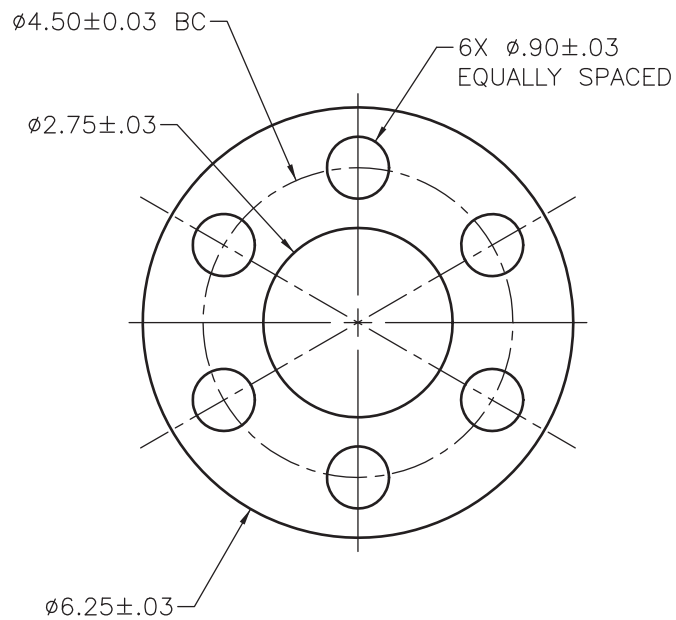
L2 PROBLEM 39

Create the drawing of a rocker arm. Dimension the drawing using decimal units.

2X R1$\frac{1}{4}$

2X ø$\frac{3}{4}$

ø2

2X R4

2X ø1$\frac{1}{2}$

$\frac{3}{4}$

$\frac{3}{4}$

ø3

3

3
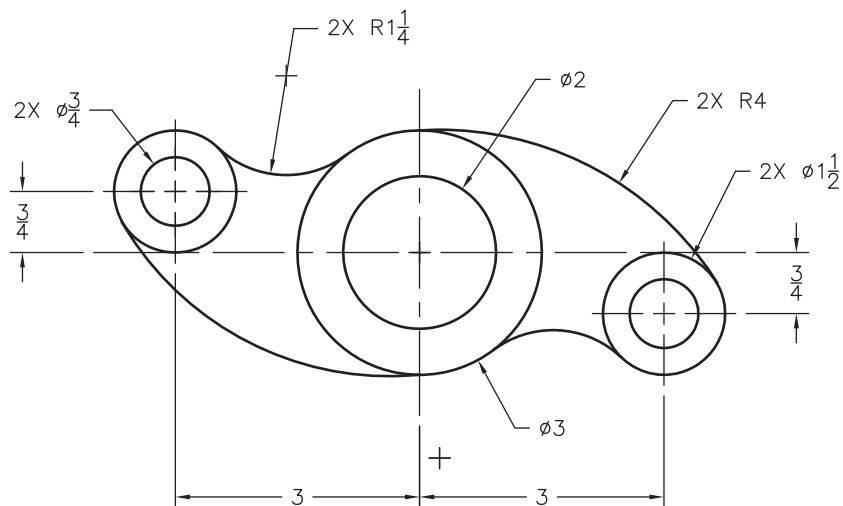
L2 PROBLEM 40

Create and fully dimension the drawing of a Geneva plate. Use a drawing area of 22 × 17.

8X R.250 SLOTS EQUALLY SPACED

.250 X .180 KEYWAY

ø1.750

ø1.000

45.0

R1.328 TYP   22.5

ø6.500

ø8.456

ø3.250
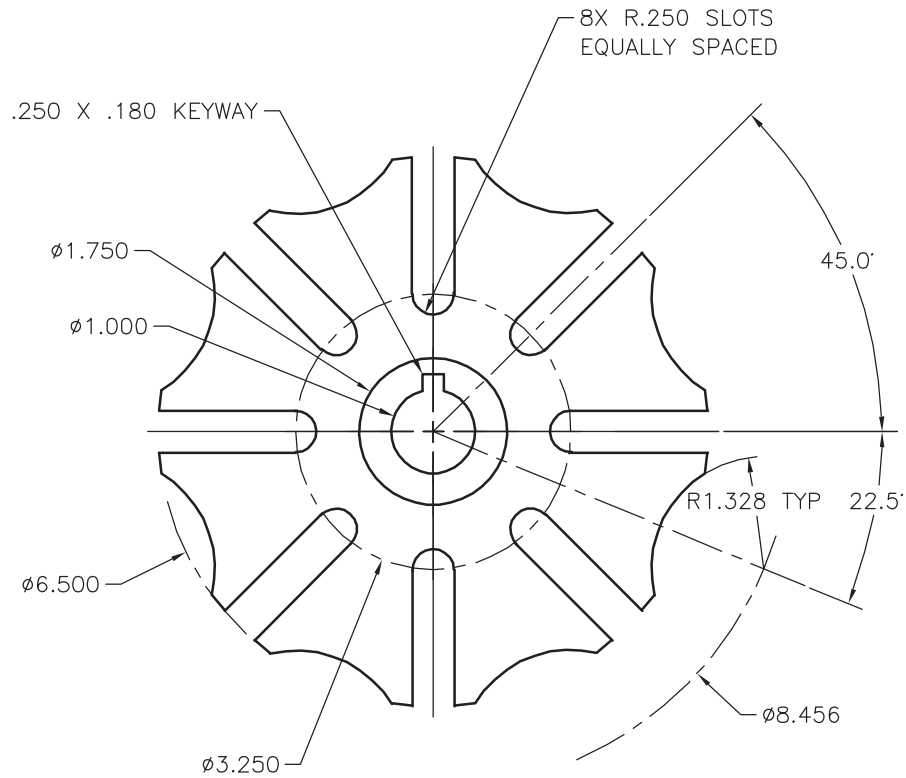
Additional Problems

L2 PROBLEM 41

Create and fully dimension the drawing of a cassette reel. Use a drawing area of 280 × 215.
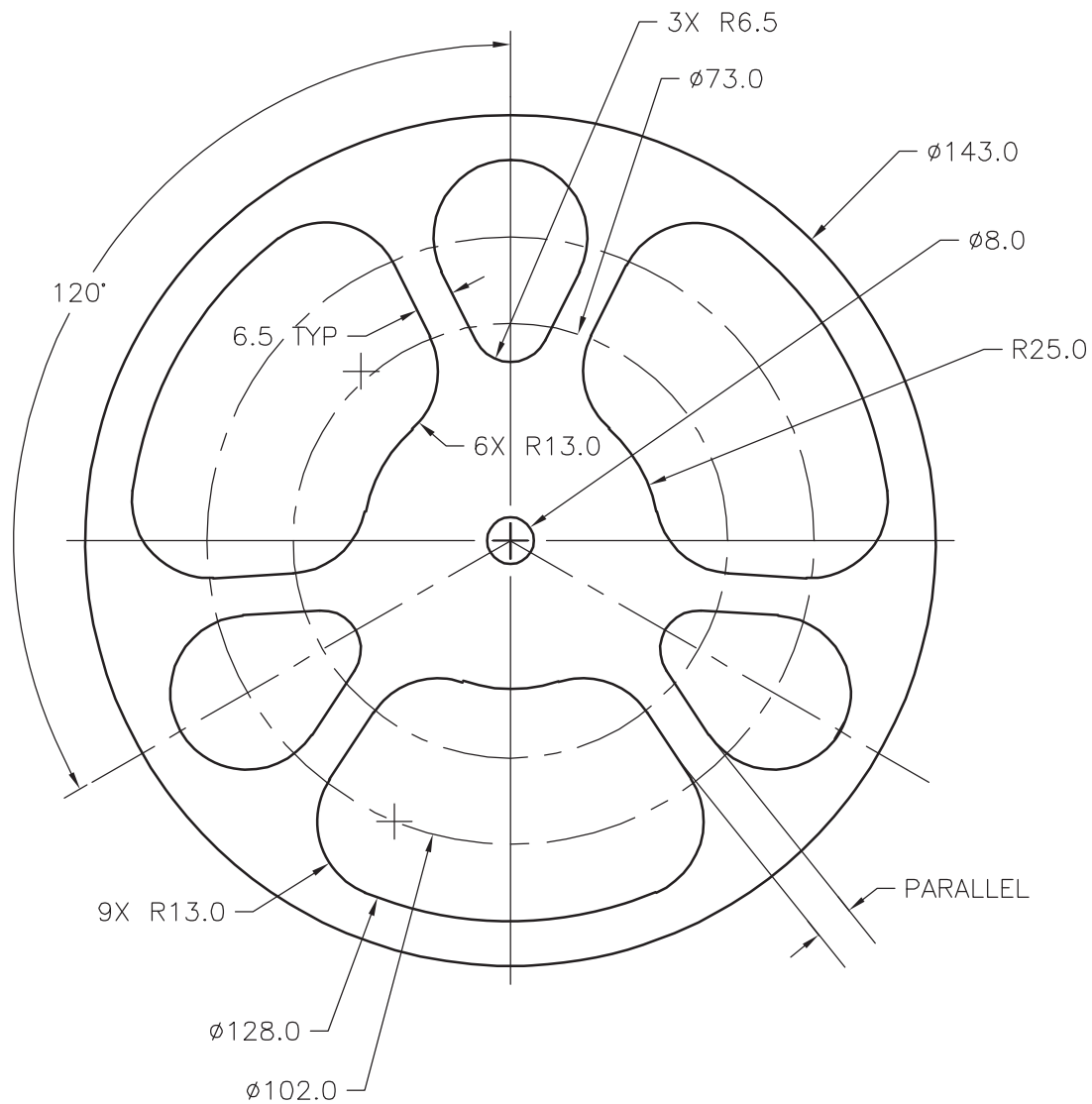
Adapted from the textbook Drafting Fundamentals *by Scott, Foy, and Schwendau*

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679

L2 PROBLEM 42

Create and fully dimension the drawing of a slotted wheel. Use a drawing area of 17×11.
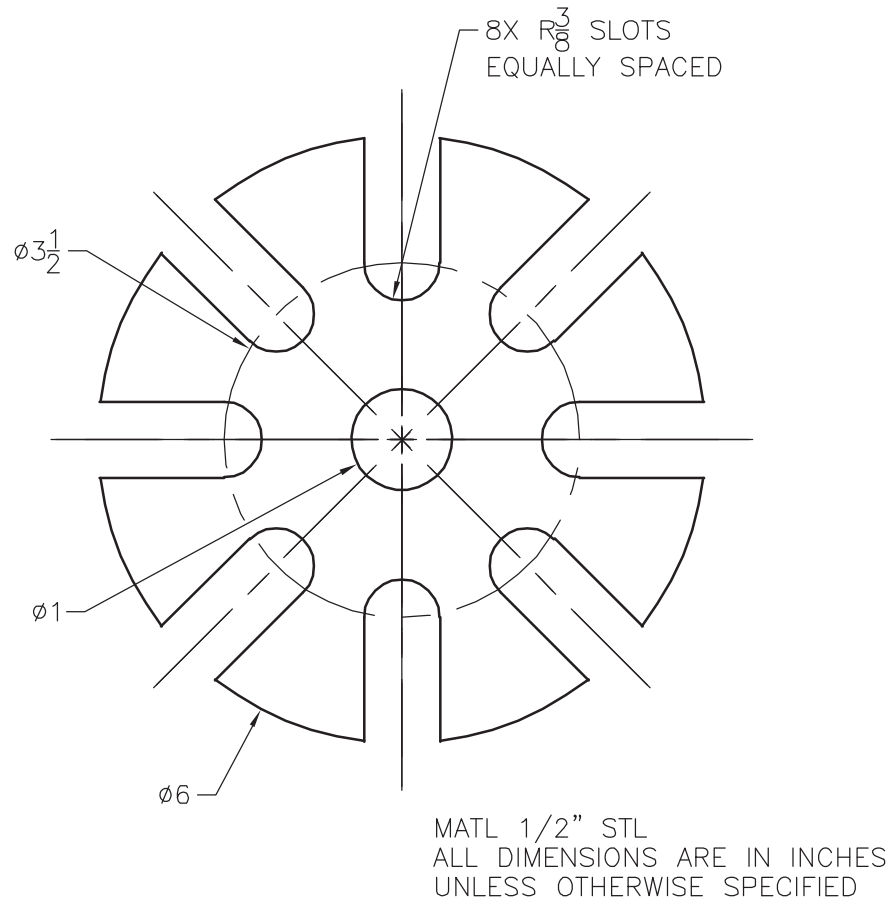
Adapted from the textbook Drafting Fundamentals *by Scott, Foy, and Schwendau*

Additional Problems

L2 PROBLEM 43

Create and dimension the drawing of a gasket.
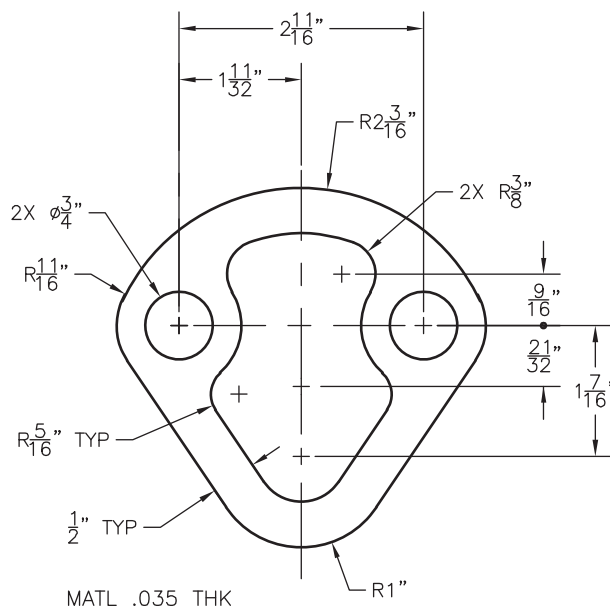
Adapted from the textbook Drafting Fundamentals *by Scott, Foy, and Schwendau*

L2 PROBLEM 44

Create and dimension the drawing of an 18-tooth cutter. Use a drawing area of 17 × 11.
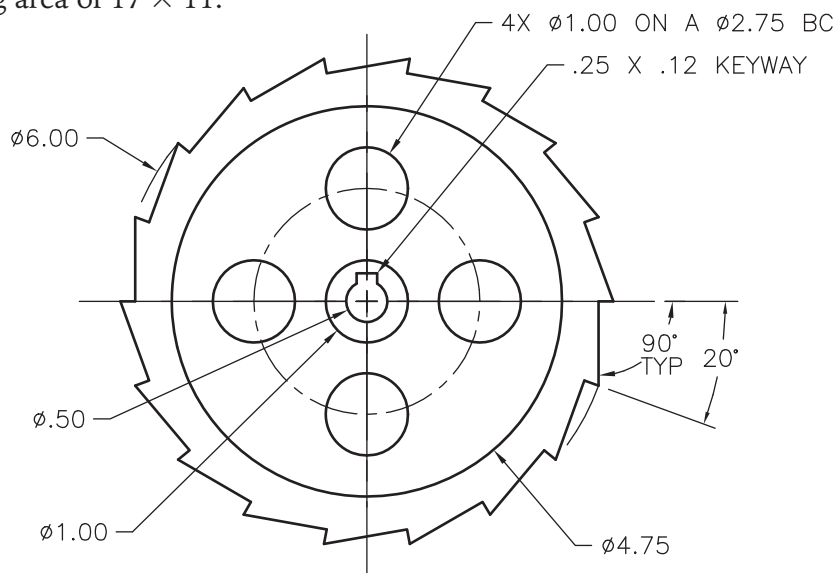
L2 PROBLEM 45

Create and dimension the top and front orthographic views of the slotted shaft.
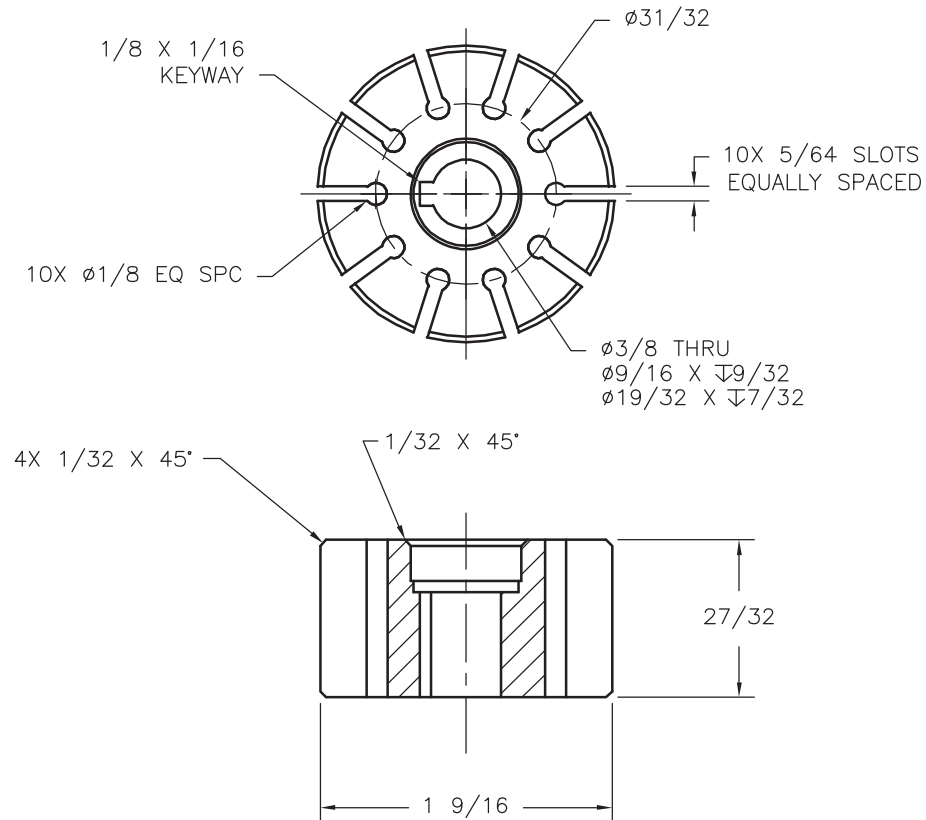
1/8 X 1/16 KEYWAY

ø31/32

10X 5/64 SLOTS EQUALLY SPACED

10X ø1/8 EQ SPC

ø3/8 THRU
ø9/16 X ⤓9/32
ø19/32 X ⤓7/32

4X 1/32 X 45°

1/32 X 45°

27/32

1 9/16

Adapted from the textbook **Drafting Fundamentals** *by Scott, Foy, and Schwendau*

Additional Problems

L2 PROBLEM 46

Create and dimension the drawing of an adjustable bracket.
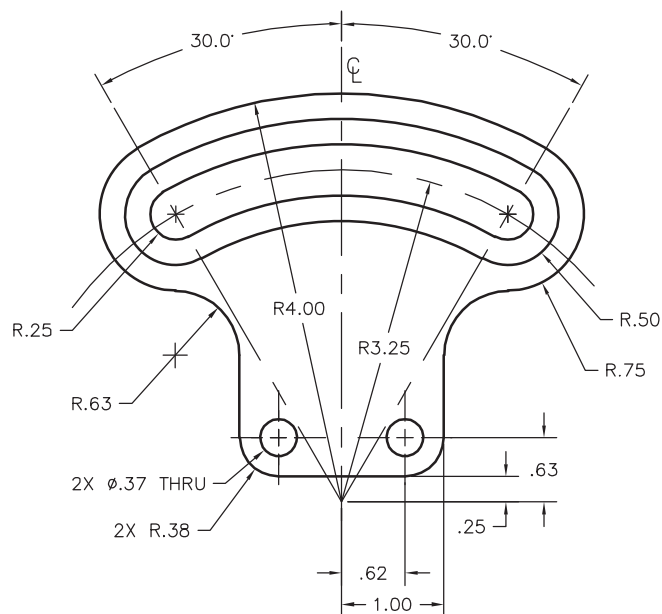
L2 PROBLEM 47

Create and dimension the drawing of a basketball backboard. Use a scale of 1″ = 1′.
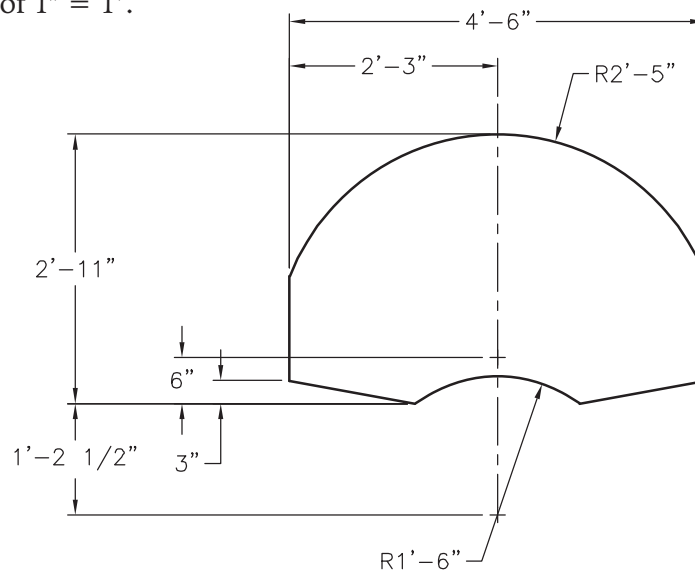
Problems 46 and 47 adapted from the textbook Drafting Fundamentals *by Scott, Foy, and Schwendau*

L2 PROBLEM 48

Create and dimension a complete drawing of the wall plaque.
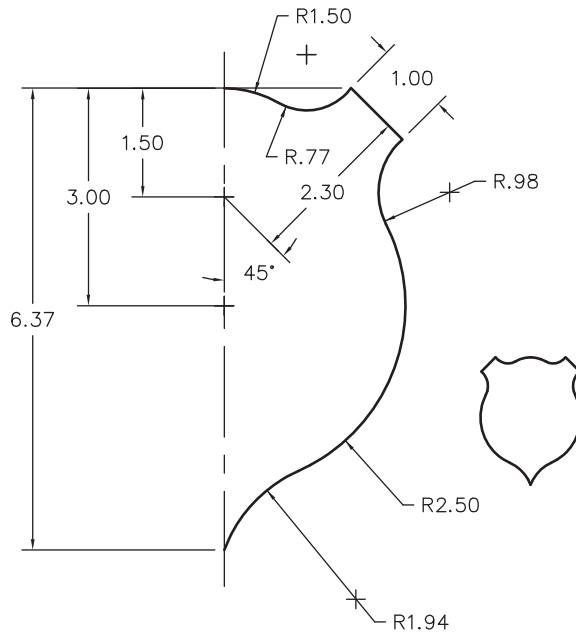
Courtesy of Mark Schwendau, Kishwaukee College

L2 PROBLEM 49

Create the electrical schematic drawing. Create blocks of each of the components and then insert them as needed.
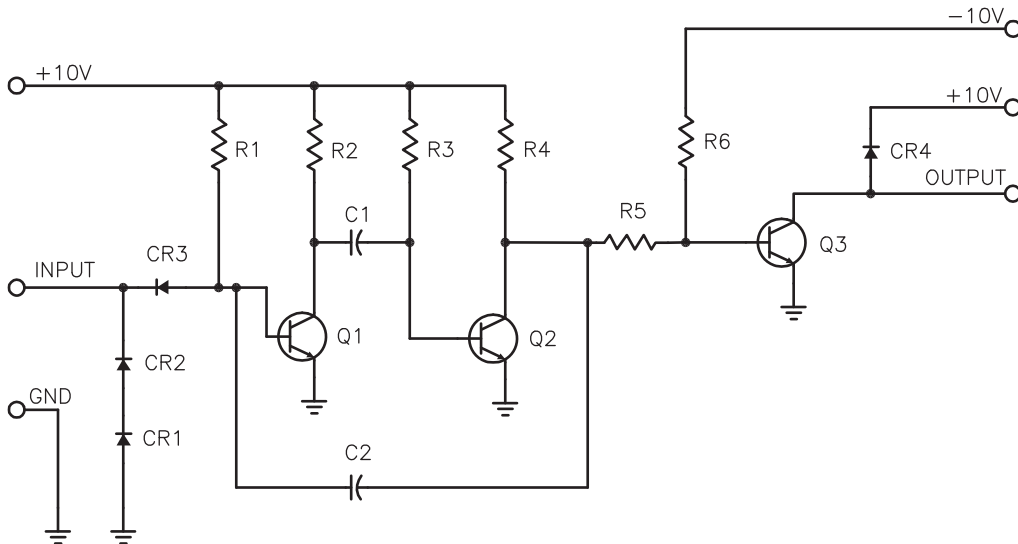
Additional Problems

L2   PROBLEM 50

Create and dimension the spacer. Use a drawing area of 17×11.
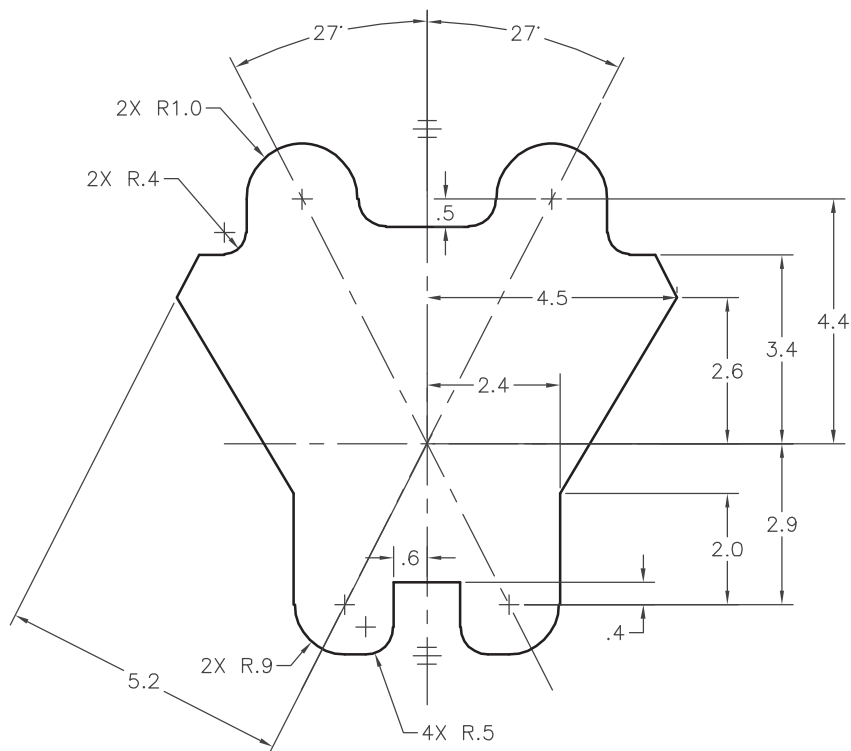
Courtesy of Mark Schwendau, Kishwaukee College

Additional Problems

L2 PROBLEM 51

Create a single view of the spacer. Dimension the drawing and add a note indicating the thickness of the spacer.
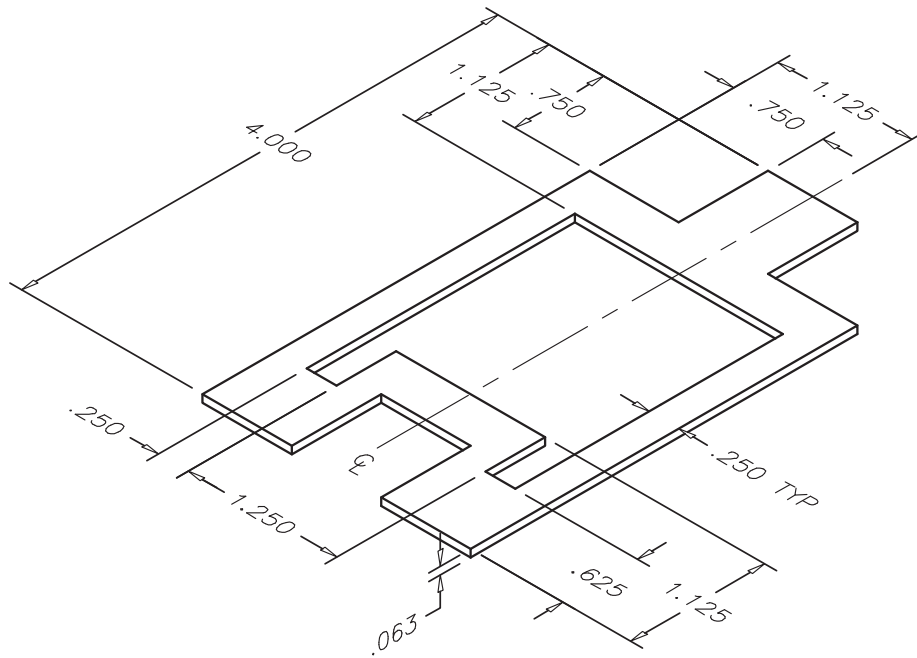
Additional Problems

L2 PROBLEM 52

Create the front and right-side views of the centering bushing. Include dimensions.

R.240

R.120 TYP

ø2.000

1.120

1.240

3X ø.375
ø.750 ⌄.25
ON A ø3.376 BC

ø1.375 ±.001

2.625

.625

ø4.750

Courtesy of Steve Huycke, Lake Michigan College

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687
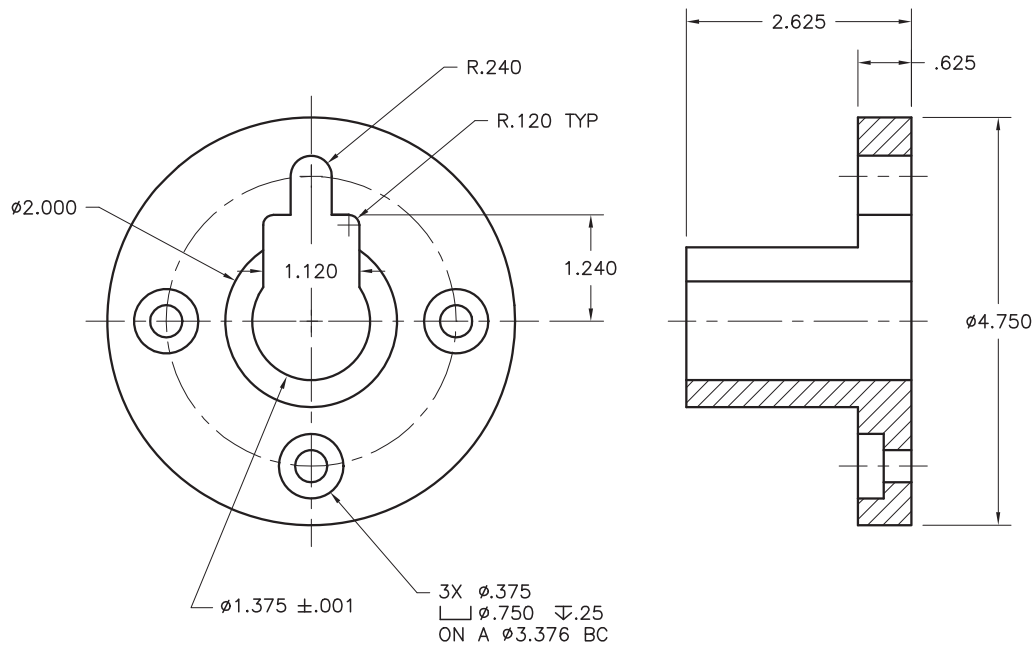
L2 PROBLEM 53

Create and fully dimension the views of the rod support. Determine the drawing area on your own, based on space requirements.
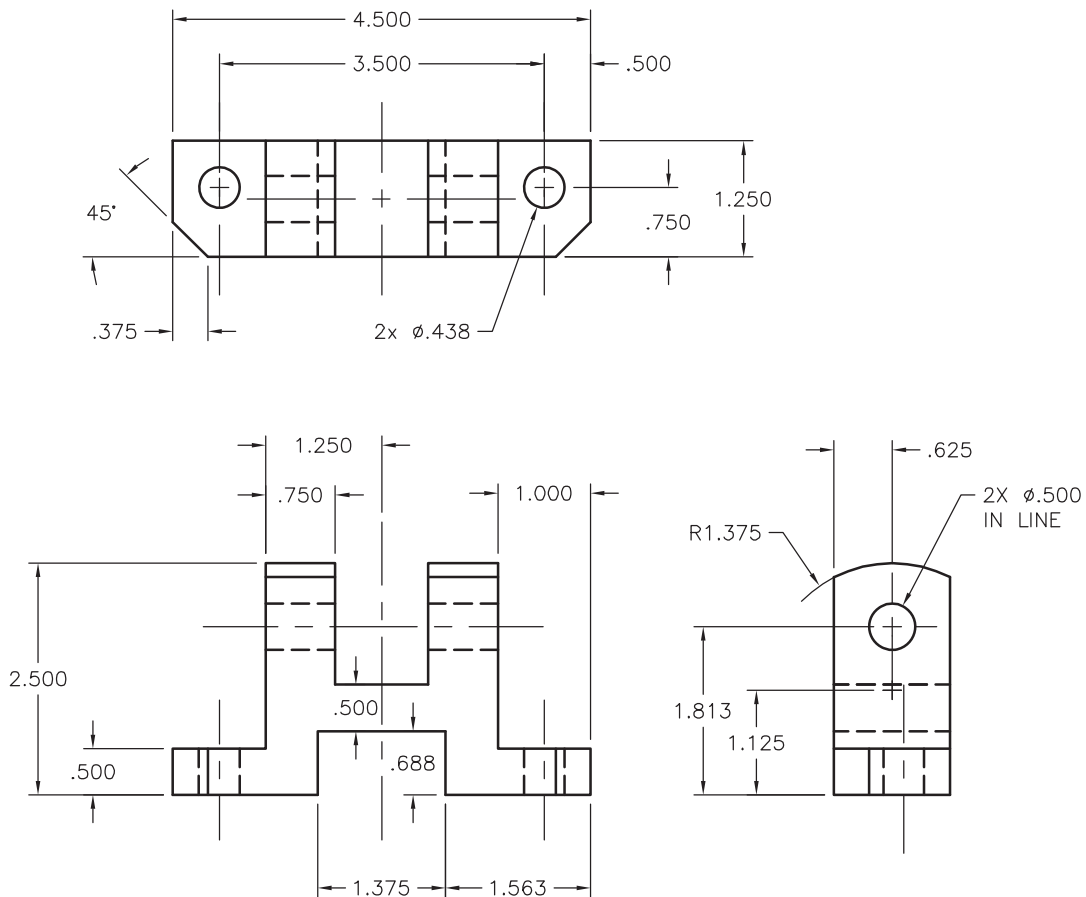
Courtesy of Steve Huycke, Lake Michigan College

Additional Problems

L2 PROBLEM 54

Create a drawing of the template and dimension it as shown.
Determine the drawing area based on space requirements.
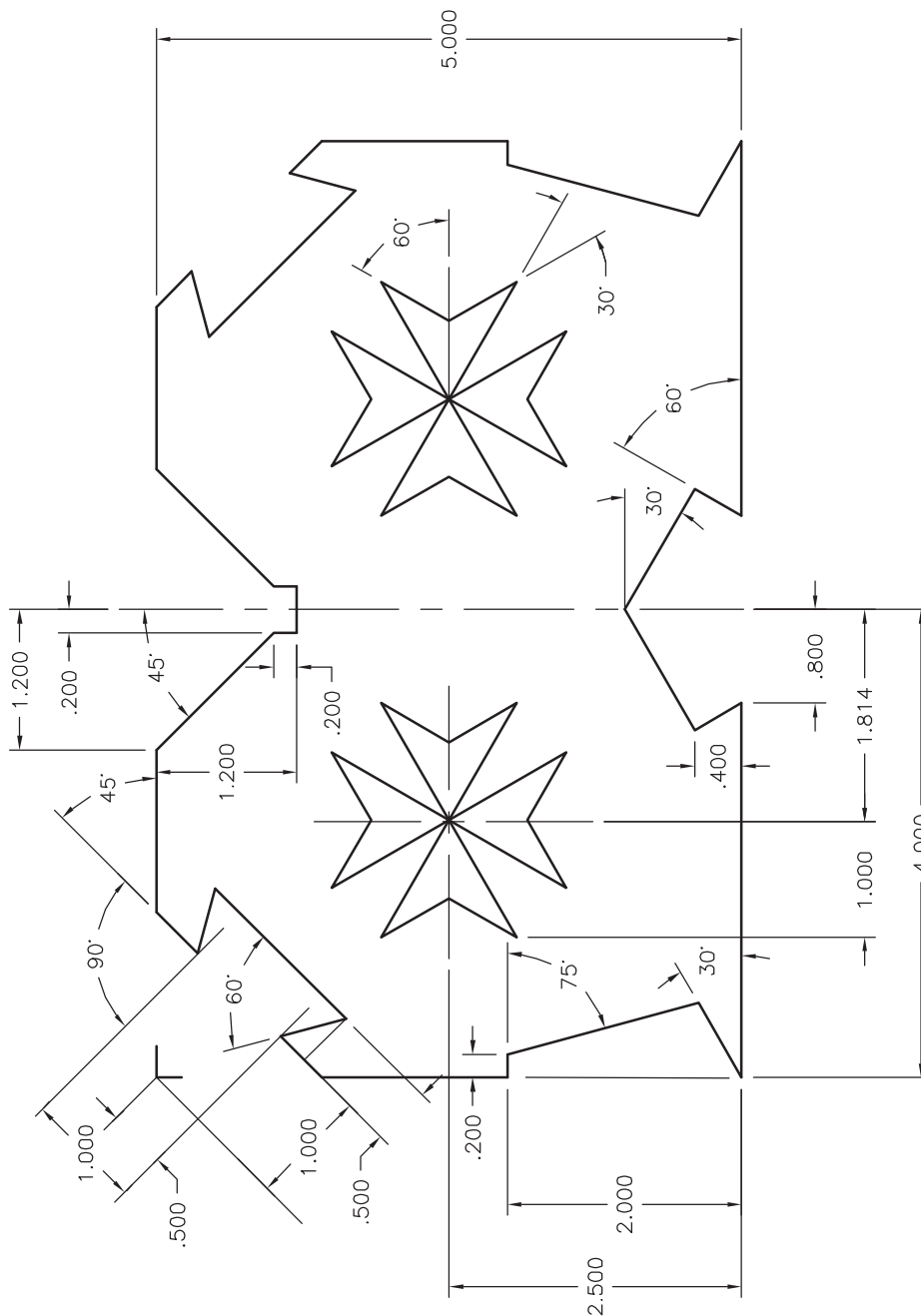
Courtesy of Steve Huycke, Lake Michigan College

L2 PROBLEM 55

Create the top view and front full section view of the arm. Be sure to include the detail.
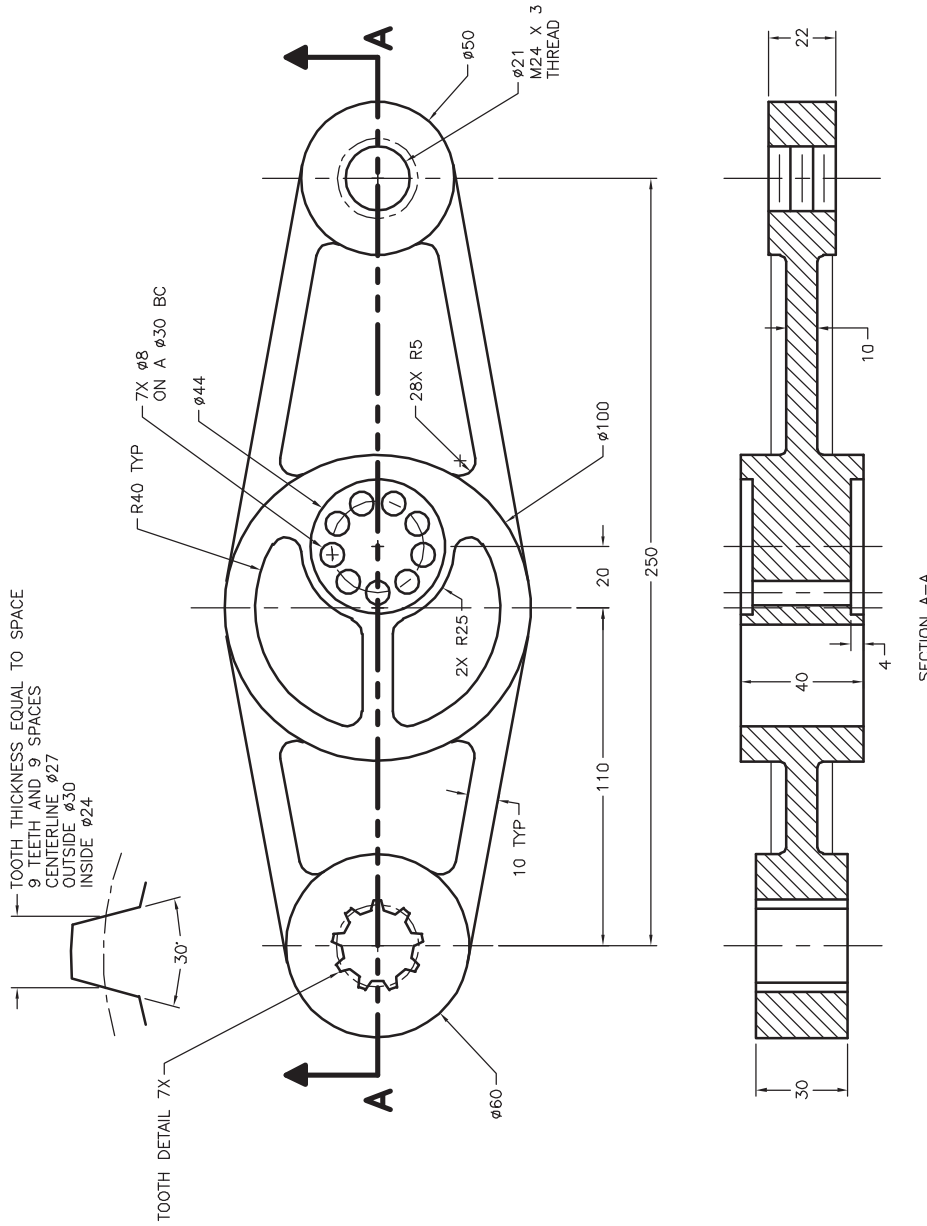
Courtesy of Alan Fitzell, Central Peel Secondary School

Additional Problems

L2 PROBLEM 56

Create and dimension the irregular curve. Determine the drawing area based on space requirements.
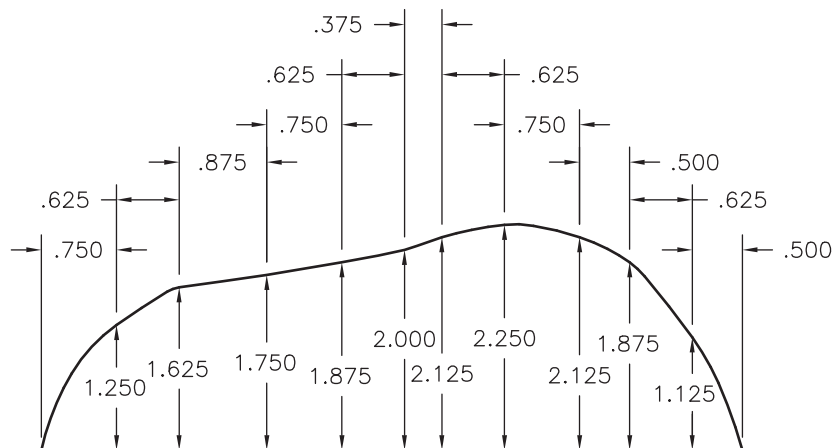
L2 PROBLEM 57
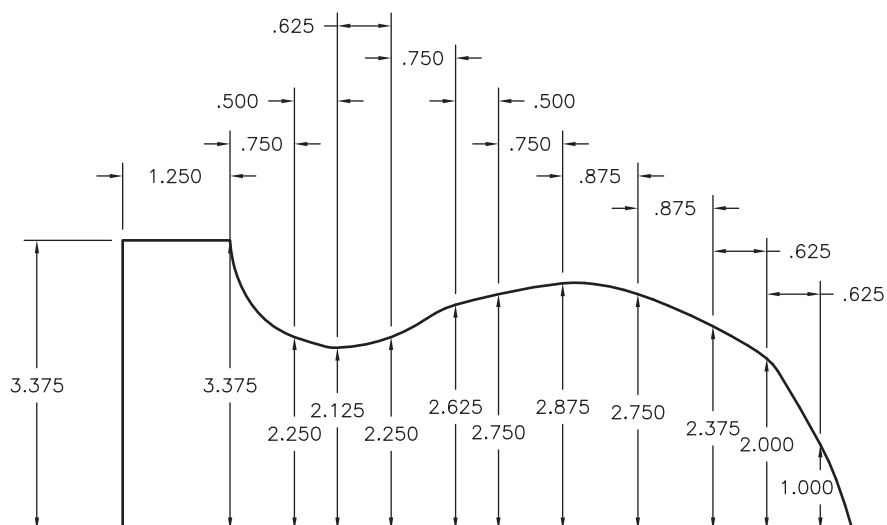
Create and dimension the irregular curve. Determine the drawing area based on space requirements.

L2 PROBLEM 58

Create and fully dimension the two views of the crank handle using decimal inches to three decimal places. Estimate all dimensions.
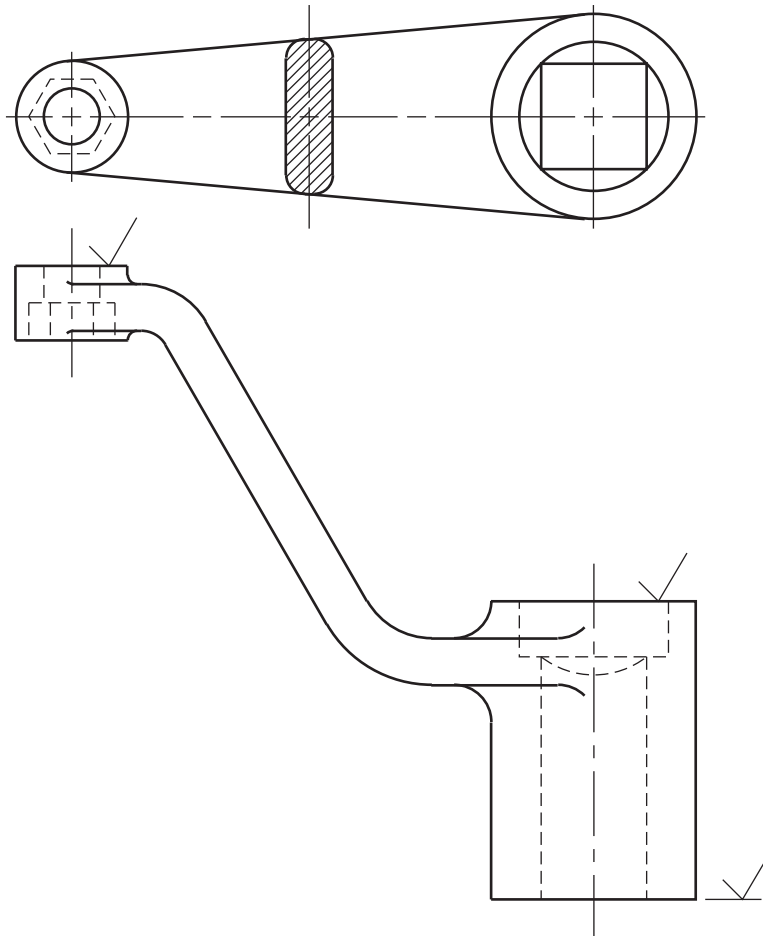
Courtesy of Joseph K. Yabu, San Jose State University

Additional Problems

L2 PROBLEM 59

Create and fully dimension the wheel cover.
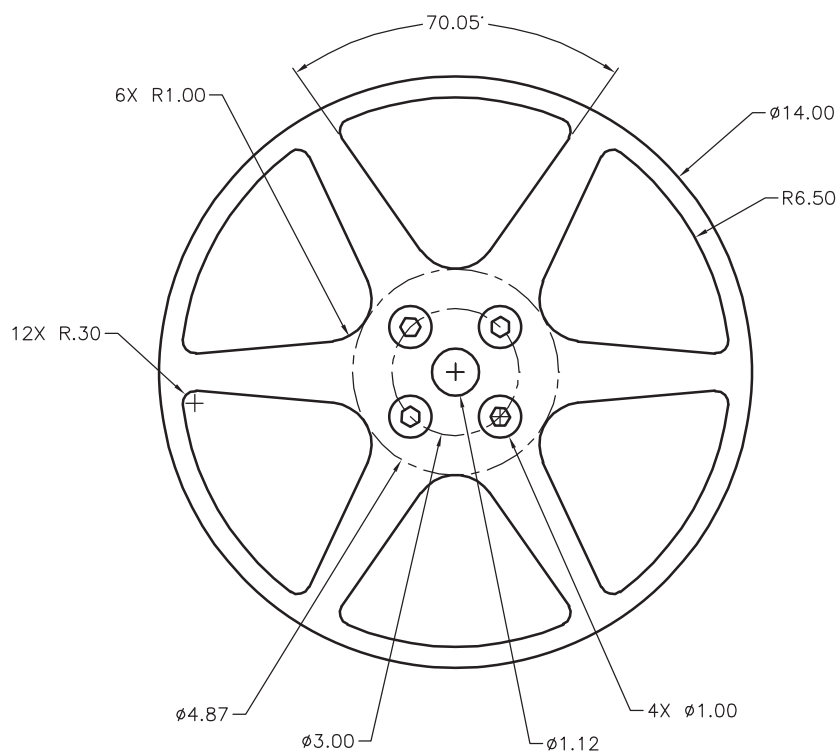
L2 PROBLEM 60

Create drawings of the wheel covers. Estimate all dimensions.
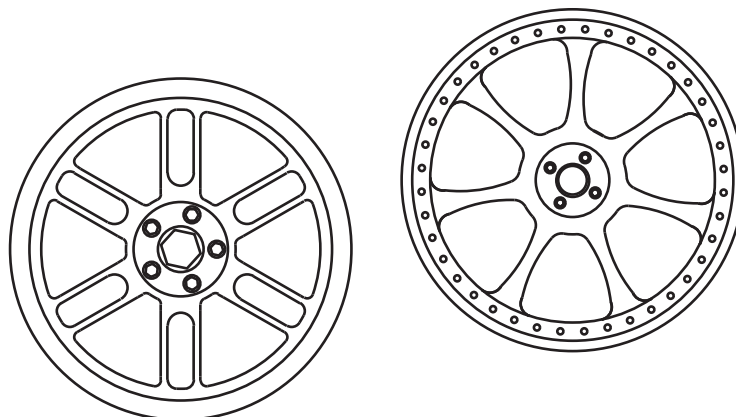
Problems 59 and 60 courtesy of Joseph K. Yabu, San Jose State University

L2 PROBLEM 61

In a steel frame building, a bracket is to be connected to a column as shown. The front and right-side views of the structural detail are given below. The W 14 × 26 wide flange beam is to be bolted to a 9 × 5 × ³⁄₈ plate which, in turn, is to be welded to the W 8 × 35 wide flange beam. The dimensions for the two beams are taken from the American Institute of Steel Construction's *Manual of Steel Construction*. Draw the two views.
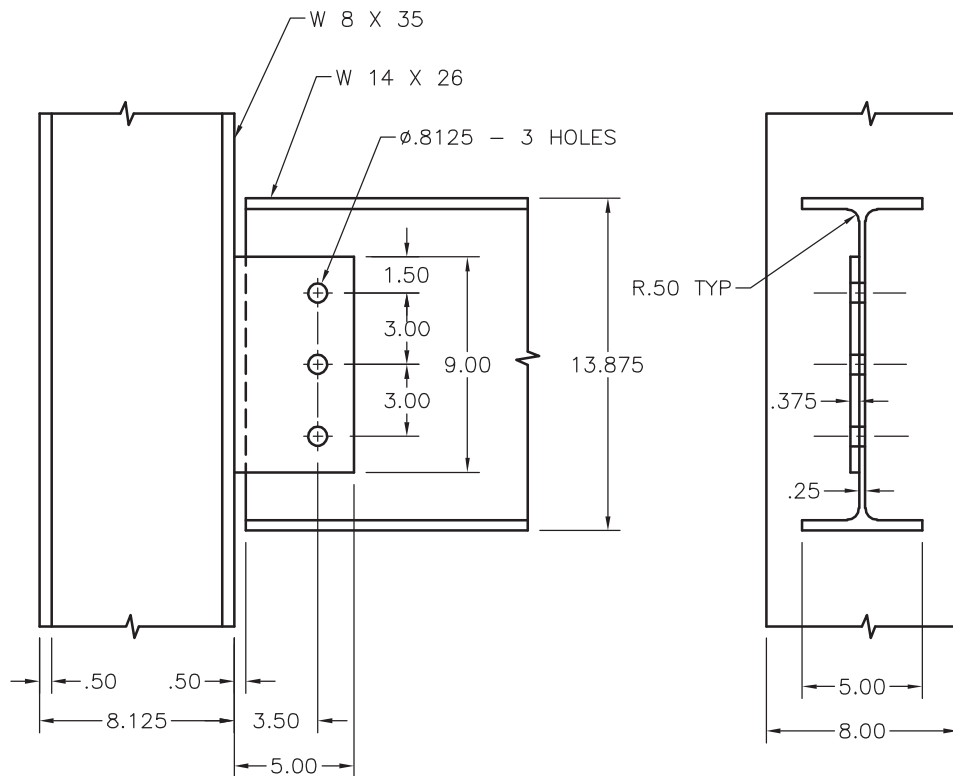
Courtesy of Gary J. Hordemann, Gonzaga University

Additional Problems

L2 PROBLEM 62

Draw the top and front views of the barrier block. Study the top view and figure out how to minimize drawing and maximize the use of replicating commands such as **COPY**, **MIRROR**, and **ARRAY**.
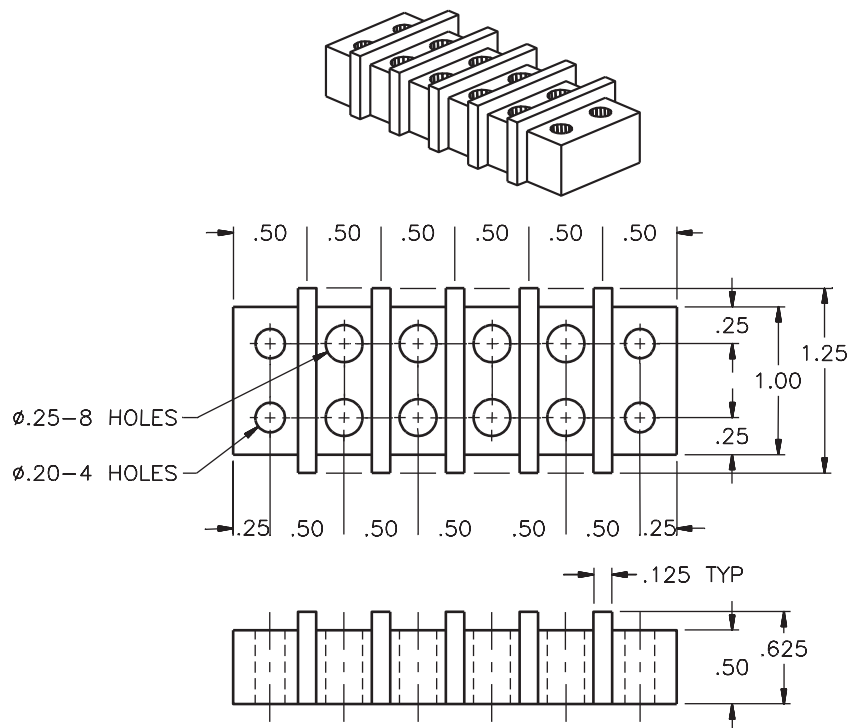
Courtesy of Gary J. Hordemann, Gonzaga University

L2 PROBLEM 63

The front view of the link shown below is amazingly easy to draw if you draw the large circle first; then draw long horizontal and vertical lines snapped to the center of the circle. Offset the lines to locate the centers of the two smaller circles; if necessary, use a zero-distance chamfer or zero-radius fillet to extend the lines until they meet. Insert the two small circles, snapped to the intersections of the lines. Offset again to construct the lines composing the lower left corner and then fillet that corner. Draw a line snapped tangent to the upper small circle and the large circle; use the **FILLET** command to insert the arcs joining the small and large circles. Finally, insert the octagon snapped to the center of the large circle.
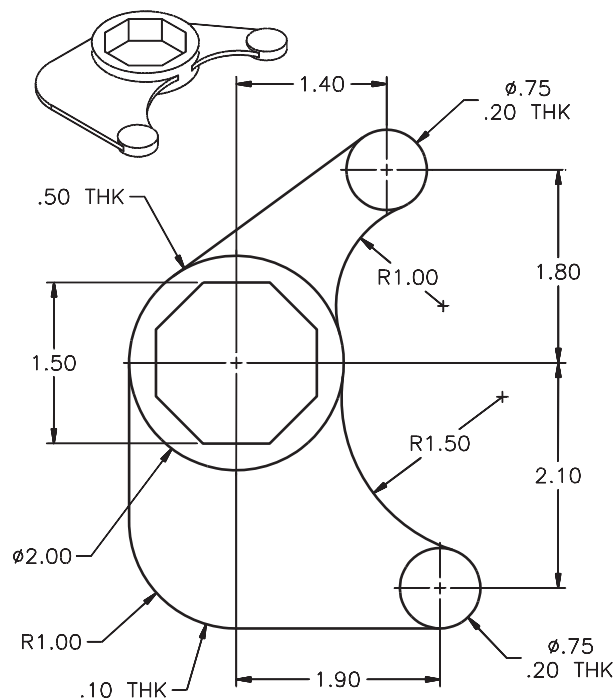
Courtesy of Gary J. Hordemann, Gonzaga University

Additional Problems

L2 PROBLEM 64

Draw the top view of the stanchion shown below.
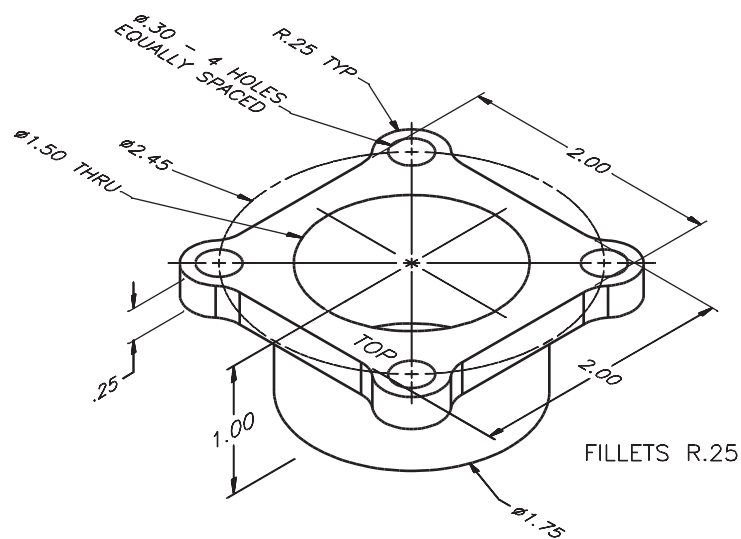
L2 PROBLEM 65

A drawing of part of the 20 scale from an engineer's scale is shown below. Using the **ARRAY** command, draw the 20 scale. Then draw 30 and 50 scales.
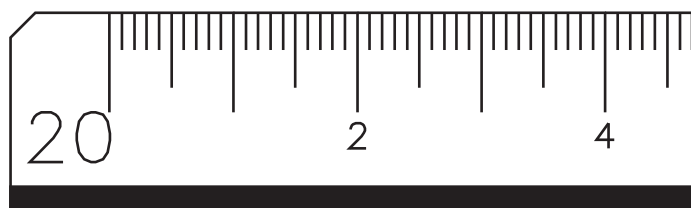
Problems 64 and 65 courtesy of Gary J. Hordemann, Gonzaga University

L2 **PROBLEM 66**

Draw the front view of the swivel link shown below. Draw half the object using **LINE**, **CIRCLE**, **OFFSET**, and **FILLET**. Construct the external fillets using circles and lines snapped to tangents. Then use **BREAK** to trim the circles to arcs. Use **MIRROR** to complete the other half.
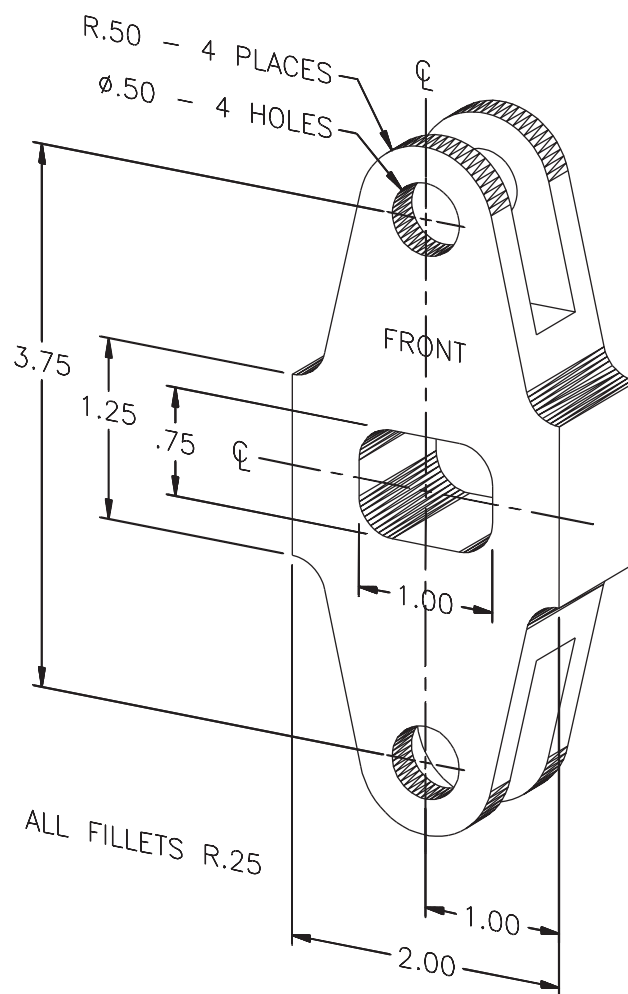
Courtesy of Gary J. Hordemann, Gonzaga University

Additional Problems

L2 PROBLEM 67

The footprints for two cordless mice on a mouse docking (recharging) station are shown below. Draw the two footprints *without* using the **FILLET** command by drawing circles for all the rounded ends. Use the **TTR** option of the **CIRCLE** command for the R3.50 arc. Then use **TRIM** to create the final objects. Would it be easier to use filleted lines? Try it by drawing the outlines again, this time without using the **CIRCLE** command.
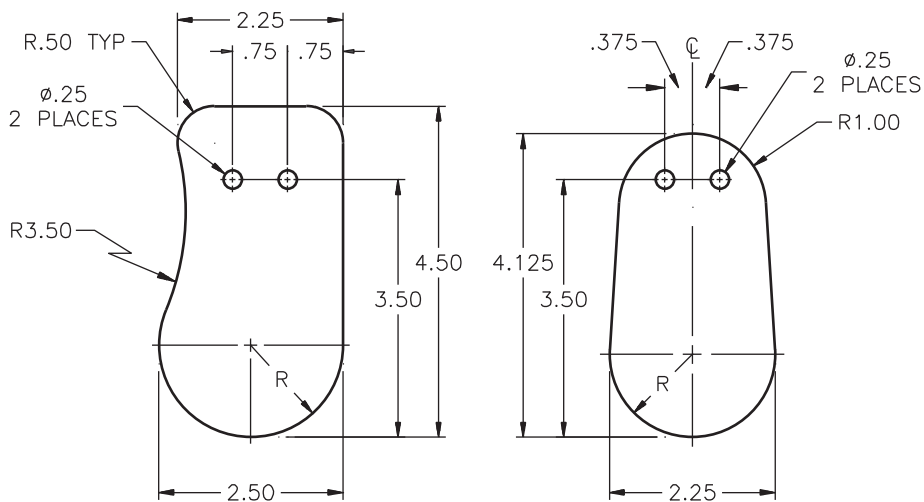
Courtesy of Gary J. Hordemann, Gonzaga University

L2 PROBLEM 68

Draw the front view of the splined separator block shown below. A suggested approach: Draw two circles bounding the spline teeth. Draw a horizontal line through the center and offset it to form the boundary of one tooth. Trim the circles and lines to form one tooth and array it about the center. Using the **OFFSET** command, create $1/12$ of the object. Use **MIRROR** to create $1/6$ of the object, and then array it about the center to finish it. When you have finished the drawing, use **SCALE** to reduce the spline teeth so that the outer diameter is **12**. Use **STRETCH** to narrow the outer gaps from **8** to **6**.
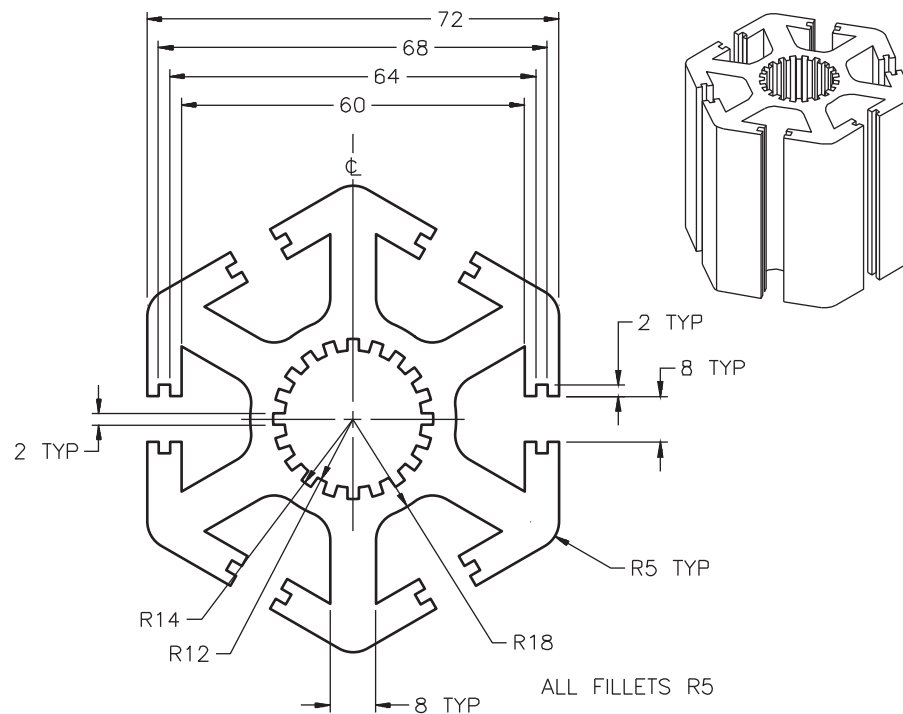
Courtesy of Gary J. Hordemann, Gonzaga University

Additional Problems

L2 PROBLEM 69

Create and dimension the two views of the cam shown below. The spacing between the dots is **.125**.
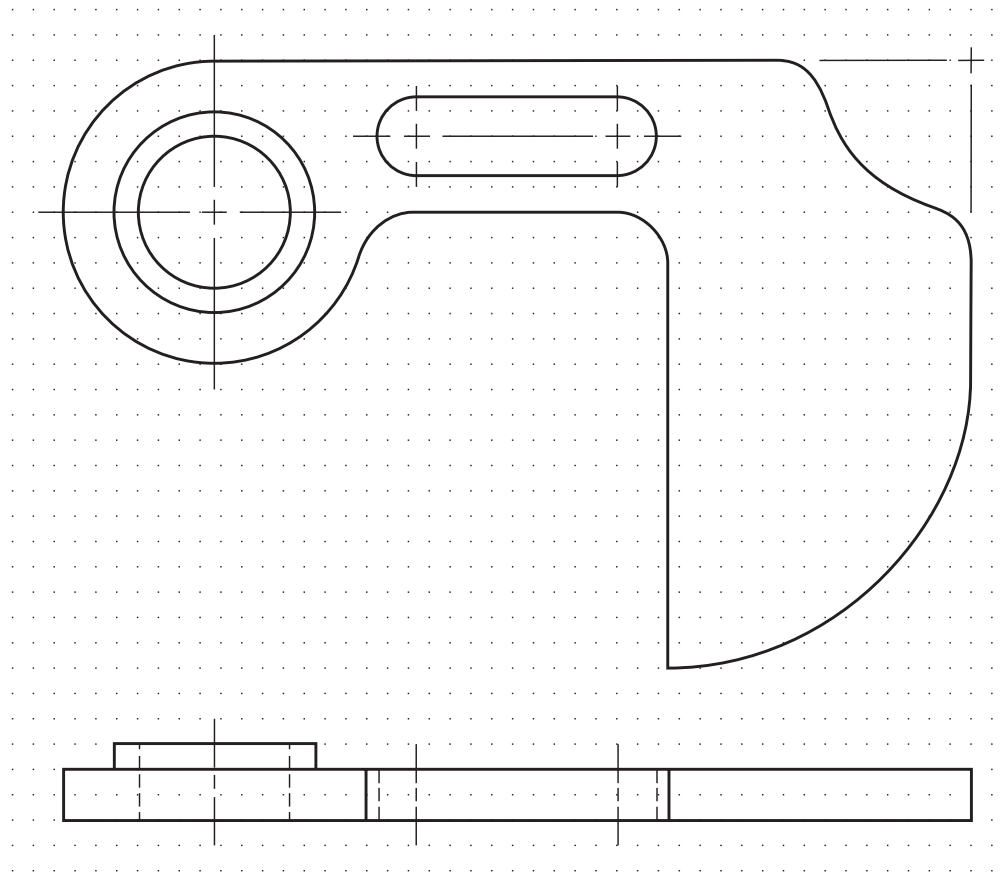
Courtesy of Gary J. Hordemann, Gonzaga University

L2 PROBLEM 70

Create a new text style called **Inside** with the following settings:

- Dimension line spacing: **.30**
- Dimension line color: **ByLayer**
- Extension line extension: **.06**
- Extension line origin offset: **.05**
- Extension line color: **ByLayer**
- Arrow size: **.12**
- Fit: **Text only**
- Unit linear precision: **0.00**
- Unit angular precision: **0**
- Suppress leading zeros
- Text style: **ROMANS** (font **romans.shx**)
- Text height: **.12**
- Text gap: **.06**
- Text color: **Blue**

Using the new dimension style, draw and dimension the two views of the clock blank shown below. The spacing between dots is **.125**.
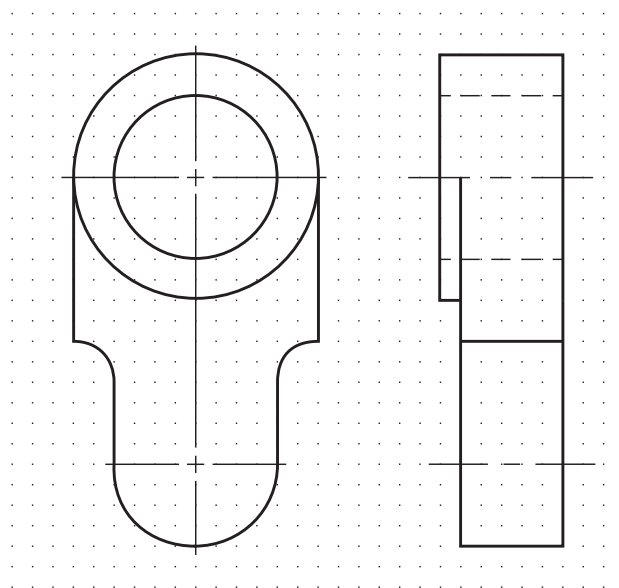
Courtesy of Gary J. Hordemann, Gonzaga University

Additional Problems

L2 PROBLEM 71

Draw each of the three thread representations shown below and write them to files using the **WBLOCK** command. The external threads are most easily drawn using the **ARRAY** command. The distance between dots in the grid is not important, but **.125** should work well.

Courtesy of Gary J. Hordemann, Gonzaga University

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Additional Problems

L2. Uses Intermediate
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703
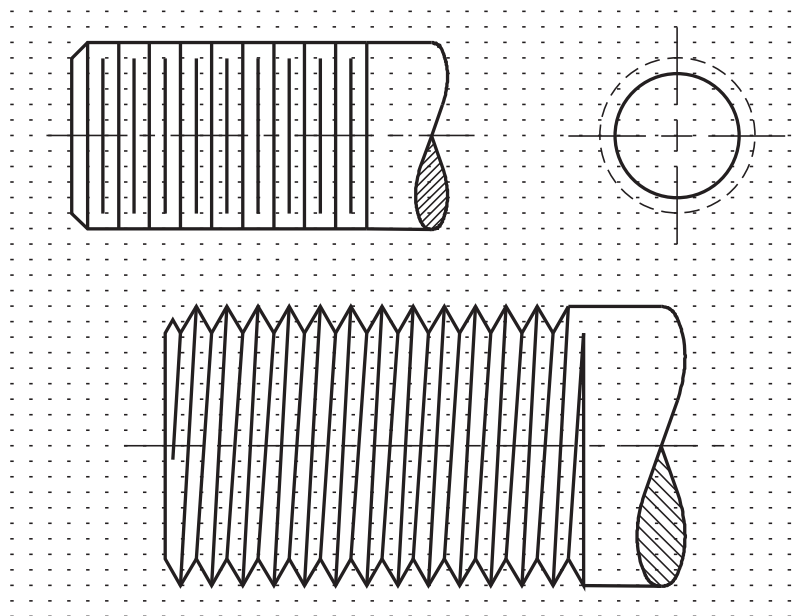
L2 PROBLEM 72

Imagine an object composed of:

  a. A cube measuring **2″** on each side.

  b. A four-sided rectangular pyramid with a height of **1″**. The pyramid sits on top of the cube and the four edges of its base align with the four edges of the cube's top surface.

  c. A hole, **1¹⁄₂″** in diameter, centered in the front cube face and having a depth of **¹⁄₂″**.

  d. Another hole, coaxial with the first but **1″** in diameter, extending through the object.

Three ways of drawing this object are shown below. They are called *obliques*. Simply put, they are created by drawing one face at its full size and shape with the receding lines at an angle of **45°**. The *cabinet* oblique has half-size receding lines; the *cavalier* oblique has full-size receding lines. The receding lines in *general* obliques vary; those on the one shown here are **0.707**-size lines. Draw the three obliques.

CABINET   CAVALIER   GENERAL
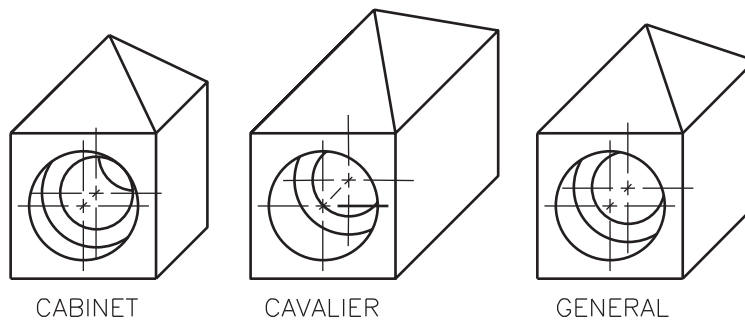
Courtesy of Gary J. Hordemann, Gonzaga University

Additional Problems

L2 PROBLEM 73

Use 3D faces and a sphere to create the object shown below. Then use solid primitives with the **UNION**, **SUBTRACT**, and/or **PRESSPULL** commands to create a completely solid version. The large cube is **2** units on each side and the small cube is **1** unit on each side. The sphere has a diameter of **.5**. The total height of the object, including the sphere, is **4** units.
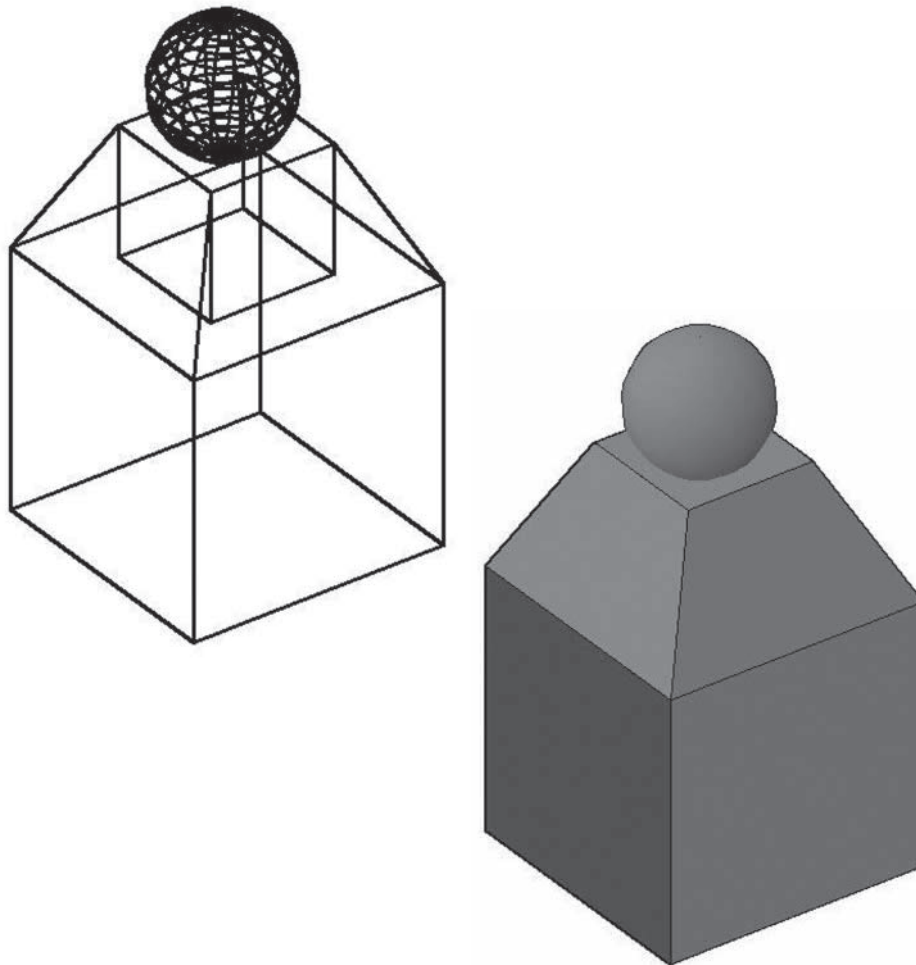
Courtesy of Carole J. Williams, South Harrison High School

L2 PROBLEM 74

Create the chess pawn piece as a solid model. Use a polyline and the **REVOLVE** command. Estimate all dimensions. Then assign a material to the pawn and render it. Save both the model and the rendered image.
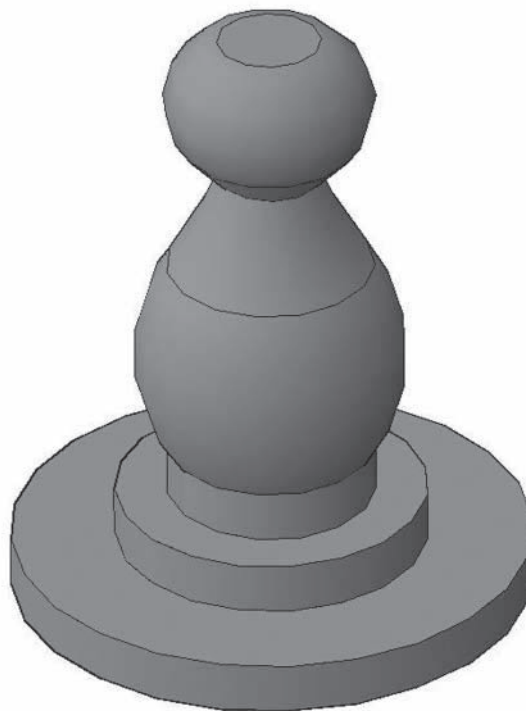
Courtesy of Carole J. Williams, South Harrison High School

Additional Problems

L2 PROBLEM 75

Create the floor plan for the first story of a residence. Determine the drawing area according to space requirements. Estimate the dimensions that are not given.
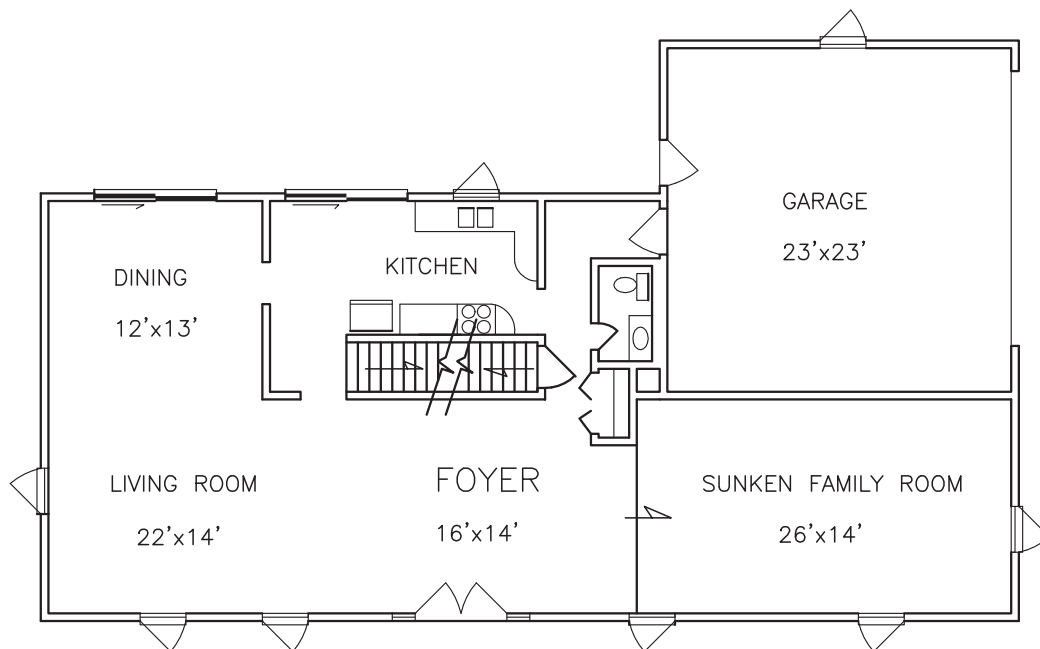
Courtesy of Mark Schwendau, Kishwaukee College

L2 PROBLEM 76

Create the wall section.

WOOD SIDING
1/2" DRYWALL
2X4 STUDS @ 16" O.C.
CARPETING
3" RANCH MOLDING
2X4 SOLE PLATE
3/4" PLYWOOD DECK
1X4 BRIDGING
2X10 JOIST
2X10 HEADER
1/2" INSUL. BOARD
2X6 SILL
1/2" ANCHOR BOLT
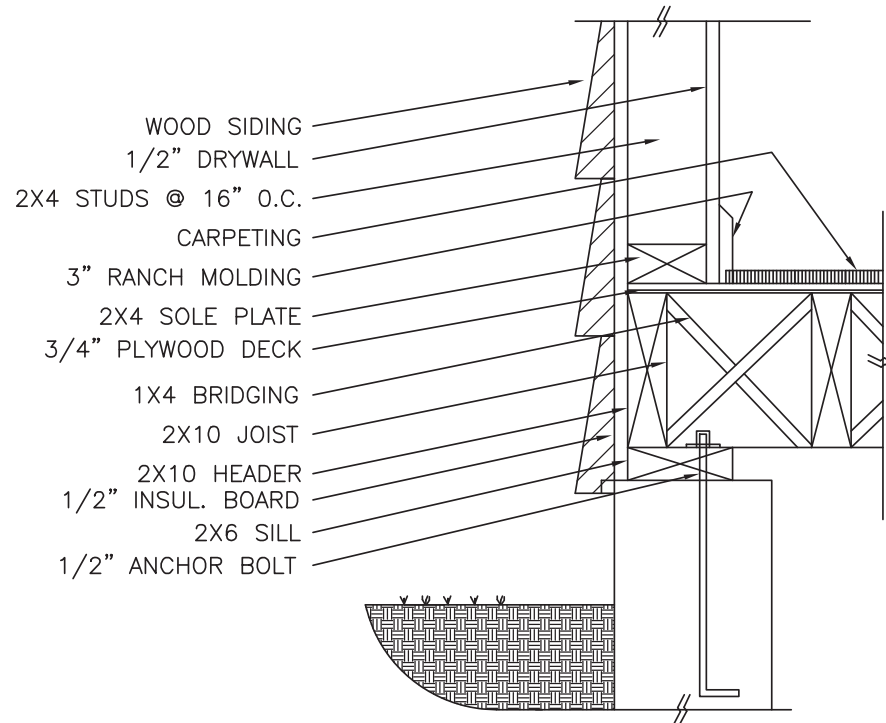
Additional Problems

L2 PROBLEM 77

Create a dimension style named **Structural** that uses feet and inches, oblique arrowheads, and text placed above the dimension lines. Draw and dimension the foundation detail shown below. The hatch patterns are **Earth**, **ANSI37**, and **AR-CONC**.
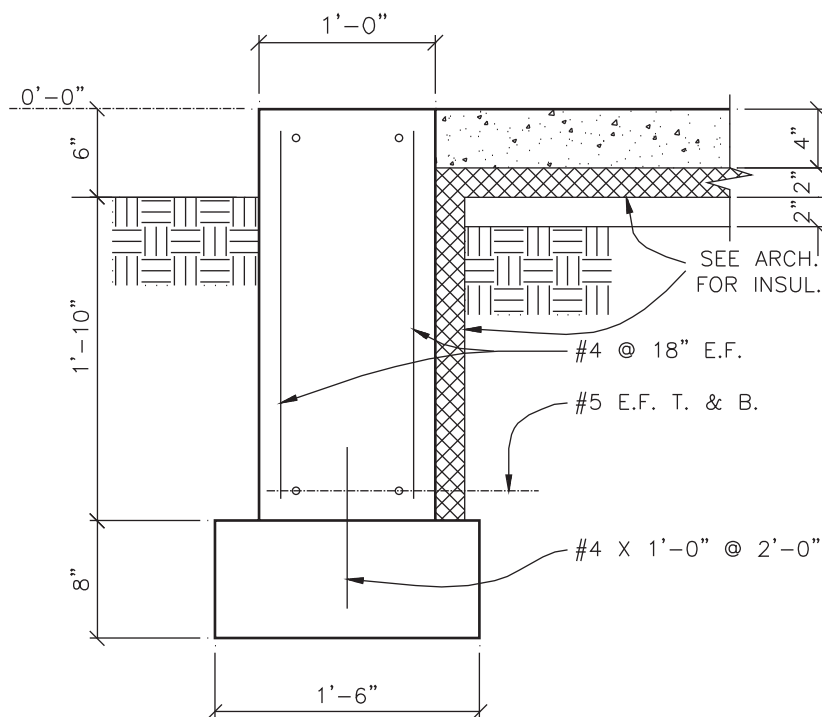
Courtesy of Gary J. Hordemann, Gonzaga University

L2 PROBLEM 78

Create and dimension an isometric view of the base.
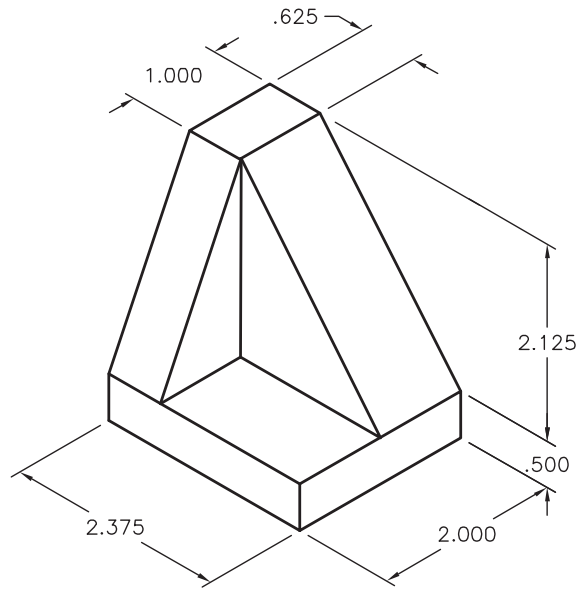
L2 PROBLEM 79

Create and dimension an isometric view of the wedge block.
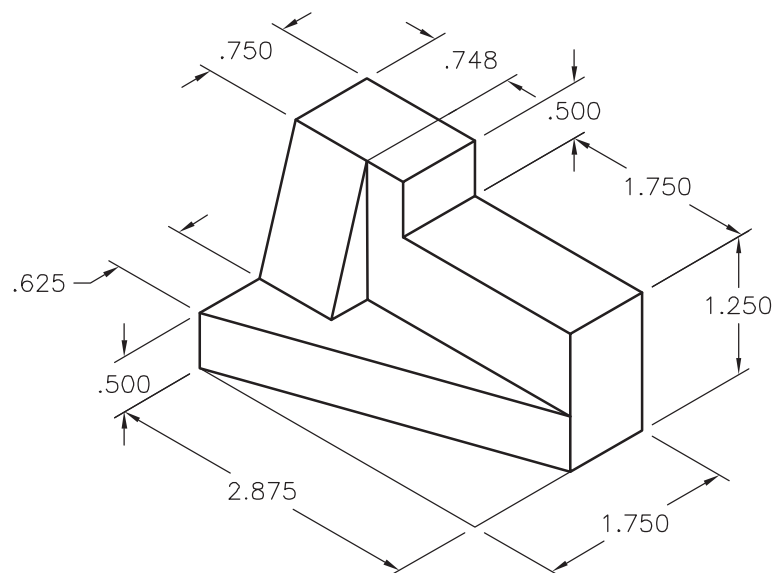
Additional Problems

L3 PROBLEM 80

Create solid models of the table and vase. Estimate all dimensions. Display the model in the **Conceptual** visual style.
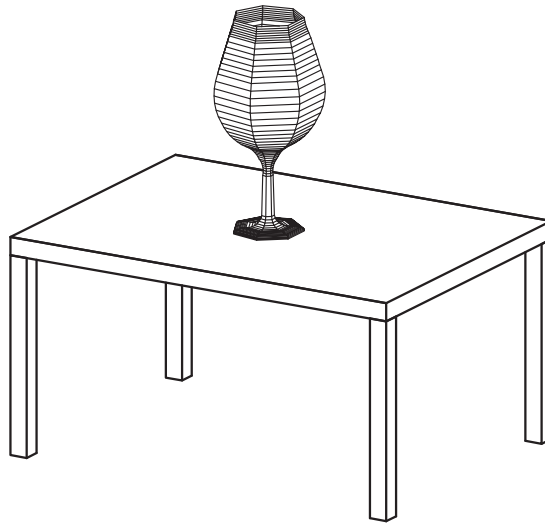
L3 PROBLEM 81

Create the top and side views of the fighter aircraft. Estimate all dimensions.
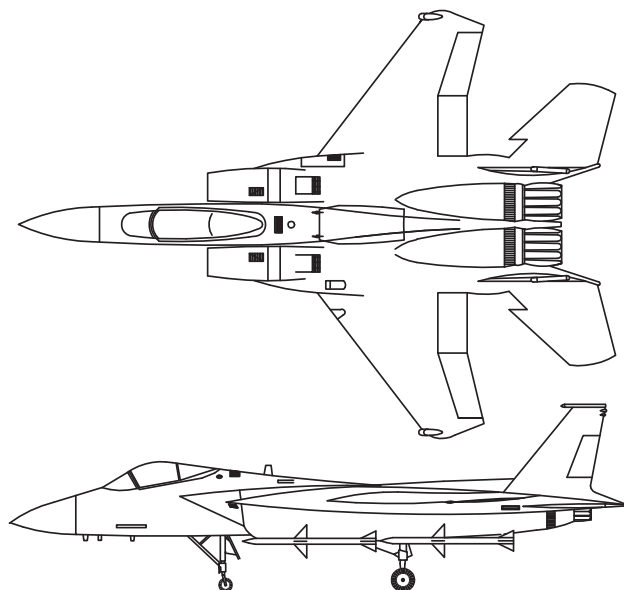
Problem 81 courtesy of Matt Melliere and Ronald Weseloh, Red Bud High School

L3 PROBLEM 82

Create the landscape drawing as shown. Create blocks of the trees and shrubs and insert them as shown. Estimate all dimensions.

Courtesy of Mills Brothers Landscape and Nursery, Inc.

712

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Additional Problems

**L3. Uses Advanced
AutoCAD and
Problem–Solving Skills**

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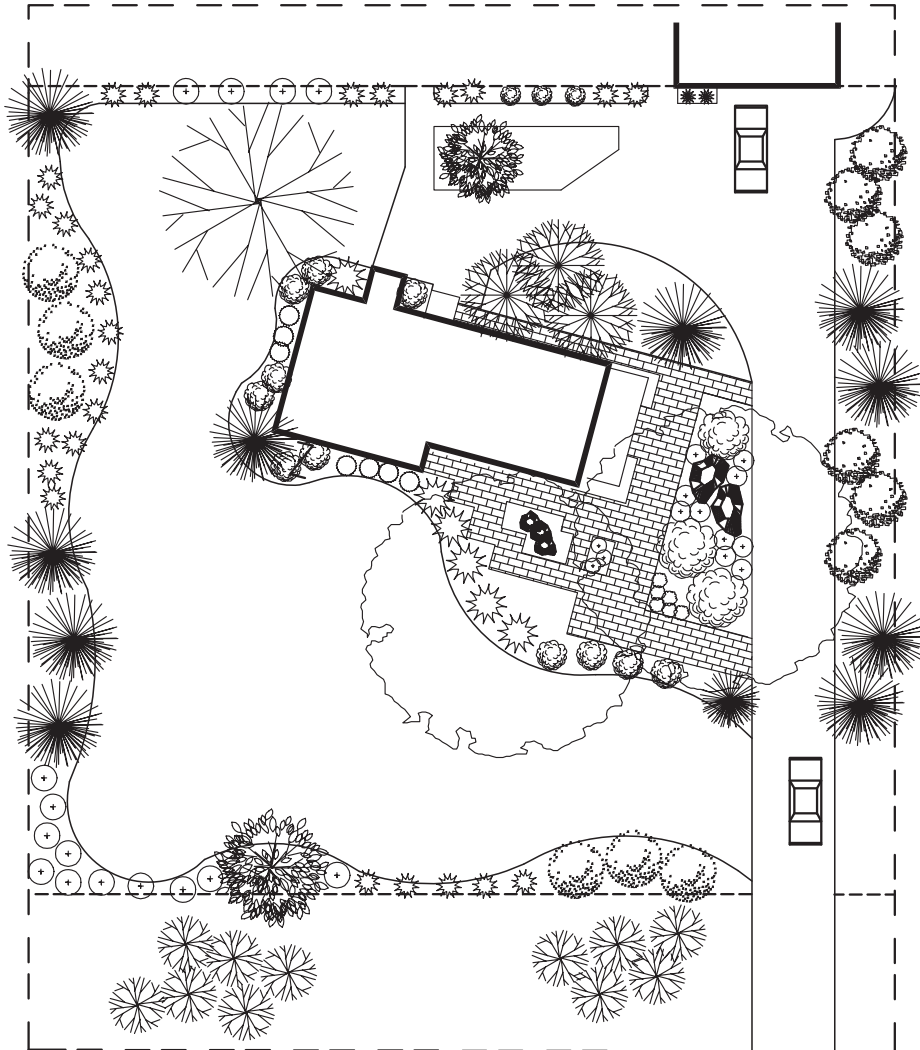
Additional Problems

L3 PROBLEM 83

Create the design elevation. Create blocks of the trees and shrubs and insert them as shown. Estimate all dimensions.
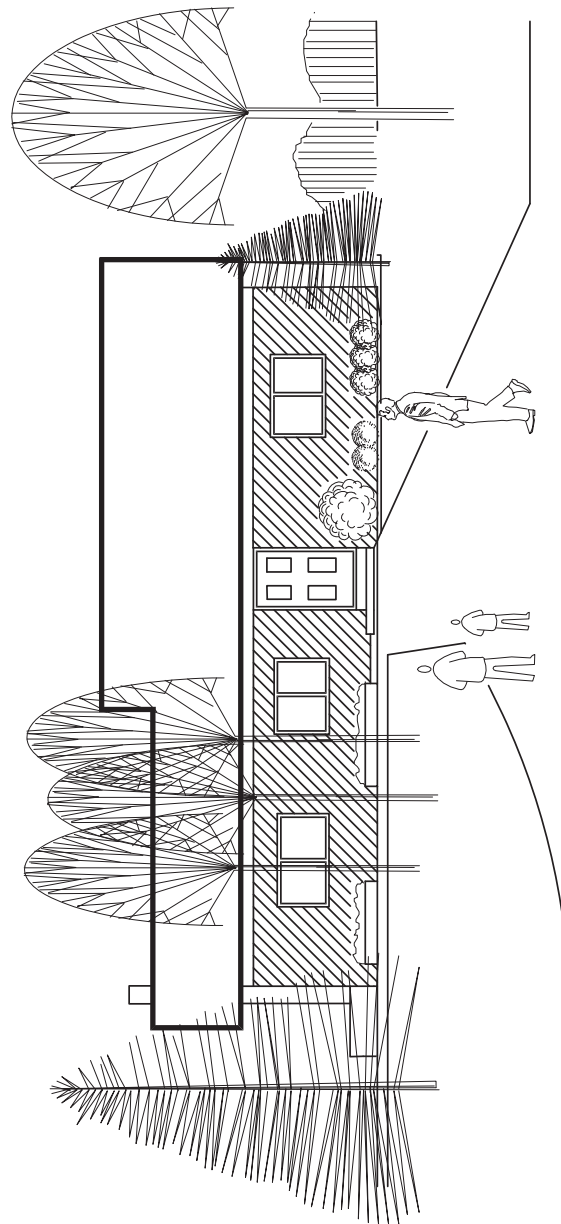
Courtesy of Mills Brothers Landscape and Nursery, Inc.

L3 PROBLEM 84

Create a solid model of the bracket. Estimate all dimensions. Display the model in the **Conceptual** visual style.
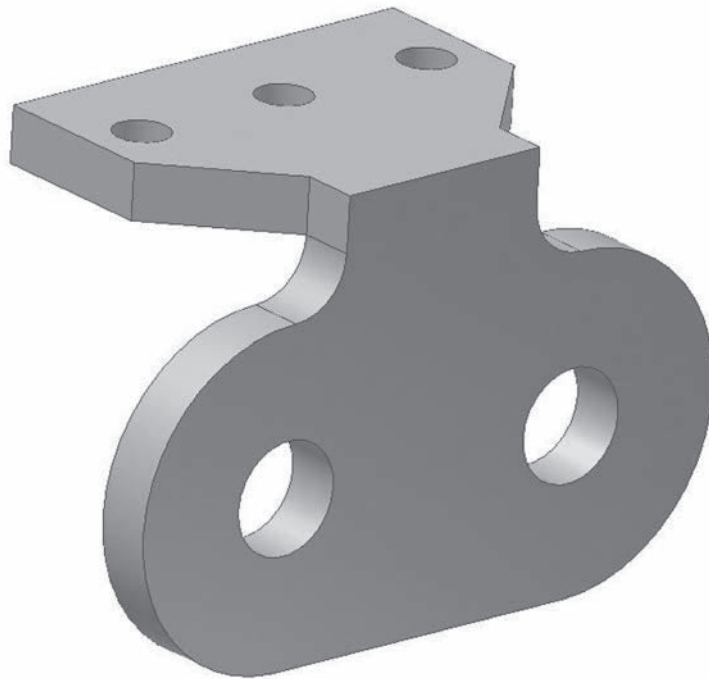
Additional Problems

L3 PROBLEM 85

Create a solid model of the 90° link. Display the model in the **Conceptual** visual style.
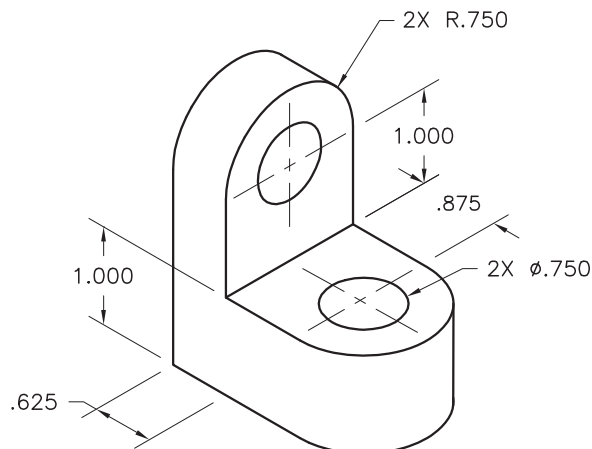
L3 PROBLEM 86

Draw the front and right-side views of the gear. Use the detail drawing to create the teeth accurately. Hatch as indicated using the ANSI31 pattern.
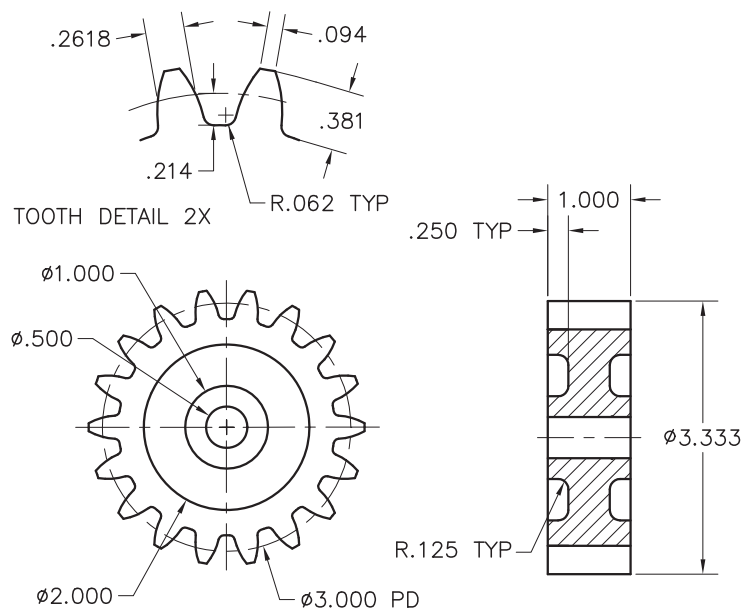
Problem 86 courtesy of Steve Huycke, Lake Michigan College

L3 PROBLEM 87

Create and fully dimension the isometric view of the strip block. Use a drawing area of 17×11.

Additional Problems

L3 PROBLEM 88

Create and dimension the drawing of a wrench. The units are millimeters. Determine the drawing area based on the dimensions of the wrench.

Adapted from the textbook Drafting Fundamentals *by Scott, Foy, and Schwendau*

L3 PROBLEM 89

Create the front and right-side views of the fitting. Then construct the auxiliary view. Dimension the front and right-side views.
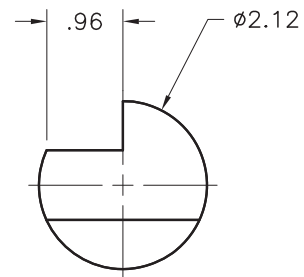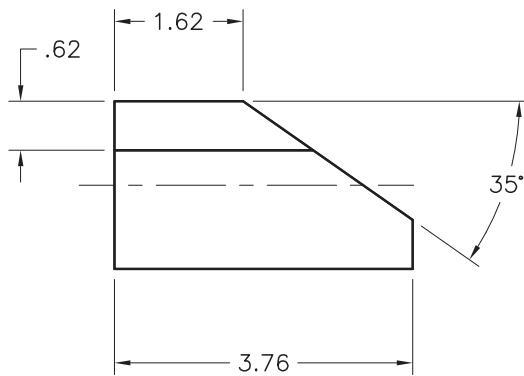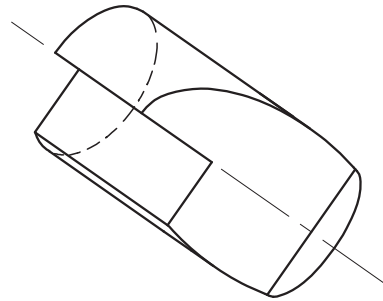
Additional Problems

L3 PROBLEM 90

Create the front and right-side views of the block. Then construct the auxiliary view. Dimension the front and right-side views.

2X 109°

2.00

35°

3.36

1.47

1.89
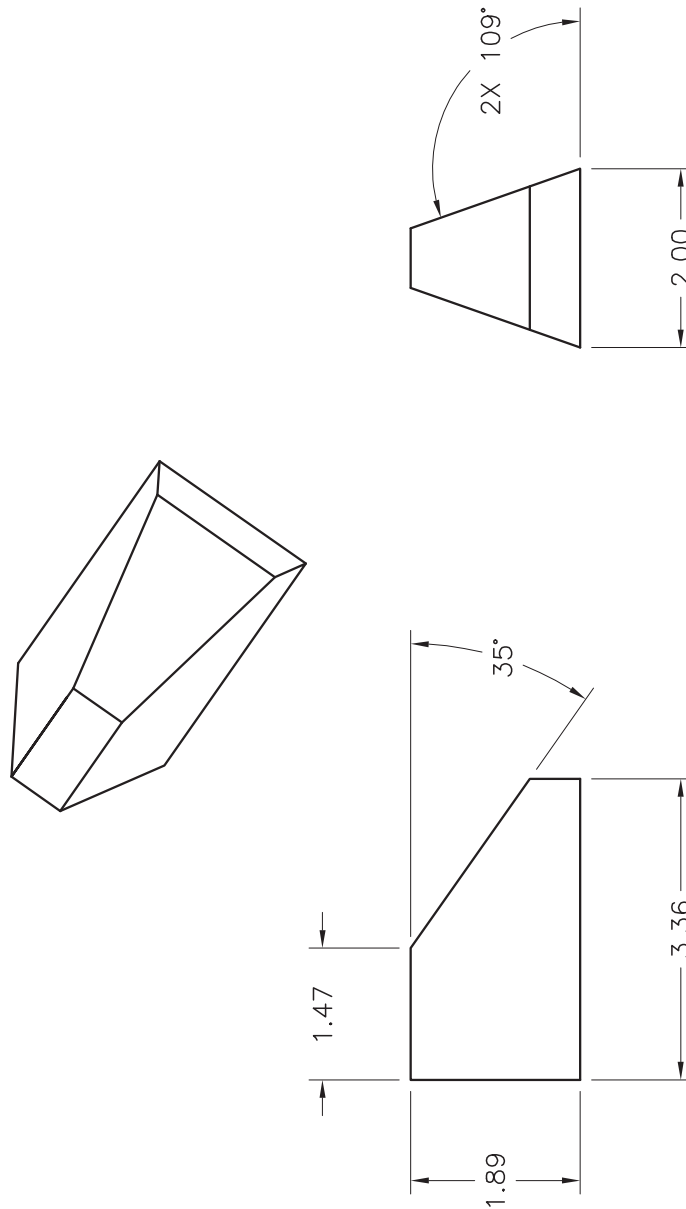
Courtesy of John F. Kirk, Kirk & Associates

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719

L3 PROBLEM 91

Create the front and right-side views of the casting. The right-side view should include a half section. Fully dimension both views.

ALL FILLETS R.10
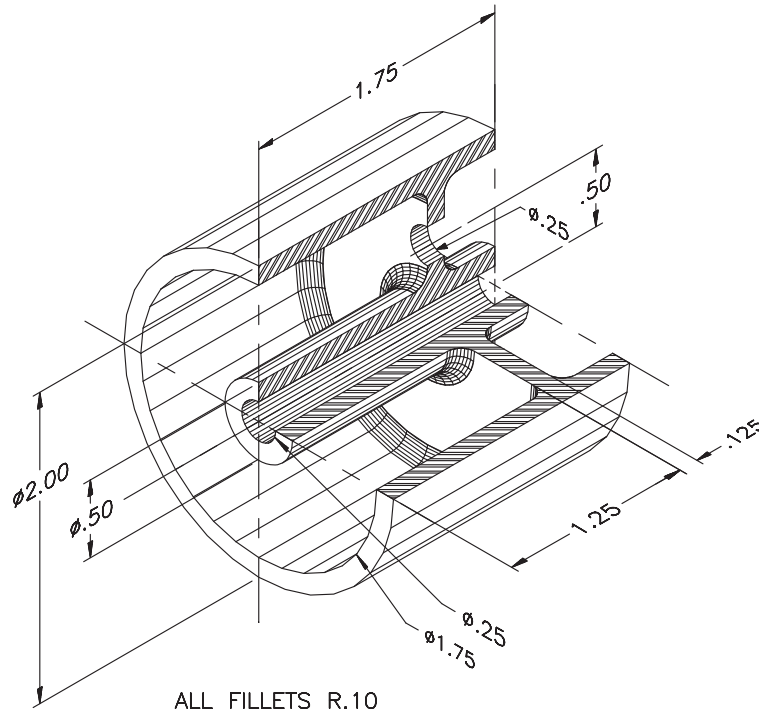
Courtesy of Gary J. Hordemann, Gonzaga University

Additional Problems

L3 PROBLEM 92

Create the top and front views of the alternator bracket. Use a drawing area of 17 × 11.
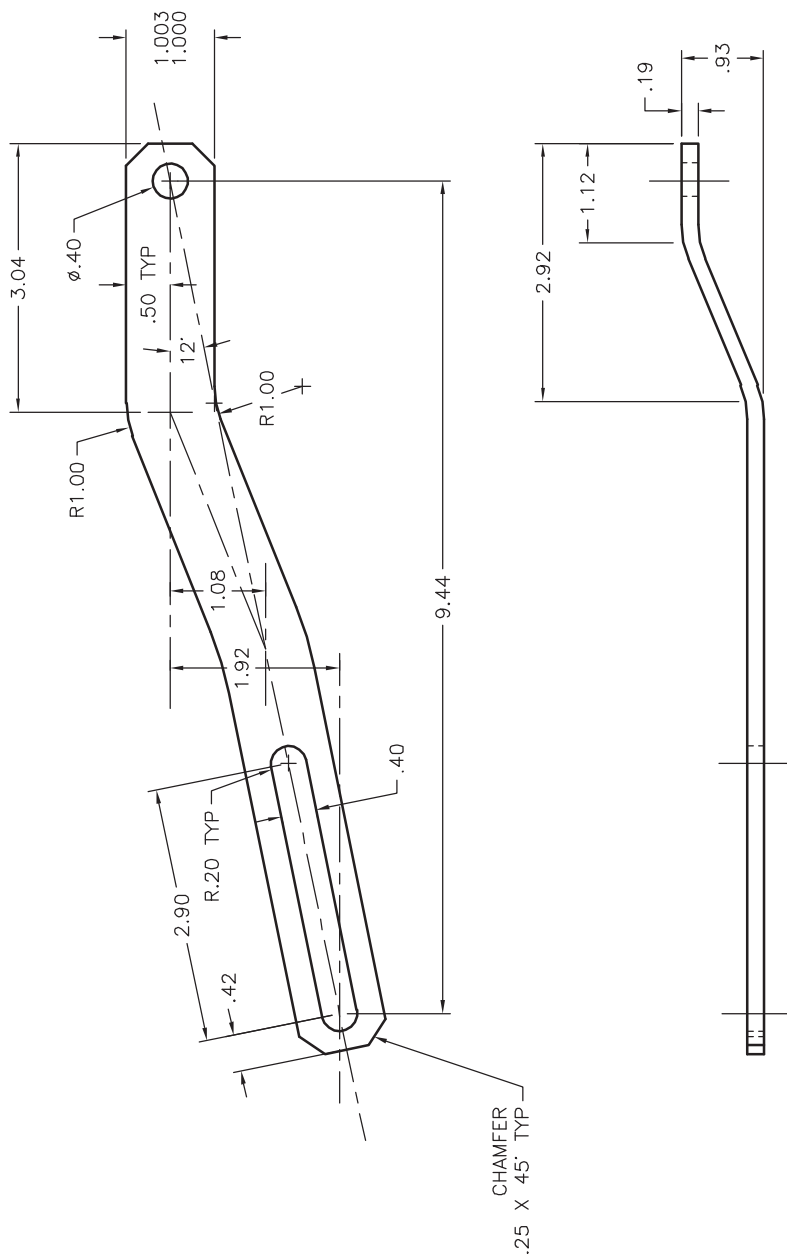
Adapted from the textbook Drafting Fundamentals *by Scott, Foy, and Schwendau*

L3 PROBLEM 93

Create the necessary orthographic views of the angle block and fully dimension the views. Add an isometric view in the upper right area of the drawing.
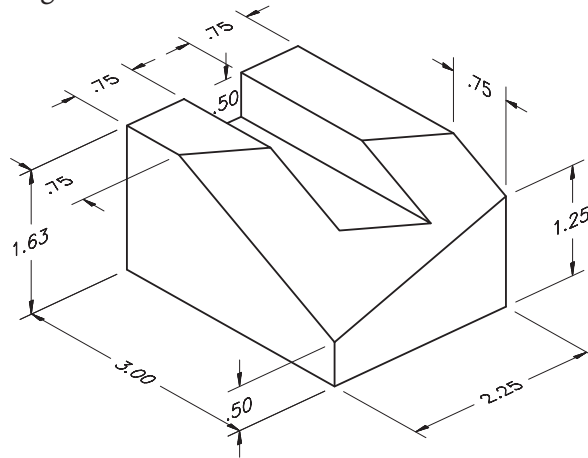
Adapted from the textbook Drafting Fundamentals *by Scott, Foy, and Schwendau*

L3 PROBLEM 94

Create a solid model of the swivel stop. First extrude the basic profile. Then use **PRESSPULL** to create the holes. Display the model in the **Conceptual** visual style.
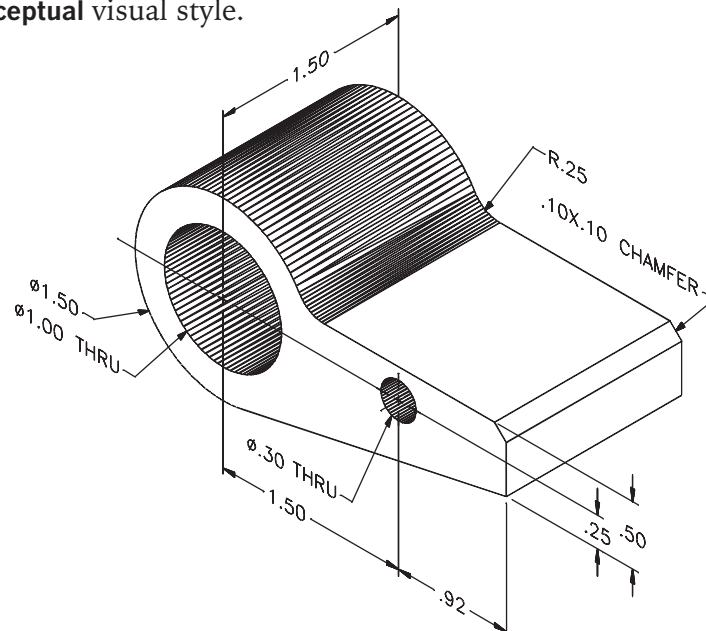
Courtesy of Gary J. Hordemann, Gonzaga University

Additional Problems

L3 PROBLEM 95

Create a solid model of the impeller. Display the model in the **Conceptual** visual style.

Courtesy of Gary J. Hordemann, Gonzaga University

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723
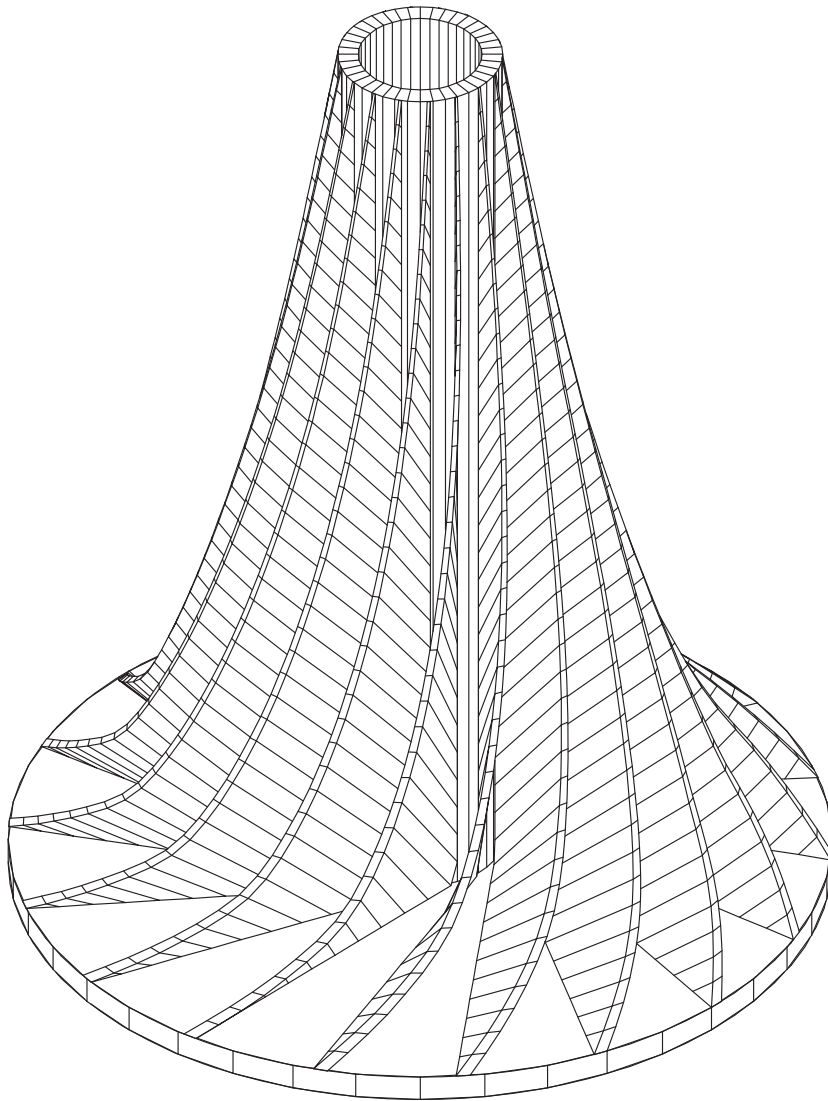
L3 PROBLEM 96

Create a solid model of the block.
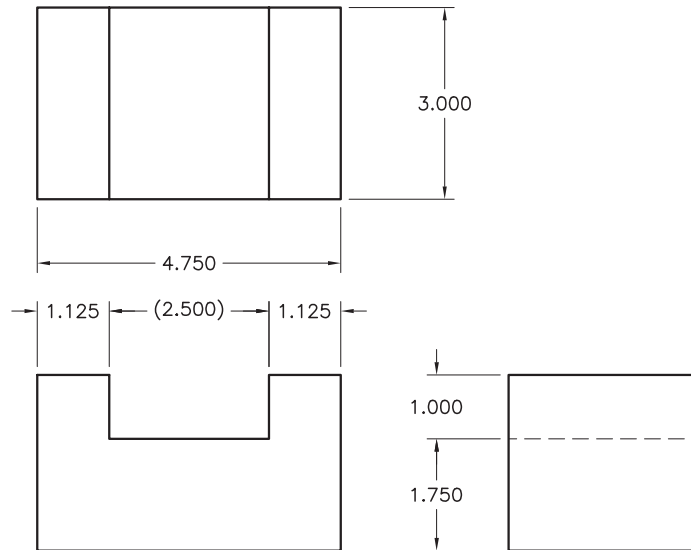
L3 PROBLEM 97

Create a solid model of the block. Then produce orthographic views
from the solid model.
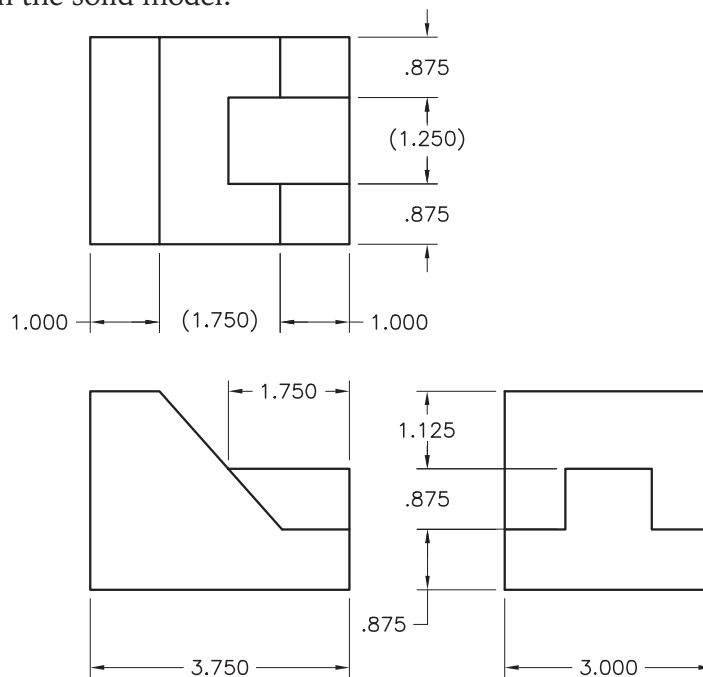
Additional Problems

L3 PROBLEM 98

Create the views of the 45° elbow as shown and fully dimension them. Determine the drawing area based on space requirements.

4X R.56   .38   ø1.62 CORE   4X ø.38

4X .38 SLOTS
EQUALLY SPACED
ON A ø3.38 BC

ø3.88   .19   1.315   2.63 TYP

□ 3.75

NOTE:
ALL FILLETS AND ROUNDS
TO BE R.12

R4.88   45°   .50 TYP
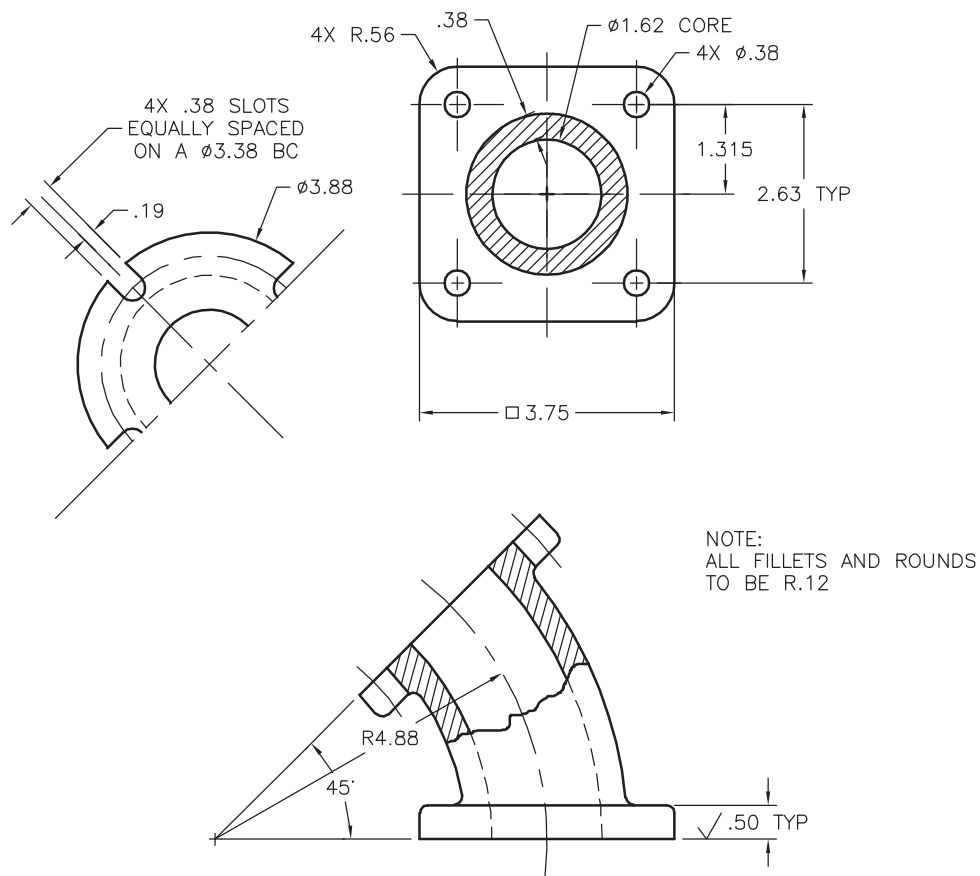
Courtesy of Steve Huycke, Lake Michigan College

L3 PROBLEM 99

Create the front and full section views of the sprocket and fully dimension them. Units are in millimeters. Determine the drawing area based on space requirements.
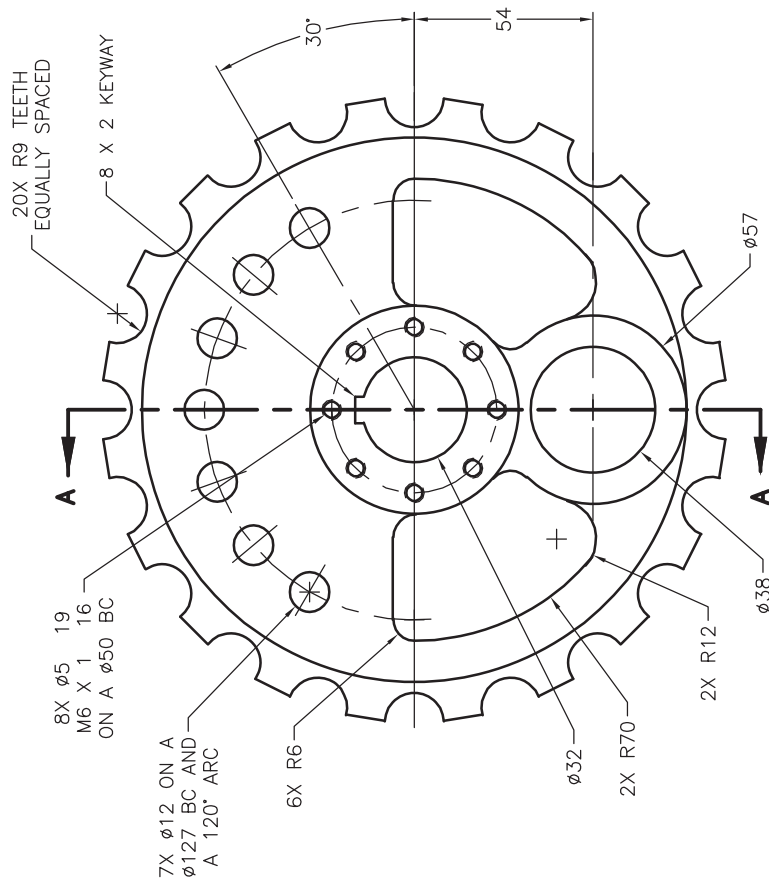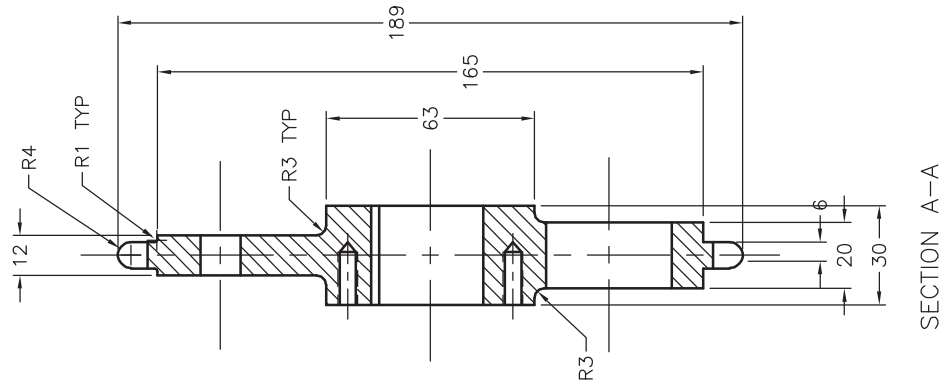
Courtesy of Alan Fitzell, Central Peel Secondary School

Additional Problems

L3 PROBLEM 100

Create the front and full section views of the tool and dimension them. Also include the detail as shown. Determine the drawing area based on space requirements. Units are in millimeters.
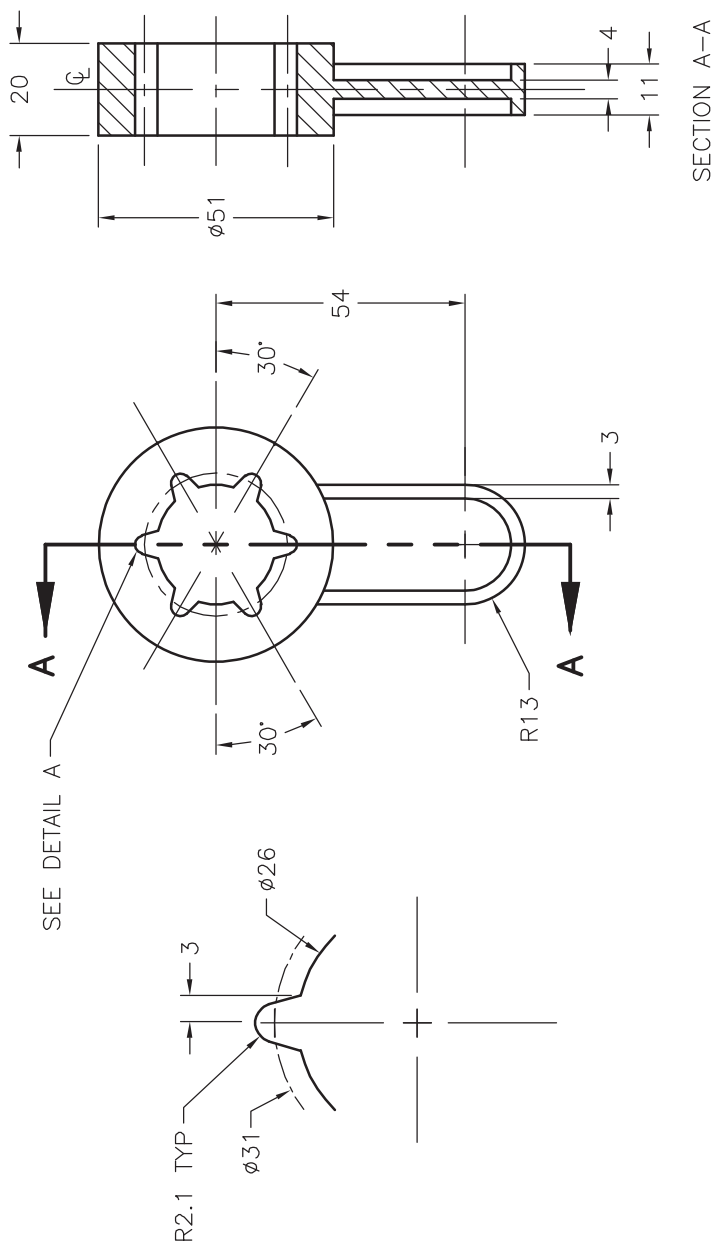
Courtesy of Julie H. Wickert, Austin Community College

L3 PROBLEM 101

Create the entire set of drawings of the control bracket. Determine the drawing area based on space requirements.

NOTE:
ALL FILLETS AND ROUNDS R.12
UNLESS OTHERWISE SPECIFIED

MATL: CAST IRON

SECTION A—A

SECTION B—B
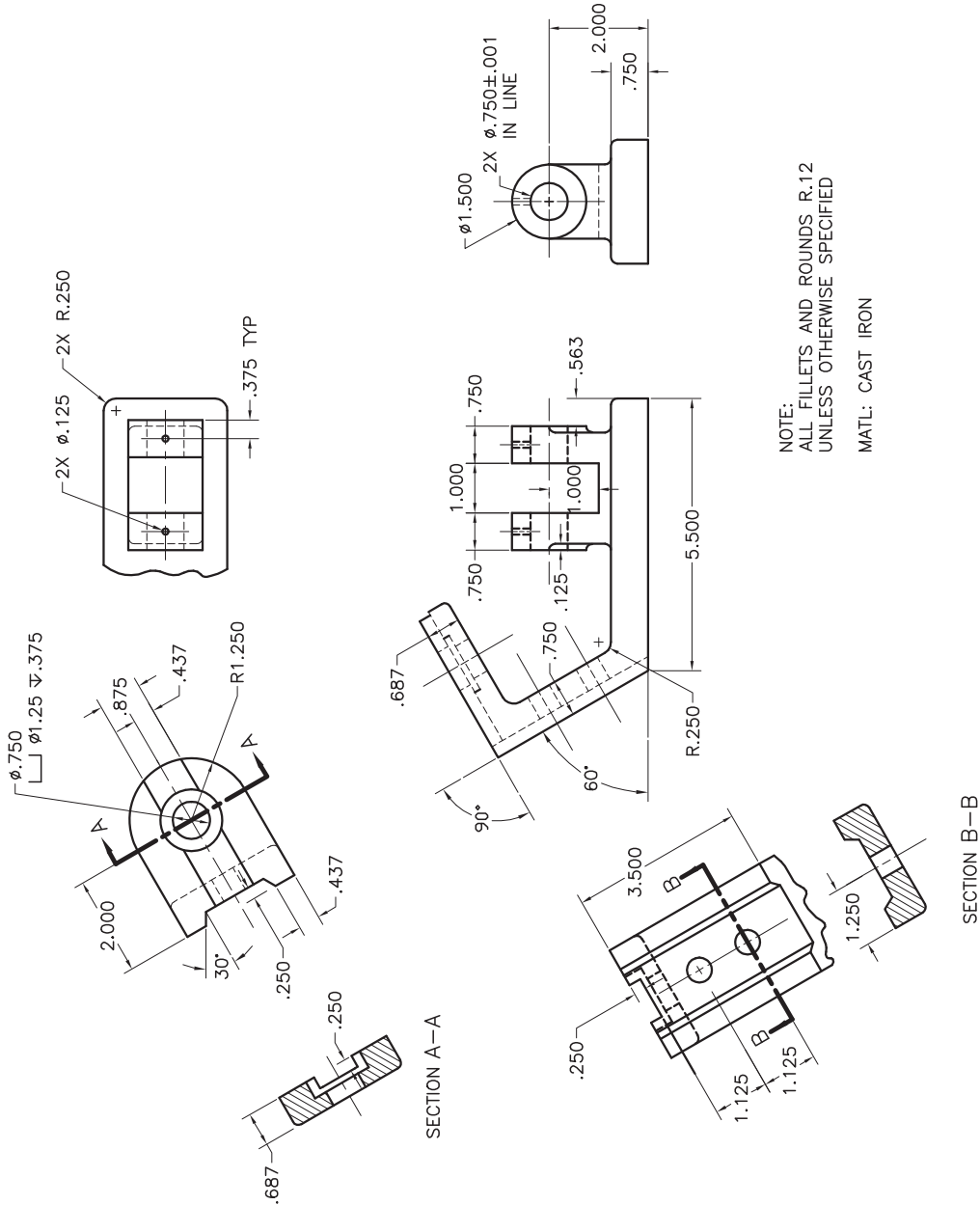
Courtesy of Steve Huycke, Lake Michigan College

Additional Problems

L3 PROBLEM 102

Create the drawing of the fork lift as well as the bill of materials. Estimate all dimensions.

3pt. HITCH
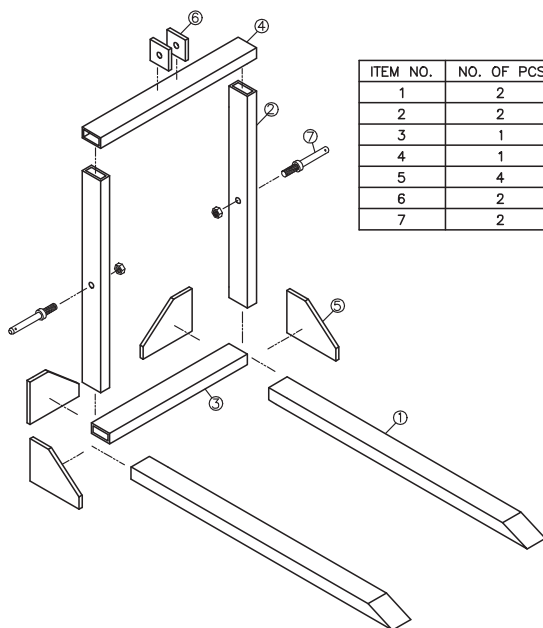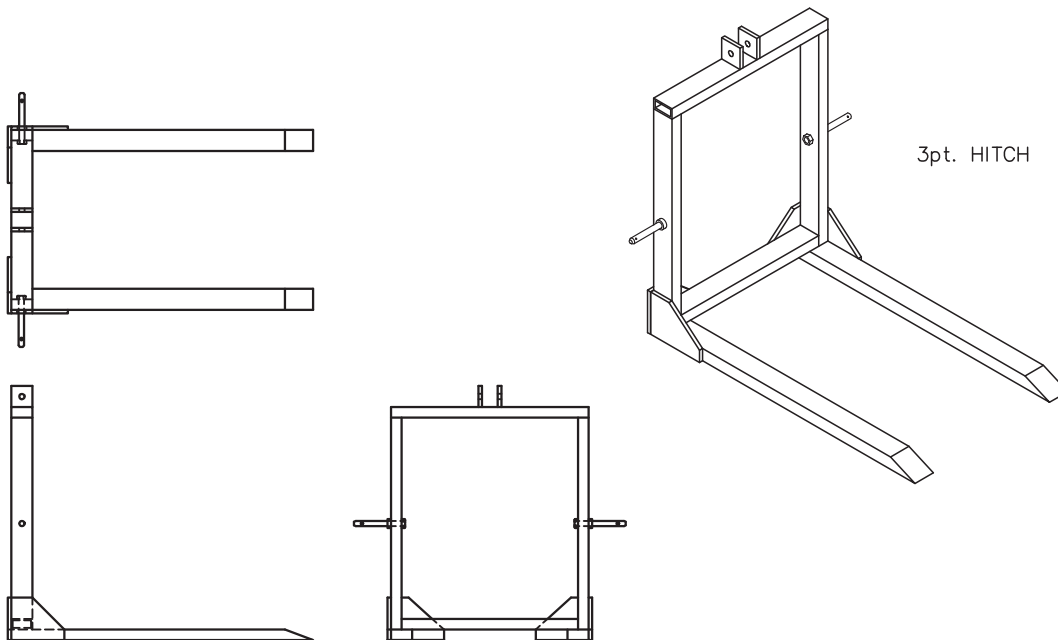
ITEM NO.	NO. OF PCS	SIZE	DESCRIPTION
1	2	42.75	1/8–1.5–3 TUBING
2	2	30	1/8–1.5–3 TUBING
3	1	22.5	1/8–1.5–3 TUBING
4	1	25.5	1/8–1.5–3 TUBING
5	4	3/8–6–8	ANGLE BRACES
6	2	1/2–3–3	UPPER ARM SUPPORTS
7	2	7.25 – 3/4 HEX. NUT	PIN HITCH–STANDARD PART

Courtesy of Craig Pelate and Ron Weseloh, Red Bud High School

L3 **PROBLEM 103**

Create the top, front, right-side, and isometric views of the picnic table. Do research to find appropriate sizes and dimensions.
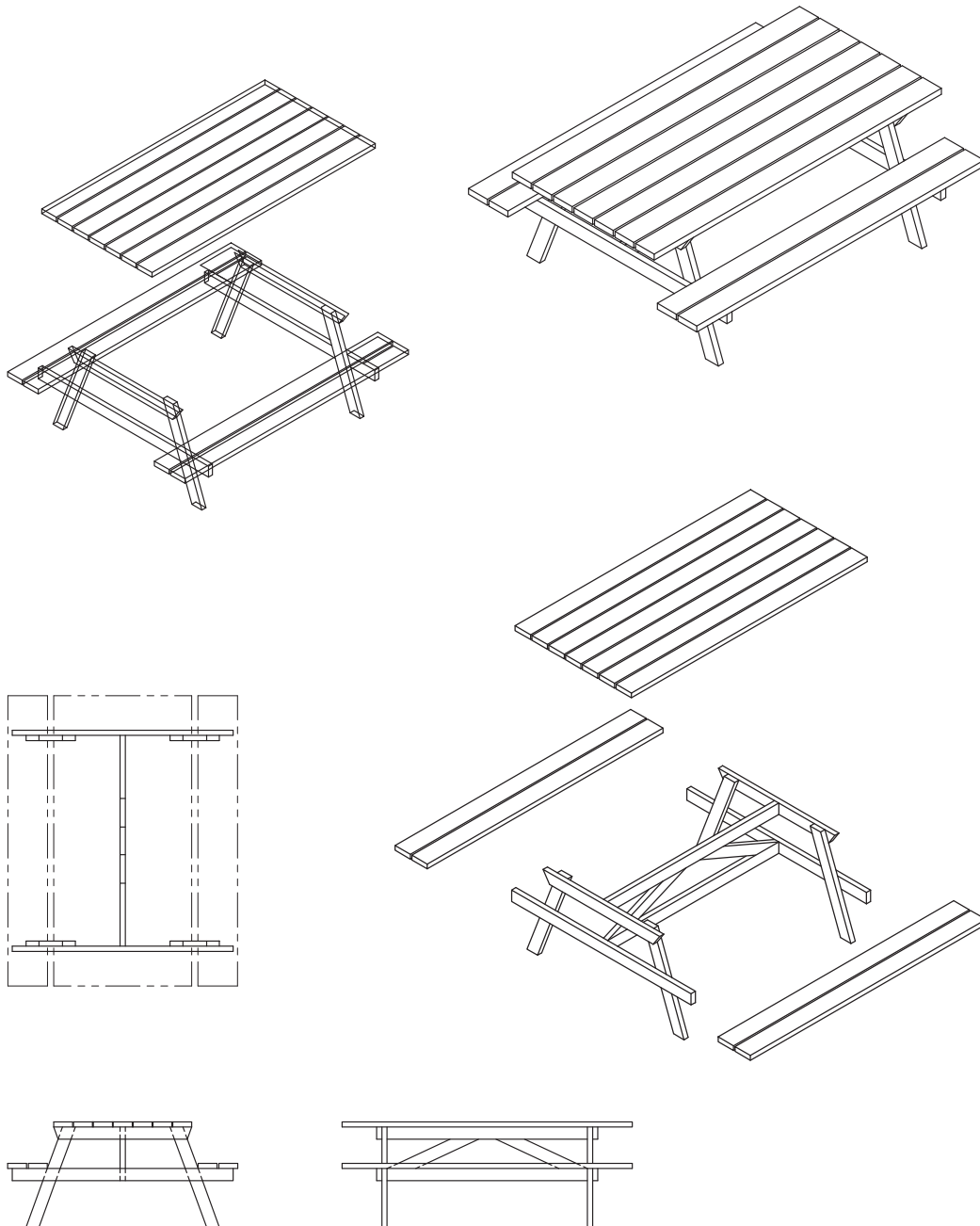
Courtesy of Dan Cowell and Ron Weseloh, Red Bud High School

Additional Problems

L3 PROBLEM 104

Create the front and side views of the structural bracket and dimension them as shown. Also include section and isometric views. Determine the drawing area based on space requirements.

W14 X 176

Pc WT18 X 85

8

5

1 1/2

14X ⌀15/16
@3 = 1'-6

1 1/2

4
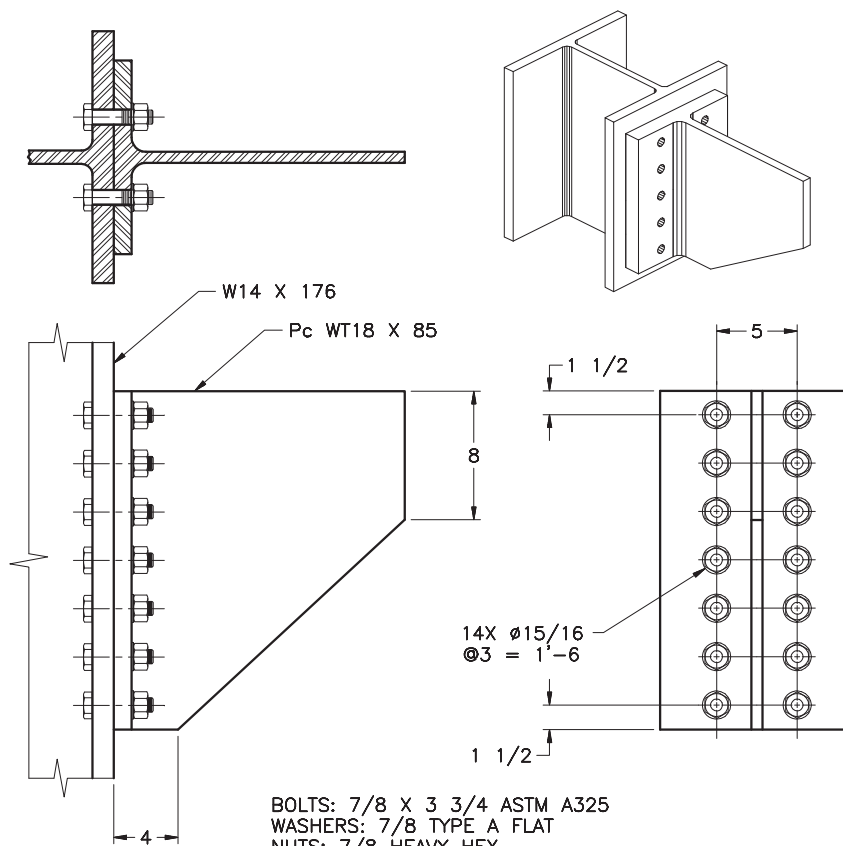
BOLTS: 7/8 X 3 3/4 ASTM A325
WASHERS: 7/8 TYPE A FLAT
NUTS: 7/8 HEAVY HEX

FOR DIMENSIONS, SEE: MANUAL OF STEEL CONSTRUCTION,
AMERICAN INSTITUTE OF STEEL CONSTRUCTION

Courtesy of Gary J. Hordemann, Gonzaga University

L3 PROBLEM 105

Draw the top view of the spacer shown below. Assume that all of the ribs have a width of **.2** and are symmetrical about the center lines. Try this approach: Create the hexagon and offset it. Add one set of circles and array them. Draw lines joining the centers of the circles and polygons. Then offset the lines, fillet them, and array them. *Note:* The fillets are between lines and circles and between lines and the outer polygon, *not* between lines.

.20 TYP

Ø1.00 — 6 PLACES

.75

Ø.50 — 6 HOLES EQUALLY SPACED

Ø4.00

ALL FILLETS R.20
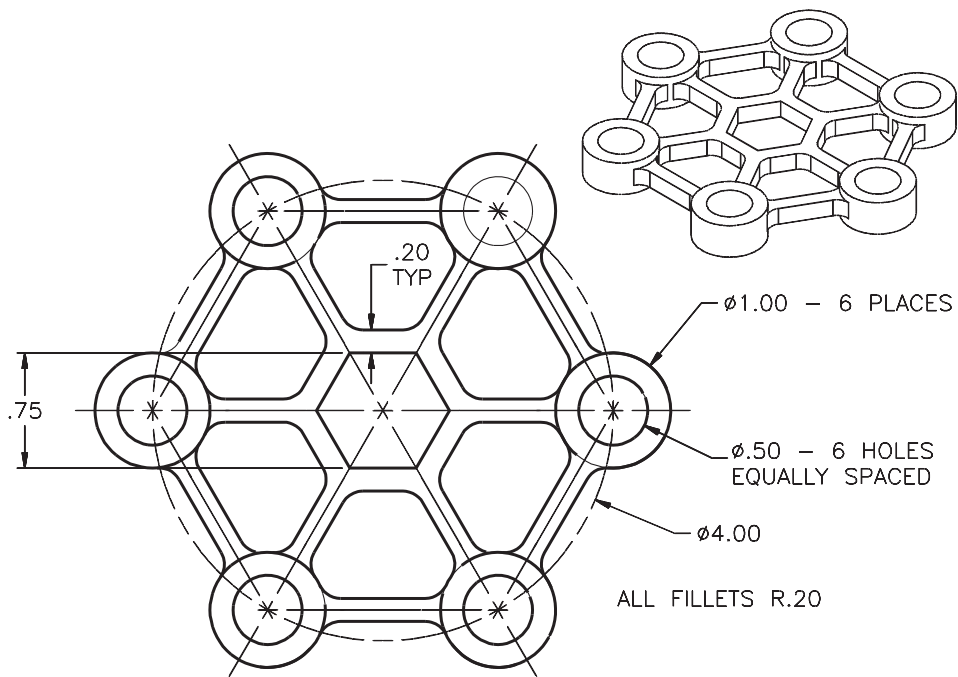
Courtesy of Gary J. Hordemann, Gonzaga University

Additional Problems

L3 PROBLEM 106

Create a solid model of the bracket using the dimensions shown in the orthographic views below. View the model in the **3D Wireframe** visual style. Create orthographic views from the model.

Courtesy of Mark R. Stevenson

L3 PROBLEM 107

Create and dimension a solid model of the locking sleeve according to the dimensions shown below. Display the model in the **Conceptual** visual style.
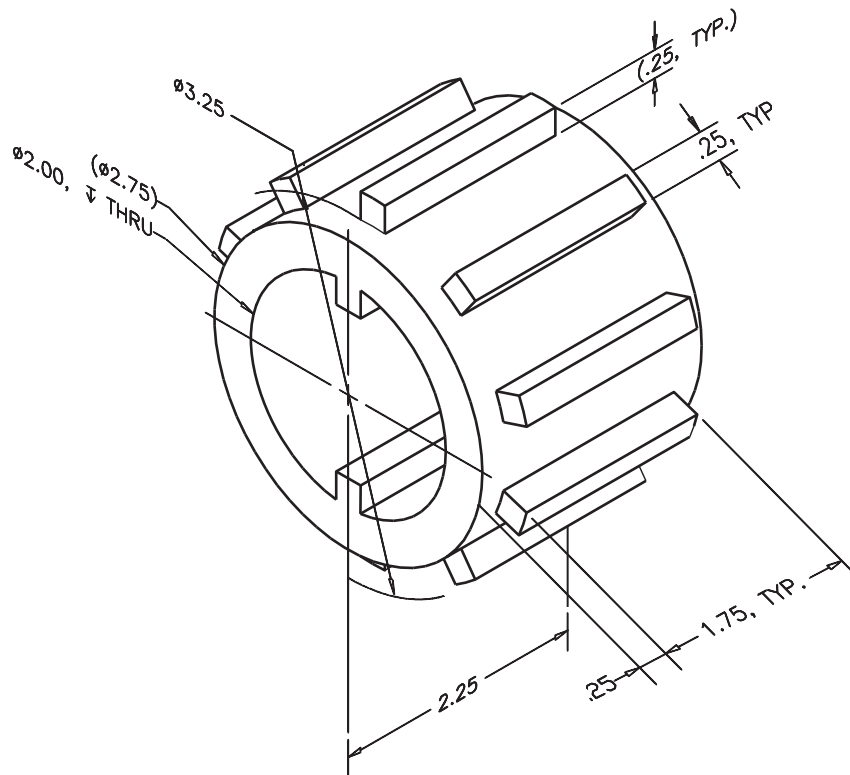
NOTES:

1.) 10 EVENLY SPACED OUTSIDE TEETH.
2.) 2 EVENLY SPACED INSIDE TEETH (.25X.25X2.25)

Courtesy of Mark R. Stevenson

Additional Problems

PROBLEM 108

Create a solid model of the roller tube block. The overall dimensions of the block are **2 × 2 × 3**. The hole on the top surface has a depth of **1**, and the holes on the sides are thru holes. All of the holes have a diameter of **1**. Display the finished model in the **Conceptual** visual style. Rotate the model to view it from several viewpoints.
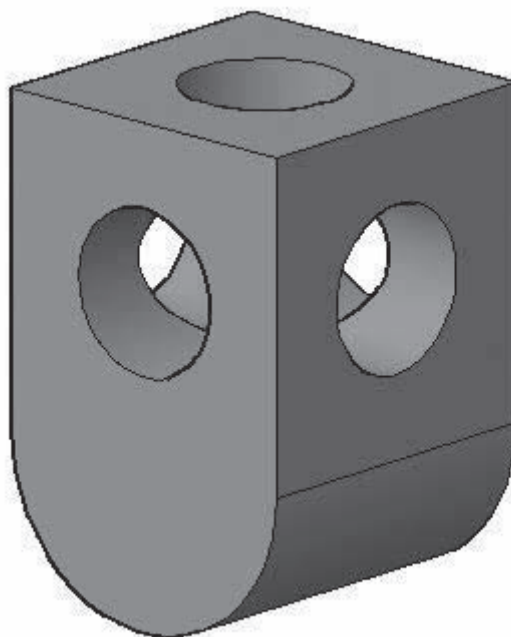
Model courtesy of Gary J. Hordemann, Gonzaga University

L3 PROBLEM 109

Using the **REVOLVE** command, create a solid model of the V-belt pulley shown below. Note that by revolving around the axis, you can create a hole without using the **SUBTRACT** or **PRESSPULL** command.
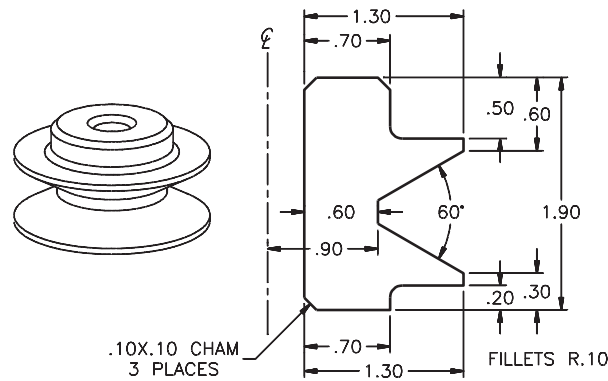
Courtesy of Gary J. Hordemann, Gonzaga University

L3 PROBLEM 110

Create top, front, and isometric views of the fence. Determine the drawing area based on space requirements. Dimension the top and front views.
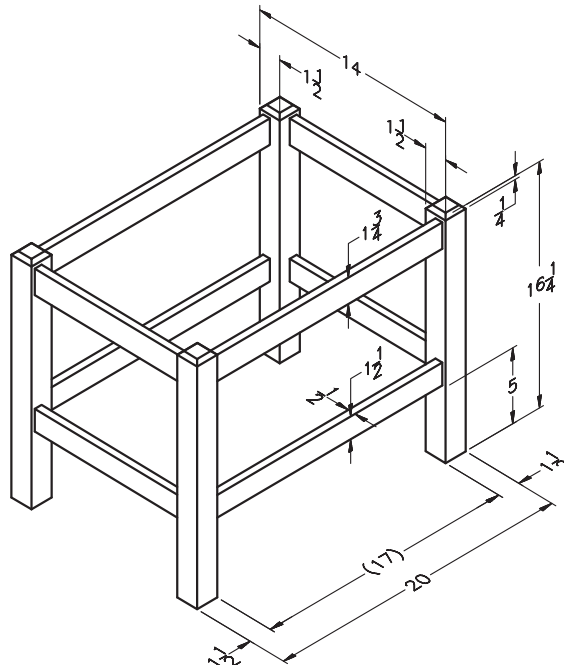
Adapted from the textbook Drafting Fundamentals *by Scott, Foy, and Schwendau*

Additional Problems

L3 PROBLEM 111

Create a solid model of the surveyor's transit. Estimate all dimensions. Display the model in the **Conceptual** visual style.
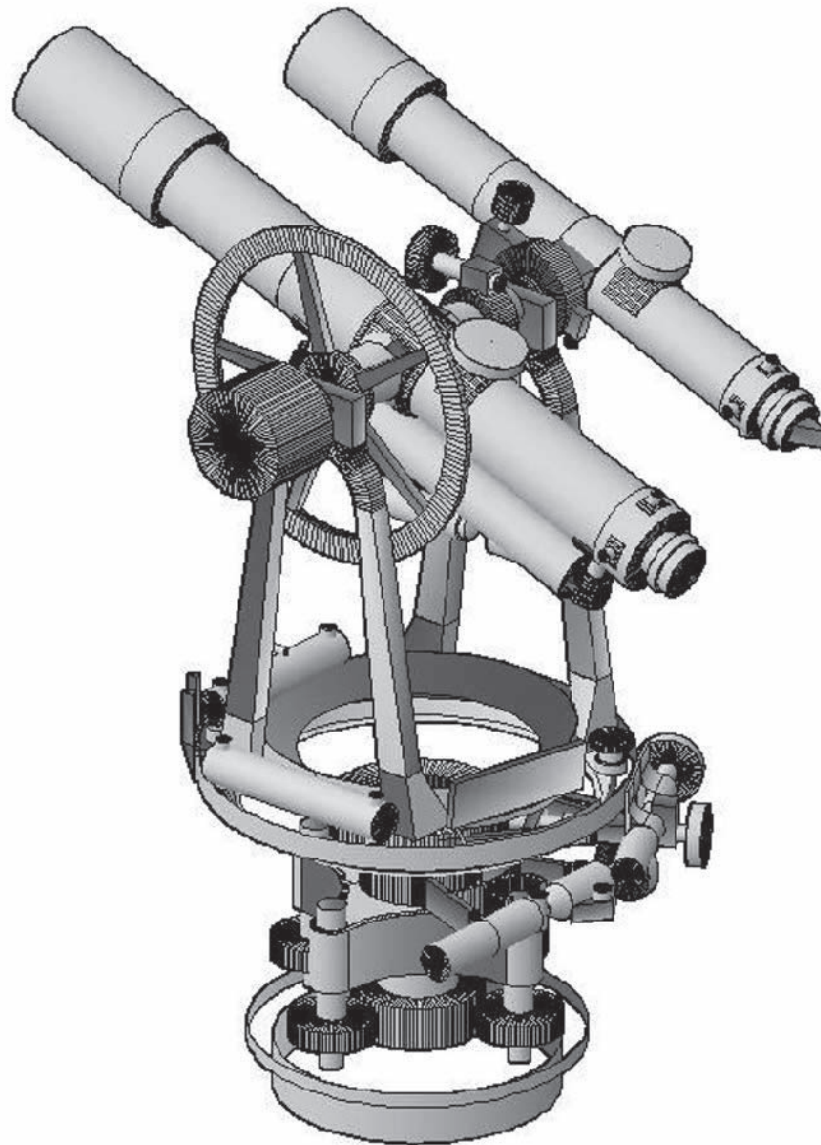
Model courtesy of Riley Clark

L3 PROBLEM 112

Create the architectural elevation of the cathedral. Estimate all dimensions.

Courtesy of David Sala, Forsgren Associates, from World Atlas of Architecture *(G.K. Hall and Company)*

Additional Problems

L3 PROBLEM 113

Create the wall section.

BUILT–UP GR. ROOF
3/4" RIGID INSUL.
1 1/2" 24 GA. METAL DECK
G.I. GUTTER, GRAVEL GD., & 2"X8" FACIA
8"
2 1/2"
WD. PL AS RQD.
$10\frac{1}{8}$"
4"X6"X3/8" ANGLE SEAT
10" Z PURLIN @ 3'–9" O.C.
10
WALL TIES @ 1'–4" O.C. VERT.
3 1/2"X3 1/2"X3/8" ANGLE
SUSP. 3/4" PLASTER CL'G ON 3/8" ROCKLATH
6" BAT INSULATION
10'–2 3/4"
3'–11 3/8"
6"X8" PRECAST LINTEL
SEE WINDOW DETAILS
7'–3"
8'–8 3/8"
5 7/8"
1" MARBLE SILL
5'8" PLASTER ON MASONRY
RUBBER COVE BASE
1'2" EXPANSION JT.
6X6 10/10 W.W.F.
STONE SILL
2'–10 3/4"
BRICK
4"
1 1/2" STYROFOAM
POLYETHYLENE FILM
4"–6" SAND & GRAVEL
2'–4"
10' CONC. BLK.
1 1/2" STYROFOAM
10"
1'–6"

Adapted from a drawing by Paul Driscoll

L3 PROBLEM 114

Create the front and left-side architectural elevation drawings. Estimate all dimensions.
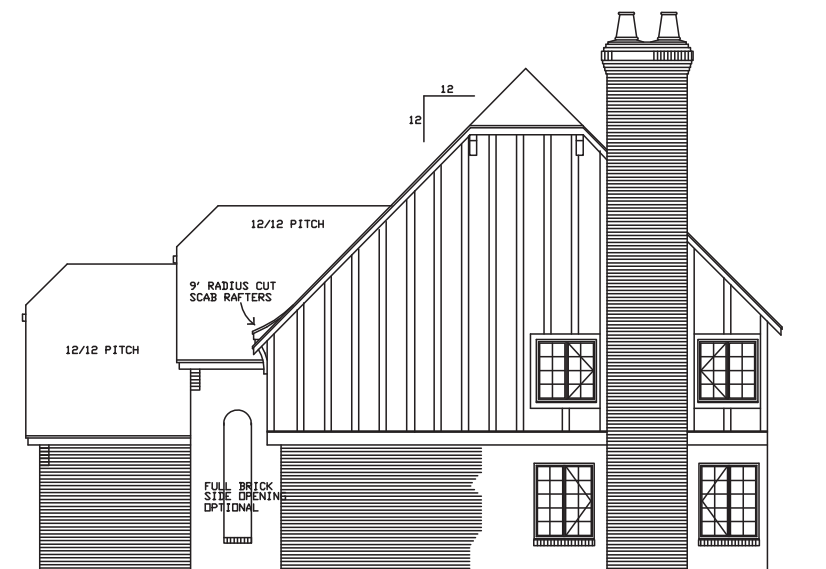
Courtesy of Rodger A. Brooks, Architect

Additional Problems

L3 PROBLEM 115

Create the back and right-side architectural elevation drawings. Estimate all dimensions.

ROOF VENTS

12/12 PITCH ROOF

REFER TO SPECIFICATIONS
FOR MATERIALS USED

SHINGLES

FULL BRICK
LOWER LEVEL

12
12

12/12 PITCH ROOF

12/12 PITCH ROOF

METAL PAN

FULL BRICK

Courtesy of Rodger A. Brooks, Architect

L3 PROBLEM 116

Create the architectural elevation drawing. Estimate all dimensions.

FRONT ELEVATION
1/8" =1' –0"
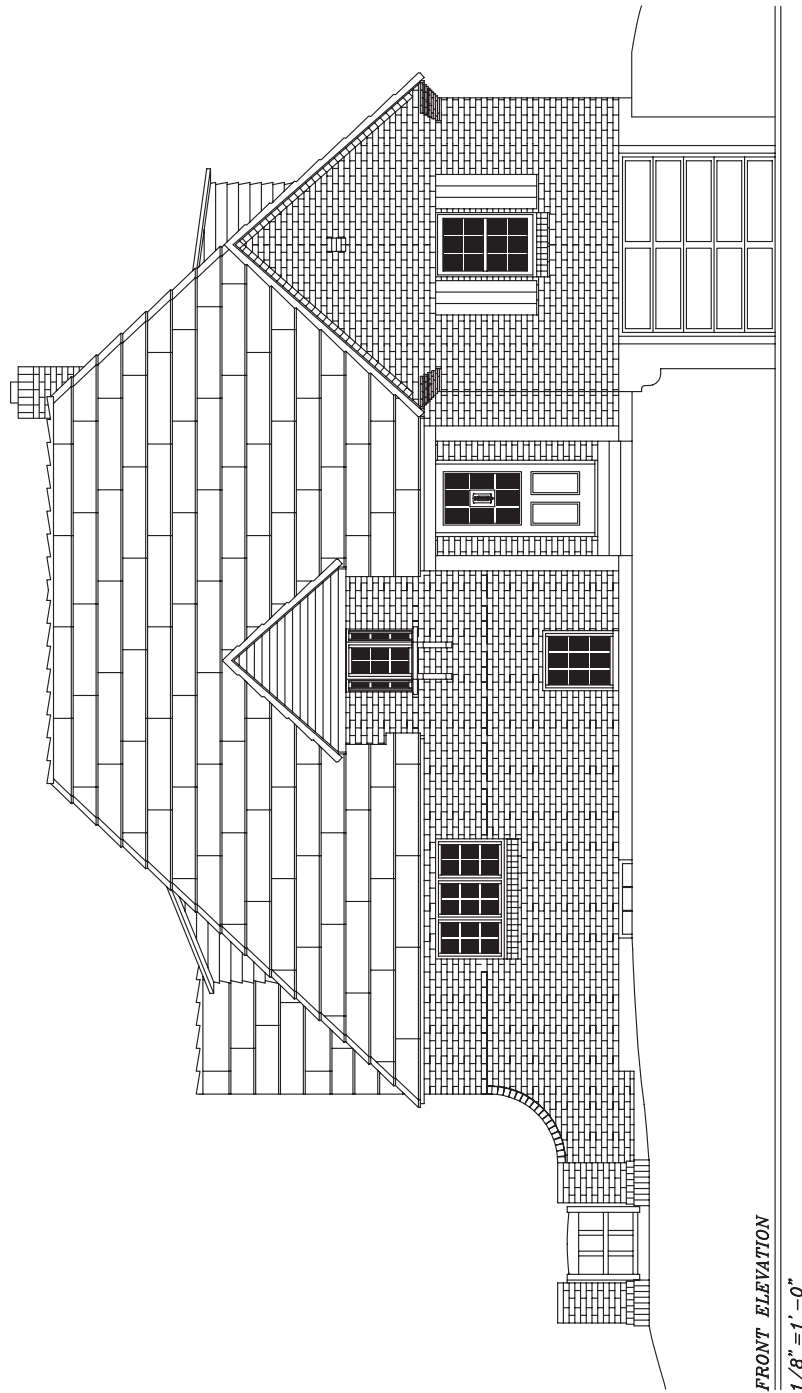
Courtesy of Gary J. Hordemann, Gonzaga University

Additional Problems

L3 PROBLEM 117

Create a solid model of the sprocket guard. Estimate all dimensions. Display the model in the **Conceptual** visual style.
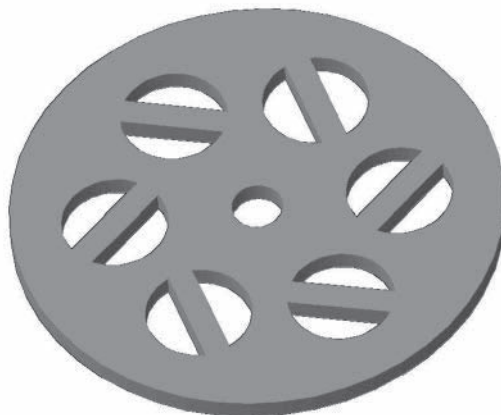
L3 PROBLEM 118

Create solid models of the parts and arrange as an exploded assembly. Estimate all dimensions. Display the assembly using the **Conceptual** visual style.
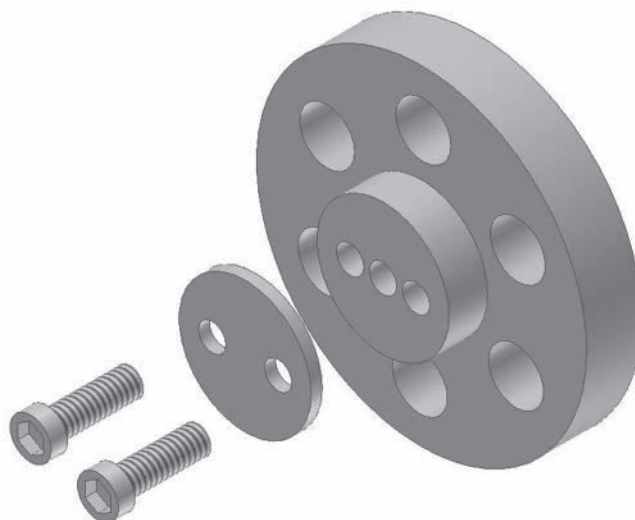
Appendix A
Options Dialog Box

AutoCAD's **Options** dialog box permits you to customize many of AutoCAD's settings. Also, the dialog box enables you to add pointing devices and printers to your AutoCAD system. Select the **Options...** button on the bottom of the **Menu Browser** to display the **Options** dialog box, as shown in Fig. A-1.
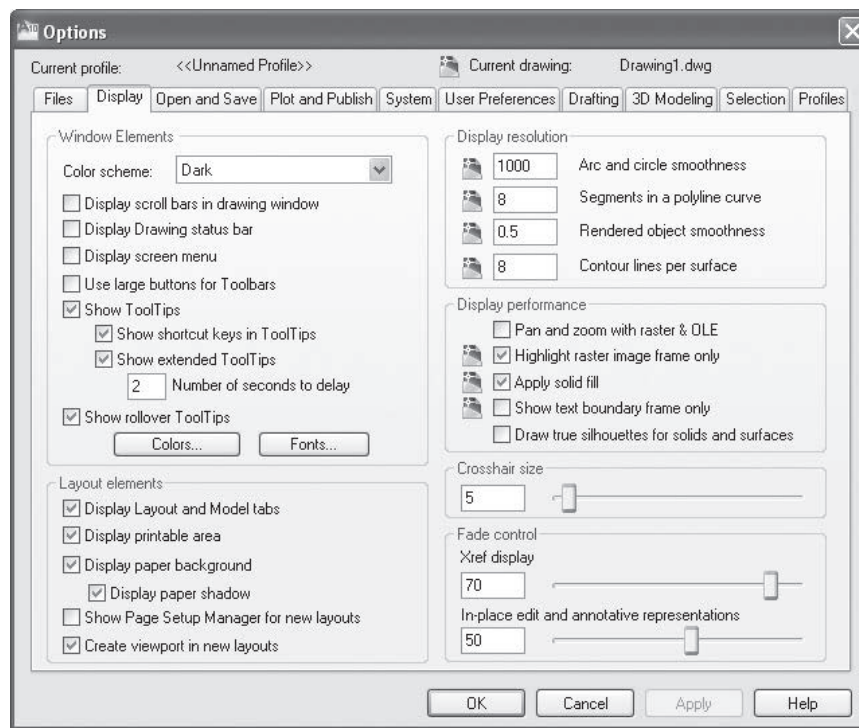
Fig. A-1

Files

The **Files** tab specifies the folders in which AutoCAD searches for text fonts, drawings to insert, linetypes, hatch patterns, menus, plug-ins, and other files that are not located in the current folder. It also specifies user-defined settings such as the custom dictionary to use for spell checking.

Display

This tab customizes the AutoCAD window. For instance, you can specify the display of scroll bars in the window. It allows you to set the colors for most items in the user interface, including the backgrounds for the **AutoCAD**

Continued

Classic and **3D Modeling** backgrounds, **Command** line background and text, and AutoTrack™ vector. You can also control the size of the crosshairs and the resolution of arcs and circles.

Open and Save

This tab controls the options related to opening and saving files. Examples include the default file type when saving an AutoCAD drawing and whether a thumbnail preview image appears in the **Select File** dialog box. The settings on this tab also allow you to set the number of minutes between automatic saves.

Plot and Publish

The **Plot and Publish** tab deals with settings and controls related to plotting and printing devices. You can set the default output device and add and configure a new device. You can also add or edit plot style tables.

System

Use this tab to control miscellaneous system settings, such as the display of 3D graphics. This is also where you change the current pointing device.

User Preferences

This tab helps you optimize the way you work in AutoCAD. For example, you can customize how the right-click features in AutoCAD behave. Also, you can indicate the source and target drawing units when dragging and dropping drawing content. You can also set lineweights and edit the scale list from this tab.

Drafting

The **Drafting** tab controls many editing options. For instance, you can make adjustments to the AutoSnap features, such as changing the color and size of AutoSnap markers. Here, you can also change AutoTrack settings, including whether to display polar tracking vectors. You can also customize settings for light and camera glyphs.

Wohlers: Applying
AutoCAD® 2010

Back Matter

Appendix A: Options
Dialog Box

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745

3D Modeling

This tab allows you to control the appearance of the 3D crosshairs and the 2D and 3D UCS icons. You can also control the number of isolines used in models, customize walk, fly, and animation settings, and control the appearance of the **ViewCube** and **SteeringWheels**.

Selection

The **Selection** tab controls settings related to methods of object selection. Examples include turning noun/verb selection on or off and adjusting the size of the pickbox and grip boxes. You can also set the color of selected and unselected grip boxes.

Profiles

This tab controls the use of profiles. In this context, a profile is a user-defined configuration that allows each user to set up and save various personal preferences, such as which toolbars are present and where they appear on the screen. Profiles allow several people to share a single computer without having to change the settings individually each time they use it.

If no profiles have been defined for a computer, you can define one by picking the **Add to List...** button. A dialog box appears that allows you to name the profile and give it a description. Note that creating a new profile does not automatically select it as the current profile. You must pick the **Set Current** button to set the active profile.

When more than one profile exists, any changes you make to AutoCAD's settings are automatically saved to the current profile. To change the current profile, display the **Options** dialog box, pick the **Profiles** tab, select the profile you want to make current, and pick the **Set Current** button.

Appendix B
Managing AutoCAD

Organizing an AutoCAD installation requires several considerations. For example, you should store files in specific folders. Name the files and folders using standard naming conventions. If you don't, you may not be able to find the files later. It is also very important that you back up regularly. If you overlook these and other system management tasks, your productivity may decline. Also, you may lose work, causing problems that you could otherwise avoid.

AutoCAD users and managers alike will find the following information helpful. If you manage an AutoCAD system with care, you should never have to create the same object twice. This comes only with careful planning and cooperation among those who will be using the system.

Bear in mind that the process of implementing, managing, and expanding an AutoCAD system evolves over time. As you and others become more familiar with AutoCAD and the importance of managing files and folders, refer back to this appendix.

System Manager

One person (or department) within the organization should have the responsibility for managing the AutoCAD systems and overseeing their use. This person should be the resident CAD champion and should answer questions and provide directions to other users of the systems. The manager should oversee the software, documentation, and hardware and should work with the users to establish procedural standards for use with the systems.

Management Considerations

The system manager should consider questions like those listed below when installing and organizing AutoCAD. The questions are intended to help guide your thinking, from a management perspective, as you become familiar with the various parts of AutoCAD.

- How can I best categorize the files so that each folder does not grow to more than 200 files total? Folders with a larger number of files become difficult to navigate.
- If I plan to install three or more AutoCAD stations, should I centralize the storage of user-created files and plotting by using a network and file server? A network and a centralized file system are recommended.

After AutoCAD is in place and you are familiar with it, you will create many new files. The following questions address the efficiency with which you create and store these files.

- Are there template files (or existing drawings) on file that may serve as a starting point for new drawings? It is a good practice to benefit from past work.

- Where should new drawing files be stored, what should they be named, and how can users locate them easily? Develop organizational standards that are clear and easy to follow.

- Are there predefined libraries of symbols and details that I can use while I develop a drawing? Do any of the symbol libraries that come with AutoCAD meet my needs?

- Is a custom pull-down menu or toolbar available that lends itself to my drawing application?

- Are AutoLISP routines or other programs available that would help me perform certain drafting operations more easily?

If you feel uncomfortable about your answers to these questions, there is probably room for improvement. The following discussion is provided to help you organize and manage your CAD systems more effectively.

> **NOTE** Generally, the following discussion applies to almost all AutoCAD users and files. However, there are inevitable differences among users (backgrounds and interests, for example), drawing applications, and the specific hardware and software. Take these differences into consideration.

Software/Documentation

The AutoCAD system manager spends considerable time organizing and documenting files, establishing rules and guidelines, and tracking new software and hardware developments.

File Management—Know where files are located and the purpose of each. Understand which ones are AutoCAD system files and which are not. Create a system for backing up files and folders, and back up regularly. Emphasize this to all users. Delete "junk" files.

Continued

Template Files—Create an easy-to-use system for the development, storage, and retrieval of AutoCAD template files. Allow for ongoing development and improvement of each template. Store the template files in a folder dedicated to templates so that they are accessible by other users.

Document the contents of each template file by printing the drawing status information, layers, text styles, linetype scale, and other relevant information. On the first page of this information, write the name of the template file, its location, sheet size, and plot scale. Keep this information in a three-ring binder for future reference to other users.

User Drawing Files—Store these in separate folders. Place drawing objects on the proper layers. Assign standard colors, linetypes, and lineweights to the standard layers. Make a backup copy of each drawing and store it on a separate disk or backup system. Plot the drawings most likely to be used by others and store them in a three-ring binder for future reference.

Symbol Libraries—Develop a system for ongoing library development. (See Chapter 35 for details on creating symbol libraries.) Plot each symbol library drawing file, and place the library drawings on the wall near the system or in a binder. Encourage all users to contribute to the libraries.

Menu Files—Develop, set up, and make available custom toolbars and pull-down menus. Store the menus in a folder dedicated to menus so that they will be accessible to others.

AutoCAD Upgrades—Handle the acquisition and installation of AutoCAD software upgrades. Inform users of the new features and changes contained in the new software. Coordinate training.

AutoCAD Third-Party Software—Handle the acquisition and installation of third-party software developed for specific applications and utility purposes. Inform users of its availability and use.

Hardware

Oversee the use and maintenance of the hardware components that make up the system. Consider hardware upgrades as user and software requirements change.

Procedural Standards

Develop clear, practical standards in the organization to minimize inconsistency and confusion. Each template file should have a standard set of drawing layers, with a specific color and linetype dedicated to each layer. For example, you may reserve a layer called **Dimension**, with color green, a continuous linetype, and a lineweight of **.3 mm**, for all dimensions. This will avoid confusion and improve consistency within your organization. Develop similar standards for other AutoCAD-related practices.

AutoCAD Standards Checker

AutoCAD offers a method of checking arrowhead size, center marks, zero suppression, text height, and other items for organizational standards. The following steps outline this method.

1. Open a drawing file that you would consider a model example of how drawings should appear at your organization.
2. Using **Save As…**, save the drawing in **AutoCAD Drawing Standard (DWS)** format.
3. Open or create a new drawing and then open the **CAD Standards** toolbar.
4. Pick the **Configure…** button.
5. In the **Configure Standards** dialog box, pick the **Add Standards File** button (the one containing the plus sign) and open the **DWS** file you saved in Step 2.
6. Pick the **Check Standards…** button.

AutoCAD audits the current drawing against the **DWS** file and displays the results in the **Check Standards** dialog box.

7. Pick the **Settings…** button, check the **Automatically fix non-standards properties** check box, and pick **OK**.
8. Pick the **Next** button repeatedly until the **Checking Complete** box appears.

AutoCAD fixes the current drawing to conform to the standards contained in the **DWS** file.

9. Close the **Checking Complete** and **Check Standards** dialog boxes.

Appendix C
Drawing Area Guidelines

Scale Type	Sheet Size	Approx. Plotting Area	Scale	Drawing Area
Architect's Scale	A: 12″ × 9″ B: 18″ × 12″ C: 24″ × 18″ D: 36″ × 24″ E: 48″ × 36″	10″ × 8″ 16″ × 11″ 22″ × 16″ 34″ × 22″ 46″ × 34″	$1/8″ = 1′$ $1/2″ = 1′$ $1/4″ = 1′$ $3′ = 1′$ $3′ = 1′$	96′ × 72′ 36′ × 24′ 96′ × 72′ 12′ × 8′ 16′ × 12′
Civil Engineer's Scale	A: 12″ × 9″ B: 18″ × 12″ C: 24″ × 18″ D: 36″ × 24″ E: 48″ × 36″	10″ × 8″ 16″ × 11″ 22″ × 16″ 34″ × 22″ 46″ × 34″	1″ = 200′ 1″ = 50′ 1″ = 10′ 1″ = 300′ 1″ = 20′	2400′ × 1800′ 900′ × 600′ 240′ × 180′ 10,800′ × 7200′ 960′ × 720′
Mechanical Engineer's Scale	A: 11″ × 8″ B: 17″ × 11″ C: 22″ × 17″ D: 34″ × 22″ E: 44″ × 34″	9″ × 7″ 15″ × 10″ 20″ × 15″ 32″ × 20″ 42″ × 32″	1″ = 2″ 2″ = 1″ 1″ = 1″ 1″ = 1.5″ 4″ = 1″	22″ × 17″ 8.5″ × 5.5″ 22″ × 17″ 51″ × 33″ 11″ × 8.5″
Metric Scale	A: 279 mm × 216 mm B: 432 mm × 279 mm C: 55.9 cm × 43.2 cm D: 86.4 cm × 55.9 cm E: 111.8 cm × 86.4 cm	229 mm × 178 mm 381 mm × 254 mm 50.8 cm × 38.1 cm 81.3 cm × 50.8 cm 106.7 cm × 81.3 cm	1 mm = 5 mm 1 mm = 20 mm 1 cm = 10 cm 2 cm = 1 cm 1 cm = 2 cm	1395 × 1080 8640 × 5580 559 × 432 43.2 × 27.95 237.6 × 172.8

Note: 1″ = 25.4 mm

Appendix D
Dimensioning Symbols

Geometric Characteristic Symbols

Type of Tolerance	Symbol	Name
Location	◎	Concentricity
	⌖	Position
	≡	Symmetry
Orientation	∠	Angularity
	//	Parallelism
	⊥	Perpendicularity
Form	⌭	Cylindricity
	▱	Flatness
	○	Circularity (roundness)
	—	Straightness
Profile	⌒	Profile of a line
	⌓	Profile of a surface
Runout	↗	Circular runout
	↗↗	Total runout
Supplementary	Ⓜ	Maximum material condition (MMC)
	Ⓛ	Least material condition (LMC)
	Ⓟ	Projected tolerance zone

Basic Dimensioning Symbols

Symbol	Type of Dimension	Symbol	Type of Dimension
Ø	Diameter	⌄	Countersink
R	Radius	⌴	Counterbore/Spotface
SR	Spherical radius	⤓	Deep
SØ	Spherical diameter	×	Places, times, or by
( )	Reference		

Appendix E
AutoCAD Fonts

AutoCAD provides several standard fonts, which have file extensions of **SHX**. You can use the **STYLE** command to apply expansion, compression, or obliquing to any of these fonts, thereby tailoring the characters to your needs. (See Chapter 20 for details on the **STYLE** command.) You can draw characters of any desired height using any of the fonts.

Examples of some of the fonts supplied with AutoCAD are listed in this appendix, along with samples of their appearance. With the exception of **monotxt.shx** (not included in the samples below), each font's characters are proportionately spaced. Hence, the space needed for the letter "i," for example, is narrower than that needed for the letter "m."

Each font resides in a separate file. This is the "compiled" form of the font, for direct use by AutoCAD. Examples of standard and TrueType fonts are shown in this appendix.
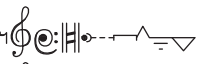
TrueType Fonts

AutoCAD uses several TrueType fonts and font "families" (groups of related fonts). In the TrueType fonts, each font in a family has its own font file. The examples below are shown in outline form, as they appear by default in

TrueType Fonts

Name	Appearance
Arial Narrow	ABCDEFGHIJKLMNOPQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz0123456789
Dutch801 Rm BT	*ABCDEFGHIJKLMNOPQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz0123456789*
Lucida Console	ABCDEFGHIJKLMNOPQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz0123456789
Swis721 BdOul BT	ABCDEFGHIJKLMNOPQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz0123456789
UniversalMath1 BT	ΑΒΨΔΕΦΓΗΙΞΚΛΜΝΟΠΘΡΣΤΘΩϬΧΥΖ αβψδεφγηιξκλμνοπϑρστθωφχυζ″ + − × ÷ = ± ∓°′
Wingdings	✂✃✄✆✇✈✉✍✎✏☺☻☹✺✻❀✿✾⚜✠✡✢✣✤✥✦✧✪☆ ♋♌♍♎♏♐♑♒♓✝✞✟●○■□▢□□□□◆♦❖❖◆⊠⊡⌘□□

Standard Fonts

Name	Appearance
txt.shx	ABCDEFGHIJKLMN☐PQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz
romans.shx	ABCDEFGHIJKLMNOPQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz
romand.shx	ABCDEFGHIJKLMNOPQRSTUVWXYZ abcdefghijklmnopqrstuvwxyz0123456789
italicc.shx	*ABCDEFGHIJKLMNOPQRSTUVWXYZ* *abcdefghijklmnopqrstuvwxyz0123456789*
scripts.shx	*ABCDEFGHIJKLMNOPQRSTUVWXYZ* *abcdefghijklmnopqrstuvwxyz0123456789*
gothice.shx	**ABCDEFGHIJKLMNOPQRSTUVWXYZ** **abcdefghijklmnopqrstuvwxyz0123456789**
syastro.shx	☉☿♀⊕♂♃♄♅♆♇☾☽✶☊☋♈♉♊♋♌♍♎♏♐♑♒♓≈ ✷''⊂⊃∩∈→↑←↓∂∇⌃`˜א§†‡Ⅎℒ£®©
symusic.shx	˙.˒°○●♯♮♭━-ˣˠ𝄞◉:𝄀𝄽·━⌐⌐‿▽ ˙.˒°○●♯♮♭━-ˠˣ𝄞♪:𝄢☉☿♀⊕♂♃♄♅♆♇

AutoCAD. To display solid characters, set the **TEXTFILL** system variable to **1**. You may also change the print quality by changing the value of the **TEXTQLTY** system variable.

Standard Fonts

AutoCAD's standard **SHX** fonts include both text and symbol files that have been created as AutoCAD shapes. You can change the appearance of these fonts by expanding, compressing, or slanting the characters.

Glossary of Terms

Note: Numbers in parentheses are chapter numbers.

A

absolute points Specific coordinate points entered directly (without using relative positioning). (7)

additive manufacturing A process of joining liquids, powders, filaments, or sheet materials to form parts from 3D model data in plastics, metals, ceramics, or composite materials, usually layer upon layer as opposed to subtractive manufacturing methodologies. (48)

alignment grid A non-printing grid that can be set at any desired spacing in either **Standard** or **Isometric** drawing mode. (12)

alignment paths Temporary, non-printing lines that appear at predefined or customizable angles to help you create objects more accurately. (7)

alternate units A second set of distances shown inside brackets on some dimensioned drawings. (29)

aperture box A small box that defines an area around the center of the crosshairs within which an object or point will be selected when you press the pick (left) button. (10)

arcball An object that displays on the screen when **3D Orbit** is active; it allows the user to manipulate the drawing view in realtime. (39)

area The number of square units needed to cover an enclosed 2D shape or surface. (31)

array An orderly arrangement of objects. AutoCAD can create rectangular and circular, or polar, arrays. (17)

aspect ratio The ratio of height to width of a rectangular object or region. (12)

associative dimensions Dimensions that update automatically when you change the size of objects by stretching, scaling, etc. (28)

associative hatch A hatch object that updates automatically when its boundaries are changed; a hatch created with the **BHATCH** command. (19)

attributes Text data stored in blocks. (36)

attribute tag A variable that identifies each occurrence of an attribute in a drawing. (36)

attribute values The information stored by attribute tags in specific instances of an attribute. (36)

audits Examines the validity of a drawing file. (9)

B

baseline dimensions A set of dimensions that have a common baseline to reduce measurement error that can accumulate from "stacking" dimensions. (27)

basic dimension A dimension to which allowances and tolerances are added to obtain limits. (30)

bisect Divide into two equal halves. (48)

block A collection of objects that you can group to form a single object. (33)

Boolean logic (44, see *Boolean mathematics*.)

Boolean mathematics The mathematical principles behind the creation of Boolean solid models. (44)

Boolean operations AutoCAD operations such as union, subtraction, and intersection, which are based on Boolean mathematics and are used to create composite solids. (44)

B-splines Splines created using the **PEDIT** or **SPLINE** command. (See *splines*.) (15)

button In AutoCAD, the small areas in toolbars and palettes that, when pressed, enter an associated command or function. (1)

C

camera The point of view that moves around the target in **3D Orbit**. (39)

Cartesian coordinate system A system used in geometry that assigns a specific coordinate pair (*x,y*) for 2D or triplet (*x,y,z*) for 3D to points in space so that each point has a unique identifier. (7)

cascading menu A submenu in a pull-down menu that offers further tool choices. (3)

cells The individual boxes in a table. (22)

center lines Imaginary lines that mark the exact center of an object or feature; shown by a series of alternating long and short line segments. (24)

chamfers Beveled edges. (16)

circular array An array in which objects are arranged radially around a center point. Also called a *polar array*. (17)

circumference The distance around a circle. (31)

clipping plane A plane that slices through one or more 3D objects, causing the part of the object on one side of the plane to be omitted. (39)

colinear Two or more line segments lying on the same infinite line. (18)

color palette A selection of colors, similar to an artist's palette of colors. (24)

comma-delimited file A file format that allows a user to write attributes to an ASCII text file. Entries in this file format are separated by commas. (37)

command aliases Command abbreviations that usually consist of the first one to three letters of a command name. (4)

Command line A text area in AutoCAD where you can key in commands and options and view information. (1)

compass When the **VPOINT** command is active, the compass is a globe representation that allows the user to orient the drawing in 3D space. (39)

composite solid A solid composed of two or more solid objects. (44)

concentric Sharing a common center point. (5)

constrain Assign limits or values for parameters in dynamic blocks. (34)

construction lines Line objects in AutoCAD that extend infinitely in both directions. (12)

context-sensitive A menu that changes depending on which option is selected. (2)

context-sensitive help Help related to the currently entered command. (8)

control panels Sections within the **Dashboard**, which appears in the **3D Modeling** workspace, that hold related buttons and controls. (5)

control points Points on or near a curve that exert a "pull," influencing the shape of the curve. (15)

coordinate pair A combination of one *x* value and one *y* value that specifies a unique point in the 2D coordinate system. (7)

coordinates Sets of *x,y* (for 2D drawings) or *x,y,z* (for 3D models) values used to describe specific locations in space. (7)

copy and paste A feature that allows you to copy objects from one drawing and insert them into another. (33)

crosshairs A special cursor that consists of two intersecting lines and a pickbox; used to select objects and pick points in the drawing area. (2)

crosshatch (See *hatch*.)

D

data extraction The process of gathering attribute data and placing it into an AutoCAD table or an electronic file that can be read by a computer program.

datum A surface, edge, or point that is assumed to be exact. (27)

datum dimensions (27, see *ordinate dimensions*.)

delta In mathematics, a Greek symbol used to signify change. (31)

deviation tolerancing A method of tolerancing that allows the user to set the upper and lower tolerances separately (to different values). (30)

dialog box A window that provides information and permits you to make selections and enter information. (1)

diameter The length of a line that extends from one side of a circle to the other, passing through its center. (5)

dimensioning A system of describing the size and location of features on a drawing. (27)

dimension line The part of a dimension that typically contains arrowheads at its ends and shows the extent of the area being dimensioned. (27)

dimension styles Saved sets of dimension settings that control the format and appearance of dimensions and leaders in a drawing. (28)

docked toolbar A toolbar that is attached to the top or side of the drawing area. (3)

donuts Thick-walled or solid circles created using the **DONUT** command. (5)

double-clicking Positioning the pointer on an object or button and pressing the pick button twice. (1)

drawing area The part of the AutoCAD screen in which the drawing or model appears. Also refers to the boundaries for constructing the drawing, which should correspond to both the drawing scale and the sheet size. (1)

Wohlers: Applying
AutoCAD® 2010

Back Matter

Glossary of Terms

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757

drawing database Information associated with each AutoCAD file that defines the objects in the drawing and contains specific numerical information about individual objects in the file and their locations. (31)

dynamic blocks Blocks created with certain changeable characteristics, or parameters, such as length, width, rotation, or location. (34)

dynamic input Entering commands and options at the cursor. (4)

dynamic UCS A feature that automatically moves the current UCS to a selected face of a solid model, temporarily making that face the work plane. Dynamic UCS allows you to create objects in 3D without constantly changing UCSs. (45)

E

elevations Z heights in three-dimensional space. (40)

ellipse A regular oval shape that has two centers of equal radius. (5)

entity An individual predefined element in AutoCAD; the smallest element that you can add to or erase from a drawing. Also called an *object*. (6)

extension lines The lines in a dimension that extend from the object to the dimension line. (27)

extrude To add depth to a 2D region or closed polyline. (43)

extrusion thickness Thickness in the Z axis; can be set using the **THICKNESS** variable or by applying the **EXTRUDE** command. (39)

F

feature control frames The frames used to hold geometric characteristic symbols and their corresponding tolerances. (30)

file attributes The characteristics of a file, such as its size, type, author, and the date and time it was created and last modified. (9)

fillets Rounded inside corners. (16)

filtering Including or excluding items from a selection set using specific criteria. (24)

fixed attributes Attributes whose values you define when you first create a regular (nondynamic) block. (36)

floating toolbar A toolbar that displays as a window in the drawing area. (3)

font Distinctive set of characters consisting of letters, numbers, punctuation marks, and symbols. (20)

formula An expression that performs an automatic mathematical equation in an AutoCAD table. (22)

freeze In AutoCAD, to remove a layer from view without turning it off or deleting it. (24)

full section A section that extends all the way through a part or object. (19) A view of a 3D model that slices all the way through the part, showing the interior of the part as it appears at that slice or cross section. (46)

G

geometric characteristic symbols Symbols that are used to specify form and position tolerances on drawings. (30)

geometric dimensioning and tolerancing (GD&T) Standardized format for dimensioning and showing variations (called *tolerances*) in the manufacturing process. Tolerances specify the largest variation allowable for a given dimension. GD&T uses geometric characteristic symbols and feature control frames that follow industry standard practices. GD&T standards are defined in the American National Standards Institute (ANSI) Y14.5M Dimensioning/ Tolerancing handbook. (30)

gizmo An icon that simplifies the section process for 3D commands. (45)

glyph In AutoCAD, an icon that has a specified location in the drawing area and represents one of AutoCAD's tools, such as a spotlight or camera. (47)

grips Small boxes that appear at key points on an object when you select the object without first entering a command. Grips allow you to perform several basic operations such as moving, copying, or changing the shape or size of an object. (6)

ground plane The XY plane in 3D space, on which all Z values = 0. (39)

group A named set of objects. A circle with a line through it, for instance, could become a group. Unlike blocks, groups cannot be saved as files. (32)

H

hatch A repetitive pattern of lines or symbols that shows a related area of a drawing. (19)

helix An open 3D spiral. (43)

hidden line removal The deletion of tessellation lines that would not be visible if you were viewing the 3D model as a real object. (39)

hidden lines Lines that would not be visible if you were looking at an object without "see-through" capability; shown as dashed or broken lines. (12) Dashed or broken lines used to show lines that would not be visible if you were looking at an actual object. (24) Lines removed in a plot because they would not be visible if you were looking at the actual object. (25)

I

icon The small picture on a button in AutoCAD that helps identify the function of the button. (1)

image tiles Selectable squares displayed in some dialog boxes that contain thumbnail images of tools, procedures, or options from which the user can choose. (16)

in-place text editor The text editor that allows you to edit dynamic text and mtext in place on the screen. (21)

integer Any positive or negative whole number or 0. (31)

interference The overlap of two or more objects that occupy the same area in 3D space. (45)

island An area within an object, such as a hole, that is not hatched. (19)

isometric drawing A type of 2D drawing that gives a 3D appearance. (38)

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Back Matter

Glossary of Terms

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759

isometric planes Three imaginary drawing planes spaced at 120° from each other; these planes are used to draw isometric views. (38)

J

justify Align multiple lines of text at a specific point (left, right, center, etc.) relative to the point of insertion. (20)

L

landscape A paper orientation in which the wider edge of the paper is at the top and bottom of the sheet. (25)

layers Similar to transparent overlays in manual drawing; allow AutoCAD users to display and hide information to clarify the drawing. (24)

layout An arrangement of one or more views of an object or assembly on a single sheet. (26)

leaders Lines with an arrowhead at one end that point out a particular feature in a drawing. (28)

limits The size of the drawing area, which can be specified using the **LIMITS** command or by using a **Startup** wizard. (23)

limits tolerancing A method of tolerancing in which only the upper and lower limits of variation for a dimension are shown. (30)

linear dimensions Vertical and horizontal dimensions and dimensions that are aligned with an edge of the object being dimensioned. (27)

live sectioning A setting you can specify when creating a section to provide realtime views while manipulating the grips that control where a section slices through a 3D model. (46)

lofting Creating a new solid model or surface by specifying a series of 2D cross sections. (43)

M

material condition symbols Symbols used in geometric dimensioning and tolerancing to modify the geometric tolerance in relation to the produced size or location of the feature. (30)

Maximum Material Condition (MMC) A notation specifying that a feature such as a hole or shaft is at its maximum size or contains its maximum amount of material. (30)

menu bar A row of menu items located below the title bar. (3)

mesh Unlike a solid primitive, a mesh is not closed volume. However, mesh primitives can be modified more easily and more extensively than solid primitives and can be converted to a solid at any time. (42)

mesh model Created using a mesh modeling operation. (43)

mirror To produce a mirror image of an object or set of objects. (17)

mirror line The axis or line about which objects are mirrored using the **MIRROR** command. (17)

mline (See *multiline*.) (16)

model space The drawing mode in AutoCAD in which most drawing is done. (26)

mtext A multiple-line text object created using AutoCAD's text editor. (20)

multiline A line object in AutoCAD that consists of up to 16 parallel lines created simultaneously and viewed by AutoCAD as a single object. (16)

multiview drawing A drawing that describes a three-dimensional object completely using two or more two-dimensional views. (12)

N

notes Text that refers to an entire drawing, rather than one specific feature. (See also *specifications*.) (20)

noun/verb selection An object selection method in which you select the object first, then enter a command (verb) to perform an operation on the object. (6)

O

object A predefined element; the smallest element that you can add to or erase from a drawing. Also called an *entity*. (6)

object snaps Magnet-like aids in AutoCAD that allow the user to "snap" to endpoints, midpoints, and other specific points easily and accurately. (10)

object snap tracking A feature that, when used in conjunction with object snap, provides alignment paths that help the user produce objects at precise positions and angles. (10)

offsetting Creating a new object (such as a line or circle) at a specific distance from an existing line or circle. (12)

opacity In AutoCAD, the value of the **VSFACEOPACITY** system variable. A value of **100** makes objects opaque; a value of **0** makes them transparent. (47)

ordinate dimensions Dimensions that use a datum, or reference dimension, to help avoid confusion that could result from "stacking" dimensions; also called *datum dimensions*. (27)

origin The point at which the axes cross in a coordinate system. (7)

ortho A mode in AutoCAD that forces all lines to be perfectly vertical or horizontal. (11)

orthogonal Drawn at right angles. (11)

orthographic projection A projection of views at right angles to each other in a multiview drawing. (12)

P

palette A special toolbar that contains a large grouping of buttons, tools, or information. (3)

panning A feature that allows the user to move the viewing window around the drawing without changing the zoom magnification. (14)

paper space The space, or drawing mode, in AutoCAD from which multiple views can be arranged and plotted on the same sheet. (26)

parallel projection The standard projection mode in AutoCAD, in which objects are shown in parallel regardless of perspective. (39)

parameters Characteristics in a dynamic block that can be changed after the block is inserted into a drawing. (34)

parametric shape editing Editing an object's shape by activating grips and entering a new size in the text box that appears. (6)

perimeter The distance around a two-dimensional shape or surface. (31)

perpendicular Lines, polylines, or other linear or curved objects that meet at a precise 90° angle. (10)

perspective projection Projection mode in which parts of an object that are farther away appear smaller, the same as in a real-life photograph.

phantom lines Lines drawn using a thick lineweight, alternating two short dashes with one long dash; used for cutting planes in sectional views. (24)

pickbox A cursor that consists of a small box that allows you to pick objects on the screen. (6)

pictorial representation A drawing of an object that offers a realistic view, compared to orthographic projections. (38)

plane An imaginary flat surface that can be defined by any three points in space or by a coplanar object. (12)

plot scale The scale at which a drawing is plotted to fit on a drawing sheet. The plot scale has no effect on the sizes of the objects in the drawing file. (25)

plot style A collection of property settings saved in a plot style table. (25)

plotters Originally, output devices that could handle very large sheets of paper; now used interchangeably with *printers*. (25)

polar array (See *circular array*.) (17)

polar method A relative method of entering points in which the user can produce lines at specific angles around a central point. (7)

polar tracking method A method of entering points in which AutoCAD displays alignment paths at prespecified angles to help the user place the points at precise positions and angles. (7)

polyline A line object in AutoCAD that consists of a connected sequence of line and arc segments and is treated by AutoCAD as a single object. (15)

polysolid A solid primitive that is similar to a two-dimensional polyline, but is three-dimensional. (42)

portrait A paper orientation in which the narrower edge of the paper is at the top and bottom of the sheet. (25)

primary units The units that appear by default when you add dimensions to a drawing in AutoCAD. (29)

primitives The basic building blocks of solid modeling, including the box, sphere, cone, cylinder, torus, wedge, pyramid, and polysolid. (42)

printers Originally small, tabletop output devices; now used interchangeably with *plotters*. (25)

projected tolerance zone Controls the height of the extended portion of a perpendicular part. (30)

pull-down menus Menus whose names appear across the top of the screen, from which you can perform many drawing, editing, and file manipulation tasks. (2)

purge In AutoCAD, selectively deleting blocks or other elements that are not currently in use in a drawing. (33)

Q

quadrant A quarter of a circle, defined by the points at exactly 0°, 90°, 180°, and 270°. (10)

quadrant points The points at 0°, 90°, 180°, and 270° on a circle. (10)

R

radial Describes a feature on which every point is the same distance from an imaginary center point. (5)

radius The length of a line extending from the center to the periphery of an arc or circle. (5)

rays Line objects in AutoCAD that extend infinitely in one direction from a specified point. (12)

realtime zooming A method of changing the level of magnification by picking the **Realtime Zoom** button and moving the pointing device. (13)

rectangular array An array in which the objects are arranged in rows and columns. (17)

regular polygon A polygon in which all the sides are of equal length. (5)

relative method A method of entering points that allows the user to base the new points on the position of a point that has already been defined. (7)

rendered An object that has shading applied. (25)

rendering A display of an object that combines visual styles, lighting, materials, and shadows to create a photorealistic image of a solid model. (47)

resolution The number of pixels per inch on a display system; this determines the amount of detail you can see on the screen. (1)

revolve Turn a 2D object about an axis to create a 3D model. (43)

roll In **3D Orbit**, moving the view around an axis at the center of the arcball. (39)

rotating Changing the angle of an object in the drawing without changing its scale. (18)

rounds Curved or rounded outside corners on an object. (16)

rubber-band effect The stretching effect when a command is active that lets you see the line, circle, etc., "grow" as you move the pointing device. (4)

running object snap modes/running object snaps Object snaps that have been preset to function automatically as you work in AutoCAD. (10)

S

scale Increasing or decreasing the overall size of an object without changing the proportions of the parts to each other. (18) A means of reducing or enlarging a representation of an actual object or part that is too small or too large to be shown on a drawing sheet. (23)

screen regeneration Recalculation of each vector, in a drawing. (15)

section drawings Drawings used to show the interior of a part. (19)

section plane indicator A transparent plane that appears to show where a section will be taken in a 3D model. (46)

selectable In AutoCAD groups, *selectable* means that when you select one of the group's members (*i.e.*, one of the circles), AutoCAD selects all members in the group. (32)

selection previewing AutoCAD's automatic highlighting of objects as you run the cursor over them. (6)

selection set All of the objects that are currently highlighted, or selected, for a command or operation. (6)

selection window A window created by picking diagonally opposite corners of an imaginary box at the **Select objects** prompt. (6)

shortcut menu A context-sensitive menu that appears when you right-click; shortcut menus contain several of the commands and functions the user is most likely to need in the current drawing situation. (4)

snap grid An invisible, user-controlled grid that restricts cursor movement to points that lie on the grid. (12)

solid model In AutoCAD, the type of model produced when a modeling operation results in a closed (watertight) volume. (43)

solid modeling Computer modeling of three-dimensional volumes of solid parts. CAD solid modeling programs that take advantage of constructive solid geometry (CSG) techniques use primitives, such as cylinders, cones, and spheres, and Boolean operations to construct shapes. CAD programs that use the boundary representation (B-Rep) solid modeling technique store geometry directly using a mathematical representation of each surface boundary. Some CAD programs use both CSG and B-Rep modeling techniques. (42)

special characters Characters that are not normally available on the keyboard or in a basic text font. (21)

specifications Text that provides information about sizes, shapes, and surface finishes that apply to specific portions of an object or part. (20)

spline leader Similar to a regular leader, except that the leader line consists of a spline curve, offering more flexibility with its shape. (28)

splines Curves that use sampling points to approximate mathematical functions that define a complex curve. (15)

status bar The line at the bottom of the AutoCAD screen that displays various types of status information about modes and settings, including snap, grid, the ortho mode, polar tracking, object snap, object tracking, and lineweight. (1)

subobject A face, edge, or corner of a solid that can be edited directly using grip tools. (45)

surface controls Feature control frames not associated with a specific dimension that refer to a surface specification regardless of feature size. (30)

surface model In AutoCAD, the type of model produced when a modeling operation results in an open 3D surface. (43)

sweeping Moving a 2D profile along a path curve to create a 3D model. Sweeps can result in solids or surfaces, depending on whether the profile is open (surface) or closed (solid). (43)

symbol library Drawing file that contains a series of blocks for use in other drawings. (35)

symmetrical deviation (See *symmetrical tolerancing*.)

symmetrical tolerancing A method of tolerancing in which the upper and lower limits of a plus/minus tolerance are equal. (30)

system variable Similar to a command, except that it holds temporary settings and values instead of performing a specific function. (10)

T

table A set of rows and columns that contain text. (22)

tangent Circles, arcs, or other objects that meet at a single point. (5)

template file A file that contains predefined characteristics and is used to create new drawings with those characteristics. (23) A file that defines text format for attribute extraction. (37)

tessellations A collection of planer facets on a curved surface that approximate curvature. (42)

3D faces Three- or four-sided planar meshes that have no width. (40)

thumbnail A small picture of a drawing that gives you a quick view of the graphic content. (1)

title block A portion of a drawing that is set aside to give important information about the drawing, the drafter, the company, and so on. (20)

tolerance A designation that specifies the largest variation allowable for a given dimension. (30)

toolbar A collection of related buttons that can be displayed or hidden by the user. (3)

tooltips One or more words that appear when you position the pointer over a button for a second or longer. The purpose of tooltips is to help you understand the purpose or function of the button. (4)

traces Thick lines commonly used on printed circuit boards; also, lines drawn using AutoCAD's **TRACE** command. (15)

transparent zooms Zooms that the user forces to occur while another command is in progress. (13)

2D representation A single profile view of an object seen typically from the top, front, or side. (5)

U

UCS icon An icon located in the lower left corner of the drawing area that shows the orientation of the axes in the current coordinate system. (41)

user coordinate systems (UCSs) Custom coordinate systems that a user can define for use in creating 3D objects and models. (41)

V

value set The set of specified values to which a constrained parameter is limited in dynamic blocks. (34)

variable attributes Attributes whose value you can change as you insert the block. (36)

vector A line segment defined by its endpoints or by a starting point and direction in 3D space. (31)

verb/noun selection The traditional method of entering a command in AutoCAD, in which the user enters a command and then selects the object or objects on which the command should operate. (6)

vertex Any endpoint of an individual line or arc segment in a polyline, polygon, or other segmented object. (15)

vertex grip A corner grip on a solid model that has corners, such as a box or pyramid. (45)

view box The window that appears in **Aerial View** to display and define the current screen magnification and viewing window. (14)

viewport A portion of the drawing area that a user can define to show a specific view of a drawing. (25)

view resolution The accuracy with which curved lines appear on the screen. (13)

visual style The visual representation of objects in the drawing area, including the way the edges are displayed and the way the object is shaded. (25) The display style of a 3D model. (39)

W

wireframe A "stick" representation of a 3D model whose shape is defined by a series of connected lines and curves. (25) A representation of a 3D object in which lines and curves are used to define its boundaries. (40)

wizard A series of dialog boxes that steps you through a sequence of steps to perform a procedure. (10)

workspace A complete user interface, including the location and appearance of the drawing area and all menus, buttons, tool palettes, and toolbars. (3)

world coordinate system (WCS)
AutoCAD's name for the Cartesian coordinate system. (See *Cartesian coordinate system*.) (7) AutoCAD's default coordinate system, in which the X axis is horizontal on the screen, the Y axis is vertical, and the Z axis is perpendicular to the XY plane (the plane defined by the X and Y axes). (41)

X

xlines A type of construction lines created by the **XLINE** command that extend infinitely in both directions. (12)

xref An ordinary drawing file that has been attached to the current (base) drawing for reference. (37)

XYZ octant The area in 3D space where the x, y, and z coordinates are greater than 0. (48)

Z

Z axis The axis in a 3D coordinate system on which depth is measured. (39)

Index

A

Absolute method, and coordinates, 82–83
Absolute points, 82
Additive manufacturing, 632
  data outputting, 636–640
Aerial view, 178
Alignment grid, 147
Alignment paths, 85
All object selection, 71
Alternate units, 397
American National Standards Institute (ANSI); *See* ANSI
ANSI (American National Standards Institute), 250
ANSI hatch patterns, 250
Aperture box, 124
Aquatic designer, 468
Arc; *See also* Arc command
  button, 52–54
  command, 52–54
  creating, 52–54
  Start, Center, Angle button, 53
  Start, End, Radius button, 52
  tangent, 53–54
  3-Point, 52, 53
Arc button, Start, End, Radius button, 52
ARC command, 52–54; *See also* Arc
  continue option, 53–54

Architectural drafter, 490
Arc-related options, 189
Area, 422–423
AREA command, 422
Array
  circular, 222
  producing, 220–222
  rectangular, 220–221
Aspect ratio, 149
Associative dimensions, 382; *See also* Dimensioning, 382
ATTDISP command, 483
ATTEDIT command, 485
Attributes
  and bills of materials, 493–495
  block manager, 486
  creating fixed, 479–482
  displaying, 483
  editing, 485
  fixed, 479–482
  introduction to, 478–479
  storing, 482–483
  table in drawing, 495
  tags, 480
  values, 480
  variable, 483–484
AUDIT command, 106–108
Audits, 106
AutoCAD
  exiting, 7
  starting, 2–3
  uses of, 1
AutoCAD Classic, 22–23
  creating and saving a file, 34–35

opening file, 35–36
  toolbars, 24–25
AutoCAD text window, 134–136
  copying and pasting, 135–136
  displaying, 135
Auto-hide, 27–28
Axis, End button, 54

B

Baseline dimensioning, 364–365; *See also* Dimensioning
Basic dimension, 406
Basic modeling; *See also* Composite solids; Primitives; Solid model; 3D drawing; 3D orbit
  adjusting, 586–596
  beveling, 593–595
  camera, 622–625
  creating surfaces, 567–568
  edge modification, 616–617
  extrusion, 562–564
  lighting options, 617–619
  lofting, 566–567
  materials and rendering, 620–621
  mesh model, 559
  mesh object modification, 596
  modifying primitives, 587–590

Wohlers: Applying
AutoCAD® 2010

Back Matter

Index

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771

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AutoCAD® 2010

Back Matter

Index

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773

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Back Matter

Index

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775

776 Wohlers: Applying
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Back Matter

Index

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Back Matter

Index

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777

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Back Matter

Index

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779

Advanced Projects

The advanced projects are designed to help you develop your creativity and problem-solving skills. Each project describes a task or job as you might encounter it in the workplace.

Every project in this section requires considerable thought and planning. These projects are suitable for long-term projects for individuals or groups. They are not intended to be short-term projects.

Pepper Mill (page AP-2)

Design, create the plans for, and produce a solid model of a pepper mill.

Drawing Table (page AP-6)

Using a group approach and external references, complete the parts of and assemble a solid model of a student drawing table.

Mancala Game Board (page AP-9)

Design, create the plans for, and produce a solid model of a game board for an ancient African game called *Mancala*.

Bolted Seat Connection (page AP-13)

Create structural details for the connection between beams in a steel frame building and then create a solid model representing the connection.

Bushing Assembly (page AP-17)

Create production drawings for a bushing assembly consisting of bushing holders, bushings, and a housing box; then create a solid model of each piece.

Architectural Plans (page AP-20)

Develop a complete set of residential architectural plans according to the client's specifications.

Buffet Tray (page AP-22)

Design a lap tray for use at a buffet or picnic; document the design, and then create solid models of the individual pieces and display them as an exploded assembly.

Shackle Assembly (page AP-25)

Create models of each piece of the shackle assembly and assemble them; slice the assembly and remove half so that the interior is visible.

Clothing Rack (page AP-29)

Create solid models of the pieces of this wall-mounted clothing rack and arrange the models into an exploded assembly.

Protective Packaging (page AP-34)

Design and develop plans for protective packaging for shipping a product.

Advanced Project

PEPPER MILL

Shown in Fig. AV1-1 is a rendered model of a disassembled pepper mill. The complete assembly is composed of the following eight parts:

- 1 plastic top
- 1 plastic barrel to hold the peppercorns
- 1 stainless steel stator
- 1 rotor
- 1 threaded shaft
- 2 nuts
- 1 spring

An exploded assembly view of the eight parts is shown in Fig. AV1-2. The shaft and one nut are permanently affixed to the top. When the top is turned, the shaft and rotor rotate, grinding the peppercorns between the rotor and stator.

Your supervisor has asked you to improve the design by creating a more attractive top and barrel. Do not modify the shaft/rotor/stator/spring/nut assembly, except to adjust the length of the shaft to accommodate the new

Fig. AV1-1

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Advanced Projects

Pepper Mill

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783
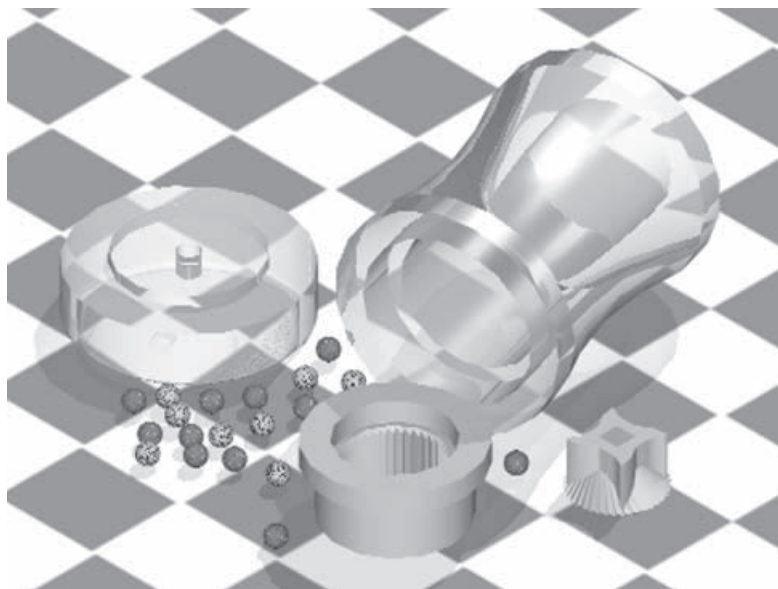
Advanced Projects

design. (The dimensions of these parts are given in Fig. AV1-4 on page 743.) Concentrate instead on the shape of the pepper mill and the material from which it is made. Consider ergonomic factors: Does your new shape make the pepper mill easier to hold and use?

PART 1

Using the dimensions given, create a new design for the top and barrel. Perform the following tasks:

- Draw the two-dimensional views of all the parts, inserting all hidden and center lines.

- Dimension all of the parts.

- Draw the front view of the assembly with the parts exploded. All of the parts should be drawn as full sections, using the hatch patterns appropriate for the materials. Omit hidden lines, but include the center line.

- Label each part with a circle ("balloon") containing its part number and accompanied by its title and material.

- Add a border and title block.

The complexity of the rotor teeth makes drawing this part especially challenging. A dimensioned single tooth is shown in Fig. AV1-3. While it is a simple object, it is rotated about its axis 10° outward from the center of the rotor, rotated 20° clockwise about the midpoint of its lower edge, and leaned 35° toward the center of the rotor. This twisting and turning results in a drawing complexity that is beyond the scope of this textbook. Consequently, when drawing the rotor, approximate its appearance by following the dimensions given in Fig. AV1-3.
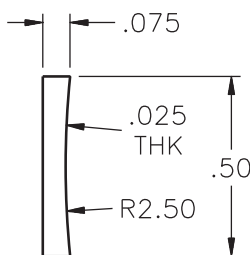
Fig. AV1-2
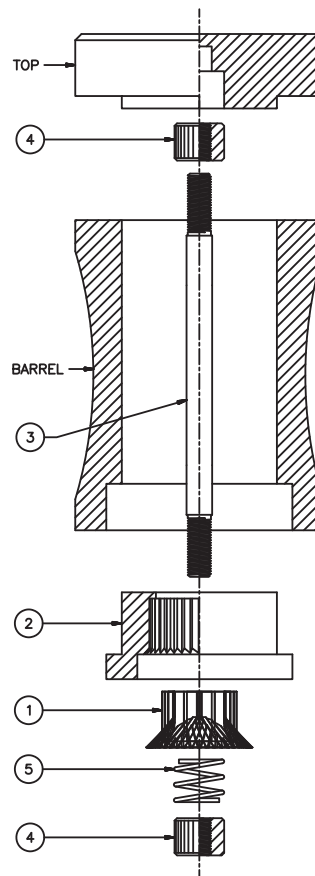
Fig. AV1-3

Advanced Projects

After creating the top and barrel (freehand sketches may help), you should establish how you will arrange the views of the objects in the drawing. There are so many parts to this assembly that it will probably pay you to draw rectangles representing the objects and their dimensions, and then use them to lay out the parts and make decisions about scales and how many pages will be required. Given the smallness of the parts, you should plan to plot them to at least double scale.

Most of the dimensions of the parts are "friendly"; that is, they have a common multiple. This makes the use of snaps and grids particularly helpful. Some of the parts, such as the rotor and stator, are especially suitable for the **ARRAY** and **MIRROR** commands. (You should always examine parts for symmetries so you can make use of AutoCAD's powerful replication commands.)

With all your preliminary settings completed and the format of your drawings decided (and assuming that you already have a drawing template file containing most of the layers, text style, linetypes, and dimension style), you should outline the steps you will take to create the drawing. The general strategy should be:

1. Draw the needed views of all of the parts. As is often the case with objects that have circular features, it is easiest to begin with the views in which circular features appear as circles or arcs. Then use these views to construct the others.
2. Use copies of the appropriate views to create the exploded assembly.
3. Dimension the orthographic views.

PART 2

Make solid models of the pieces of your design. Then perform the following tasks:

- Arrange the solids in an exploded assembly and display them in a view in which you are looking upward at the assembly from below.
- Determine the volumes and the surface areas of all the pieces.
- Look up the densities of the materials of your objects (you may have to go to the reference section of a suitable library or search the Internet); use the densities to determine the weights of the parts.
- Render the assembly using suitable materials.

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Advanced Projects

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785

Advanced Projects

① ROTOR

② STATOR

③ SHAFT

⑤ SPRING — ⌀.03125

④ NUT — 2 REQD

Fig. AV1-4

Pepper Mill project courtesy of Gary J. Hordemann, Gonzaga University
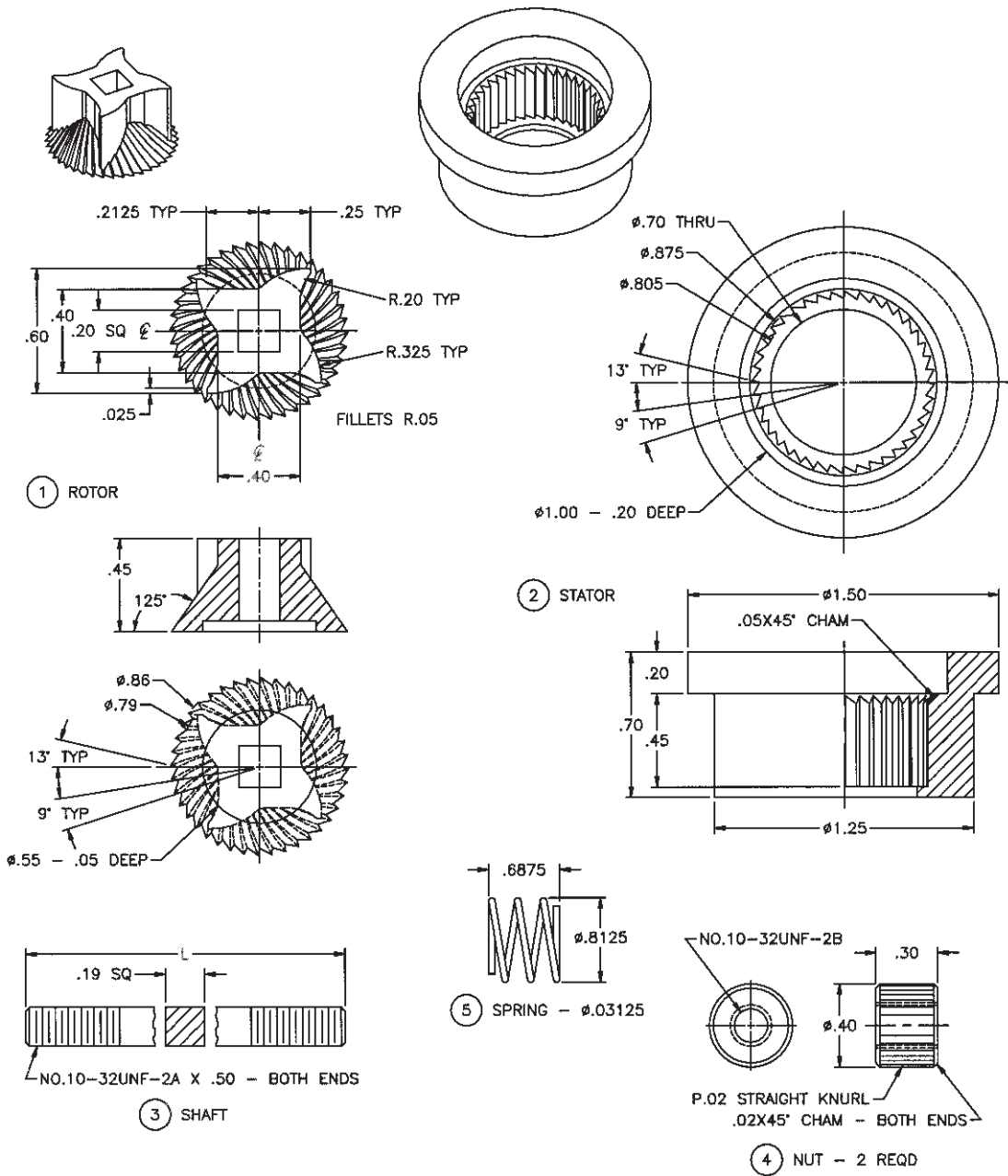
Advanced Project

DRAWING TABLE

Create the solid model of a student drawing table as shown in Fig. AV2-1. Consider making this a group project. Determine which group member will be responsible for each part of the table. When the individual parts have been created, place the files in a central location. As an optional activity, the group should consider using AutoCAD's external referencing capability to ensure proper fitting of the parts.

As a group project, this project works best with a team of 4 people. Assign each person one of the following pieces of the assembly:

- table top
- table drawer
- frame assembly (legs and feet)
- mounting assembly for the top (tubular support and screw heads)

Fig. AV2-1

Advanced Projects

As with any solid model, there are several ways to create a model of the drawing table. Study the parts and choose the method you think will work best. If it doesn't work as well as you'd like, you may want to

Fig. AV2-2

consider a different method. The following illustrations and tips may give you some ideas about how to approach each part of the project. For example, one way to create the table frame is to draw a path for the legs and then use the **EXTRUDE** command. You may want to consider using the **REVOLVE** command to create the feet of the table legs.
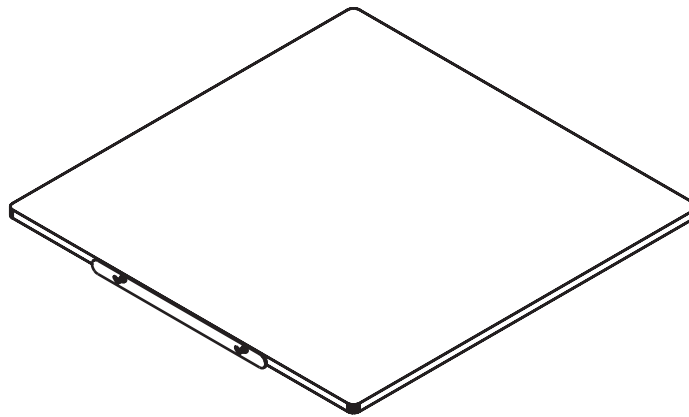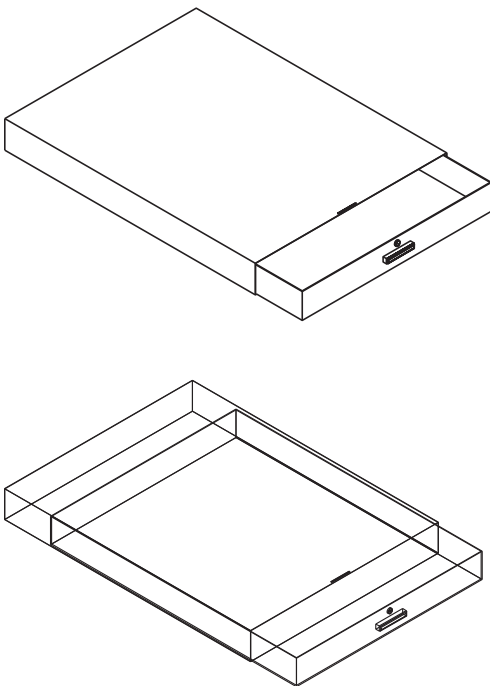
Fig. AV2-3

Fig. AV2-2 shows a hidden-line view of the table top. Solid primitives of various types may be useful in creating parts such as this.

The drawer handle, hinges, and brackets may also include primitives. Two views of the drawer are provided in Fig. AV2-3. The wireframe view may give you an idea of how the drawer should fit together. Don't forget to hollow out the inside of the drawer. Measure a real drawer to determine the thickness of the four sides of the drawer.

The support assembly, which attaches the table top to the frame, consists of a tubular supporting bracket and holder and

Continued

Advanced Projects

Philips-head screws. The tubular bracket is shown in Fig. AV2-4. Fig. AV2-5 shows a closeup of a screw head, and Fig. AV2-6 shows a wireframe view of the assembly in position under the table top.

Fig. AV2-4

Fig. AV2-5

Fig. AV2-6

Drawing Table project courtesy of Bill Fell, AHST High School
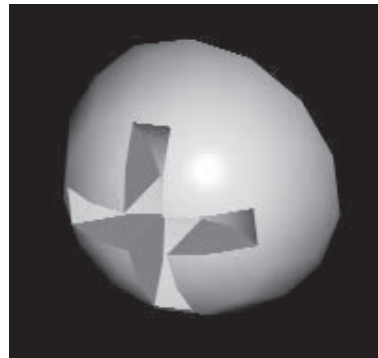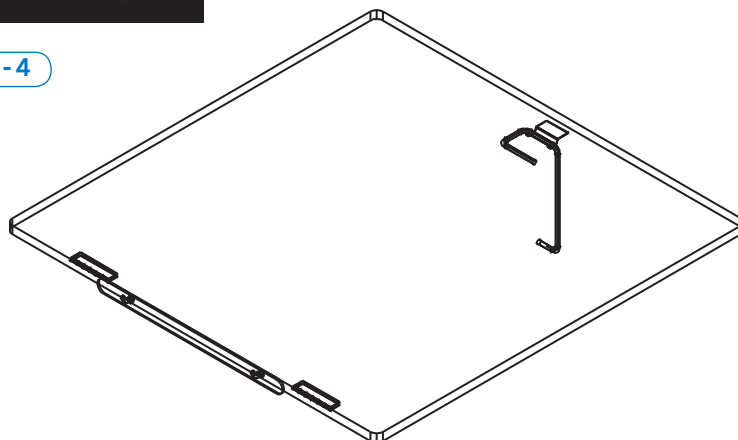
Advanced Project

MANCALA GAME BOARD

Mancala is an ancient bean game originating in Africa. It can be played with beans, pebbles, marbles, or any other small objects, which are placed into holes (pockets) dug into the ground or routed out of a board, as shown in Fig. AV3-1.

This project has three parts. In Part 1, you will create all the orthographic views necessary to build a *Mancala* board. In Part 2, you will design your own version of a *Mancala* board. In Part 3, you will create a solid model of the *Mancala* board from Part 1.

PART 1

Your task in this part is to create completely dimensioned views of the board. A number of views can be drawn, but the obvious choice is the top and front views. The fillet and depth dimensions of the pockets further suggest that the front view be drawn as a half or full section.

The board is to have the dimensions shown in Fig. AV3-2. The positioning

Fig. AV3-1

Continued
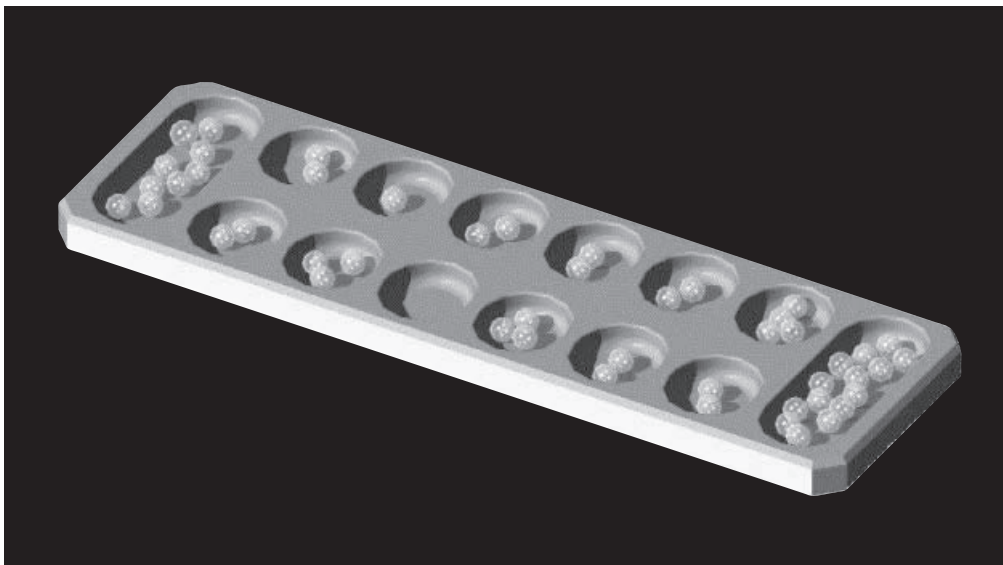
Advanced Projects
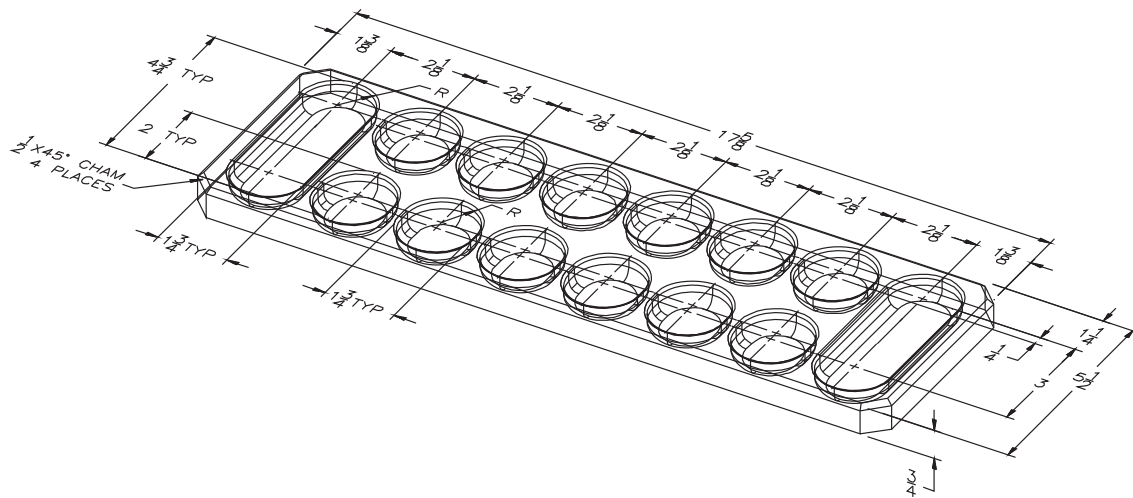
Fig. AV3-2

of the pockets on the board is to be symmetrical. Additional dimensions you will need are:

- Chamfer around the top edge: **.125**″
- Depth of the pockets: **.500**″
- Pocket top fillet radius: **.062**″
- Pocket bottom fillet radius: **.250**″

It is probably easiest to begin with the top view and then use it to help construct the front view. You should always begin a drawing by looking for symmetries because AutoCAD is very adept at replication. You can usually save a great deal of time (and boredom!) by taking advantage of commands such as **ARRAY**, **MIRROR**, **OFFSET**, and **COPY**. The *Mancala* board is particularly suitable for such commands—the entire top view of the pockets could be drawn by replication of just one line and one arc. Another thing you should examine when starting a drawing is the dimensions. Is there a lowest common multiple for at least most of the dimensions? If so, you should use this number for your initial snap and grid settings. What would be good initial snap and grid settings for this project?

The final preliminary considerations for this drawing should be the size of the drawing area and the ultimate size of the sheet on which the drawing will be plotted. Assuming a size A sheet (8.5 × 11), a good drawing limit size for

this board is 22 × 17. A final plot scale of **1 = 2** will then give you a nicely sized final plot. Your dimensioning style should then contain a general overall scale setting of **2**.

With these settings completed—and assuming that you have begun with a drawing template containing the appropriate layers, text style, linetypes, and dimension style—you could proceed to draw the top view as follows:

- Draw an arc and a line for the large pocket.
- Use **MIRROR** to finish the pocket.
- Use a copy of the arc, a shortened line, and **MIRROR** to create a small pocket.
- Make five copies of the small pocket using a rectangular array.
- Draw one-half of the outer edge of the board, using **FILLET** or snaps to draw the filleted corners.
- Offset these lines to obtain the filleted edges.
- Use **MIRROR** to obtain the complete top view.
- Construct the front view by drawing one pocket, complete with fillets, and replicating it using a rectangular array.
- Hatch the front view.
- Insert the center lines for all of the circular features, except fillets.
- Dimension the two views.

PART 2

Create another, very different *Mancala* board. Include features that you think might be useful to *Mancala* players. See the *Mancala* instructions on the next page for ideas. You may also consider adding the following features:

- handles
- legs
- edges for stacking multiple boards
- a lid
- one or more drawers to store the beans

Create orthographic and isometric views of the board you design.

PART 3

Create a solid model of the board from Part 1. Render the board using different materials such as wood, marble, and brass. Add beans, marbles, or other playing pieces (marbles are shown in Fig. AV3-1) and assign appropriate materials to them also. Be sure to create and position lights so that the board is visible and attractive when you render it. (Remember that you will need to render using either the **High** or **Presentation** render quality to display the materials correctly.)

Rules for *Mancala*

Place four beans in each of the twelve smaller pockets. Each of the two players has six of the small pockets and one of the large pockets. The first person to play picks up the four beans in any one of his or her six pockets. Moving counterclockwise, the player then places one bean in each pocket of the twelve and in his or her own large pocket, but not the opponent's.

If the last bean played falls into the player's large pocket, he or she gets another turn. If it falls into an empty pocket on the player's own side of the board, the bean captures all the beans in the opponent's pocket directly across from that pocket. The capturing bean plus all of the captured beans are placed into the player's large pocket.

The game ends when one player runs out of beans in his or her small pockets. When this happens, the other player places all the beans remaining in his or her small pockets into the large one. The player with the most beans in his or her large pocket wins.

Mancala Game Board project courtesy of Gary J. Hordemann, Gonzaga University

Advanced Project

BOLTED SEAT CONNECTION

When steel frame buildings are designed, a structural engineer prepares structural details, which are representations of the connections between beams. The bolted seat connection shown in Fig. AV4-1 is an example of such a connection.
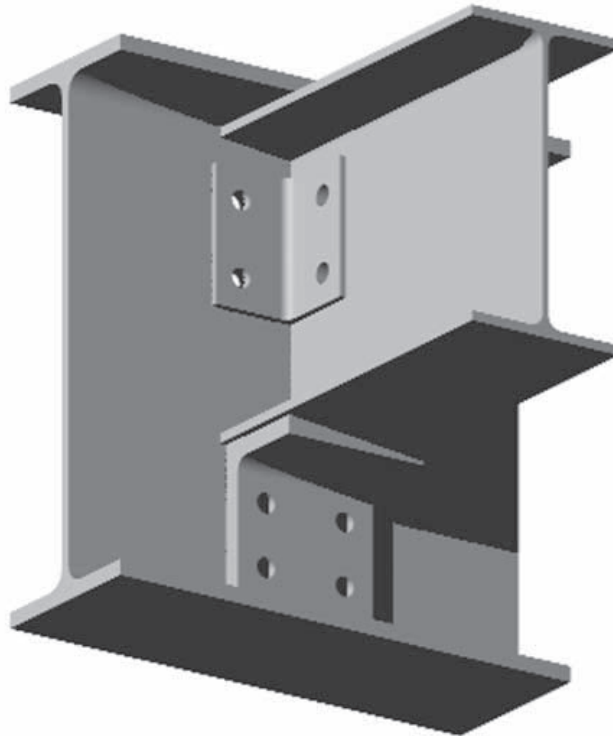
The specifications for structural connections are contained in the *Manual of Steel Construction* published by the American Institute of Steel Construction (AISC). This project uses AISC specifications and is adapted from an example in *Structural Steel Detailing,* also published by AISC.

Bolted seat connections are commonly used to connect grid filler beams to supporting girders (Fig. AV4-2 on page 752). The filler beam is supported by a seat angle, often merely sitting on the angle without bolts or welds. The other angle is used only for torsional stability.

Continued

Advanced Projects

GIRDER BEAM

FILLER BEAM

ANGLE

SEAT ANGLE
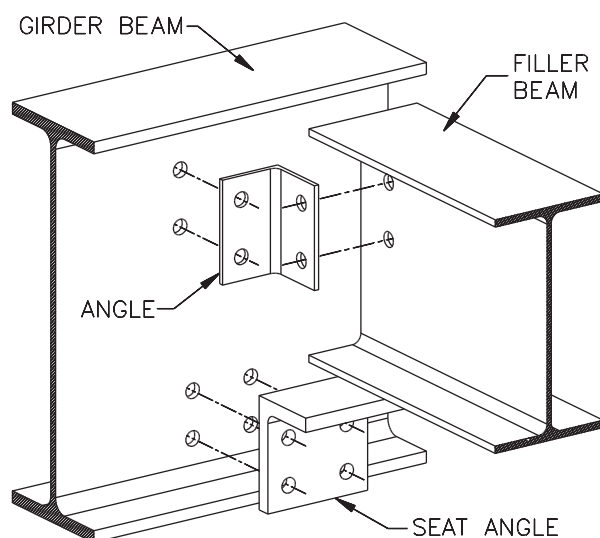
Fig. AV4-2

In its specifications for beams, AISC provides dimensions and physical property data according to beam shape codes. The beams shown here would usually be "W shape," and their identifying codes would be listed in that category.

For the connection in this project, the four members are:

- Girder: W21 × 62 wide flange beam
- Filler: W12 × 26 wide flange beam
- Seat Angle: 6 × 4 × $^3/4$ × 6
- Angle: $3^1/2$ × $3^1/2$ × $^1/4$ × $5^1/2$

PART 1

Create the front and right-side views of the beam connection. A partially dimensioned front view is shown in Fig. AV4-3. The remaining dimensions, extracted from the *Manual of Steel Construction,* are given in Fig. AV4-4 (page 754).
A few other facts you will need:

- Seat angle fillet radius: **.5**″
- Angle fillet radius: **.375**″
- Hole diameters: **.8125**″
- Clearance between the end of the filler beam and the girder: **.5**″
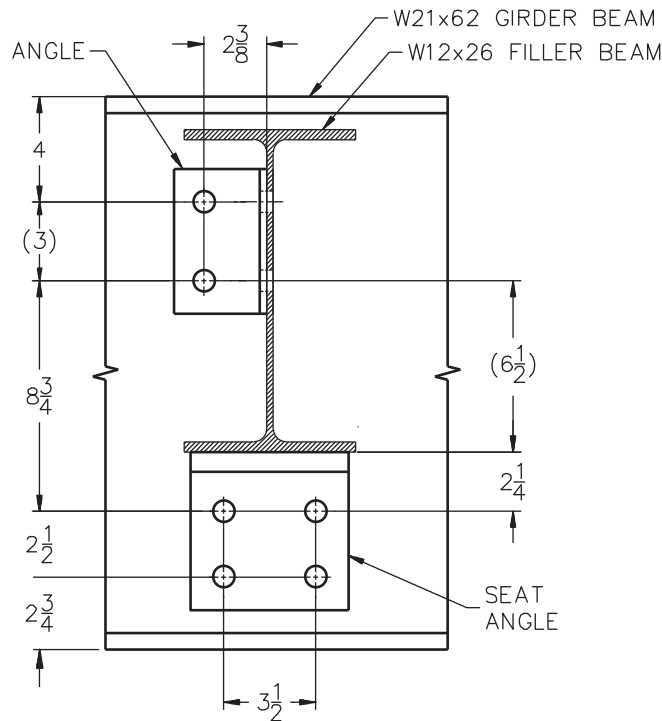
Fig. AV4-3

Structural details typically do not show hidden lines, so leave them out. Center lines should be shown.

This is the type of problem for which snaps and grids are of little use. Furthermore, trying to draw these objects directly, *i.e.*, by using **LINE** and **CIRCLE** with coordinates, would be foolhardy. This project is particularly suited to AutoCAD's construction commands **OFFSET**, **EXTEND**, and **TRIM**.

Other preliminary considerations of importance are the size of the drawing area and the size of the sheet on which the drawing will ultimately be plotted. Assuming a size A sheet (8.5 × 11), a good drawing area for this drawing is 44 × 34. A final plot scale of **1 = 4** will then give you a nicely sized final plot.

Assuming that you have begun with the proper layers, linetypes, and other routine settings, you could create this drawing as follows:

- Start with the front view.
- Draw a vertical line representing the center line of the front view.

Advanced Projects
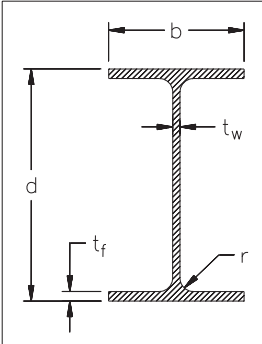
	W Shape Dimensions from AISC *Manual of Steel Construction*				
Designation	Beam Depth d (in)	Web Thickness t_w (in)	Flange Width b (in)	Flange Thickness t_f (in)	Fillet Radius r (in)
W 21 x 62	20.99	0.400	8.240	0.615	0.750
W 12 x 26	12.22	0.230	6.490	0.380	0.500

Fig. AV4-4

- Draw a horizontal line representing the top edge of the girder.
- Use **OFFSET** to create all the other lines, including center lines for the holes.
- To avoid getting confused by having too many offset construction lines, pause occasionally to trim the lines to their final lengths.
- Use the front view to construct the right-side view, again making extensive use of the **OFFSET** command.

PART 2

If you have access to a copy of the AISC *Manual of Steel Construction,* draw the two views of the bolted column and bracket shown in Problem 104 in the "Additional Problems" section of this book. Leave out the fasteners; they usually are not shown.

PART 3

Create solid models of the connection members from Part 1. Render the models using materials that make the members resemble steel.

Bolted Seat Connection project courtesy of Gary J. Hordemann, Gonzaga University

Advanced Project

BUSHING ASSEMBLY

The assembly shown in Fig. AV5-1 with nominal dimensions is composed of five parts: two bushing holders, two bushings, and a box.
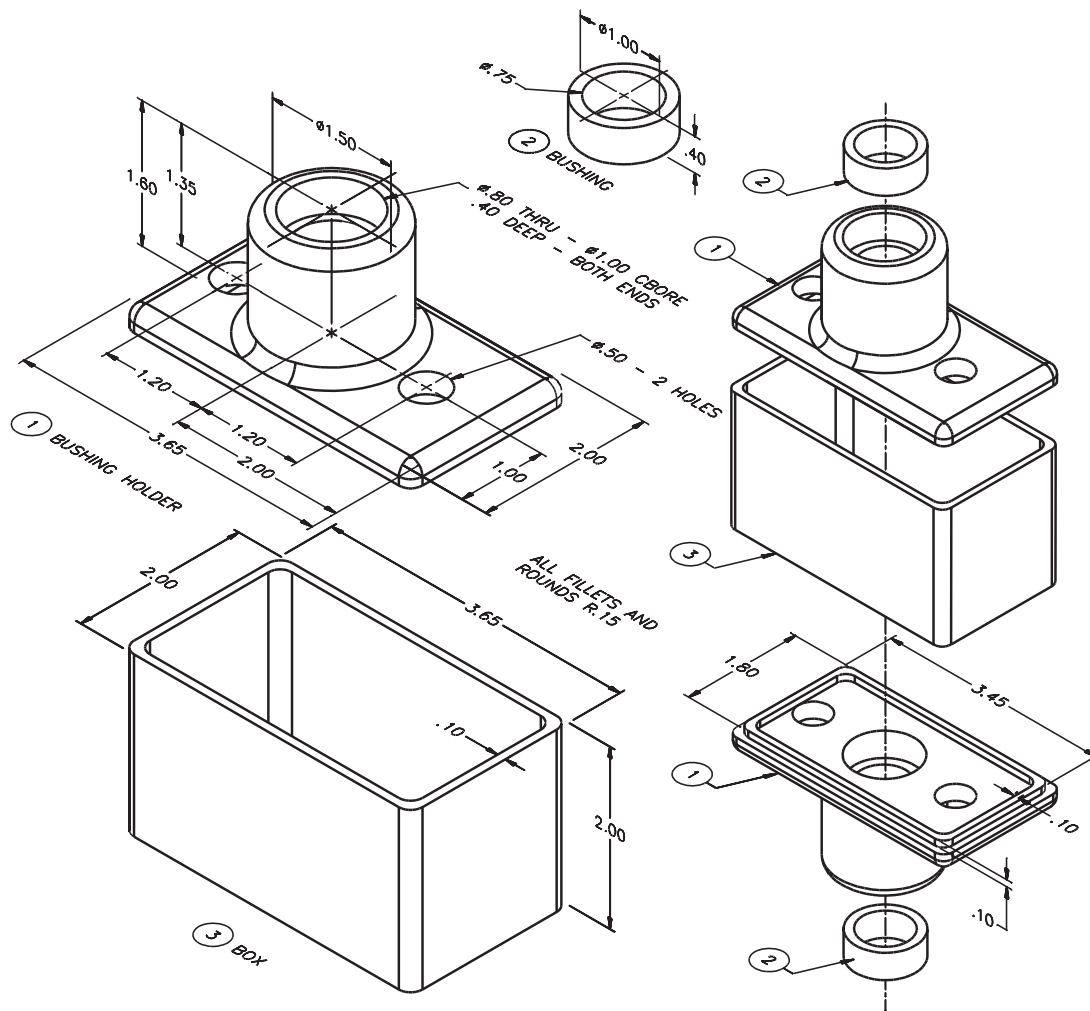
Fig. AV5-1

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PART 1

Create a full set of drawings to describe the assembly, as follows:

- Draw the necessary two-dimensional views of all the parts, inserting all hidden and center lines.
- Dimension the objects, including tolerances, as specified below.
- Draw the front view of the assembly with all of the parts put together. The front view should be drawn as a full section. Assume the holders, box, and bushings to be made of steel, cast iron, and bronze, respectively. Because hidden lines are usually not included in simple sections, leave them out. Include all center lines.
- Add a border and title block.
- Add notes and the properly toleranced dimensions to specify:
  - Overall tolerance of **±.020″** on all dimensions unless otherwise specified
 - Bushing outside diameter: **1.000″ – 1.0008″**
  - Holder inside diameter for bushing: **1.0014″ – 1.0019″**
  - Bushing inside diameter: **.7500″ – .7512″**
  - Perpendicularity tolerance of holder bushing center line to base: **.002″**

Assuming that some sort of shaft is to be supported by the bushings, what tolerances would you add to ensure that the two bushings align properly?

The three different objects are most easily drawn by beginning with their top views. You should first determine how many views of each object will be required to describe it completely.

Plan how you will arrange the views and the assembly on the sheet. Will you need more than one sheet? What plot scale or scales will you use? Although it would probably be overkill with so few parts, it often helps to draw rectangles representing the outside dimensions of the views and move them around on the drawing to get a preliminary idea of the layout. In planning the layout, be sure to allow plenty of room for dimensions.

When you have finished the dimensioning, ask yourself whether you could redraw it using only your dimensions; this will help you discover any missing dimensions. Since such redrawing usually goes very quickly, you might even take the time to redraw it. (It helps to let a day or so lapse between dimensioning and redrawing.) As a practical matter, you should end up with at least as many dimensions as on the original problem statement.

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These objects have "friendly" dimensions and therefore should be drawn using grids and snaps. Determine the lowest common multiple of the dimensions and use that for your snap setting. Use twice that for your grid setting. With all of your preliminary settings completed and the format of your drawings decided (and assuming that you have begun with a prototype drawing containing all of the appropriate layers, text style, linetypes, and dimension style), you may wish to proceed as follows:

• Draw the top view of the holder and use it to construct any other views you need.

• Draw the box and bushing views, starting with the top views.

• Use copies of the appropriate views of the three objects to construct the sectioned assembly.

• Dimension the orthographic views of the three pieces.

• Place a circle containing a part number (balloon) with each piece; use a circle four times larger than the number and insert the number using the **Middle** text option and the **Center** object snap.

• Use the **QLEADER** command to add "balloons" to the assembly.

PART 2

Create solid models of all the pieces in this assembly. Use the models to create an exploded assembly drawing similar to that shown on the right in Fig. AV5-1.

Bushing Assembly project courtesy of Gary J. Hordemann, Gonzaga University

Advanced Project

ARCHITECTURAL PLANS

You are a senior designer for an architectural firm. A client wants to see design proposals for a new home to be built on property she already owns. A copy of the plot plan is shown in Fig. AV6-1. The client has given you the following list of specifications:

- two-story residence
- 4 bedrooms, 2 baths
- walk-in closets in all bedrooms
- brick construction or brick facing
- attached 2-car garage
- noise level as low as possible in the bedroom area(s)
- total square footage not to exceed 2800 square feet

PART 1

Design a residence that meets these specifications. Assume a utility easement of 10′ on the north and east sides of the lot. Then create a complete set of plans, including (but not necessarily limited to) a site plan, floor plans for each floor, elevations, and a landscaping plan. Include all the information the client needs to make a decision about whether to accept this design. Note that the dimensions shown in the plot plan have been rounded to the nearest foot.

PART 2

Create a 3D model of the home you have designed. Include interior features such as kitchen and bathroom cabinets and standard appliances such as refrigerator, range, dishwasher, and water heater. Prepare a demonstration for the client that includes walking or flying through the model.

Advanced Projects

193'

145'

95°

←N

250'

102°

90°

160'

175'
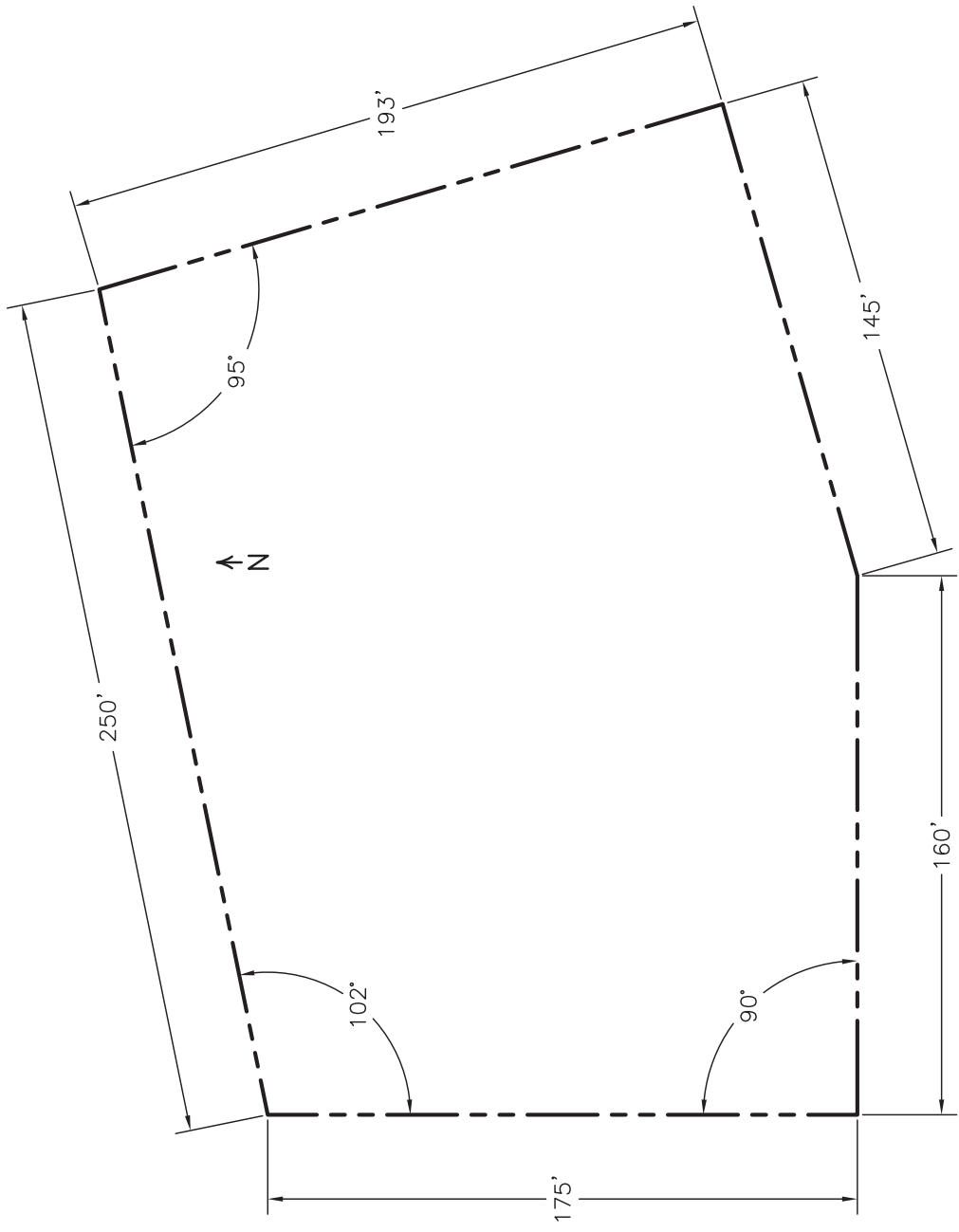
Fig. AV6-1

Advanced Projects

Advanced Project

BUFFET TRAY

Two examples of a lap tray for use at a buffet or picnic are shown in Fig. AV7-1. Note that each example has handles and other adaptations to make it easier for the user to control various dishes and implements.

Fig. AV7-1

PART 1

Design your own buffet/picnic tray. In doing so, nearly everything is left up to you. The only constraints are:

- The tray is to contain depressions and/or holders for items such as drink containers, plates, silverware, etc.
- The tray is to have handles that are separate pieces; they may or may not be removable or rotatable.

Document your creation by drawing completely dimensioned orthographic views. The thinness of the object, plus the need for dimensions on the

Advanced Projects

depressions and their fillets, make section views appropriate. The nature of the tray also suggests the use of offset sections. Thus, you should:

- Draw complete orthographic views of the tray body, the handles, and any other objects that make up your tray.
- Use section views as needed.
- Dimension the objects.
- In preparing the orthographic views, show each piece individually. Identify each piece with a name and an associated part number inside a circle (balloon).
- Draw a section view of the assembly with the connection between the tray and handle magnified.

An example set of dimensioned drawings for the tray shown in the top of Fig. AV7-1 is given in Fig. AV7-2 (page 762). Use this example as a guide in preparing the drawings.

After you have thought out the design (a pencil sketch may help), prepare to draw it by first considering its dimensions. Is there a common multiple of the dimensions that will make it convenient to use grid and snap? What will be the ultimate size of the sheet on which the drawing will be plotted? Will you need more than one sheet? Set the drawing area and dimension and text scales accordingly.

PART 2

Prepare solid models of the pieces of the tray you designed or the tray shown in the sample design.

After creating the solids, perform the following tasks:

- Display the solids as an exploded assembly displayed in the **Conceptual** visual style and the **Southeast Isometric** view.
- Determine the volumes and surface areas of the tray and handles.
- Select your materials and look up the density of each. You may have to go to the reference section of a suitable library or search the Internet. Use the density to determine the weights of the parts.

The modeling should probably be started in the top view, and **PRESSPULL** will likely be the "workhorse" command. Tubular handles like those shown in the bottom tray in Fig. AV7-1 can be created by extruding a circle along a polyline.

Advanced Projects

Try rendering the model with different materials, such as wood or plastic. Be sure to consider the weight of the tray built with each material or combination of materials. Which materials will be the most practical to use for the tray you designed?
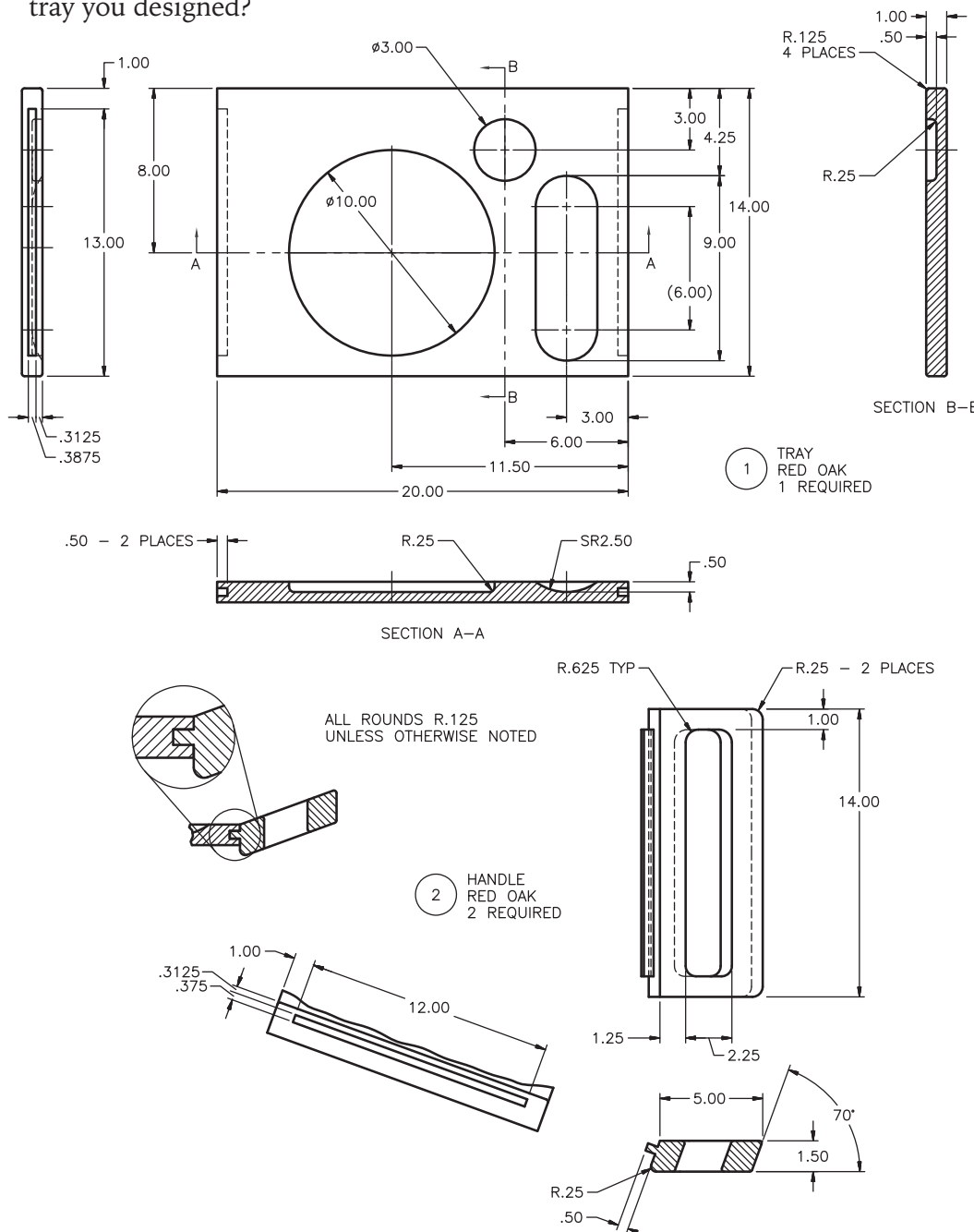

Fig. AV7-2

Buffet Tray project courtesy of Gary J. Hordemann, Gonzaga University

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Shackle Assembly

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805

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SHACKLE ASSEMBLY

The shackle assembly shown here is composed of seven parts. Five of the parts (the chain, two yokes, and two clevis pins that secure the chain to the yokes) are shown in Fig. AV8-1. Not shown are two cotter pins that, when inserted into the holes in the pins, keep the pins from working loose.

The dimensions of the yoke and clevis pin are given in the drawing in Fig. AV8-2 (page 764). The clevis pin dimensions are those specified for a nominal $1/2''$ standard clevis pin by the American National Standards Institute (ANSI) and published as ANSI B18.8.1-1972, R1983 by the American Society of Mechanical Engineers (ASME).

Each link of the chain has a diameter of **.375**″ and outside measurements of **2.125**″ × **1.375**″ Using these dimensions, perform the following tasks:

- Create solid models of the five components shown and arrange them into an exploded assembly similar to the one shown in Fig. AV8-1.
- Determine the volumes of all pieces, including the chain.
- Assuming the pieces to be steel, with a nominal density of .28 lb/cu in., determine their weights.
- Determine the surface area of a yoke.
- Put the solids together and slice the assembly with a cutting plane oriented along the axes of the pins; remove half so the inside of the assembly can be seen.
- Repeat the slice and remove operations with the cutting plane perpendicular to the axes of the pins.

Fig. AV8-1

Continued

Advanced Projects

1.50

.50

.50

1.00

1.75

ø.25 THRU
ø.75 CBORE
.25 DEEP
BOTH ENDS

ø.50 THRU
2 HOLES

ALL ROUNDS R.10

.25

.50

1.00

1.3800
1.3600

.1600
.1400

1.2230
1.2030

R .0150
.0050

ø .6300
.6100

ø .4400
.4300

ø .4960
.4910

ø .1510
.1360

.0890
.0630

.0500
.0300 X 45° CHAM
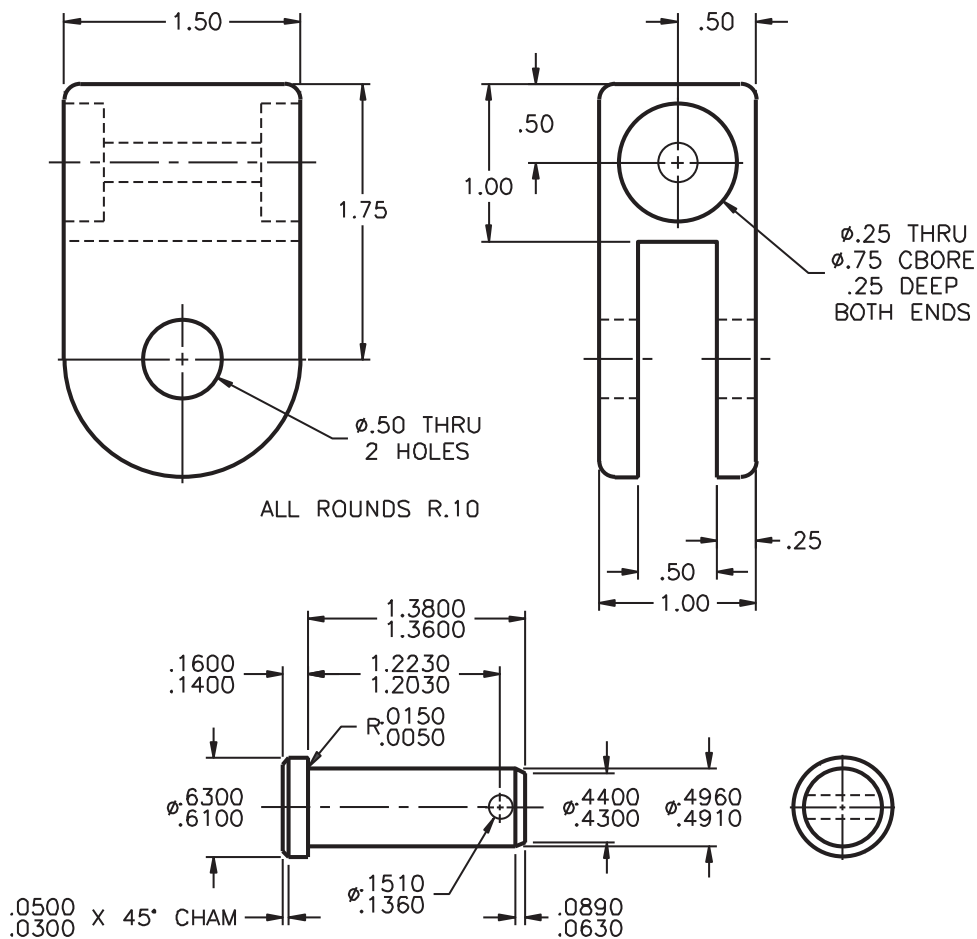
Fig. AV8-2

When creating the solids, it is prudent first to choose the starting view. You should select a view such that the world coordinate system standard orthographic views of top, front, right-side, etc., correspond to those same views of the object. This will minimize the confusion over what you are looking at when you switch views. Assume that this assembly is meant to hang vertically, so it has a natural top view. Make the view facing the heads of the clevis pins the front view. The dimensioned drawing of the yoke shown in Fig. AV8-2 shows its front and right-side views.

After settling on the orientation of the solid, take a little time to consider how you are going to create it. There are usually several ways to build a solid, and some may be very difficult and time-consuming.

Advanced Projects

One way to create the yoke is to follow this procedure:

- Draw the front view profile, making it a closed polyline.
- Extrude the polyline to the full depth of **1**″.
- Insert and subtract a cylinder.
- In the right-side view, insert and subtract cylinders to create the counter-bored hole.
- Insert and subtract a rectangle or use the **PRESSPULL** command to create the cavity between the arms.
- Fillet the outside edges.

Some variations on this procedure include:

- Extrude the profile to a depth of **.25**″ to create one arm; then use a duplicate for the other arm, joining the two arms with a box.
- Make the counterbored hole by drawing and revolving its profile in the front view.
- Draw and extrude the yoke profile in the right-side view. Then use **FILLET** to obtain the geometry common to both extrusions. (This approach is perhaps the most elegant.)
- Create the space between the arms by making three slices and unions. (This approach, however, is not recommended.)
- You could even create the yoke without using **EXTRUDE**, **PRESSPULL**, or **REVOLVE** by unioning and subtracting a series of cylinders and boxes. This approach, too, is impractical in reality.

Clearly, there are several practical ways to create this assembly, as well as several very impractical ways.

You should also think through the sequence of operations. The most common error resulting from an incorrect sequence is a hole that does not go all the way through the object.

Finally, always start a drawing at a known and easy-to-remember point such as **0,0,0** or **2,2,2**. This is particularly critical when you are working in 3D space.

If you wish to proceed with "elegance," create the yoke as follows:

- Draw the profile of the front view of the yoke and extrude it to **1**″.
- Insert a cylinder and subtract it.

*Continued*

Advanced Projects

- In the front view, draw the half-profile of the counterbored hole and revolve it.
- In the right-side view, draw and extrude the profile, without the holes.
- Fillet the edges of both objects.
- After making sure the two extrusions are coincident, obtain the final object using **INTERSECT**.

You may create the clevis pins from unioned cylinders or from a revolved profile. The chain link can be made from a circle extruded around a polyline (consider using multiple viewports) or from a sliced torus unioned with two cylinders.

You can model the two cotter pins by extruding a half-circle along a polyline. Select the appropriate pin from a table of ANSI standard cotter pins. Often, you can find a table such as this in the appendix of a graphics or drafting textbook.

Shackle Assembly project courtesy of Gary J. Hordemann, Gonzaga University

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Clothing Rack

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809

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CLOTHING RACK

Shown in Fig. AV9-1 is a wall-mounted rack for drying clothing. It is composed of six rods inserted into a large cylinder, which is housed in a square tube. The tube has cutouts that allow the rack to be folded downward when the rods and cylinder assembly are slid to the right. In this view, we are looking upward from below the object.
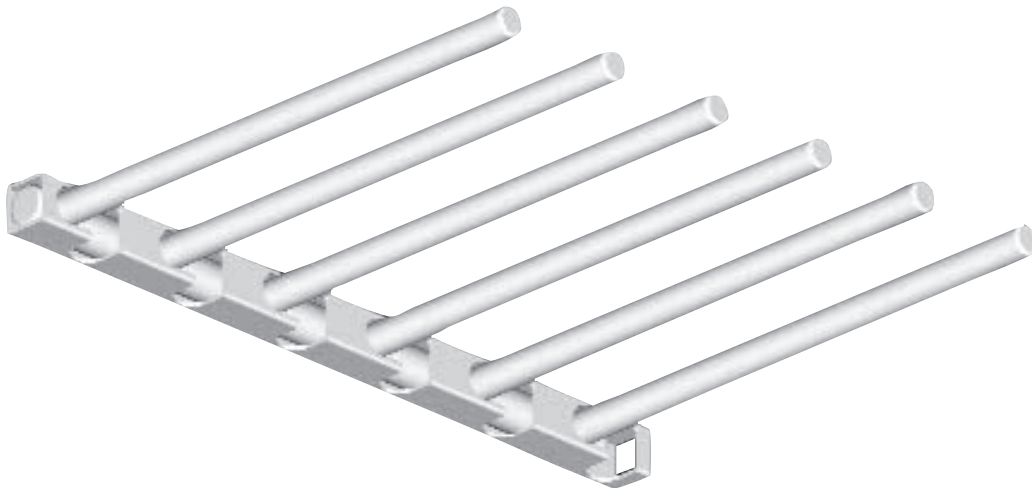
Fig. AV9-1

PART 1

The dimensions of the components are shown in Fig. AV9-2 on page 768. Using these dimensions, perform the following tasks:

• Create solid models of the eight pieces and arrange them into an exploded assembly.

• Determine the volumes and surface areas of all eight pieces.

• Assuming the tube to be aluminum with a density of 160 lb/cu ft, determine its weight.

• Assuming the rod assembly to be oak with a density of 45 lb/cu ft, determine its weight.
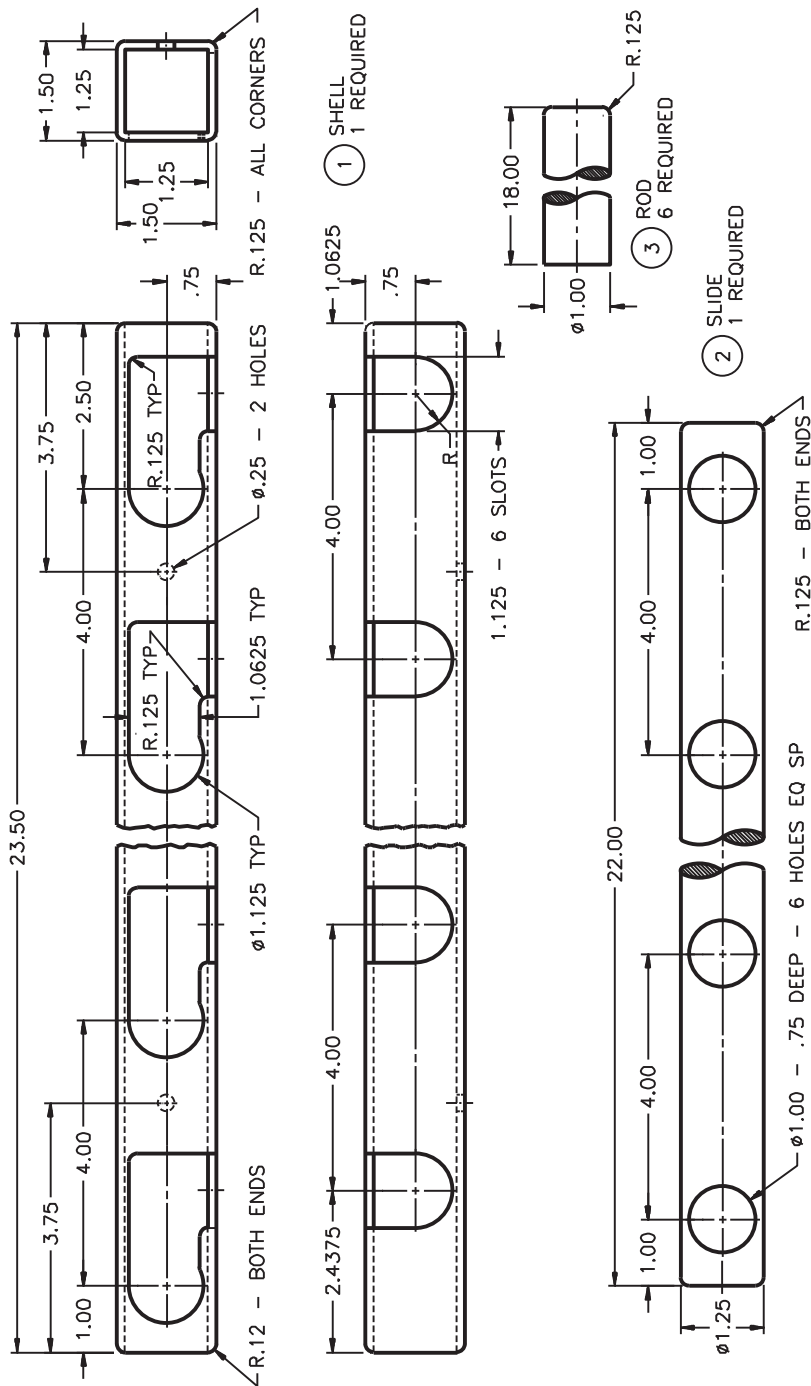
Continued

Advanced Projects

Fig. AV9-2

Advanced Projects

When creating three-dimensional objects, you will find it least confusing to select the proper view before you begin. Try to align the object with the world coordinate system standard orthographic views of top, front, right-side, etc. The standard views will then make sense when you switch from view to view. Many objects—such as this one—have a natural top and front, and you should plan to orient the object accordingly. Following this orientation, the dimensioned drawing in Fig. AV9-2 shows the front, bottom, and right-side views.

Before starting the model, consider the different ways to carry out this project. Always look for patterns and symmetries so that you can take advantage of AutoCAD's replication commands. This object is ideal for use of the **ARRAY** command. The modeling of this assembly is straightforward, being primarily a collection of cylinders. The square tube at first glance appears complex, but if you realize that cavities are usually best modeled by creating the positive of the cavity and then subtracting it, the creation of this object becomes much easier.

After you have laid out your approach to the modeling, begin the drawing at some known and easy-to-remember point, such as **0,0,0** or **1,1,1**. The least difficult approach to this model is probably to begin with the rods. Doing so, you could proceed as follows:

- Switch to the front view and insert a cylinder. **Dynamic UCS** should be on.
- Fillet one end of the cylinder.
- Use **ARRAY** to make five copies on **4″** centers.
- In the right-side view, begin the slide by inserting a cylinder; fillet both ends.
- In the front view, insert and array a set of small cylinders for the six holes.
- Subtract the small cylinders from the slide.
- Begin the shell in the right-side view, inserting two squares; then fillet the outer one.
- Extrude and subtract the squares.
- In the top view, create the profile of the cavity, joining all of the pieces with the **Join** option of the **PEDIT** command.
- Extrude the profile to at least the thickness of the shell.
- Switch to the front view and repeat the process for the profile of the cavity in that view.
- Array and subtract the two profiles.

Continued

Advanced Projects

- Create the two mounting holes in the back of the shell.
- Arrange the objects in an exploded assembly.

PART 2

Using AutoCAD's isometric mode, create an isometric exploded assembly of the rack. Using a leader, tag each part with a balloon. A balloon is a circle containing the part number of the piece. The balloon diameter should be four times the height of the numeral.

Drawing the shell in isometric mode will be a bit of a challenge. Sometimes you can create the isometric view more easily by first drawing an orthographic view such as the front view. Then switch to one of the four standard isometric views to complete the drawing. When switching views, you will want to stay in the world coordinate system. Because different parts of the drawing may end up on different planes, you will discover that **TRIM** and **EXTEND** may not work properly. However, **TRIM** can usually be made to work if you change the coordinate system so that the current view is an XY plane. Once created, the isometric may be transferred into any other view by cutting and pasting. (Do not forget always to change the UCS to the current view before cutting and again before pasting.)

PART 3

The rack as specified in Part 1 has a few shortcomings. First, the mounting holes are not located properly—they should be in line with the cavities so the installer can use a tool on whatever fastener is in the hole. Second, if the mounting fasteners have hex or round heads, the heads will interfere with the movement of the rods/slide assembly. The fasteners should have flat heads, but then the holes should be countersunk. Third, the rectangular tubing might be too expensive.

Address these problems by creating a new design. Use a round tube for the shell. Make the rods and slideout of $3/4''$ PVC pipe. Use $3/4''$ PVC pipe elbows and Ts to connect the pipe pieces, and $3/4''$ PVC pipe caps on the ends of the rods. A trip to the local hardware store will provide you with the outside dimensions of these pipe fittings.

Using the drawings on page 768 as a guide, create a complete set of dimensioned drawings of your design. Remember, when you are drawing orthographic views, draw only as many views as necessary to specify the object completely, and only draw one copy of each part. Tag each part with a balloon specifying the part number, material, and how many are required in the full assembly.

In a drawing with so many parts, you should first plan how you will arrange the parts on the paper. This will help you figure out how many sheets to use and what the plot scales will be. Be sure to allow sufficient room for the dimensions.

PART 4

Using AutoCAD's isometric mode, create an isometric exploded assembly of the rack you designed in Part 3. Using leaders, tag each part with a balloon. Create the isometric drawing as a separate drawing file. Then insert the isometric into the drawing from Part 3.

PART 5

Make solid models of the pieces of your design. Then perform the following tasks:

- Arrange the solids in an exploded assembly and display them in a view in which you are looking upward at the assembly from below.
- Determine the volumes and surface areas of all the pieces.
- Select a material for the shell and look up its density and that of PVC; use the densities to determine the weight of the shell and of the rods/slide assembly.
- Render the rack using suitable materials.

Clothing Rack project courtesy of Gary J. Hordemann, Gonzaga University

Advanced Projects

Advanced Project

PROTECTIVE PACKAGING

Designers for packaging companies typically design custom packaging to protect various products. The packaging company's clients depend on the expertise of these designers to provide safe, effective shipping containers and other packaging. In this project, you will develop protective packaging for a client's new product, shown in Fig. AV10-1.

The device is made of a titanium alloy to reduce weight. The client has specified that the packaging must:

• protect the device from sudden impacts that may result from being dropped or shifting in an airplane cargo hold.

• provide an attractive appearance that will be appealing to the client's potential customers.

Design custom packaging for this instrument and create drawings to show how the packaging will protect it. Create assembly drawings as necessary to show the client how to assemble the packaging and how to pack the instrument properly.
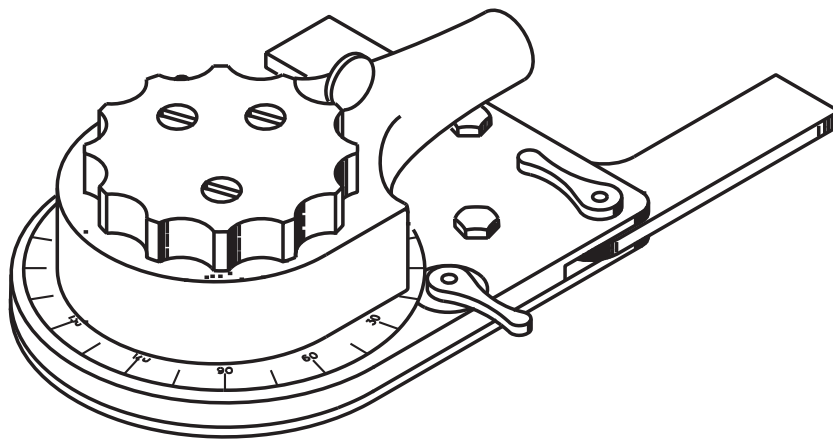

Fig. AV10-1