Chapter
7

Computer-Aided Drafting and Design

Objectives

After studying this chapter, you should be able to:

- Explain how computer technology is revolutionizing drafting, design, and engineering.
- Describe the basic features and operation of a computer-aided drafting program.
- Explain the various commands used to create objects in CAD.
- Describe the tools used to modify CAD drawings.
- Identify the various display functions used in CAD programs.
- Describe the typical components in a CAD program help system.
- Explain the importance of CAD file management and identify common storage techniques.
- List different types of CAD software and their applications.
- Describe the common tools and processes used in parametric modeling.
- Describe the basic features of advanced CAD systems.

Drafting Vocabulary

absolute coordinates
array
assembly
attributes
base feature
base view
bill of materials
block
Cartesian coordinate system
chamfer
computer-aided design/ computer-aided manufacturing (CAD/CAM)
coordinates
degrees of freedom (DOF)
direct distance entry
feature-based modeling
fillet
geometric constraint
grid
hatch pattern
layers
linetype

model dimensions
object snap
orthogonal mode
parameters
parametric modeling
part
pixels
polar coordinates
raster objects
reference dimensions
reference plane
relative coordinates
rendering
resolution
round
scanner
scene
schedule
sketch
snap
solid modeling
subassembly
surface modeling
symbol library
template
user coordinate system
vector objects

Before You Begin

Read the chapter title and tell a classmate what you have experienced or already know about the topic. Write a paragraph describing what you would like to learn about the topic. After reading the chapter, share two things you have learned with the classmate.

Employability

Being an Efficient Employee

Employers want employees who are punctual, dependable, and responsible. They want their employees to be capable of taking initiative and working independently. Other desirable employee qualities include organization, accuracy, and efficiency.

No matter what your job is or how long you have had it, you can always improve your efficiency. Efficiency is a measure of how much you can do—and how well you can do it—with the resources you have. Like the world in general, the workplace is constantly changing. Look critically at the work you do, whether it is schoolwork or work at a part-time or full-time job. Examine the methods you use to get your work done. Methods that have been useful in the past can sometimes be replaced with newer, more efficient methods made possible by new technology.

General tips that may help you improve your efficiency include:

- At the beginning of each day, think about what you need to accomplish that day and set your priorities accordingly.
- Consider your energy levels during the day and match your energy level to your tasks. For example, if you think better in the morning, do tasks that require the most thought or brainpower in the morning.
- Organize your work environment. Identify items you need on a daily basis and keep them within easy reach. Not having to spend time searching for these items not only saves time, but also helps you retain your focus on the task at hand.

Activity

Perform a self-analysis using the tips provided here. How well are you doing? In what ways can you improve your efficiency? Set yourself three written goals that you believe will increase your efficiency within the next month. At the end of the month, review your list. Did you achieve your goals?

Computer graphics continues to revolutionize drafting and engineering, **Figure 7-1**. Computer-aided drafting and design systems play an integral part in the entire engineering process, from design and drafting to analysis and presentation. They aid in designing mechanical products, buildings, and many other types of manufactured and constructed items.

There are many benefits to using CAD as a design and drafting tool. Traditional drafting tasks are greatly simplified with the electronic tools of a CAD system. CAD programs provide setup tools, drawing and editing commands, and customization methods to maximize accuracy and proficiency. There is a wide range of programs to choose from depending on the nature of the work. This chapter introduces the various applications of CAD drafting and discusses the common tools used to create CAD drawings.

Overview of Computer Graphics and CAD

Computer graphics was first employed for aerospace design in the 1950s. It is now a required tool of industrial technology. Until the advent of CAD, engineers, designers, and drafters had to imagine and then evaluate a three-dimensional object that was drawn in two dimensions on a flat sheet of paper.

The only way a design could be verified in three dimensions was to make a wood, clay, plaster, or plastic model. This is an expensive and time-consuming process.

The emergence of CAD provided designers with a dynamic new tool. CAD technology permits more time for creative work because it eliminates the repetitive tasks required in traditional drafting.

CAD is a computer graphics technology that allows users to generate and display designs electronically, **Figure 7-2**. Many people are able to use a computer drawing program after a suitable training period. However, the principles of drafting are fundamental to both traditional drafting and CAD. A working knowledge of basic drafting standards, techniques, and procedures is absolutely necessary to be an effective CAD drafter.

The main function of a CAD designer or engineer is to define the basic shape of a part, assembly, or product in 2D or 3D form. The process involves many changes and refinements before a design is finalized. With a CAD system, design changes can be made and evaluated quickly.

Computer graphics programs fall into one of two classifications. These classifications are based on how the actual images are created.

AlexLMX/Shutterstock.com

Figure 7-1. Computer-aided drafting systems are used for a wide variety of applications in design and engineering. With the proper CAD program, it is possible to generate realistic three-dimensional models representing complex mechanical products.

Inga Ivanova/Shutterstock.com

Figure 7-2. CAD drafters use tools in the software to create and display designs electronically.

Images in a computer graphics program may be created with vector objects or raster objects. Drawings created in a CAD program are made up of vector objects. *Vector objects* are made up of lines (vectors) and arcs and are defined with point coordinates in space. See **Figure 7-3A**.

Raster objects are defined using tiny shapes of data called picture elements, or pixels. See **Figure 7-3B**. *Pixels* are arranged in a fixed, precise manner. Each pixel is the same size and shape. The number of pixels making up an image defines the *resolution*, or visual quality, of the image. Raster objects are also known as bitmap graphics. Image editing programs are commonly used to alter bitmap graphics. Televisions and computer monitors are examples of devices that use raster displays.

A vector-based drawing can be converted to bitmap form in several different ways. The most basic way is to export a drawing file from a CAD program as a bitmap file. The file can then be imported into a different program for editing. For example, a 3D model created with a CAD program may be converted to a bitmap file and edited to add special effects, such as lighting and shadows. If a vector drawing is output as hard copy, it can be converted to bitmap form with a *scanner*. A scanner is an automatic digitizing device. It analyzes the lines, circles, and other graphic elements of the drawing and converts the objects into computer data.

Referring to **Figure 7-3**, vector images are edited by modifying the individual lines and arcs making up the drawing. Raster images are modified by editing the individual pixels making up the image. In creating or editing a vector object, such as a line, it can be helpful to visualize the object as an entity defined with coordinates. Most CAD programs provide a drawing grid that can be used to define objects with coordinates. Objects can be drawn to the desired size by defining coordinates or other parameters or by scaling an original object to a different size. Coordinate entry, scaling, and other types of drawing functions in CAD programs are discussed in this chapter.

How CAD Works

A CAD drawing project starts with the generation of a geometric model of the proposed design. The base outline or profile of the design is created first and then features are added. CAD models are created in 2D or 3D form, depending on the type of program being used and the purpose of the project.
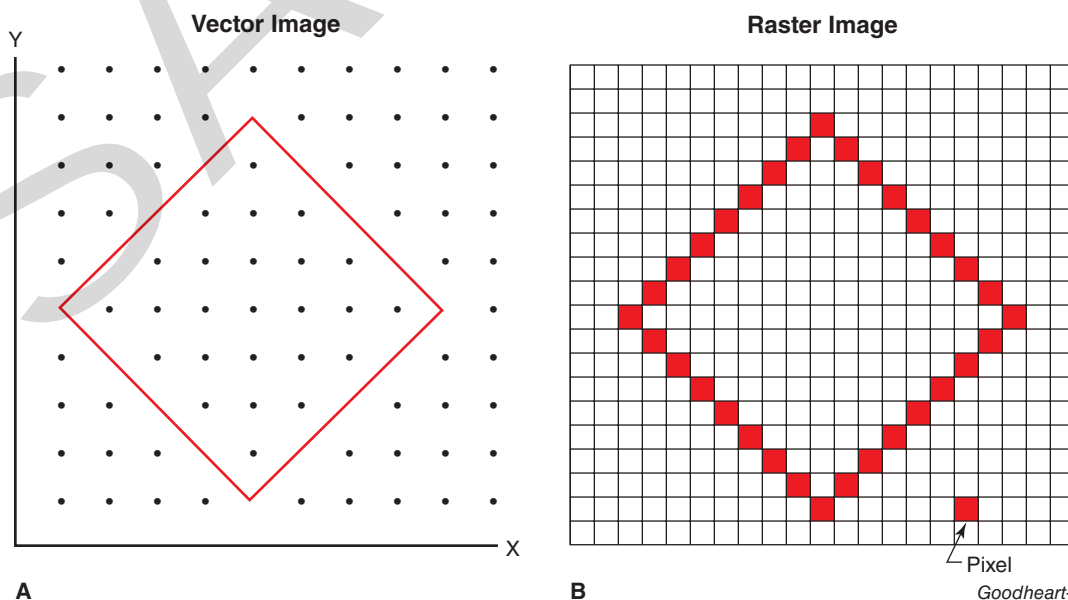

Figure 7-3. A comparison of vector and raster displays.

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Instructions are given to a CAD program through the use of commands. Commands can be picked from "ribbon" or "command manager" tabs, selected from pull-down menus, or entered at the keyboard. See **Figure 7-4**. Toolbars and palettes may also be available for selecting commands. A typical CAD software interface normally provides a number of ways to execute commands and functions.

Once objects are drawn, they can be altered as needed. For example, an object can be moved, rotated, copied, deleted, or mirrored. An object drawn in CAD can be manipulated in a variety of ways, making it unnecessary to draw the object again.

Dimensions and text can be added to a CAD drawing by selecting the appropriate command or function and entering the required information. The drafter can define the text size, style, and orientation as needed. CAD functions for creating text and dimensions are discussed later in this text.

After a CAD drawing is completed, it can be output in a number of ways. CAD drawings are typically printed on a plotter, **Figure 7-5**. Inkjet plotters are most commonly used to produce CAD drawings. CAD drawings may also be output to a printer, such as a laser printer.

CAD drawings may also be converted to an appropriate file format for electronic viewing in another program. For example, a drawing may be posted by a firm on a website for clients or for another engineer working on the project. If the file is sent via e-mail to a client, the client can typically download the viewer software needed to display the file.

Regardless of the drawing and production methods required, CAD programs offer a variety of ways to manage projects from design to completion. The following sections discuss the basic features, tools, and commands used to generate CAD drawings.
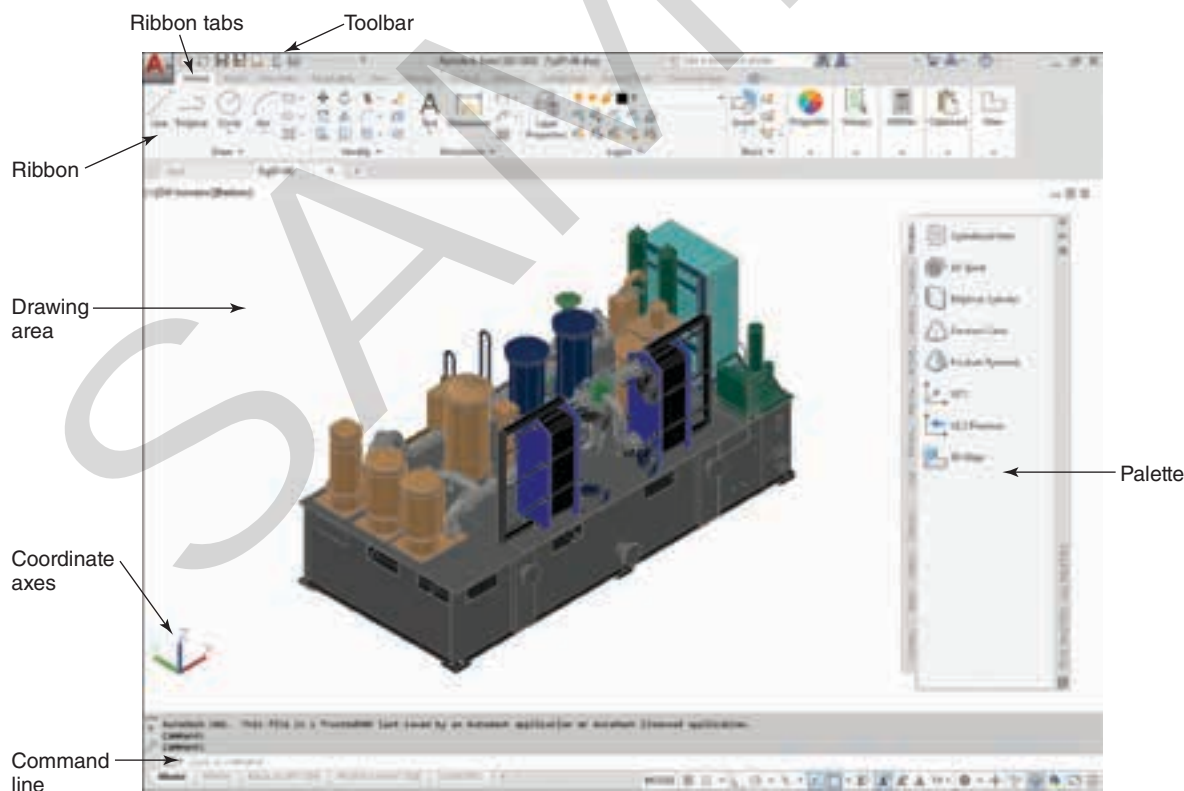
Figure 7-4. A typical CAD program drawing display. Commands can be accessed in a variety of ways to suit the user's needs.

Hewlett-Packard Company

Figure 7-5. Plotters produce hard copy from CAD drawing files.

File Management Functions

As is the case with most computer software programs, CAD programs include file management commands for common tasks. Commands such as **New**, **Open**, **Save**, and **Plot** provide basic functions for creating, opening, saving, and plotting files. Other commands are available for similar file management tasks.

The **New** command allows the user to create a new file, such as a drawing (DWG) file. The **Open** command is used to open an existing file. The **Close** command is used to close the currently active file. The **Save** command allows the user to save the latest changes in the file from RAM storage to a more permanent location, such as the hard drive or a portable media device. When first saving a file with the **Save** command, the user is prompted for a file name and location.

The **Save as** command is similar to the **Save** command, but it allows the user to save the current file under a different file name. In effect, this saves a "copy" of the file and preserves the original file. The **Save as** command can also be used to save a file as a different file type, such as a drawing standards (DWS), drawing template (DWT), or Drawing Interchange Format (DXF) file.

The **Export** command allows the user to "export" the file to additional file formats. This command is often used when it is necessary to "share" data in the drawing file with users of other programs. The **Export** command can be used to create a file in the Windows Metafile (WMF), bitmap (BMP), or stereolithography (STL) file format. Additional commands similar to the **Export** command are available for generating other types of files, such as Portable Document Format (PDF) files.

The **Page setup**, **Plot**, and **Publish** commands are used for common plotting and drawing distribution tasks. The **Page setup** command is used to specify a plotting device and page layout settings prior to plotting. The **Plot** command is used to configure device settings, specify the desired media, and plot the drawing. The **Publish** command is used for advanced plotting tasks, such as creating and "publishing" multiple-sheet drawing sets. Publishing drawing sets is common in architectural building projects, where multiple sheets representing different plan drawings can be organized into a single document. Documents of this type can then be plotted or "published" to hard copy. Another option is to "publish" the document to a file, such as a PDF file or a Design Web Format (DWF) file, so that it can be viewed outside the native program. DWF files are compressed versions of drawing files that can be distributed to other users or posted on the internet. DWF files can be opened and reviewed using special viewer software that provides viewing and markup tools.
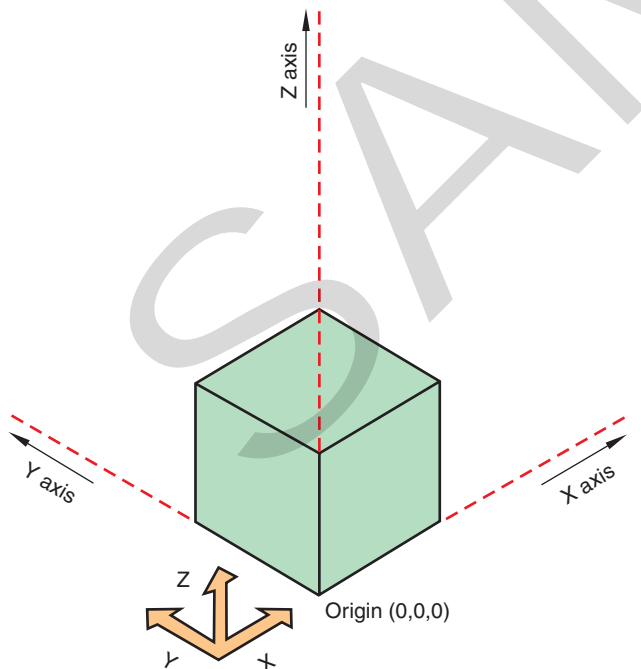
Basic CAD System Functions

One of the main benefits of CAD drafting is the added productivity and efficiency made possible by tools in the software. Many of these tools are designed to automate the common tasks associated with traditional (manual) drafting. There are a number of basic features and functions common to most types of CAD software. These include coordinate systems, drawing aids, layers, linetypes, and blocks. These features are discussed in the following sections.

Coordinate Systems

As previously discussed, objects in a CAD drawing are defined with coordinates. *Coordinates* are points representing units of real measurement from a fixed point. Most CAD programs provide a basic coordinate system and the ability to create user-defined coordinate systems. The most basic coordinate system in a typical program is the world coordinate system. Coordinates are located in relation to the 0,0,0 origin. See **Figure 7-6**. This system is based on the *Cartesian coordinate system*. In this system, objects are defined by coordinates using the X axis (horizontal axis), Y axis (vertical axis), and Z axis (the axis projecting perpendicular from the XY plane).

The horizontal and vertical axes of the Cartesian coordinate system divide the XY drawing plane into four quadrants. See **Figure 7-7**. Coordinates are entered as positive or negative, depending on the location from the origin. Referring to **Figure 7-7**, the coordinate (2,2) is located in the upper-right quadrant. This quadrant has positive X coordinates and positive Y coordinates. The coordinate (4,–3) is located in the lower-right quadrant. This quadrant has positive X coordinates and negative Y coordinates. The coordinate (–6,8) is located in the upper-left quadrant. This quadrant has negative X coordinates and positive Y coordinates. The coordinate (–3,–5) is located in the lower-left quadrant. This quadrant has negative X coordinates and negative Y coordinates.

Objects drawn with XY coordinates are sufficient for 2D drawings. A third coordinate axis, the Z axis, is required for 3D drawings. See **Figure 7-8**. This axis is used for coordinate entry above or below the XY plane. A point with a positive Z coordinate, such as (0,0,1),
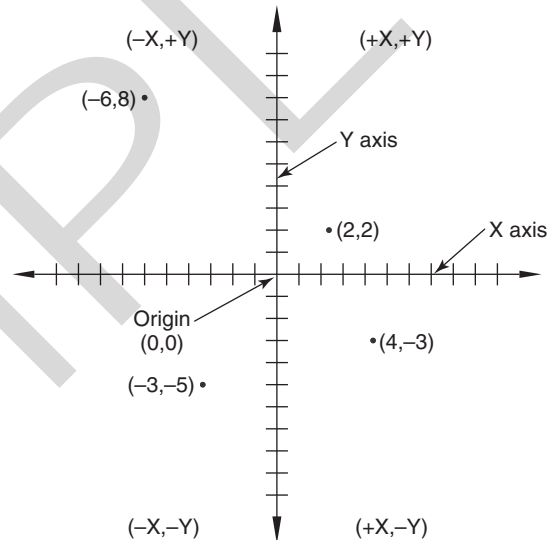
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Figure 7-7. The XY axes of the Cartesian coordinate system divide the drawing plane into four quadrants. Coordinates have positive or negative X and Y values.

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Figure 7-6. In the Cartesian coordinate system, coordinates establish points of measurement along the X, Y, and Z axes in relation to the 0,0,0 origin.

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Figure 7-8. A third axis (the Z axis) is needed to generate three-dimensional drawings like the one shown. The ability to visualize objects in 3D space is fundamental to creating 3D drawings.

is located "above" the XY drawing plane. When looking at a drawing on screen, this location can be thought of as a point projecting out of the monitor toward you. A point with a negative Z coordinate, such as (0,0,–1), is located "below" the XY drawing plane. When looking at a drawing on screen, this location can be thought of as a point projecting into the monitor away from you. In order to draw objects in 3D space with a CAD system, it is important to be able to visualize objects in three dimensions and understand how points are located using the three coordinate axes. Refer to **Figure 7-6**.

A basic way to locate points when drawing objects is to pick points on screen using the mouse. In addition, there are three common forms of coordinate entry used to specify point locations. Coordinates can be entered as absolute coordinates, relative coordinates, or polar coordinates. See **Figure 7-9**. When using *absolute coordinates*, objects are drawn using points in relation to the coordinate system origin (0,0). The absolute coordinate (2,2) indicates that the point is located two units from the origin along the positive X axis and two units from the origin along the positive Y axis. When using *relative coordinates*, objects are drawn using coordinates in relation to the last coordinate specified (or the origin, if a previous point has not been specified). For example, entering the relative coordinate (@5,4) after entering the absolute coordinate (2,2) places the next point five units along the positive X axis and four units along the positive Y axis "relative" to the absolute coordinate (2,2) or at the absolute coordinate (7,6). Refer to **Figure 7-9B**.

When using *polar coordinates*, coordinates are located at a given distance and angle. Polar coordinates are entered using a format such as (*distance<angle*). This format specifies a linear distance and angle in the XY plane relative to the previous point or the origin. Angles are most commonly measured counterclockwise, with the positive X axis representing 0°. Entering the @ symbol before the polar coordinate locates the point relative to the previous point. If the @ symbol is not entered before the polar coordinate, the point is located relative to the origin. Referring to **Figure 7-9C**, the polar coordinate entry (5<45) locates the point five units away from the origin at an angle of 45° in the XY plane. The polar coordinate entry (8<120) locates the point eight units away from the origin at an angle of 120° in the XY plane.

Each of the different types of coordinate entry can be used when creating a drawing. The entry used typically depends on the object drawn. For example, it may be useful to use absolute coordinates when a given point is known, or when drawing straight lines. Relative coordinates may be used when locating points from a common feature. Polar coordinates are useful for drawing inclined lines and round objects.

User Coordinate Systems

As previously discussed, the default world coordinate system in a CAD program has the origin located at 0,0,0. This is typically sufficient for most 2D drawings, since coordinates for 2D objects can be drawn on the XY drawing plane without specifying a third coordinate along the Z axis.

When creating drawings in 3D, however, it is often useful to change the world coordinate system to a different coordinate system to establish a different drawing plane. A *user coordinate system* is a user-defined coordinate system with an orientation different from that of the world coordinate system. As with the world coordinate system, coordinates are located in relation to a fixed origin. The origin used may be a specific point on an object, such as a corner or center point. See **Figure 7-10**.

User coordinate systems greatly simplify the 3D drawing process. There are also viewing tools and drawing commands that are useful for 3D drawing. Viewing tools and 3D-based modeling methods are discussed later in this chapter.

Absolute Coordinates

Relative Coordinates

A

B

Polar Coordinates

C
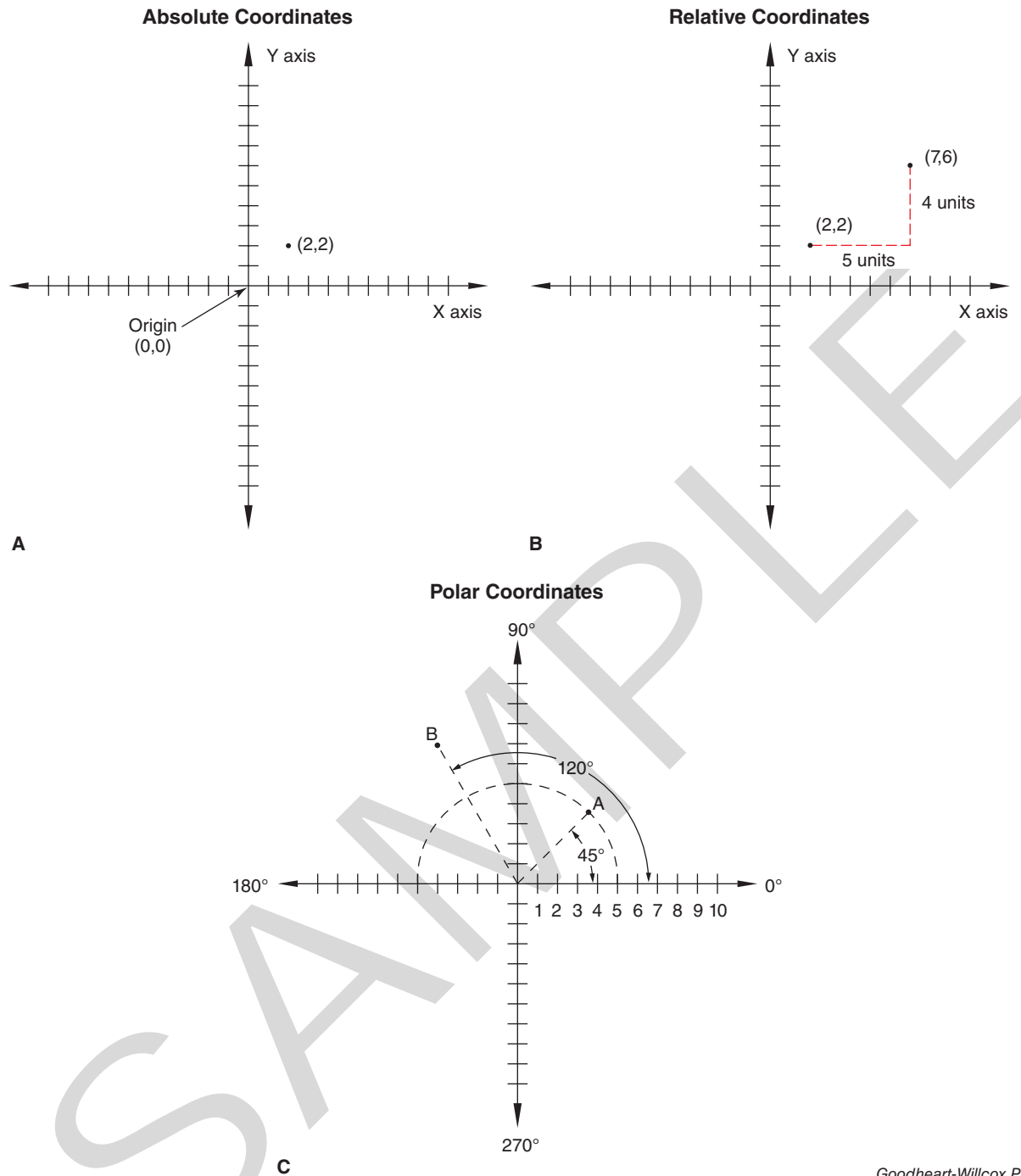
Figure 7-9. The three common types of coordinate entry used to specify point locations. A—The absolute coordinate (2,2) is located 2 units along the positive X axis and 2 units along the positive Y axis from the 0,0 origin. B—Entering the relative coordinate (@5,4) locates the next point 5 units along the positive X axis and 4 units along the positive Y axis "relative" to the absolute coordinate (2,2) or at the absolute coordinate (7,6). C—Polar coordinates are located at a specified distance and angle. Angular values are measured counterclockwise. Point A represents the polar coordinate entry (5<45) relative to the origin. Point B represents the polar coordinate entry (8<120) relative to the origin.
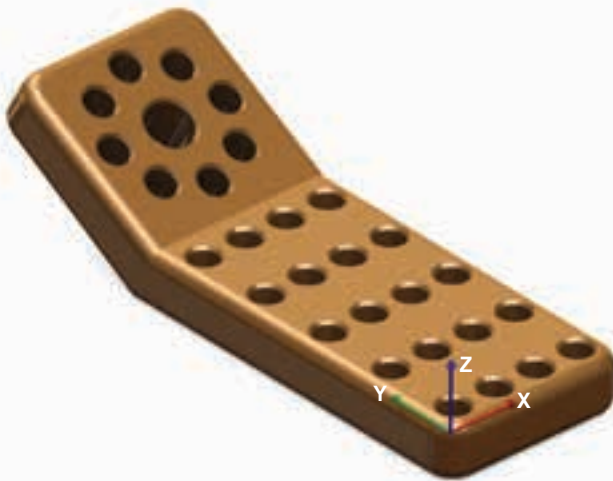
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Figure 7-10. A user coordinate system is used to establish a drawing plane in 3D space so that objects can be drawn on the surface of the plane. The XYZ coordinate axes identify the orientation of the drawing plane. In this 3D model, the XY drawing plane is oriented so that objects can be added to the top surface of the part. Note the direction of the axes.

Drawing Aids

In addition to coordinate systems, CAD programs offer a number of drawing aids that simplify the drawing process. These features make it easy to specify distances and locate coordinates when drawing objects. The typical drawing aids in a CAD program include grid and snap, object snap, orthogonal mode, and direct distance entry.

In CAD, a *grid* is a network of uniformly spaced points used to determine distances. Displaying a grid is similar to using graph paper in manual drafting. The grid spacing may be set to any value that simplifies the process of locating points at specific increments. When used, the grid display is for reference only. It does not print when the drawing is plotted.

Snap is a function that allows the user to "snap" (align) the cursor to specific increments, or snap points, in an invisible grid. Snap can be used with or without the grid display turned on. However, it is common to set the snap spacing in conjunction with the grid spacing. For example, if the grid spacing is set to .50″, snap may be set to .25″ so that the cursor can be "snapped" to the grid points as well as the midpoints between grid lines.

Object snap is a function that allows the cursor to be "snapped" to specific locations on an existing object. It can be set to one or more modes that determine the type of location. This is useful for drawing objects using points on an existing object, such as a line or circle. For example, the **Endpoint** object snap mode allows you to snap the cursor to an endpoint of an object, such as a line. The **Midpoint** object snap mode is used to snap the cursor to the midpoint of an object. The **Center** object snap mode is used to snap the cursor to the center of a circle or an arc. Object snap can also be used to draw objects that are parallel, perpendicular, or tangent to other objects.

Orthogonal mode simplifies the task of drawing horizontal and vertical lines. When orthogonal mode is enabled, the cursor is confined to horizontal and vertical movement on the drawing plane. Inclined lines cannot be drawn when using this mode. Orthogonal mode is useful for drawing lines at 90° angles, such as the outlines making up a drawing border.

Direct distance entry is used to specify accurate distances when drawing lines by moving the mouse in the desired direction and entering a measurement value. This function is commonly used in conjunction with orthogonal mode to draw horizontal or vertical lines at a specific distance. However, direct distance entry can also be used to draw a line at a specific distance and angle by using the appropriate angular entry. This is often faster than using coordinate values to locate points when drawing lines.

Layers and Linetypes

In manual drafting, it is common to have different plan drawings on separate sheets for complex drawing projects, such as an architectural building project. When this is the case, the sheets are overlaid on top of each other so that the different drawings can be viewed separately. In CAD drafting, drawings can be managed in a similar way through the use of *layers*. Layers can be thought of as a system of "overlays" similar to the drawing sheets used in manual drafting. Layers are groupings of objects that can be turned on or off to

distinguish drawing content. See **Figure 7-11**. Using layers, several different displays can be shown within a single drawing file. It is very common, for example, to create separate layers for visible lines, construction lines, section lines, text, and dimensions. The display of each layer can be turned on or off as desired. In this way, certain portions of the drawing can be "hidden" while displaying other features. In architectural projects, different plans are commonly placed on different layers within a single drawing file. The floor plan, foundation plan, and plumbing plan, for example, may each be assigned to a separate layer. This provides a way to plot different displays from a single drawing.

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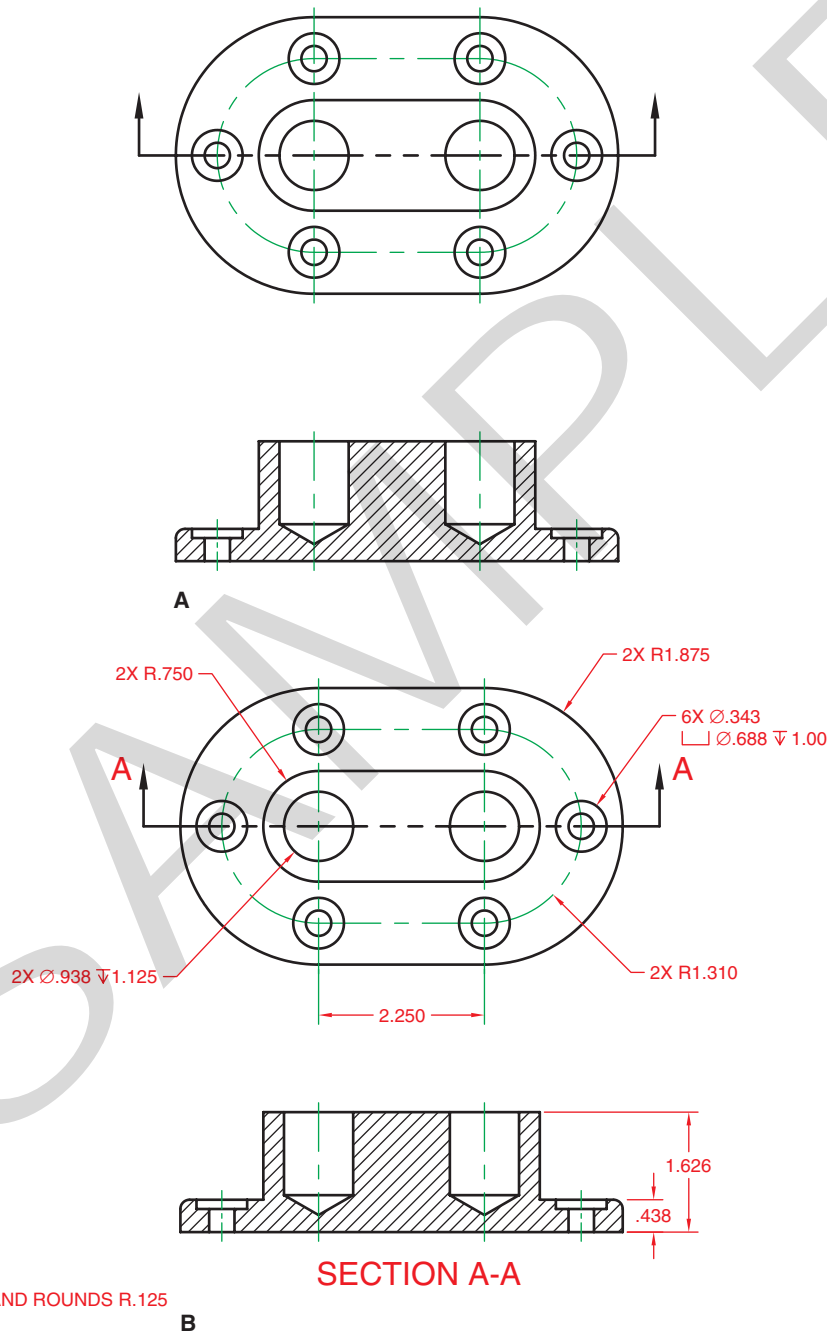
Figure 7-11. Layers are used in CAD drawings to distinguish different drawing content. A—The visible lines, centerlines, section lines, and cutting-plane line in this drawing are on different layers. Each layer has the appropriate linetype. The Centerline layer is assigned the color green. All other layers are assigned the color black. B—The dimension layer is turned on to show dimensions. This layer is assigned the color red.

Layers are typically named to reflect their content. For example, all of the dimensions in a drawing may be assigned to a layer named "Dims." In addition, each layer may be assigned its own color. In many cases, company or school standards determine layer naming conventions and specify how to organize drawing content.

Layers may also be assigned different linetypes to distinguish content. A *linetype* is a setting used to describe a line definition in the Alphabet of Lines. Examples of linetypes include the Object, Centerline, and Hidden linetypes. When a line is drawn with a specified linetype, it has the same characteristics as the equivalent line in the Alphabet of Lines. Each linetype may have its own lineweight setting to reflect the plotting thickness desired. When using a plotter with pens, the line thickness is determined by the size of the plotter pen.

Drawing specifications such as layer and linetype settings should be determined before starting a project. Saved settings for layers, linetypes, object snaps, and other CAD functions can be specified in a drawing template and used each time a new drawing is started. Templates and setup commands are discussed later in this chapter.

Blocks

One of the basic advantages of CAD is never having to draw the same object twice. For example, objects can be copied and reused once they are drawn. Copying objects is discussed later in this chapter. Another way to avoid drawing objects repeatedly is to use blocks. *Blocks* are objects designed for multiple use in drawing projects. Once something is drawn and saved as a block, it can be inserted into a drawing as many times as needed. This is a powerful function that greatly increases drawing productivity. Blocks are typically created for commonly used symbols. For example, blocks are used on architectural plan drawings to represent items such as windows and doors.

Many companies store hundreds of blocks in symbol libraries. A *symbol library* is a collection of related drawing symbols. See **Figure 7-12**.

To create a block, the symbol is drawn and saved with a name. It can then be inserted into drawings. Many manufacturers and drafting firms provide blocks of their products that can be downloaded from websites on the internet.

In some CAD programs, blocks may be saved with attributes. *Attributes* are text strings of information about the related block
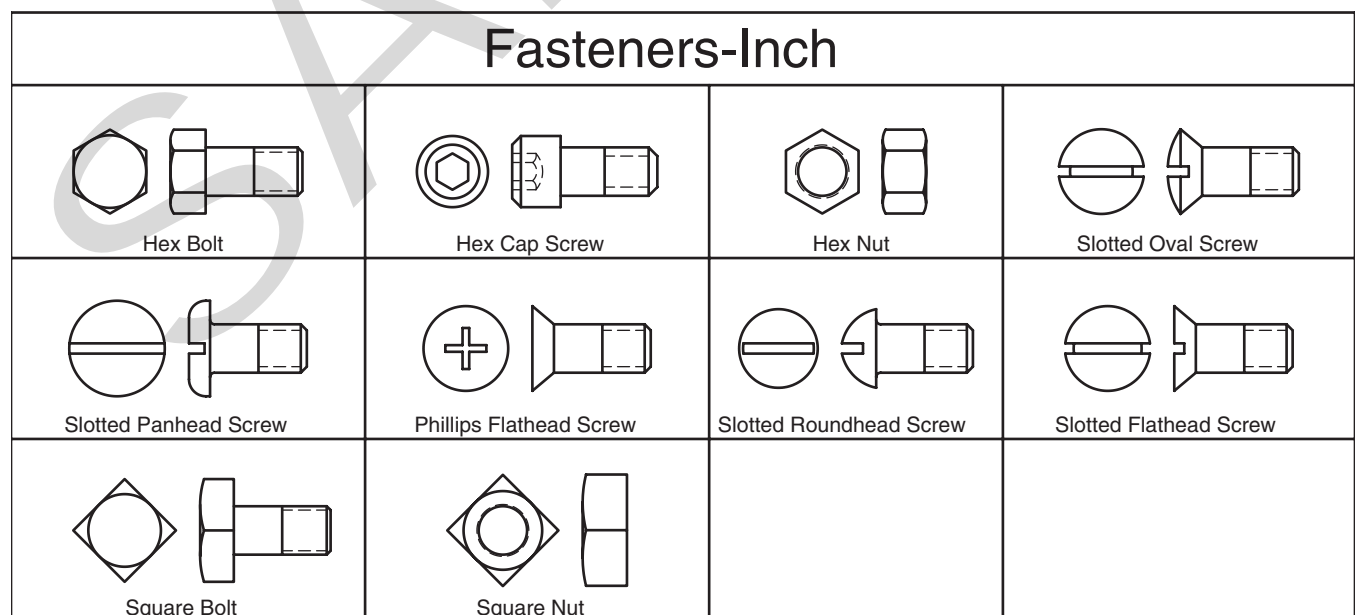
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Figure 7-12. A symbol library of fasteners used in mechanical drafting.

symbol. For example, attributes may be created to identify a product number, price, size, or material. Attributes can be displayed along with the symbol on the drawing, or they may be set invisible. Attributes are commonly used to create schedules. A *schedule* is a chart or table used to list manufacturing and purchasing information about parts and products on a drawing.

Drawing Setup Functions

CAD drawings should be planned carefully prior to beginning work. Making drawing settings ahead of time increases efficiency. Drawing setup is an important part of the CAD drawing process and should become a normal routine when beginning a project.

Linear and angular measurements made with CAD commands are typically based on the current unit settings. A drawing can be set up to use decimal inch, architectural, metric, or engineer's units. Units are typically set with the **Units** command. The unit format selected depends on the drafting discipline.

Regardless of the unit format used, objects are drawn at full size in CAD drafting. This means that an object 2″ × 3″ is drawn at that size. The drawing scale determines the size of the drawing when it is plotted. The drawing scale is based on the size of the drawing media to be used and is determined before beginning a drawing. The drawing scale affects the size of dimensions and text. A number of settings are typically made in relation to the scale factor, such as linetype scaling and text height.

Layers are normally also created during the drawing setup process. Linetypes are assigned to layers as needed. This provides a means to organize a wide variety of content.

Many of the drawing aids previously discussed can be saved in a drawing template. A *template* is a file with standard user settings used to start a new drawing file. Different templates can be created for different drafting disciplines. A typical template includes settings for the unit format, sheet size, and drawing scale. It also includes predefined text styles,

dimension styles, layers, and block definitions. The use of templates saves drawing time and allows drafters to focus on the drawing project at hand.

Creating Objects

There are a variety of ways to create objects using CAD. In most cases, the creation of an object begins with a command. While a command is active, objects may be created by specifying coordinates, such as absolute coordinates, or parameters, such as linear measurements and radius or diameter values. Coordinates and parameters may be entered at the keyboard or specified dynamically on screen with the cursor.

Most of the basic geometric shapes discussed in Chapters 3 and 6 can be drawn quickly with drawing commands. The following sections discuss the common methods used to create basic geometric shapes in CAD.

Drawing Lines

Lines are most commonly drawn with the **Line** command. A line may be drawn horizontal, vertical, or inclined by specifying coordinates at the keyboard or by using the cursor to pick points on screen. A line requires two coordinates, **Figure 7-13**. Additional coordinates may be entered within a single command sequence to create as many line segments as needed. As previously discussed, lines may be displayed using different line conventions by applying the proper linetype. Before drawing the line, the layer with the appropriate linetype setting must be set as the current layer.
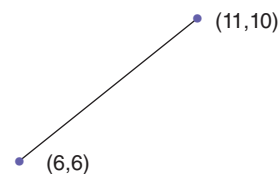
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Figure 7-13. A line is created by drawing a segment between two coordinates.

Drawing Circles and Arcs

Curves making up CAD-generated circles and arcs are defined mathematically by the program based on the coordinates entered. Circles are typically drawn by specifying the center point and a radius or diameter, **Figure 7-14A**. A circle may also be drawn by specifying points along the perimeter of the circle or by entering a radius and selecting two lines or two circles to which the circle should be tangent. The **Circle** command is most commonly used to draw circles. The center point location and radius value may be entered at the keyboard or picked on screen.

Arcs can be drawn with the **Arc** command. A number of methods are usually available. Arcs typically require a center point, radius, and endpoint. See **Figure 7-14B**. Arcs may also be drawn by specifying three points, or a starting point, a center point, and a third entry, such as a chord length.

Drawing Ellipses

Ellipses are drawn with the **Ellipse** command. An ellipse has a center point, a minor axis, and a major axis. See **Figure 7-15**. The axes divide the ellipse into four quadrants. Points for the axis endpoints and center point can be entered at the keyboard or picked on screen.

Elliptical arcs (portions of an ellipse) can also be drawn by specifying start and end angles after locating the axis endpoints. The start and end angles represent points on the curve relative to the angular locations of the axis endpoints. The angles are measured counterclockwise from the first axis endpoint, which is designated as 0°. For example, an arc drawn with a 0° start angle and 180° end angle represents half an ellipse.

Drawing Polygons

The **Polygon** command can be used to draw regular polygons. First, the number of sides is entered. Specifying three sides draws an equilateral triangle. Specifying four sides draws a square. Regular pentagons, hexagons, and octagons can also be drawn.

After entering the number of sides, the center point is specified. The command sequence includes prompts for the user to inscribe or circumscribe the polygon. See **Figure 7-16**. As you learned in Chapter 6, an inscribed polygon is drawn within a circle. A circumscribed polygon is drawn about and is tangent to a circle. After specifying the orientation of the polygon, the radius of the circle is entered.

Drawing Hatch Patterns

Hatch patterns are used to fill a closed area with a predefined pattern representing a material or a special graphic symbol. Hatch patterns are frequently used to represent materials in section views. Section views are used to show internal features of an object and are discussed in Chapter 11. There are different types of hatch patterns corresponding to specific materials, such as steel, brass, concrete, and wood. A commonly used hatch pattern is the general purpose section line symbol, which consists of continuous lines drawn at a 45° angle and approximately .125″ apart. See **Figure 7-17**. The general purpose section line
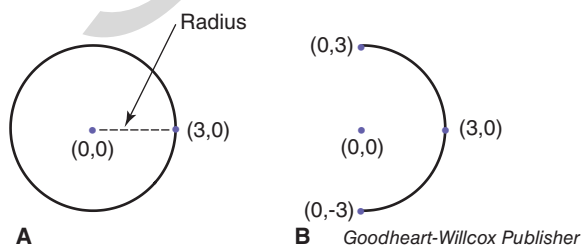

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Figure 7-14. Drawing circles and arcs. A—Circles are defined with a center point and a radius or diameter. B—Arcs are commonly defined with a center point, a starting point, and an endpoint, or with points along the arc.
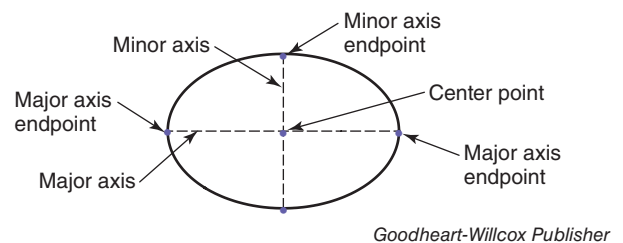

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Figure 7-15. An ellipse is drawn by specifying endpoints for the minor and major axes or by specifying a center point and the axis endpoints.

Inscribed Polygon   **Circumscribed Polygon**

A   B   *Goodheart-Willcox Publisher*
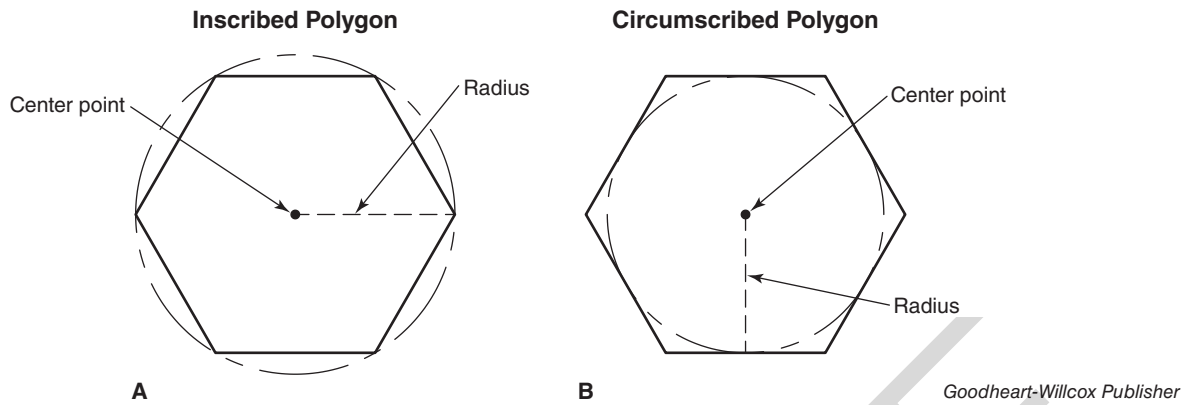
Figure 7-16. Drawing a polygon. Shown are examples of drawing a hexagon (a six-sided polygon). A—An inscribed polygon is drawn by specifying the number of sides, a center point, and the radius of the circle within which the polygon is inscribed. B—A circumscribed polygon is drawn by specifying the number of sides, a center point, and the radius of the circle about which the polygon is circumscribed.

symbol is often used when the material specification is indicated in another location on the drawing.

Hatch patterns are created with the **Hatch** command. The first step in creating a hatch pattern is to select an internal point of the area to be hatched. An alternative is to select an object that forms a boundary around the area to be hatched. After making a selection, a preview of the hatch pattern appears. At this point, you can specify a different hatch pattern or adjust the default settings of the existing pattern. Options are typically available for adjusting the hatch pattern scale and angle.
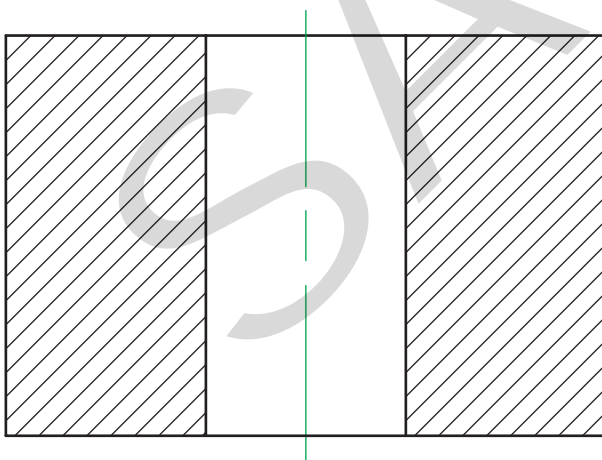
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Figure 7-17. Hatch patterns are used to represent materials in section views. The general purpose section line symbol, used for hatching in this example, consists of continuous lines drawn at 45° and spaced approximately .125″ apart.

Numerous patterns are available including different material symbols, color fills, and gradients.

Editing and Modifying Objects

One of the most important advantages of a CAD program is the ability to easily modify objects once they are drawn. Objects can be moved, copied, rotated, and scaled using the appropriate commands. These commands may be identified by the software as editing commands or modifying commands (or both), depending on the CAD system you are using. In addition to these basic commands, there are a number of other editing methods that can be used to construct objects. This provides great flexibility. Some of the most common editing and modifying commands in CAD programs are discussed in the following sections.

Moving Objects

It is often necessary to relocate objects after they are drawn. This can be done with the **Move** command. After selecting one or more objects to move, you must specify a base point for the selection set. This may be the corner point of a rectangle or the center point of a circle. You are then asked for a displacement point. You can pick a point on screen or enter coordinates, or you can move the mouse and enter a distance at the keyboard. The objects

making up the selection set are then moved automatically to the new location.

Copying Objects

Copying objects is similar to using the **Move** command, except the original objects are not altered by the operation. Objects can be copied using the **Copy** command. After selecting the objects to copy and the base point, a displacement point is specified. The selected objects are then copied to the new location.

Depending on the software you are using, the **Multiple** option may be active when you access the **Copy** command. This option allows you to copy the original object or selection set of objects to multiple locations in a single command sequence.

Rotating Objects

Rotating an object changes the angular position of the object with respect to the current orientation. Objects can be rotated using the **Rotate** command. When using this command, the selected objects are rotated about the base point specified. The objects may be rotated clockwise or counterclockwise.

If the objects to rotate are already rotated to a given angle when you enter the **Rotate** command, you can enter the angle as a reference angle. For example, if the objects are drawn at 45°, enter 45° as the reference angle. Then, enter the desired angle of rotation.

Scaling Objects

An object can be reduced or enlarged to a different size by a given scale factor. This is accomplished with the **Scale** command. After selecting the object to be scaled, the base point and scale factor are specified. A scale factor of 0.5, for example, would be used to reduce the size of an object to one-half its original size.

Undoing a Command

CAD programs typically provide a command that allows you to "undo" a previous operation. If you enter an incorrect value for a scale

or move operation, for example, you can reverse the action by using the **Undo** command. This command typically allows you to undo several preceding commands, one by one. However, the commands must be undone in sequence.

Redoing a Command

The **Redo** command is used to reverse an action that was undone with the **Undo** command. For example, if you move an object and then use the **Undo** command, the object returns to its original location. However, if you decide that the object should be placed back where you moved it, you can use the **Redo** command to reverse the "undo" operation and return the object to where it was moved. The **Redo** command must be used immediately after using the **Undo** command.

Erasing Objects

The **Erase** command provides a quick way to remove unwanted objects from a drawing. After you select the objects to erase, the command automatically removes them from the drawing. The **Undo** command can be used to restore an object that has been erased unintentionally.

Arraying Objects

An *array* of objects can be created by orienting multiple copies of a selected object in a pattern. This operation is useful when the same object appears in multiple locations in a regular pattern in the drawing (for example, when a pattern of holes is machined in a round part). Arrays may be created in rectangular or polar arrangements with the **Array** command. See **Figure 7-18**. A rectangular array is created by entering the base point, number of rows, number of columns, and the spacing between rows and columns. The number of columns and rows determines the number of objects in the array. A polar array is created by specifying a center point, the number of objects in the array, and an angular value determining the amount of rotation. Entering 360° creates a full rotation of objects about the center point.
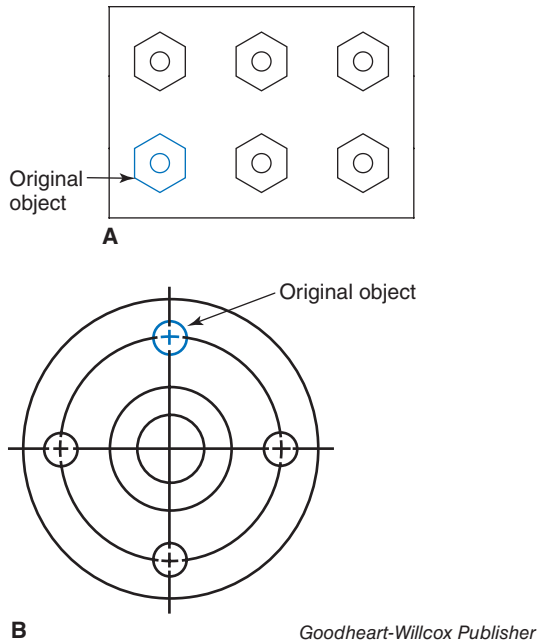
A

B   *Goodheart-Willcox Publisher*

Figure 7-18. Arrays are created by orienting copies of original objects in a regular pattern. A—Rectangular array. B—Polar array.

Mirroring Objects

When drawing symmetrical objects, it is sometimes useful to create a mirror image. This operation allows you to select an object and make a mirror copy. This can save time when you want to draw half of an object and complete it by "mirroring" it. See **Figure 7-19**. The **Mirror** command is used to mirror an object. To use this command, the objects to be mirrored are first selected. Then, a mirror axis is specified. The axis represents a line about which the objects are "reflected." The command sequence typically allows you to keep or delete the original objects selected before mirroring.

Creating Rounded and Angled Corners

Rounded and angled corners are often drawn in mechanical drafting. A *round* is an arc representing an outside rounded corner. A *fillet* is an arc representing an inside rounded corner. See **Figure 7-20**. A *chamfer* is an angled line drawn where two straight lines would normally meet at a corner. Rounds, fillets, and chamfers are used to smoothen sharp edges.

Original objects   **Objects after mirroring**

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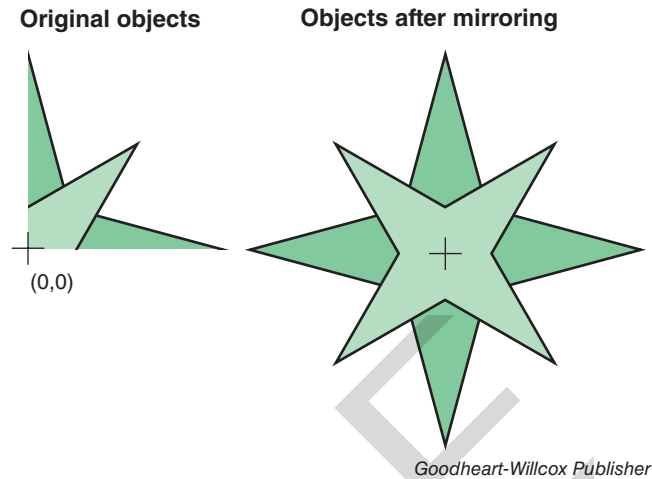
Figure 7-19. Mirroring an object creates a mirror copy about a mirror axis. In this example, the original image is mirrored twice using the X and Y axes.
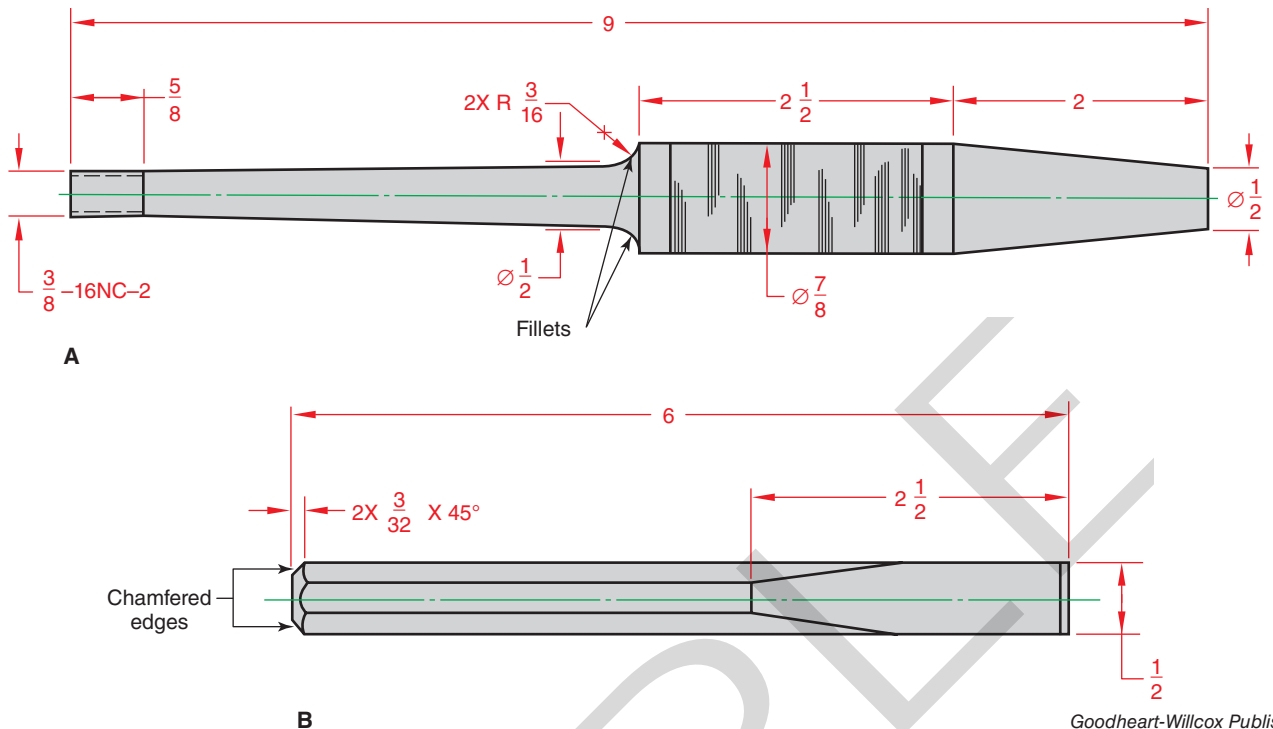

Goodheart-Willcox Publisher
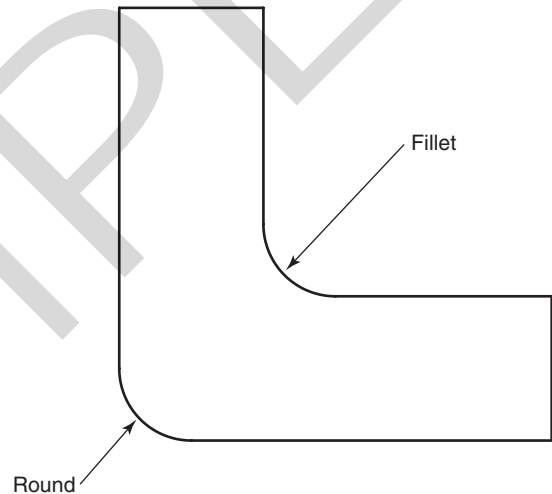
Figure 7-20. Fillets are used to strengthen inside corners and rounds are used to remove sharp edges.

See **Figure 7-21**. Rounds and fillets can be drawn with the **Fillet** command. After entering the command, the fillet radius is set. Then, the two lines, circles, or arcs forming the intersection are selected. The original objects are then trimmed and the arc is automatically drawn.

A chamfer can be drawn with the **Chamfer** command. After entering the command, the chamfer distances from the two lines to the corner are set. A 45° chamfer is created with equal distances. Next, the two lines are selected. As with the **Fillet** command, the lines are trimmed automatically when the chamfer is drawn.

Figure 7-21. Creating rounded and angled features. A—Fillets are created by specifying a fillet radius and selecting the two objects forming the intersection. B—Chamfers are created by specifying the chamfer distances and selecting two lines.

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Trimming and Extending Lines

Many objects in a CAD drawing are made up of lines. During the drawing process, it is sometimes necessary to "clean up" areas of the drawing where lines must intersect accurately. Trimming is useful when two lines overrun past a corner. Extending is useful when a line must be lengthened to meet an edge or other entity.

The **Trim** command is used to trim lines, arcs, and circles. After entering the command, a cutting edge must be specified. This represents the point to where the object is trimmed. Next, the object to be trimmed is selected. The portion of the object extending past the cutting edge is automatically removed. When using the **Trim** command, you can select as many cutting edges and objects as needed to complete the trim operation.

The **Extend** command is used to extend lines and arcs to meet other objects. After entering the command, a boundary edge must be specified. This represents the point to where the object is extended. Next, the object to be extended is selected. The object is then extended to the boundary to create a larger entity. As with the **Trim** command, you can select as many objects as needed to complete the extend operation.

Using Display Commands

There are a variety of ways to display drawing content on screen when working on a CAD drawing. Display commands are used to change the magnification of the drawing, change the viewpoint, and establish drawing views, such as the views in a multiview drawing. Multiview drawings are discussed in Chapter 9.

When working on a CAD drawing, it is often necessary to "zoom" into certain portions to view features. The **Zoom** command provides this capability. After entering the command, you can zoom into a portion of the drawing by windowing around the display. The windowed portion is then shown at a greater magnification. You can also enter a magnification scale factor to reduce or enlarge the display relative to the current display. A third option allows you to zoom the view in real time by using the pointing device to move the cursor upward (to enlarge the view magnification) or downward (to reduce the view magnification).

When you want to move the drawing across the screen to view areas outside of the current display without changing the magnification, you can use the **Pan** command. Panning adjusts the view in real time. The drawing is panned by using the pointing device to move the cursor in the direction desired.

More advanced viewing commands are available with 3D drawing programs. While command names and navigation methods will vary, 3D viewing commands typically allow you to rotate a 3D model in three dimensions by using the pointing device. The view is changed in real time, allowing you to view different features across the model dynamically.

Using Measurement, Object Property, and Drawing Status Commands

During the drawing process, it is common to check measurements of existing objects to verify accuracy and confirm that design requirements are met. Measurements are also needed to determine the correct dimensions when constructing new features in relation to other features. CAD programs typically provide commands to make measurements of existing objects and determine other information, such as object properties and current drawing file data.

Measurement commands allow you to quickly determine common measurements, such as linear distances and area and perimeter calculations. The **Measure Geometry** command allows you to select two points and measure the distance and/or angle between them. You can also use this command to determine the radius or diameter of a circle or arc by selecting the object. When using the **Measure Geometry** command to determine the area or perimeter of a closed object or a specific area, you must select the object or pick the points defining the area.

Common object properties, such as layer and linetype settings, can be identified by using the **Properties** command. After entering the command and selecting an object, the program

Academic Link

Drafting and Electronic Communication

The evolution of computer technology and electronic communication has had a major effect on the way drawings are made and presented. Not long ago, manual drawings were the primary means of communicating manufacturing information to trade workers. Today, drawings are created with computer-aided drafting programs and distributed electronically. They are used by workers to program computer-controlled machine tools. Computers then interpret the information and manufacture parts. This is accomplished through various phases of communication—including interaction between the drafter and the computer and the computer and a machine.

There are literally hundreds of CAD software programs that have been used to design products for industrial use. Computer animation programs make it possible to communicate an entire design of a product before it is manufactured or built. Internet technology makes it possible to send, receive, evaluate, and modify drawings in a very short period of time.

When compared to manual drawing techniques, CAD tools have made it much simpler to communicate information. However, it is important to understand that as with other communication tools, CAD is only a tool. The same drawing skills, visualization techniques, and concepts practiced in manual drafting must be understood in order to use CAD accurately and successfully.

displays various information, including the object coordinates and the layer, linetype, and color settings assigned to the object.

Other commands are available for determining additional information about objects in the current drawing and data associated with the drawing file. The **List** command allows you to select one or more objects and list information from the database about each object. The **Status** command allows you to identify the drawing limits and other statistics, including the number of objects in the drawing, the current drawing settings, and the amount of free space on the drive where the file is stored. The **Time** command allows you to display the current date and time, the drawing time in the current session, and the date and time the drawing was created.

Using Help System Functions

As is the case with most types of computer software programs, CAD software programs generally include a help system to provide user help. In some cases, a printed User's Guide may accompany the software when purchased. However, most CAD programs include software-based and/or online help systems. The tools provided do not vary much from those provided with other software programs. The help system in a CAD program is typically accessed with the **Help** command or by pressing the [F1] function key. See **Figure 7-22**.

Typical CAD program help systems include listings of help topics, search functions, and question-and-answer tools. These features provide a number of ways to seek answers to questions encountered while using
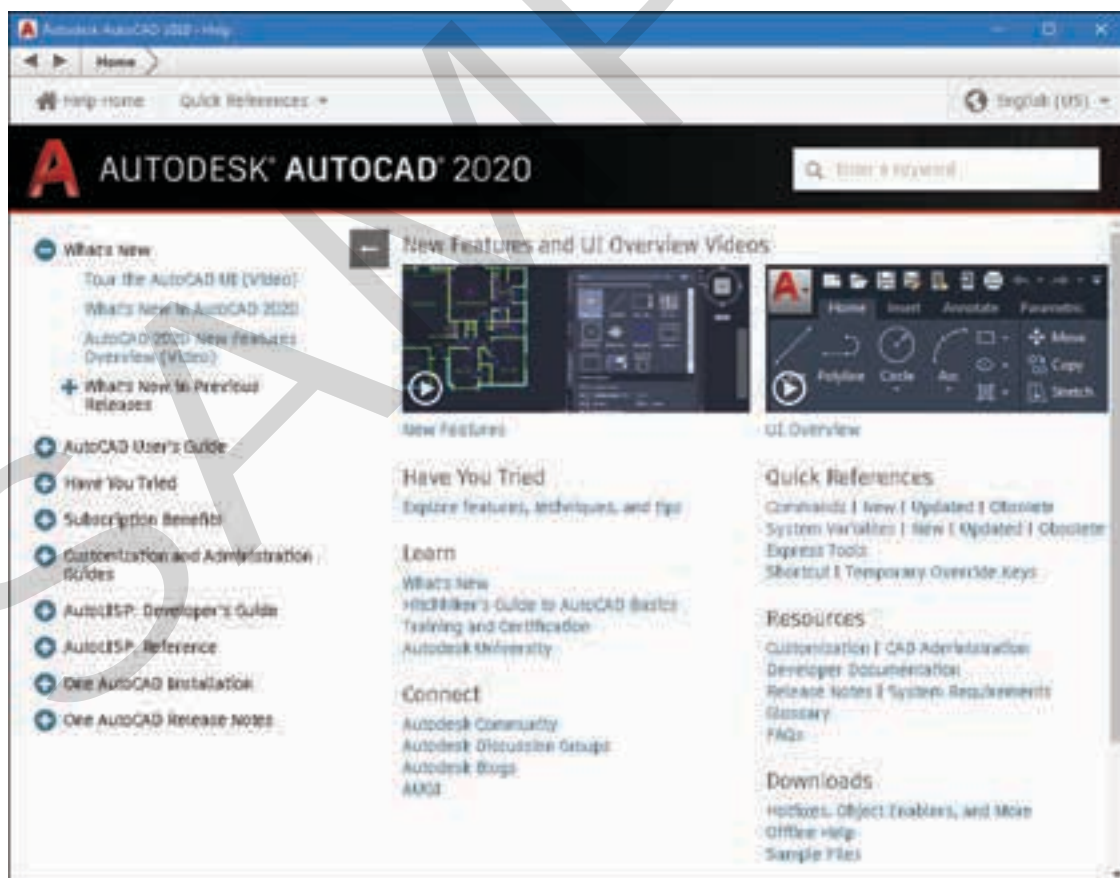
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Figure 7-22. A typical CAD program help system provides a comprehensive resource for locating help topics.

Careers in Drafting

Drafting/CAD Teacher

What would I do if I were to become a drafting/CAD teacher? Because I would be a type of career and technical education teacher, I would use many different methods of presenting information and concepts relating to drafting and computer-aided drafting (CAD) in order to help students learn how to solve design problems. I would assist students in learning how to properly create drawings that are used to manufacture or construct objects.

What would I need to know? I would need to be able to properly organize a classroom and know how to present the information, concepts, and skills that I need to teach my students. I would need to know how to use various manual drafting tools as well as computers and CAD software. I would need to be familiar with a variety of tools, techniques, and procedures as they relate to various specialized fields of drafting—such as mechanical, architectural, electrical, and civil drafting. I would need to know how to work effectively with diverse groups of students. I would also need to know how to design classroom presentations; plan, evaluate, and assign lessons; prepare, administer, and grade tests; evaluate oral presentations; and assist students in setting career goals.

For whom would I work? I could work for a public school district, usually at a secondary level. I could also work for a public or private technical school, trade school, college, or university. Some drafting/CAD teachers also work for large corporations to keep professionals current with the latest technologies, practices, and procedures.

What sort of education would I need? While in high school, I would need to take courses in drafting, mechanical drawing, CAD, and computer science, along with required high school courses. All 50 states require public school teachers to be licensed and have a minimum of a bachelor's degree from an approved teacher education program. Some states require that teachers earn a master's degree within a specified period of time after being hired. Many drafting/CAD programs are designated as career and technical education programs, and most states require that teachers of career and technical education programs be credentialed. Job experience as a drafter or CAD drafter is very helpful in acquiring a career and technical education credential.

What are my chances of finding a job? In general, the chances of finding a job are about average. Employment for teachers is expected to grow at the same pace as other occupations. However, there are several factors regarding job opportunities for drafting/CAD teachers. First, many teachers now teaching drafting and CAD are nearing retirement age and will need replacements. Second, the job market is sorely lacking in qualified drafting/CAD teachers not already employed.

How much money could I expect to make? Teacher pay varies greatly because of numerous variables. Generally speaking, I could expect to make more if I taught in a metropolitan area than in a rural setting. Recent statistics identify the average income of a public high school career and technical education teacher to be $62,460. Private school teachers generally earn less than public school teachers. Degree held, locale, and amount of experience are the three largest contributing factors to the level of pay.

If I decide to pursue a different career, what other fields are related to drafting and CAD? Because teaching drafting and CAD is much like teaching applied geometry, teaching math would be similar. General technology education also encompasses some of the content taught in drafting/CAD courses. Many projects completed in the average drafting program also have close ties to science and art. Thus, I could become a teacher of math, science, art, or technology education, and still be teaching in a related field.

Discussion Questions

- What do you think would be the benefits of becoming a drafting/CAD teacher?
- Do you think becoming a drafting/CAD teacher would be a good career choice for you? Why or why not?

the software. The help system, if used properly, can help the CAD drafter become much more proficient in using the software.

A typical help system is organized into a series of documents called "guides" and "references." The typical documents available include a User's Guide, a Command Reference, an Installation Guide, a Licensing Guide, and a Customization Guide. In addition, the help system typically includes an index. The index is an alphabetized, detailed list of topics related to the software and its use. When the index is active, the user can input a keyword in a text box and automatically "jump" to the related topic in the listing.

Additional features and functions are normally available when using the help system. The **Search** function allows you to input a term or topic to search. The system then conducts a search and produces a list of help topics based on the specified term or topic. In addition, the help system usually provides internet links to more in-depth help. For example, to learn more about a help topic or question, you may be able to access the software publisher's website, a technical support forum, or a user's forum for users of the CAD program.

CAD File Management and Storage Practices

File management plays a critical role in organizing CAD projects. Once you start working with CAD files, you will find it necessary to set up an organization of storage folders that permits quick retrieval. Establish a system that makes it easy to locate files for a given project. For example, you may want to establish a top-level folder for each project and include subfolders named to reflect their contents. Using this approach, you might decide to have a top-level folder for an architectural building project and subfolders that are named to identify the different plan drawings created in each phase of the project. A system such as this one might have subfolders with names such as First Floor, Second Floor, Electrical, HVAC, and Plumbing.

It is also important to establish consistent file naming conventions to accurately identify the contents of files. This makes it easy to locate files and improves productivity. It is common, for example, to use prefixes in file names to identify information such as the project code and drawing discipline. Your school may already have established conventions for naming files. If so, make sure to follow the conventions.

In some projects, drawing files include "references" to other files. For example, when creating an electrical plan drawing in a building project, you typically need to "reference" data from the floor plan drawing to help locate features based on the floor layout. If the floor plan is "referenced" by the electrical plan, it becomes a "dependent" of the parent file. When opening the parent file (the electrical plan), the software must be able to locate the reference file (the floor plan). If the file cannot be located, you must supply the file path. This is another reason to maintain an orderly file storage system. Make sure to organize your projects so that files referenced by other files can be located by the software.

Saving, Backing Up, and Archiving Files

Whether working with manually created drawings or CAD files, the need for secure, organized archives of original drawings is of utmost importance. Storing manually created drawings in physical files requires a secure facility, adequate organizational techniques, and plenty of space. At the same time, proper storage of CAD drawings creates an entirely different set of problems and challenges.

Since CAD drawings are electronic files, there must be a means of storage that protects against cybertheft, hardware failure, and file damage. Preventing the loss of work is a primary concern in every project. As most computer users are no doubt aware, computers do periodically crash. Hardware fails and occasionally, power fails. Hours of work can be lost in the blink of an eye if files are not saved frequently.

It is for these reasons that most CAD firms establish a protocol of file storage and backup strategies. This protocol usually requires that employees save and back up their work at regular intervals. Generally, these strategies are very in-depth routines. A good file storage protocol is designed so that everything possible is done to guarantee that valuable work and hours are not lost.

Your CAD instructor will probably set up a similar protocol for saving and backing up files. Methods will vary, but files are typically saved on a network server and backed up to another location so that valuable work is not lost. It is definitely to your advantage to follow your instructor's file storage and organizational techniques without variation. Not following an accepted protocol may result in you having to refer to the old saying, "If you do the crime, you will pay the time!" In other words, if you don't consistently save and back up your work often, there is always a chance that you will end up doing it again.

A typical protocol for achieving safe and reliable file storage may include the following:

1. Save your work every 15 minutes.
2. Save files to a network server rather than a local hard drive. Save to a network server with automatic file backup capability, if possible, to ensure file security.
3. Back up work weekly to another location separate from the network server, such as one or more portable media devices. Work can be saved to CDs, DVDs, or flash drives.

This is just a sample protocol and will probably vary based on your unique situation and your instructor's recommendations.

When work is completed on a project and the files do not need to be used on a daily basis, an electronic file archive is typically created. An electronic archive typically consists of all of the files related to a single project. This may involve hundreds of files and many different file formats. Some CAD programs provide special tools for creating file archives. No matter what method you use, there are several important points to keep in mind.

Files to be archived must be saved in a format and on electronic media that will be compatible with computer software and hardware of the future. Hence, the long-term usefulness of CAD files is of greater consequence than how much storage space they require. As with files in a currently active project, archived files must be accessible so that they can be located, opened, and edited if needed. The same file naming conventions, folder organization practices, and backup procedures used for active files should also be applied when assembling an electronic archive.

Overview of CAD Systems and Software

There are many CAD software programs available for a variety of applications. The type of program used normally depends on the application or drawing discipline for which it is used. CAD programs range in capability from simple 2D drawing programs to advanced 3D modeling and presentation programs. While basic CAD programs require relatively inexpensive hardware to drive the system, higher-end programs may require computer equipment costing several thousands of dollars.

As discussed in Chapter 4, a basic CAD system consists of a computer, monitor (display device), keyboard, pointing device, and output device. Data is stored on the hard drive of the computer or on portable media. The primary component of the CAD system is the software. The software is the set of instructions that tells the computer what to do and when to do it.

CAD software programs can be classified in several different ways. Some CAD programs provide 2D drawing capability only. These programs have many of the functions previously discussed in this chapter, with the exception of 3D-based tools. Other CAD programs provide more advanced tools for 3D modeling purposes. These programs are used to construct models defined with 3D coordinates. Models created in this manner are then typically used for additional purposes, such as testing,

analysis, and prototyping. Common 3D modeling commands are discussed in Chapter 13.

CAD programs with 3D modeling functions are typically based on a specific type of 3D modeling, such as solid modeling or surface modeling. In *solid modeling*, objects called solid models are created to represent the entire mass of an object. See **Figure 7-23**. A solid model is considered to be defined from the actual material making up the object. Solid models are used in mechanical drafting applications because they can be analyzed for properties such as mass and volume. Models created in *surface modeling* are similar to solid models, but the objects are not defined as solid. Surface models have an outer "skin" to represent exterior surfaces. See **Figure 7-24**. While a surface model is not considered as realistic as a solid model, the quality of the representation is very similar. Therefore, surface models are primarily used for presentation purposes.

Another type of 3D modeling is parametric modeling, discussed in the following sections. Parametric modeling programs that are used to create solid models are most common in mechanical engineering applications. This chapter focuses on the common tools used in these programs.

Parametric Modeling

Parametric modeling is an advanced form of 3D modeling in which the model dimensions, or parameters, are defined and modified as needed throughout the modeling process. In this type of modeling, *parameters* control feature sizes and locations and a change to one parameter affects the entire model. See **Figure 7-25**. Because parameters control the size and location of features, parametric models are sometimes referred to as "intelligent" models.

Parametric modeling programs used in mechanical design and engineering involve a different workflow in comparison to 2D CAD drawing programs and other types of CAD modeling programs. In parametric modeling, 3D models called parts are created to represent mechanical components. Each *part* is made up of the various features that define the part.

Autodesk, Inc.

Figure 7-23. A solid model represents the entire mass of an object and the material used in its construction.

Iaremenko Sergii/Shutterstock.com

Figure 7-24. A surface model is a representation of the outer "skin" of the various surfaces simulating the object.

Model courtesy of SolidWorks Corp.

Figure 7-25. Parametric modeling programs provide powerful modeling tools that allow object dimensions to be changed throughout the modeling process. Any changes made affect the entire model.

An example of a part is a shaft. In many cases, a part begins by drawing a 2D profile called a *sketch* to create the most basic feature or shape of the part, called the *base feature*. Additional features can be created from sketches or other modeling methods. Because features form the basic building blocks of a part, this process is referred to as *feature-based modeling*.

Each part is created as a separate electronic file. Several completed parts can then be used to create an assembly by assembling and mating the parts together. An *assembly* is a functional mechanism made up of multiple components (parts). An example of an assembly is a set of gears. An assembly is created as a separate electronic file and contains references of the individual part files. The part files are linked to the assembly file so that if a part file is edited, the assembly updates automatically to reflect the edit. The completed assembly can be simulated to determine the effectiveness of the design and to check for interference between parts.

Once the modeling process is completed, working drawings can be generated from the part and assembly files to document the engineering design process. Drawings are created as separate electronic files. However, the part and assembly files are linked to the drawing files so that if the model is edited, the corresponding drawing is updated to reflect the edit.

The following are typical steps used in parametric modeling to model parts and construct an assembly from the parts. These steps are discussed in more detail in the following sections.

1. Create a 2D sketch that defines the profile of the base feature of a part. Make the sketch on the appropriate plane (front, top, or side).
2. Add geometric constraints and dimensions to the sketch. Fully define the sketch.
3. Using the appropriate modeling tool, create the base feature from the completed sketch.
4. Create additional features as needed to complete the part.
5. Repeat the previous steps to create additional parts to be used in the assembly.
6. Construct an assembly model from the individual parts. Mate the parts using the appropriate assembly constraints.
7. Generate working drawings from the assembly.

Creating Sketches

Creating sketches in parametric modeling is similar to creating orthographic views, as discussed in Chapter 9. A sketch is a 2D profile of an object, such as the front or top view of the object. Therefore, the first step in making a sketch is to decide which profile of the object will be used for the sketch. This determines how the part model will begin to take shape. When making a sketch for the base feature of an object, for example, the profile is usually one of the principal views of the object. This is often the profile that shows the basic contour of the object. See **Figure 7-26**.

When starting a sketch, sketch mode is made active and you are in the sketch environment. Normally, a sketch defining the base feature is made on one of three default sketch planes. These are the front (XY), top (XZ), and right-side (YZ) sketch planes. The point where the three sketch planes intersect is the coordinate system origin. The sketch plane you select determines the orientation of the part that will be modeled and should normally correspond to the profile to be drawn. For example, if you are making a sketch that shows the front view of a part, use the front (XY) sketch plane.

In parametric modeling, commands in the software are typically referred to as *tools*. Tools used to make sketches are similar to drawing commands used in 2D CAD drafting. Common sketch tools include the **Line**, **Circle**, and **Arc** tools. The **Line** tool is used to create straight and inclined lines. The **Circle** tool is used to create circles. Circles can be drawn by specifying a center point and radius or by specifying three points on the perimeter. Circles can also be drawn tangent to other objects, such as lines and circles. The **Arc** tool is used to create arcs. Methods for creating arcs include specifying a center, start, and end point, or a start point, end point, and a point defining
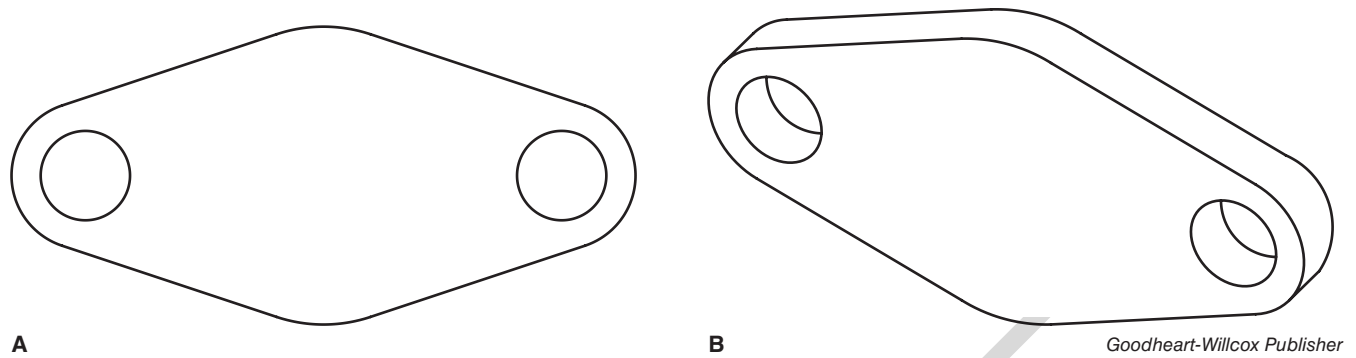
A

B

Figure 7-26. Constructing a part model in parametric modeling typically begins by identifying the base feature and creating a 2D sketch. In this example, a sketch representing the front view is created. A—The sketch is made up of lines, circles, and arcs and is created on the front plane. B—The base feature is created by extruding the sketch.

the radius. Arcs can also be drawn tangent to other objects. Additional shapes, such as rectangles, ellipses, polygons, and splines, can be drawn by using the appropriate sketch tool.

Modify tools are available when creating a sketch. For example, you can move, rotate, scale, mirror, trim, extend, or offset sketch geometry. Tools are also available for creating fillets and chamfers. Modify tools are often useful for making quick adjustments and increasing productivity. For example, if you are making a sketch of a symmetrical object, draw half of it and use the **Mirror** sketch tool to create the other half. A construction line can be drawn to serve as the mirror axis. See **Figure 7-27**.

Adding Constraints

Once the sketch has been created, the next step is to constrain the sketch. Constraining sketch geometry serves the purpose of preserving geometric and dimensional relationships when the sketch is edited or when a feature created from the sketch is edited. Sketches are constrained by adding geometric constraints and dimensions. A *geometric constraint*, usually referred to as a *constraint* or a *relation*, controls a geometric relationship between objects. For example, a parallel constraint can be used to constrain two lines so that they are always parallel to each other.

Geometric constraints can be added in one of two ways. By default, the program adds geometric constraints automatically as you create sketch geometry. For example, if you

Mirror axis
(construction line)

Original Sketch
Geometry

Sketch after
Mirroring
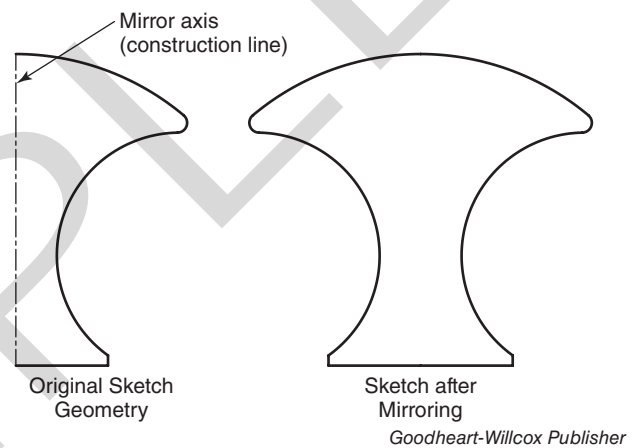
Figure 7-27. Mirroring a sketch of a symmetrical object. The mirror axis can be drawn as a construction line.

draw a straight line in a horizontal direction, a horizontal constraint is automatically added. The horizontal constraint ensures that the line will remain horizontal if the sketch is edited. Geometric constraints can also be added manually by the user after creating the sketch geometry. There are different types of constraints for controlling specific relationships. Each constraint is represented by an icon in the sketch. See **Figure 7-28**. Common types of geometric constraints include the horizontal, vertical, parallel, perpendicular, tangent, equal, concentric, collinear, and coincident constraints. The equal constraint is used to constrain two objects to the same size or length. For example, if a circle has a diameter of 1″ and a second circle has a different diameter, an equal constraint can be assigned to constrain the two circles to the same diameter. If the size of one circle is changed, the other
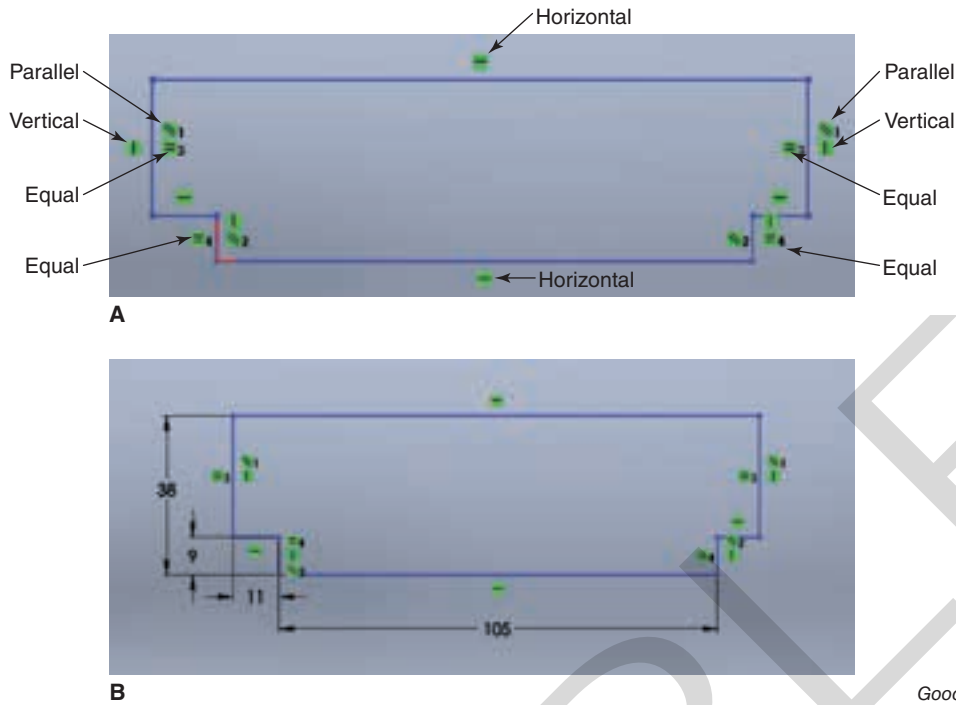
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Figure 7-28. Constraints are added to a sketch to control geometric and dimensional relationships. A—Geometric constraints are represented by icons. B—The sketch after adding dimensional constraints.

circle is changed to the same size. Referring to **Figure 7-28A**, parallel constraints have been applied to the vertical lines on the left and right sides of the sketch. The lines will remain parallel regardless of changes made to the lines or to other sketch geometry. In addition, the lines have equal constraints to keep them the same length if they are edited. Notice that each sketch entity can have one or more geometric constraints. The display of constraints can be turned off if it becomes difficult to see or select the sketch geometry.

A dimension is another type of constraint and is used to control the size and location of sketch geometry. See **Figure 7-28B**. When creating a sketch, it is usually desirable to draw each object to an approximate "rough" size without specifying the exact value. Then, dimensions can be added to constrain each object size to an exact value. There are different dimension tools used to place linear, angular, radius, and diameter dimensions.

In most cases, a sketch is fully constrained by adding the appropriate geometric constraints and dimensions before using the sketch to create a feature. This is referred to as "fully defining" a sketch. See **Figure 7-29**. When a sketch is fully defined, all objects in the sketch are constrained and cannot be freely moved or changed in size. In some programs, a message at the bottom of the screen provides an indication of whether the sketch is under-defined or fully defined. The display color of the geometry provides another indication of whether the geometry is fully defined. For example, unconstrained sketch geometry is displayed blue. When the geometry becomes fully defined, the color changes to black.

When a sketch contains constraints that are in conflict, the sketch status is overdefined. The sketch geometry changes to a different color, such as red. This indicates that certain constraints cannot be solved by the program and a modeling error exists.

Creating Features

When creating features, modeling mode is active in the program and feature modeling tools are available. As previously discussed, creating the base feature is usually the first step in constructing a part model. The base feature represents the most basic shape of

A

B
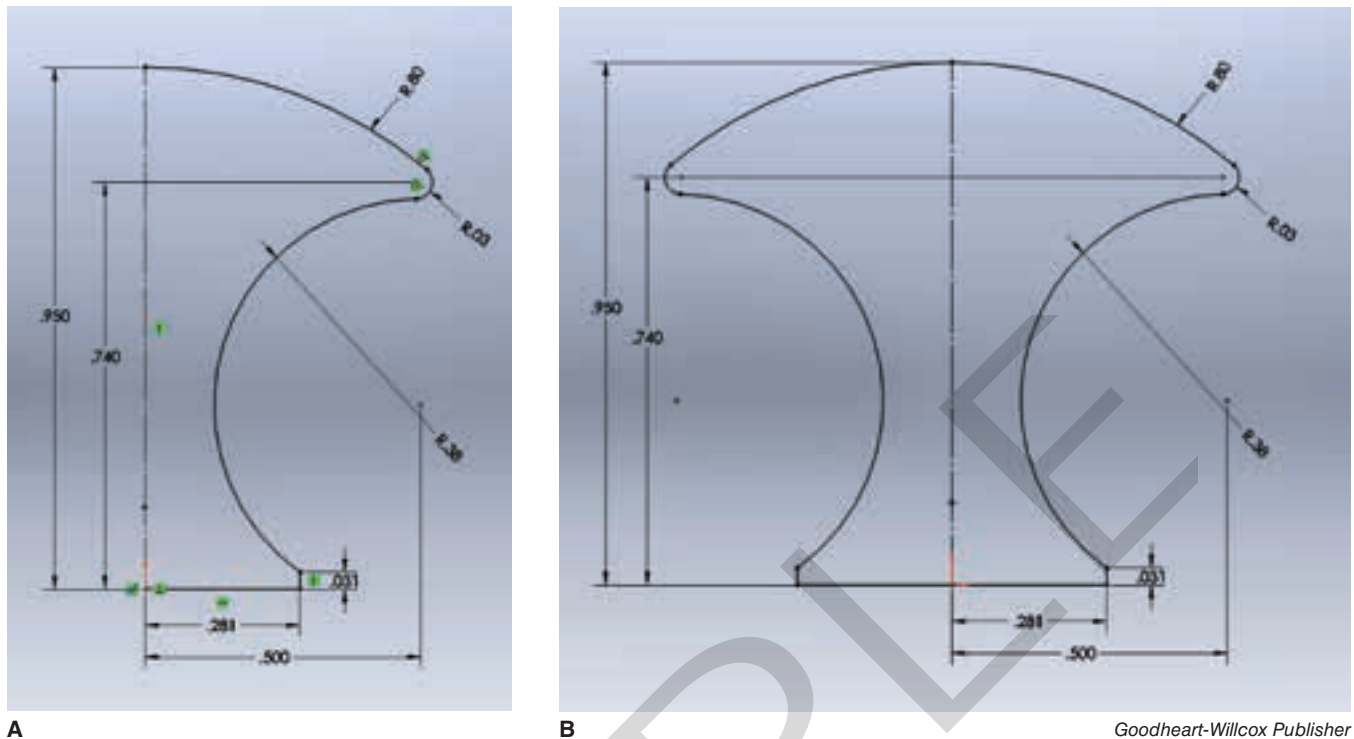
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Figure 7-29. Using geometric constraints and dimensions to fully define a sketch. A—All of the sketch geometry is displayed in black to indicate that the sketch is fully defined. The sketch geometry is fully constrained and cannot freely move. B—The sketch after mirroring. Notice that geometric relationships are maintained. The geometric constraints are hidden for display purposes only.

the part and is usually created from a sketch. A common way to create a base feature from a sketch is to extrude the sketch using the **Extrude** tool. Extruding a 2D sketch converts it to a 3D solid by "sweeping" the sketch along the Z axis to a given depth. See **Figure 7-30**. The resulting feature is parametric and can be edited to change the extrusion depth as needed. Another way to create a base feature from a sketch is to revolve the sketch geometry about an axis. In this case, the sketch is normally a 2D profile of a cylindrical object. The **Revolve** tool is used to create revolved features. Extruding and revolving operations are common in many CAD modeling programs and are discussed in Chapter 13.

Once the base feature is created, additional features, such as holes and bosses, are added to complete the part model. See **Figure 7-31**. The base feature is referred to as the parent feature. Each feature that is built onto a parent feature is called a child. A parent can exist without the child, but the child cannot exist without the parent feature. Therefore, when
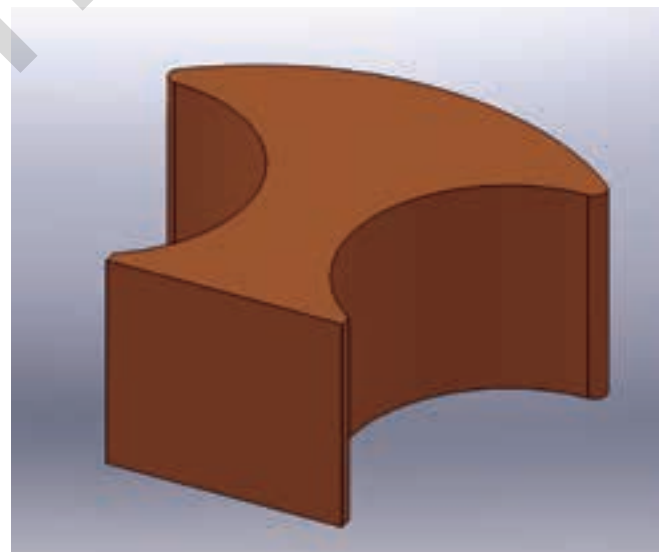
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Figure 7-30. The sketch from Figure 7-29 is extruded to create a base feature. The base feature forms the basic shape of a drawer knob for a desk.

modeling the part, it is important to start with the most fundamental shape and add features in a sequence that reflects the function of the part. Common types of features include

A

B

C
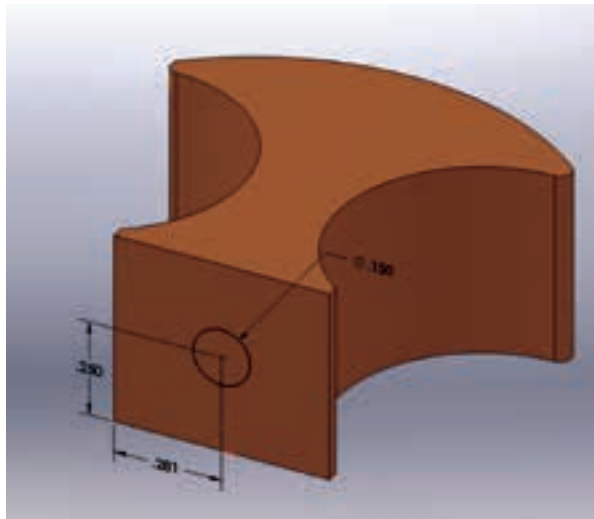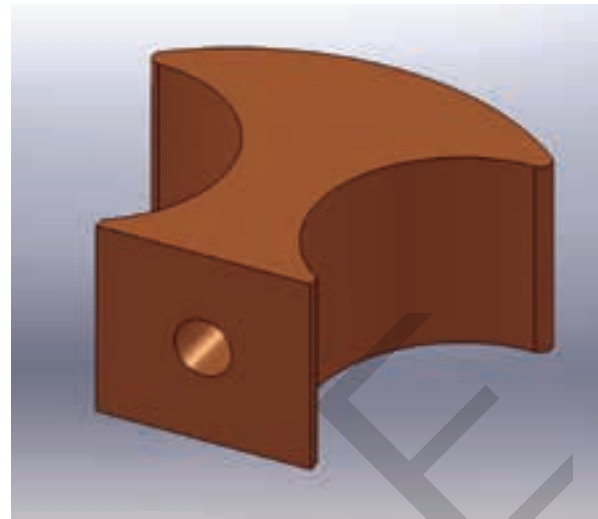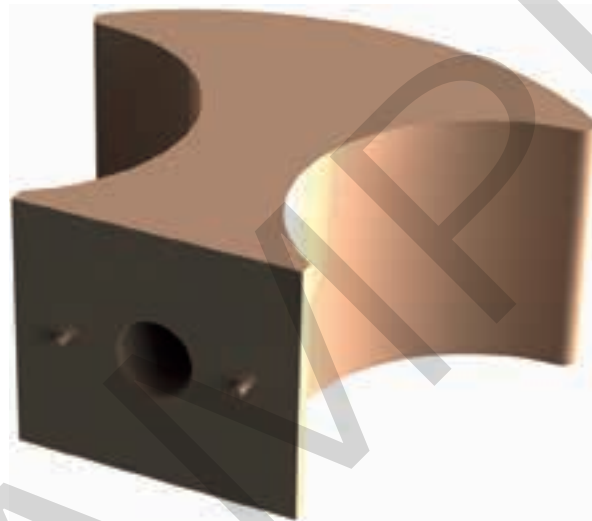
Figure 7-31. Creating additional features to complete the drawer knob. A—A sketch is made on the bottom surface to create the hole cutout. B—The part after using the sketch to create the hole cutout. C—A rendering of the completed drawer knob after adding two locating pins.

"cutouts" such as holes or slots. There are also more advanced features, such as rib features and shell features. Another way to model additional features is to create linear, circular, or mirror patterns of existing features.

Often, the existing surfaces of a solid model are used as working planes for creating feature sketches. For example, the bottom surface of the drawer knob in **Figure 7-31** is used to create the sketch for the hole cutout. Sketches can also be created on one of the default sketch planes or on a user-created construction plane, typically referred to as a *reference plane*.

Features can be edited by modifying the feature dimensions, or parameters, associated with the corresponding feature. This is often sufficient for minor changes to a part. More complex edits can be made by modifying the sketch associated with the feature. When using this method, the entire part updates to reflect changes made to the sketch geometry.

Design Intent

Because of the different processes involved and the nature of feature-based modeling, design plays a key role in parametric modeling.

It is important to consider the design intent throughout the modeling process and plan each stage accordingly. For example, the sketch for a base feature should be made on the appropriate sketch plane. The selected sketch plane establishes the orientation of the part and helps determine how the part will fit with other parts. In addition, the origin of the sketch should be planned at a location that facilitates assembly with other parts. Referring to the sketch of the drawer knob shown in **Figure 7-29**, the top plane is used for the sketch plane of the base feature, as the top view shows the most basic profile. The origin is located at the midpoint of the horizontal line along the base, which is sufficient for this part.

Other important design factors to consider include the application of geometric constraints and dimensions. The way in which a part is constrained controls how it will behave when edited. For example, it may be necessary to edit a part in order to create a slightly different part. Identifying how the part geometry will change when edited helps determine which constraints to apply and helps the designer anticipate how to make future edits in the most efficient manner. Dimensions should also be planned to reflect the design intent. In some cases, it may be appropriate to constrain dimensional values using equations. An equation is used to control a dimension using the value of another dimension. For example, an equation can be used to constrain the length of a part so that it is always equal to twice the width. This association is maintained when the part is edited. In similar fashion, a hole diameter can be constrained with an equation so that it remains proportional to the overall length of the part when the length is changed.

When designing a part, it is often helpful to plan ahead and identify the most appropriate modeling tools to use to create features. For example, if a part is symmetrical, features can be patterned or mirrored to save time. If a feature is to be created by revolving a sketch, it is important to identify the correct profile and determine where the revolution axis will be positioned.

Careful planning in the early stages of modeling helps maintain design intent in later stages, such as when assembling parts into an assembly model. Assembly modeling is discussed next. Design intent also applies to a variety of other design factors, such as material selection, how the part or assembly will be manufactured, and maintainability of the design.

Assembly Modeling

An assembly model consists of two or more individual parts or subassemblies. A *subassembly* is a smaller unit of parts that must be assembled together before being added to a larger assembly. See **Figure 7-32**. For example, an automobile engine must be assembled before it is attached to the frame. In assembly modeling, individual parts are assembled by precisely locating and constraining the parts. The assembly is saved as a separate file from the part files. The part files are linked to the assembly file so that any edits made to a part file are reflected in the assembly file.

There are three different approaches to creating an assembly model. The first approach is referred to as *bottom-up modeling*. In this approach, each part is created individually. Then, the parts are combined by inserting them into an assembly (or into several subassemblies and a top-level assembly). The second approach is referred to as *top-down modeling*. In this approach, the assembly file is created first and

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Figure 7-32. A subassembly consists of two or more parts. This subassembly is made up of two parts, a screw body and a pin, and is used in an assembly of a C-clamp.

each part is modeled within the assembly file until the assembly is complete. Each part is still saved as a separate file. This technique is useful for larger assembly models where it helps to use the assembly geometry for locating parts accurately. The third approach to assembly modeling involves a combination of bottom-up and top-down techniques. This approach can be used when most of the parts have been constructed independently and additional parts need to be created as the assembly is being built.

Adding Assembly Constraints

As parts are added into an assembly, they are positioned accurately at the appropriate locations. Then, constraints are added to the parts. Constraining the parts is required to control the *degrees of freedom (DOF)* of each part. Each part has six degrees of freedom. There are three degrees of linear movement along the X, Y, and Z axes and three degrees of rotary motion about the X, Y, and Z axes. When a part is fully constrained, all degrees of freedom are removed. By adding the appropriate constraints, it is possible to accurately control the functional motion of parts in the assembly. For example, a grinding wheel can be constrained so that it is allowed to rotate about the appropriate axis, but all other degrees of freedom are removed.

Assembly constraints are commonly referred to as *mates* because they are used to align or "mate" parts together. They can be used to align parts face-to-face, along the same center axis, or at an angle. Assembly constraints are similar to geometric constraints used in sketches. For example, a perpendicular assembly constraint can be used to constrain a face of a part so that it remains perpendicular to the face of another part. Other types of assembly constraints include the parallel, tangent, concentric, coincident, distance, and angle constraints.

Figure 7-33 shows a mechanical assembly of a C-clamp. The components of the assembly include the clamp body, the screw subassembly shown in **Figure 7-32**, and a cap at the end of the screw subassembly. The clamp body is the first component placed in the assembly and
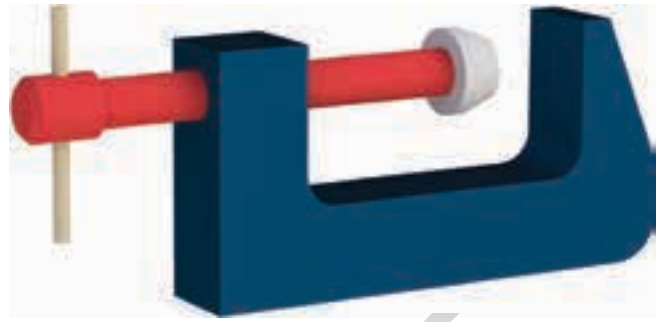
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Figure 7-33. A C-clamp assembly made up of a clamp body, a screw subassembly, and an end cap. Each part is positioned accurately and constrained to remove the appropriate degrees of freedom.

is fully constrained (all degrees of freedom are removed). The screw subassembly is assembled prior to being mated to the clamp body. The screw subassembly is constrained so that it remains concentric to the hole in the clamp body, but it is free to rotate about its axis and translate (move) through the hole. The cap is mated to the end of the screw subassembly with concentric and coincident constraints, but it is free to translate (move) through the hole with the screw subassembly.

Drawing Documentation

Part and assembly drawings are created by inserting drawing views generated by the software. Parametric modeling programs typically provide tools for generating common types of drawing views, such as orthographic views. Drawings can be created at any time during the modeling process. Typically, drawings are created from parts, subassemblies, and the top-level assembly for use in manufacturing. As previously discussed, drawings are created as separate drawing files. A drawing file is linked to the corresponding part or assembly file and updates automatically if the part or assembly is edited.

Creating Part and Assembly Drawings

In order to create a part or assembly drawing, some setup steps are required. It is often useful to start a new drawing file using a template. A drawing template contains user-defined

settings and a standard layout format, including a title block and border. Most programs provide predefined templates, but custom templates can be created for specific uses. For example, it may be necessary to create custom templates for different drafting standards or company practices. The drafting standard specified for the drawing file determines the type of projection used for orthographic views. Orthographic views can be projected using third-angle projection or first-angle projection, as discussed in Chapter 9. Third-angle projection is the preferred method in the United States.

Once the appropriate layout format and settings are established, drawing views can be inserted into the drawing file. Most programs require you to open the part or assembly file in order to lay out the corresponding views. Once a file is opened, you can select a specific view from a list of views associated with the part or assembly. In most cases, the first view inserted is the *base view*. This is often a principal orthographic view, such as the front or top view. Then, other views are projected from the base view. Views that are projected from the base view are placed in an orthographic alignment by default and are constrained to the position of the base view. See **Figure 7-34**.

In addition to orthographic views, other types of views can be projected. For example, section, auxiliary, detail, and isometric views can be created using the appropriate tools. These types of views are discussed in later chapters. Assembly drawings often include one or more exploded views, which display the assembly parts "exploded" to show how they fit together. See **Figure 7-35**.

Once drawing views are inserted, they can be dimensioned. There are typically two methods for adding dimensions to a drawing. One method is to add *model dimensions*, which are automatically generated by the software. Model dimensions are parametric and are known as *driving dimensions* because they drive the model geometry. If a model dimension is edited, for example, the model geometry is updated in both the part and drawing files. Another method used to dimension a drawing

is to add *reference dimensions*, also referred to as *drawing dimensions*. Reference dimensions are created manually by the user and are not parametric. Instead, they are driven by the model geometry. A reference dimension is updated when the model is edited, but changing a reference dimension in a drawing does not change the model geometry. When adding dimensions, it is common to use a combination of model and reference dimensions. Model dimensions generated by the software do not typically provide a complete description of the model. In addition, model dimensions often need to be moved or hidden in order to conform with standard dimensioning practices.

Notes and callouts can be added to the drawing to provide additional information. There are also tools for adding other items to a drawing, such as center marks and centerlines, geometric dimensioning and tolerancing (GD&T) symbols, surface texture symbols, and a revision block.

When working with assembly drawings, it is common to add a *bill of materials*. This is a table in the drawing listing all parts in the assembly. The bill of materials includes the name and number of each part, along with a part description, the quantity required, the part material, and any additional required information, such as the manufacturer name and pricing. When a bill of materials is inserted, the information is entered into the table automatically by the software. The bill of materials is linked to the model by default and updates to reflect any changes to the assembly. For example, if a part is replaced or a part description is changed, the bill of materials is automatically updated.

External References

An external reference is a drawing, an image, or other file format that can be referenced into a host drawing. The external reference will remain independent of the host drawing and is updated as changes are made to the referenced file. The data from the external reference appears in the host drawing for reference only.
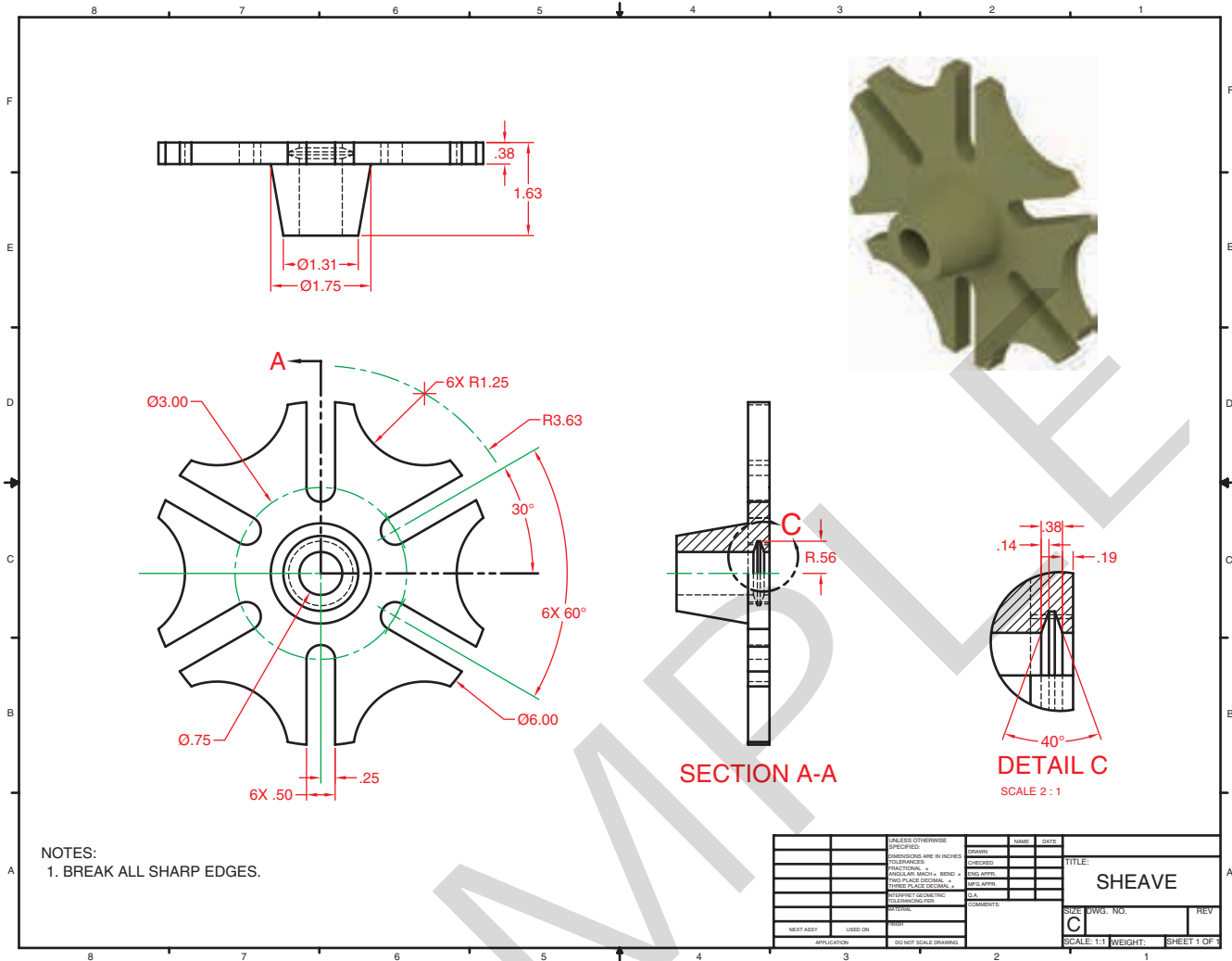
Figure 7-34. Drawing files are created by inserting drawing views into a layout. The drawing views are placed in an orthographic alignment by default. After placing the base view, additional views can be projected. The views in this drawing include two orthographic views (a front view and a top view), as well as a section view, detail view, and isometric view (shown in color). The section view and isometric view were projected from the front view. The detail view was projected from the section view. Dimensions and notes were added to complete the layout.

Advanced CAD Systems

Some CAD modeling software is specifically designed for linking mechanical designs to manufacturing. There are also advanced CAD modeling programs used to create photorealistic renderings and animations. The following sections discuss some of the features provided by advanced CAD software packages and systems.

CAD/CAM

Computer-aided design/computer-aided manufacturing (CAD/CAM) combines CAD with automated manufacturing operations. In this type of manufacturing, computer numerical control (CNC) machines control the manufacturing processes. Computer data that is input to the machine is used to control the movement of machine tools. After a part is

Iaroslav Neliubov/Shutterstock.com

Figure 7-35. Exploded views are typically used in assembly drawings to show how the parts fit together.

drawn in the CAD program, the mathematical shape and size description of the part, or design data, is calculated by the program for manufacturing. The data is used to cut the material and make the part, **Figure 7-36**.

Advanced Rendering and Animation Programs

Most CAD modeling programs typically include rendering capability. In CAD terms, a *rendering* is a highly realistic representation of a model with lighting, shadows, and other visual effects applied. Renderings are common in mechanical and architectural drafting, because a mechanical assembly or an interior room can be shown in great detail to a potential client.

In high-end CAD applications, advanced rendering programs are used to create very realistic displays. See **Figure 7-37**. These programs provide lighting, materials, and environmental effects to greatly enhance the appearance of a model. A model set up for rendering in this manner is known as a *scene*. When the scene is rendered, material finishes and other effects are calculated based on the lighting used. Computer rendering programs require greater amounts of resources in order to perform the calculations needed. The original model may be created within the program, or it may be imported from a different modeling program with less powerful rendering capability.

Some modeling and rendering programs allow you to animate models so that they can be shown in actual operation. Models created in this manner are assigned movement parameters for frame-by-frame animations. Animated models are useful in the architectural field. For example, an architectural project can be presented with an animated tour through a building for a design proposal.

One of the more advanced applications of CAD technology is the use of computer-generated animation and imaging for special effects in films. Computer drawing and animation programs are also used for applications in the medical field. While originally conceived for engineering purposes, CAD-based graphics are now used in a variety of industries. Career opportunities in CAD drafting and design will continue to expand as the technology develops.
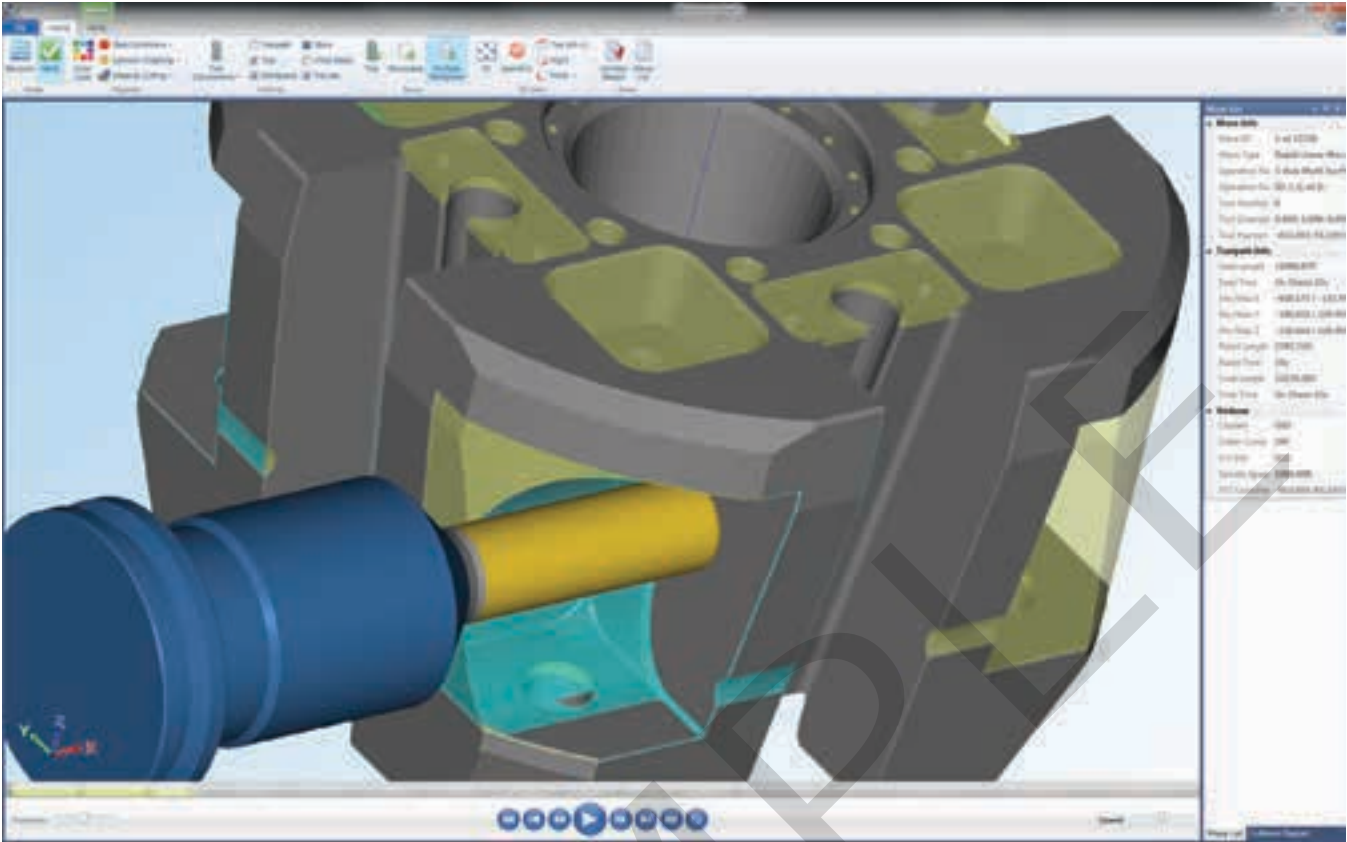
Figure 7-36. CAD/CAM programs use CAD-generated drawings to supply tool data for manufacturing processes.

Figure 7-37. Realistic scenes can be created in advanced rendering programs with special tools used to apply lighting and materials.

Summary

- Computer-aided drafting (CAD) software allows engineers, designers, and drafters to visualize parts in three dimensions without creating a physical model and eliminates the repetitive tasks associated with manual drafting.
- CAD programs have file management commands that provide functions for creating, opening, saving, and plotting files.
- CAD drawings are most commonly defined using the X, Y, and Z axes of the world coordinate system.
- The grid, snap, orthogonal mode, and direct distance entry functions simplify the process of specifying distances and locating objects.
- Layers, blocks, symbol libraries, and templates all save time and allow the drafter to focus on drawing the object.
- The **Line**, **Circle**, **Arc**, **Ellipse**, **Polygon**, and **Hatch** commands are used to draw the respective object.
- The **Move**, **Rotate**, **Scale**, **Erase**, **Trim**, and **Extend** commands are used to edit and modify objects.
- The **Copy**, **Array**, **Mirror**, **Fillet**, **Chamfer**, **Trim**, and **Extend** commands save the drafter time by automating common or repetitive tasks.
- CAD programs range from simple 2D drawing programs to advanced 3D solid modeling and surface modeling systems.
- Parametric modeling uses sketch tools to create the base feature, constraints to control geometric relationships, and feature creation commands like **Extrude** and **Revolve** to create part models.
- Multiple part models can be connected or fit together to form an assembly model.
- Computer-aided manufacturing (CAM) programs convert CAD models into instructions for computer numerical control (CNC) machines.
- Realistic renderings and animations can be created with advanced CAD systems.

Test Your Knowledge

Answer the following questions using the information provided in this chapter.

1. Drawings created in a CAD program are made up of line and arc objects called _____ objects with defined point coordinates in space.
2. _____ coordinates are located at a given distance and angle from another point.
3. Describe the difference between the world coordinate system and a user-defined coordinate system.
4. A network of uniformly spaced points used as a drawing aid to determine distances is known as what?
5. What are *layers* and what function do they serve in CAD drafting?
6. What is a *template*?
7. Circles are typically drawn by specifying a _____ and a radius or diameter.
8. What editing command is used to orient multiple copies of an object in a regular rectangular or polar pattern?
9. Describe how to create a round or fillet with the **Fillet** command.
10. What display command is used to create a magnified view by windowing a portion of a drawing?
11. Identify two ways to access the help system in a CAD program.

12. Explain a typical protocol for file saving and storage established by a CAD firm.
13. What is the difference between a solid model and a surface model?
14. Define *parametric modeling*.
15. In parametric modeling, the most basic feature or shape of a part is referred to as the _____ feature.
16. Explain the difference between *bottom-up modeling* and *top-down modeling*.
17. What is a *base view*?
18. Define *rendering*.

Applying Your Knowledge

1. Obtain samples of various types of drawings prepared using a computer. Prepare a bulletin board display with these drawings.
2. Visit an industrial drafting firm that uses a CAD system. Interview a computer workstation operator. How does the CAD system save time? How does it assist with repetitive work? How does it improve productivity? What did the system cost the business? Report to the class what you observed and learned.
3. Visit a computer store that sells CAD equipment. Request literature on the equipment sold. Prepare a bulletin board display with the material.

Communicating about Drafting

In small groups, discuss with your classmates—in basic, everyday language—your knowledge and awareness of CAD drawings. Conduct this discussion as though you had never read this chapter. Take notes on the observations expressed. Then review the points discussed, factoring in your new knowledge. Develop a summary of what you have learned about CAD and present it to the class, using the terms that you have learned in this chapter.

Drawing Problems

Draw the problems shown on the following pages. Follow the directions in each problem and use the dimensions provided. Dimensions are in inches unless otherwise indicated.
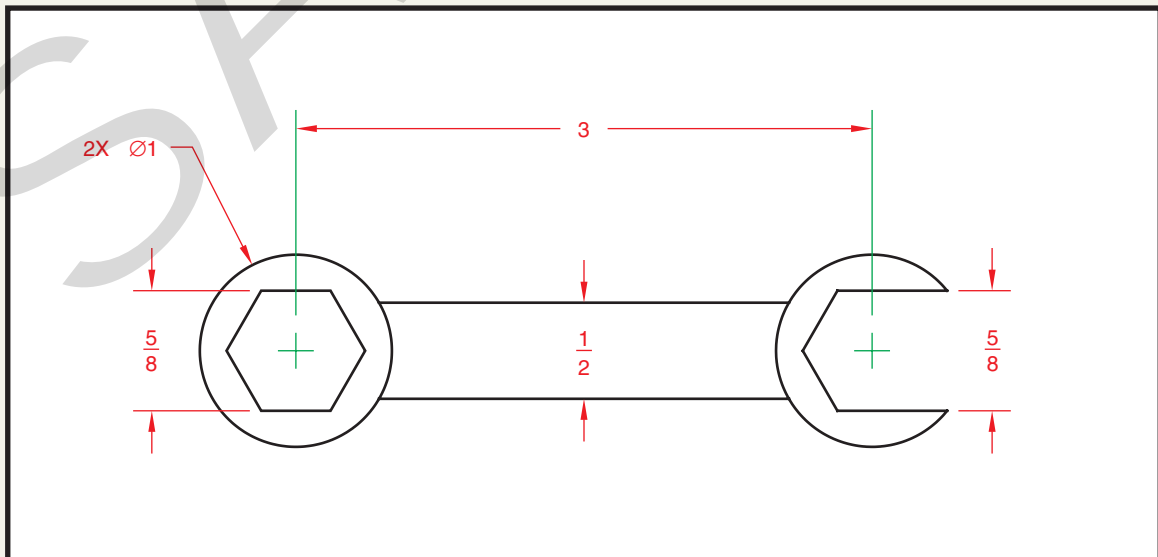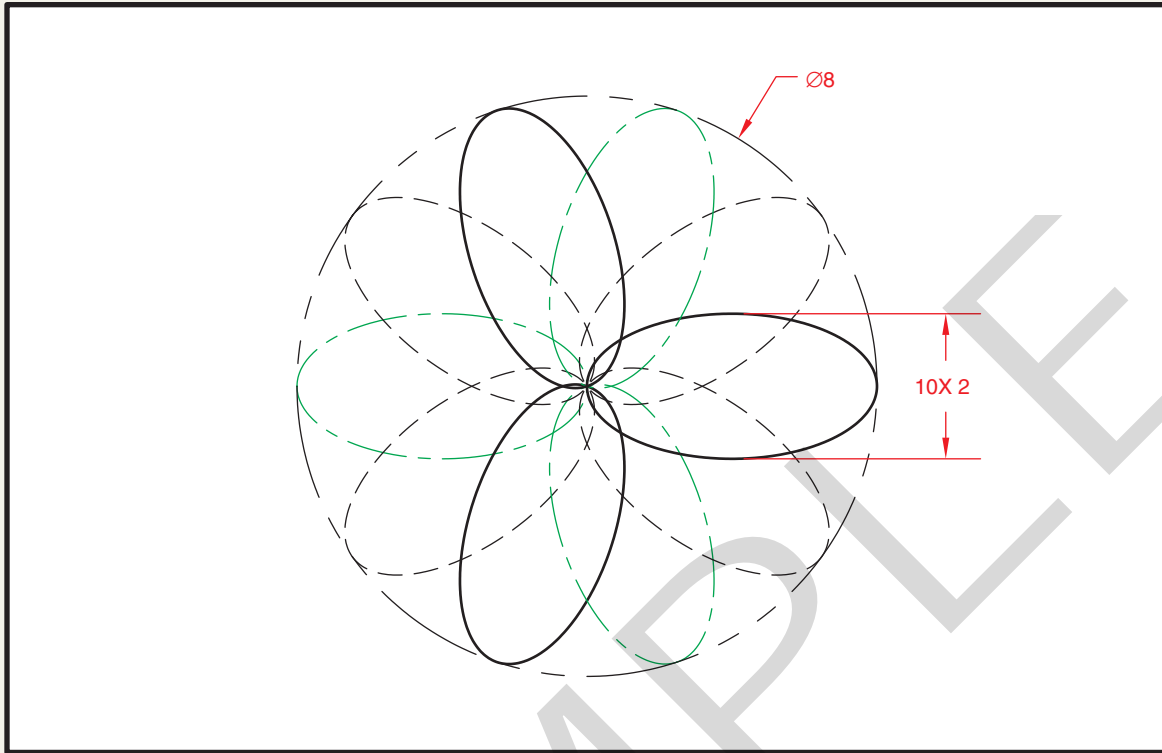
Drawing Problems
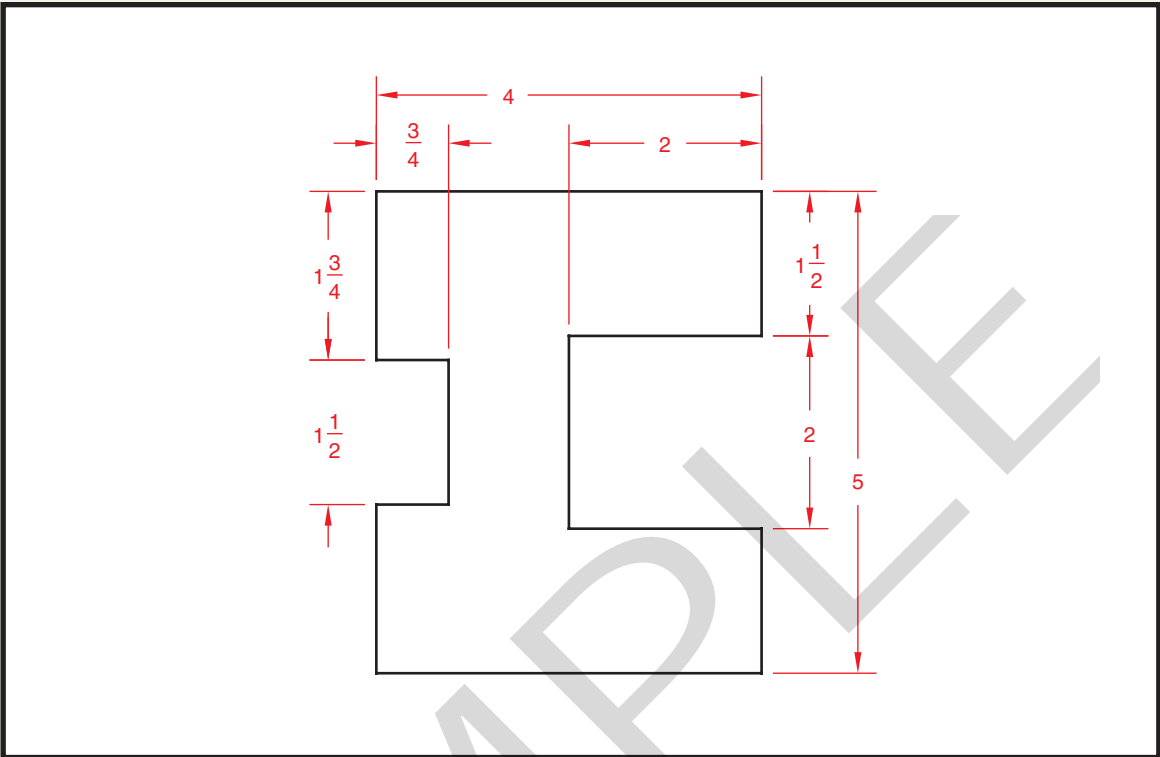
Problem 7-1. ★ Shim. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
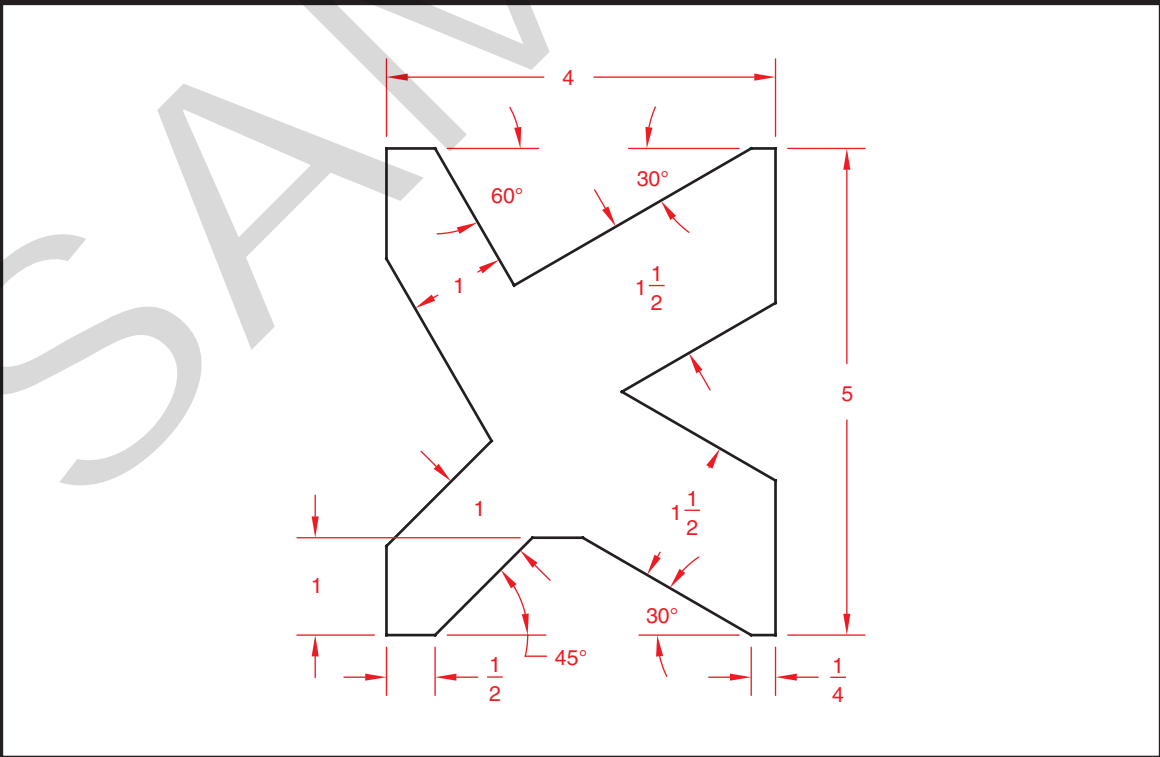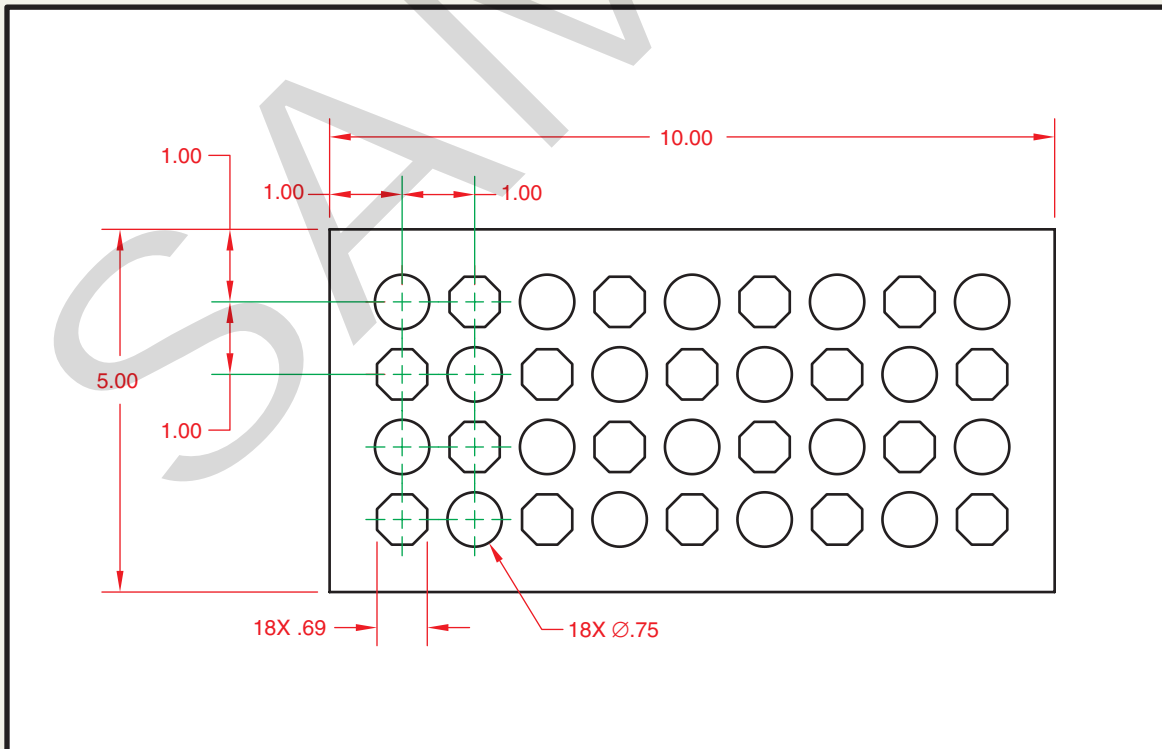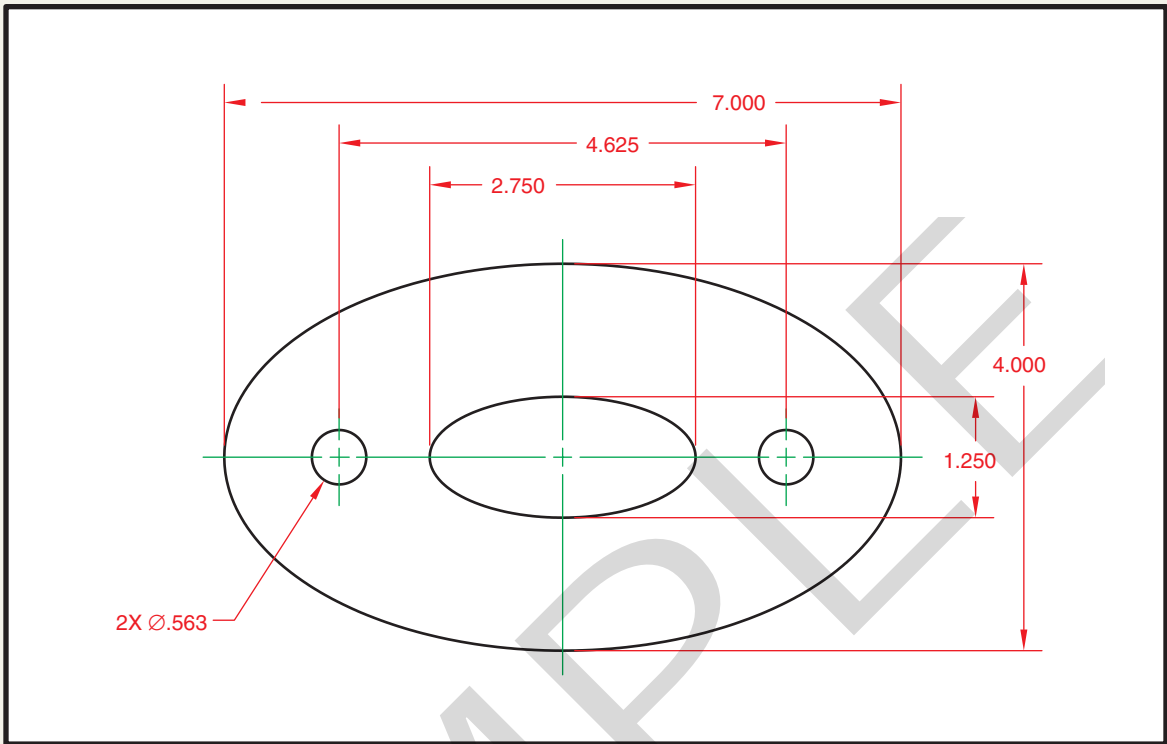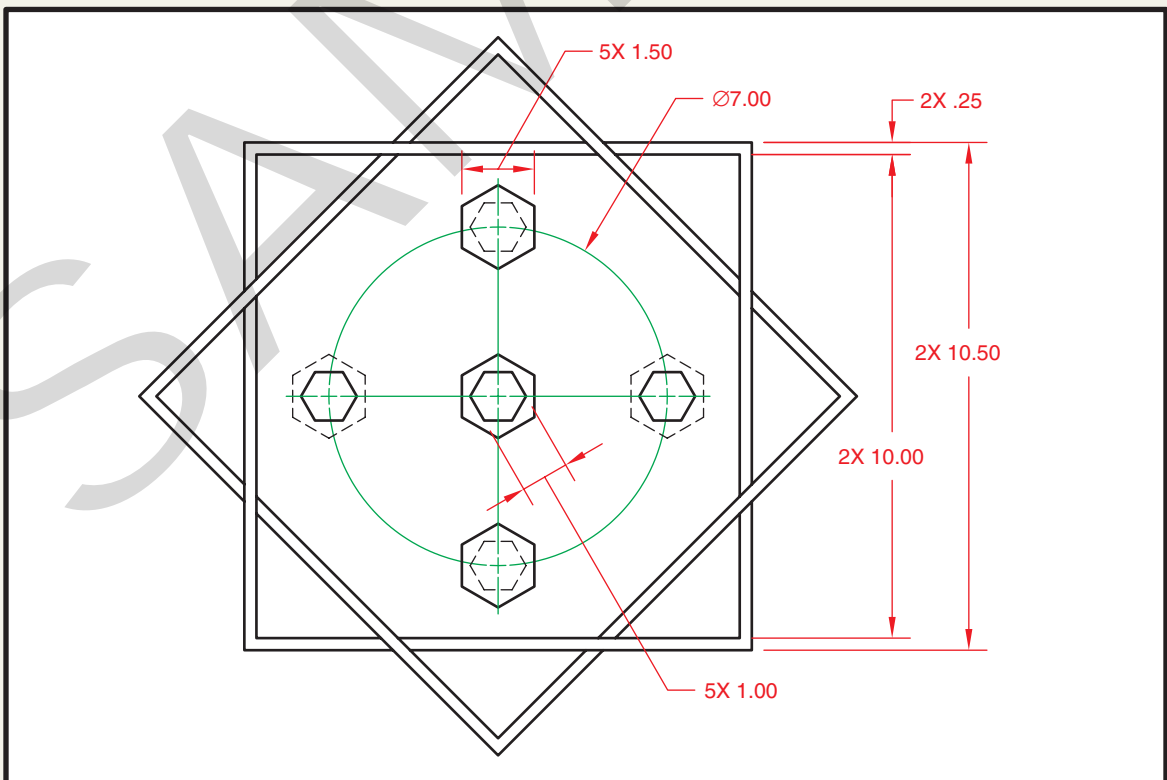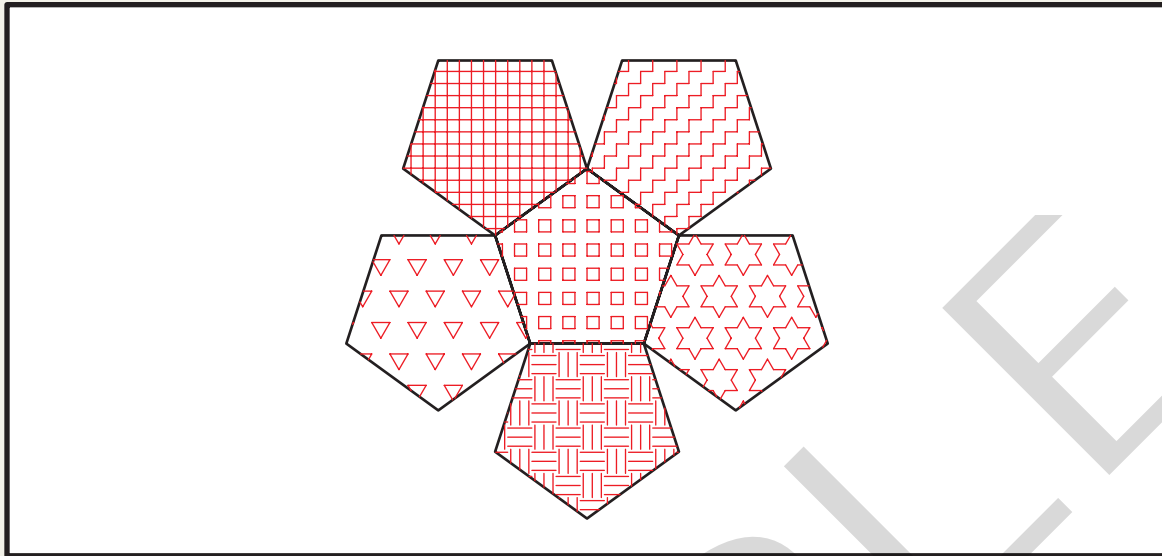

Problem 7-2. ★ Template. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
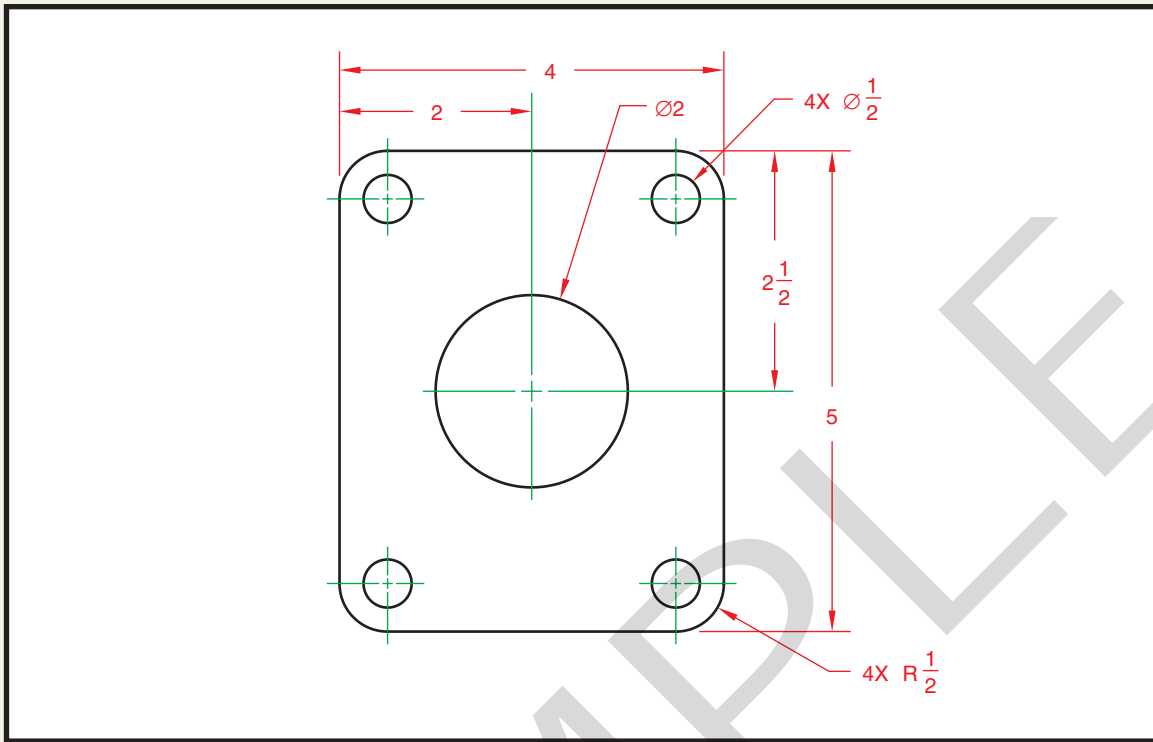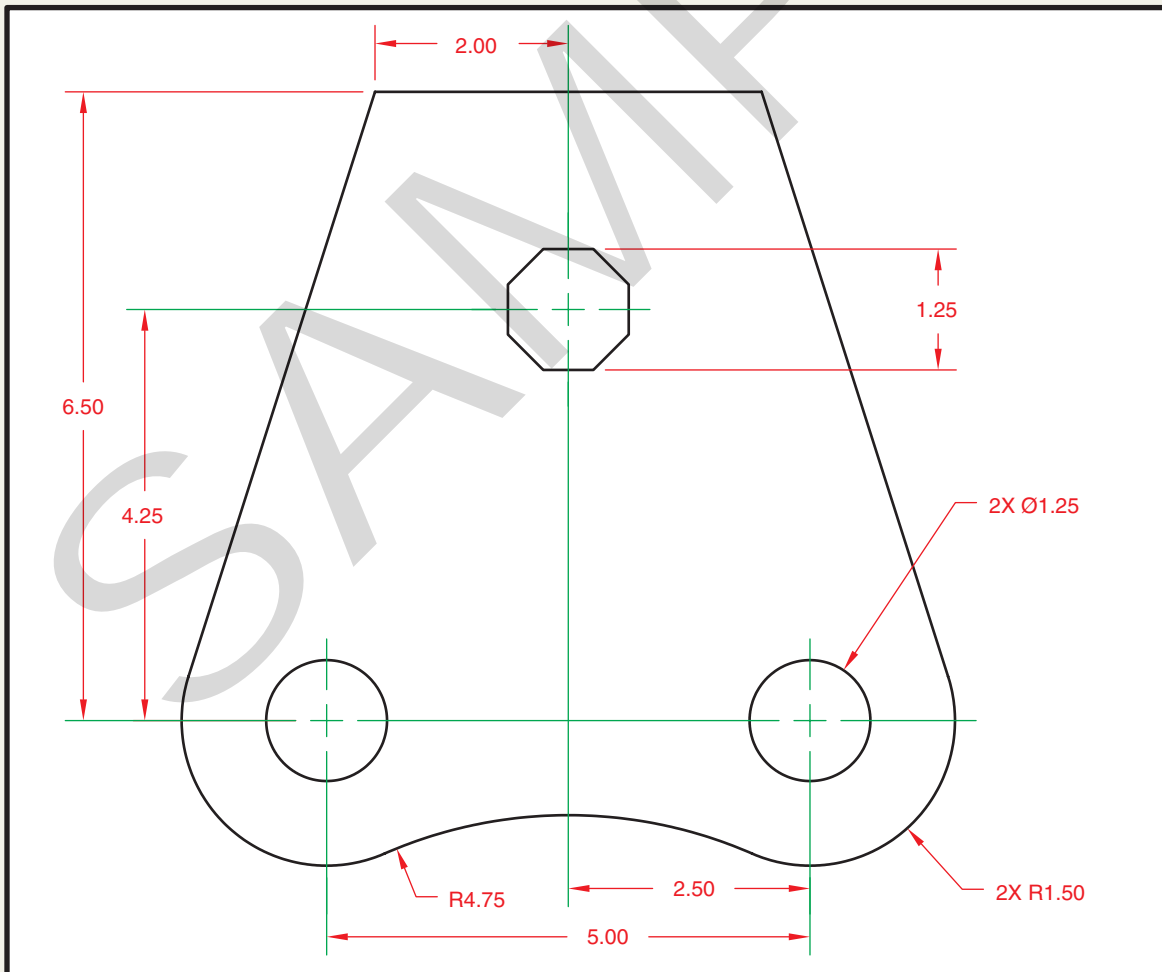
Problem 7-3. ★ Gasket. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
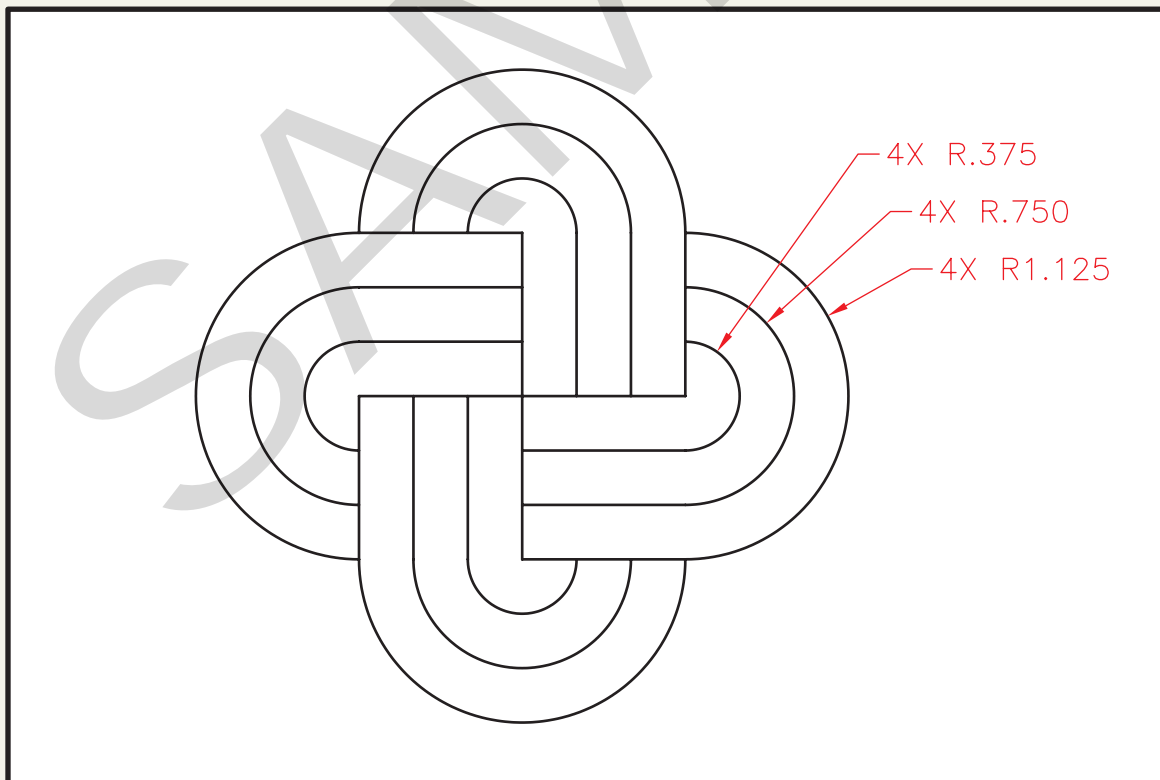
Problem 7-4. ★ Knot. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
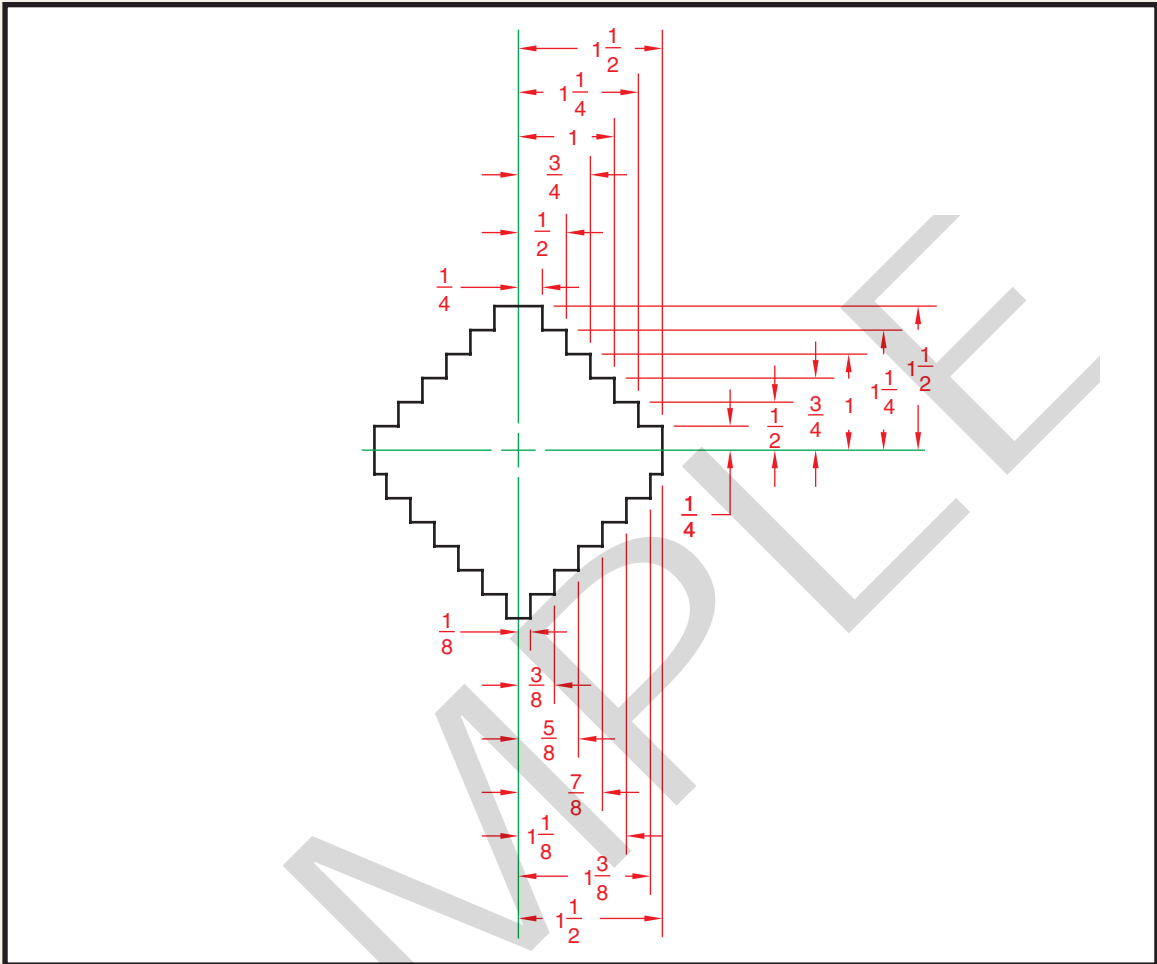
Problem 7-5. ★ Gage. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
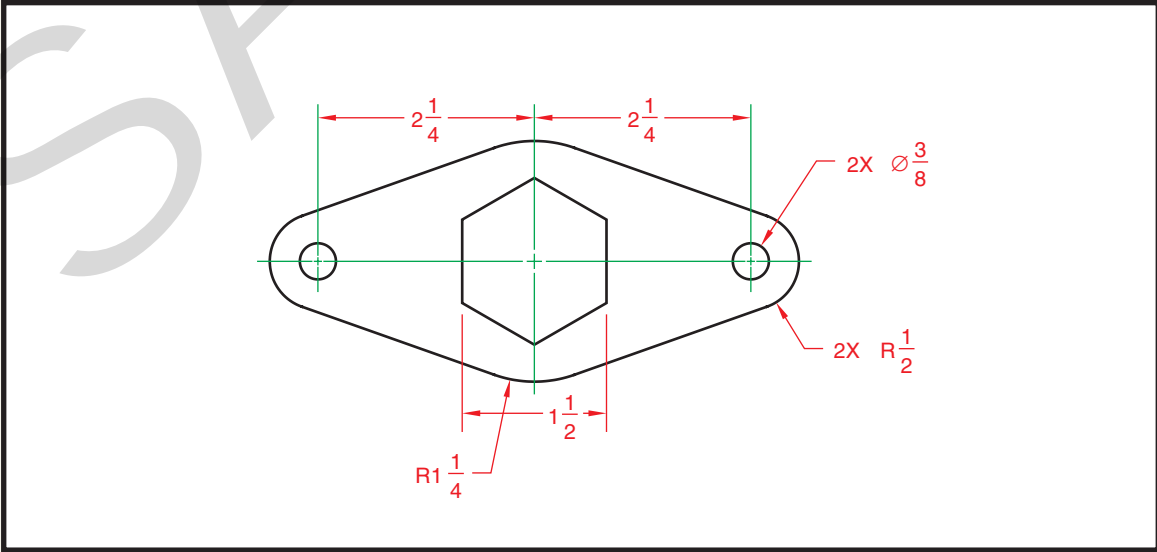
Problem 7-6. ★★ Gasket. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
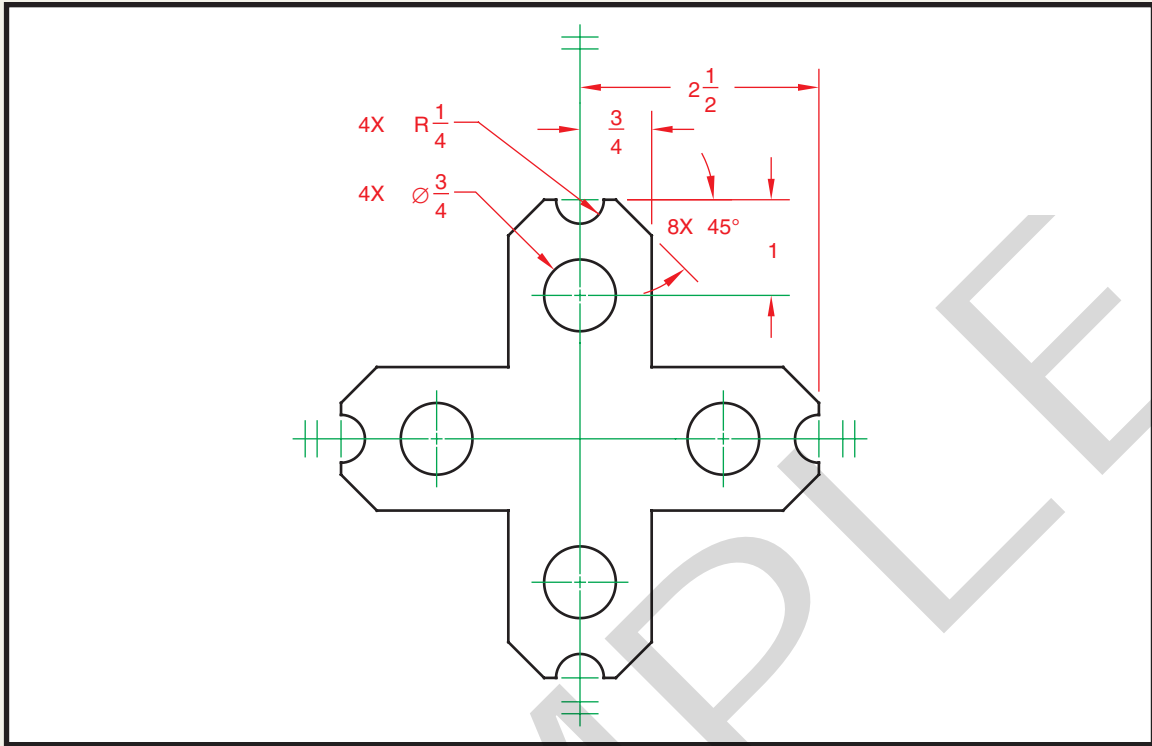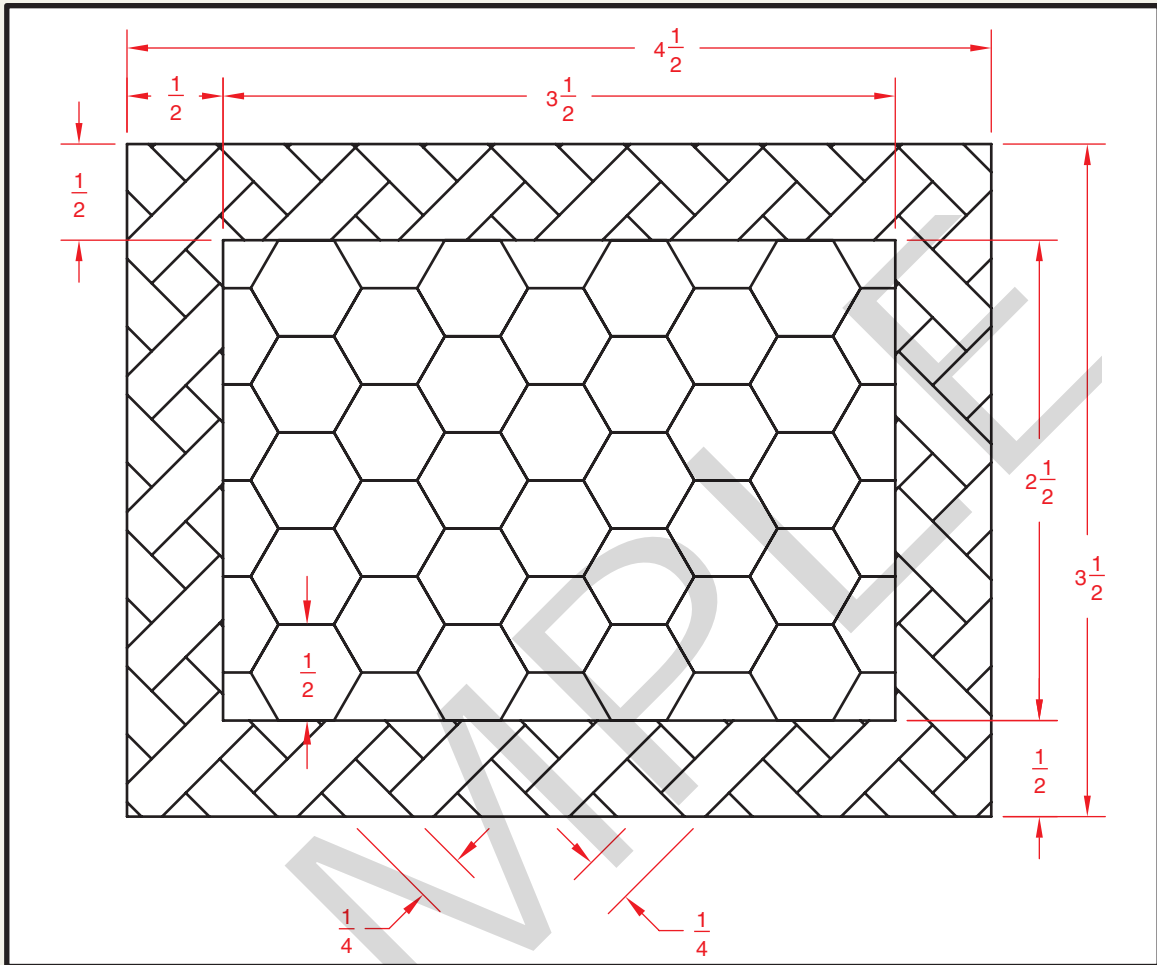
Problem 7-7. ★★ Guide. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
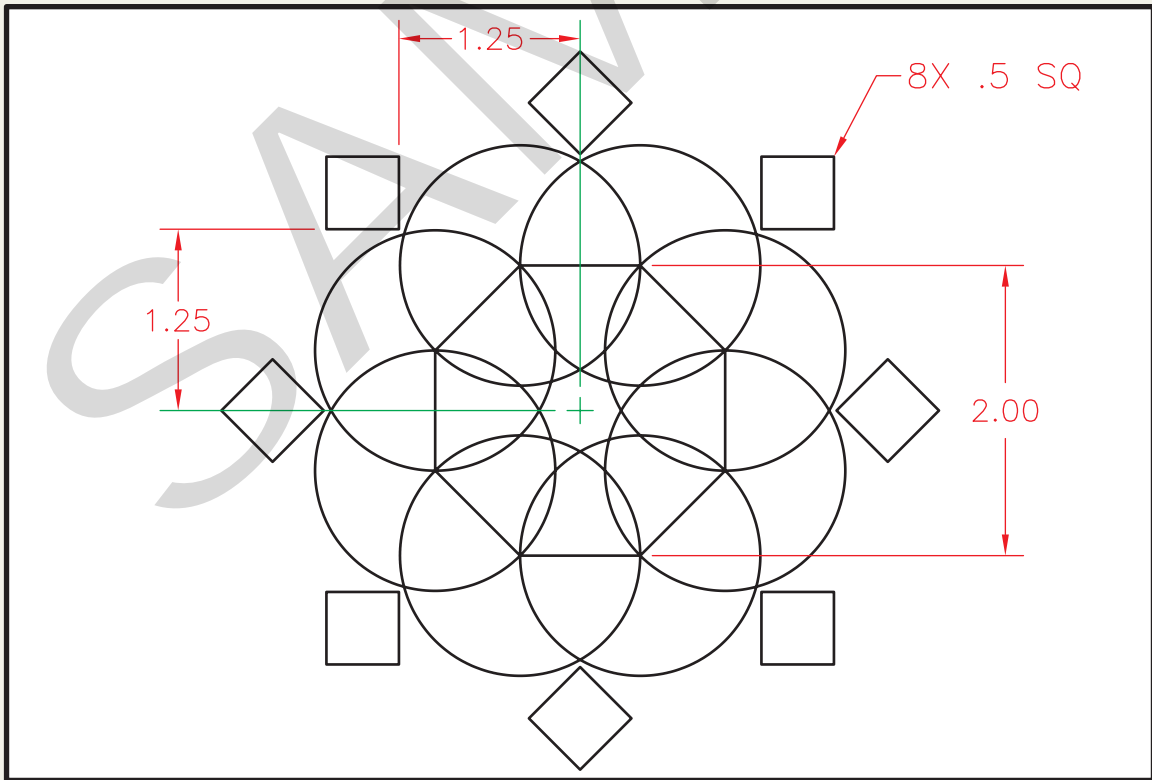
Problem 7-8. ★★ Geodesign. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
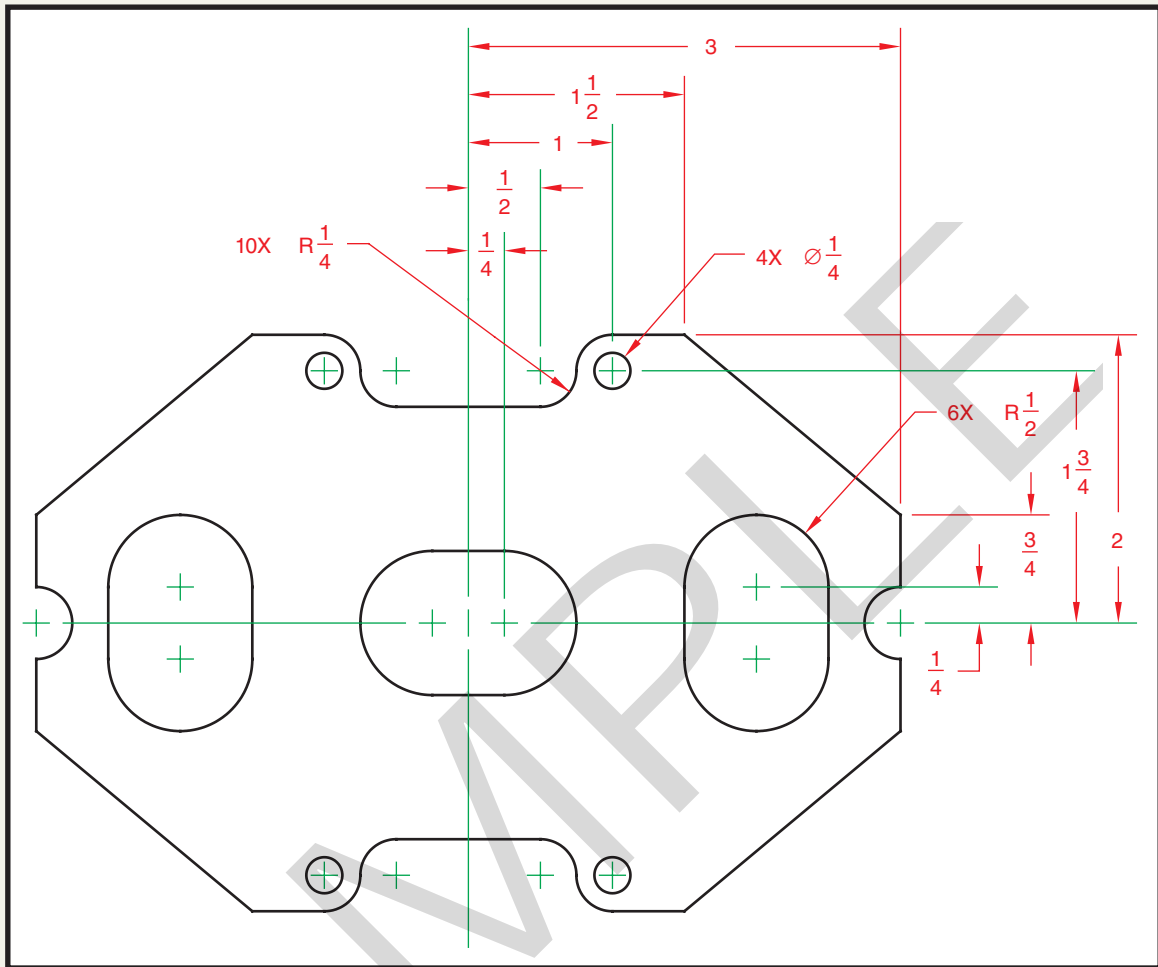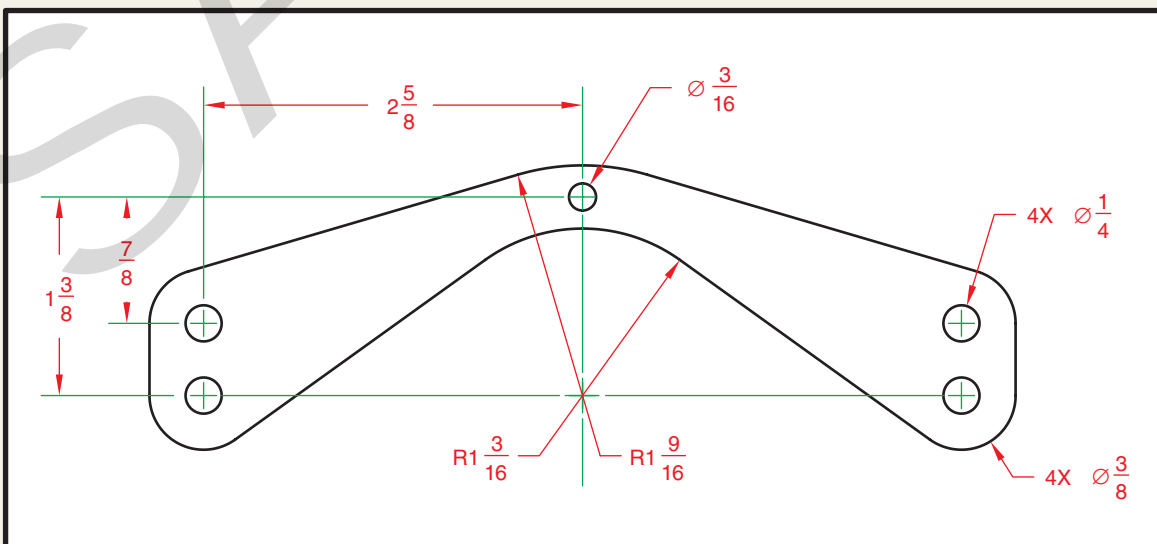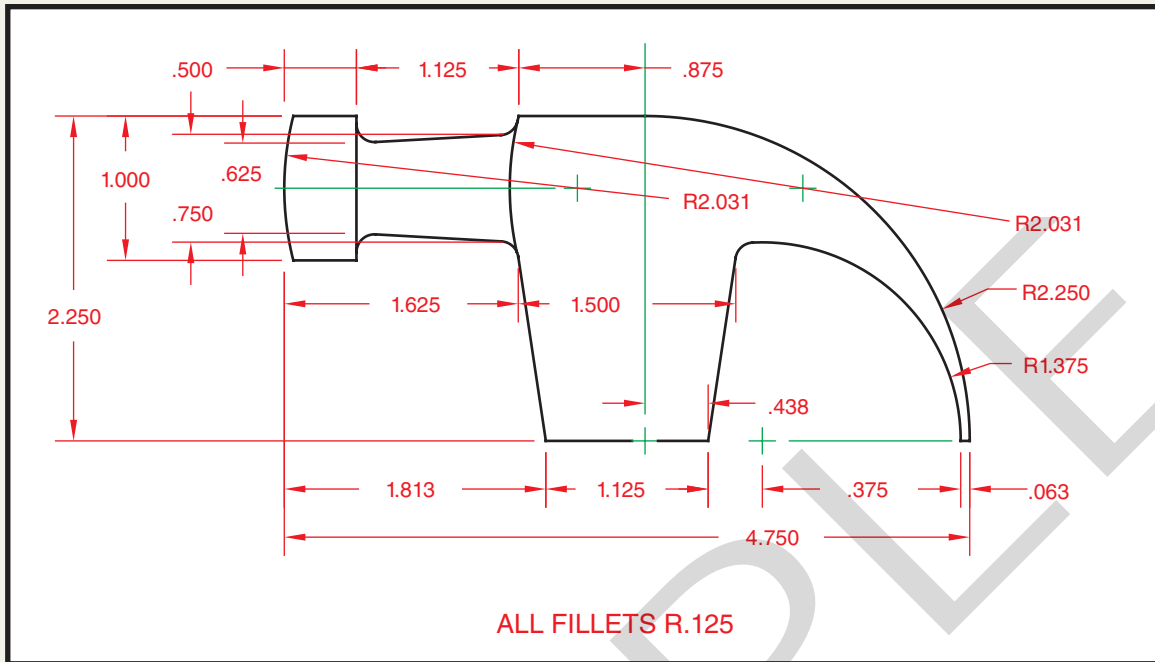
Problem 7-9. ★★ Spacer. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.

Problem 7-10. ★★ Wrench. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.

Problem 7-11. ★★ Spiral Ellipse. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.

Problem 7-12. ★★ Peg Board. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
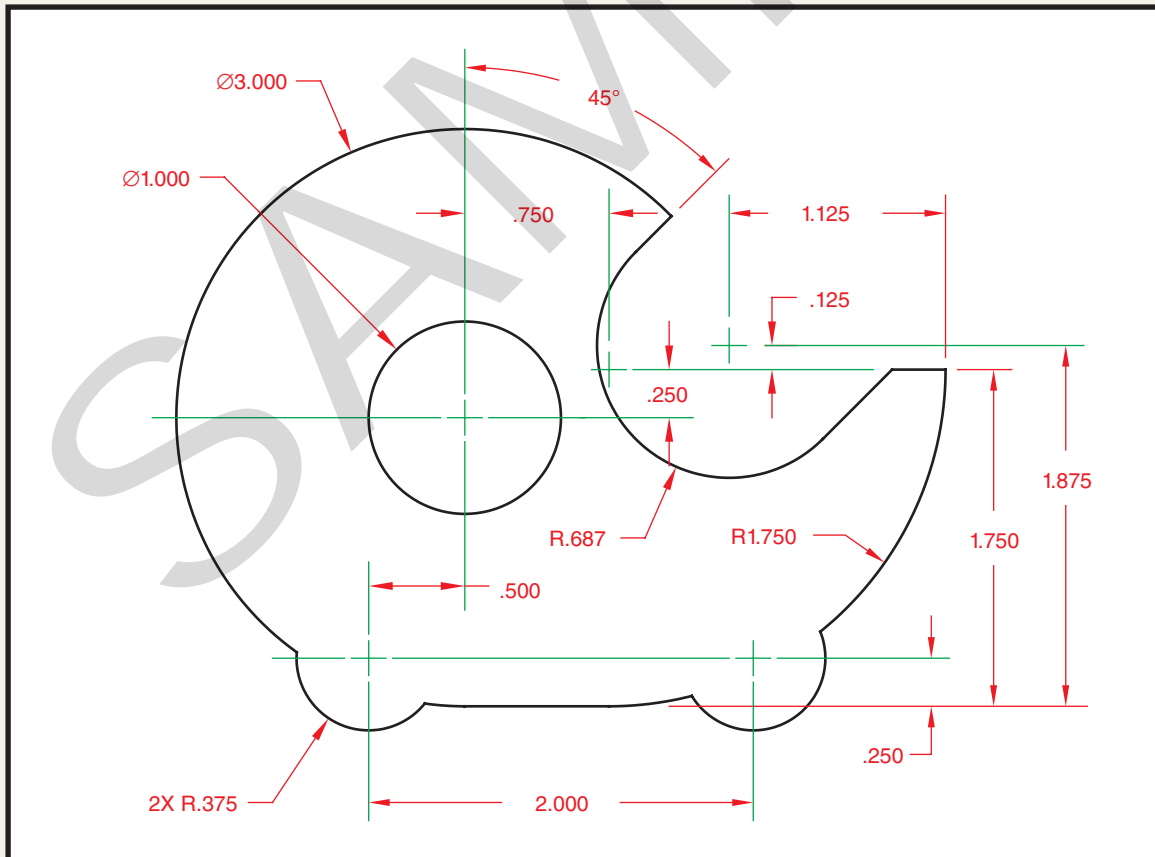
Problem 7-13. ★★ Filler Gasket. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.

Problem 7-14. ★★ Hexagon Frame. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
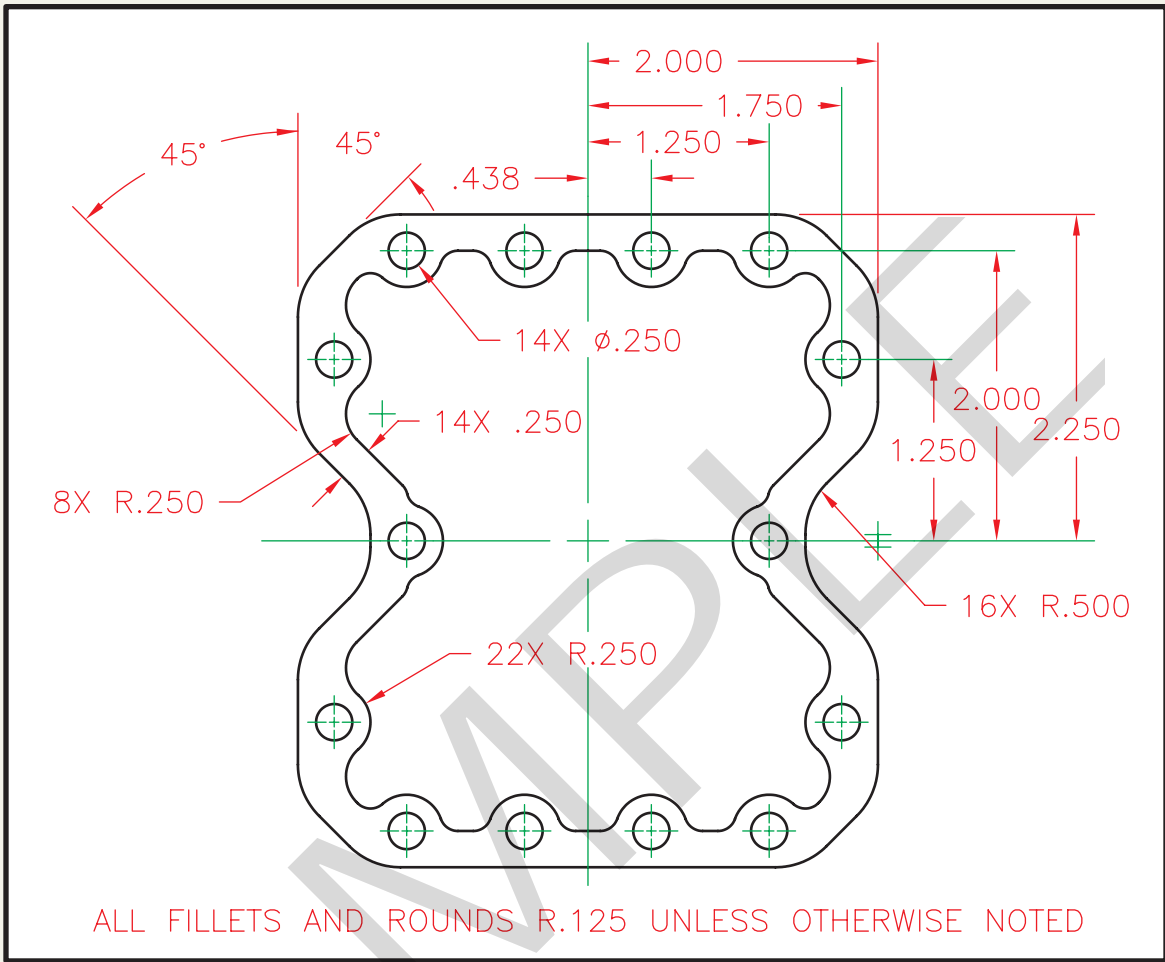
Problem 7-15. ★★ Hexagon Hatch. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Use the appropriate hatch pattern to hatch each hexagon as shown. Do not dimension.

Problem 7-16. ★★ Link Plate. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.

Problem 7-17. ★★★ Framed Mesh. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.

Problem 7-18. ★★★ Saddle Bracket. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
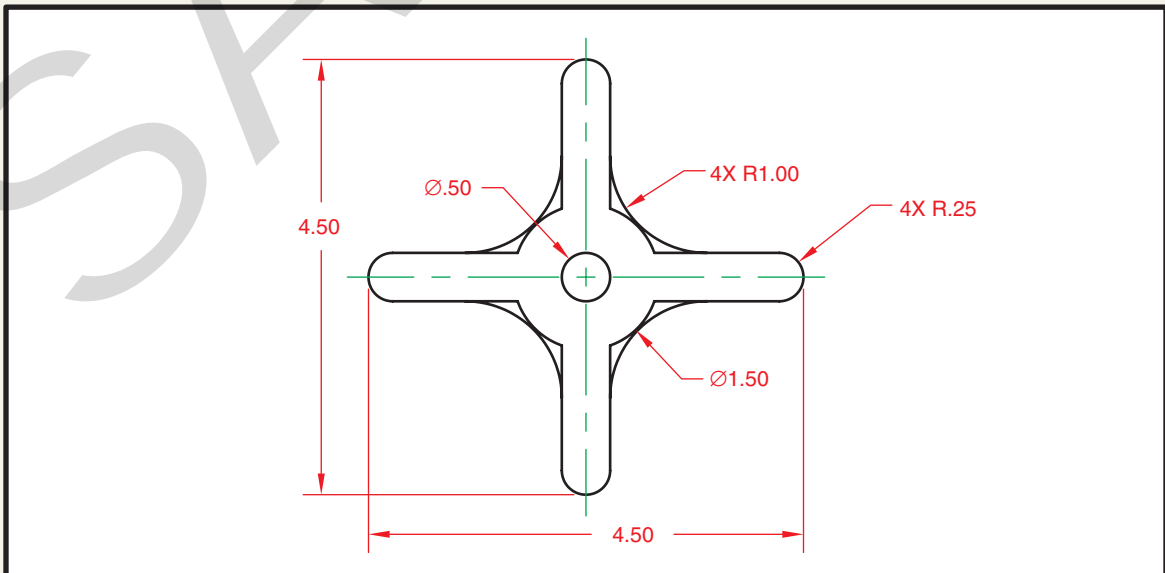
Problem 7-19. ★★★ Hammerhead. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
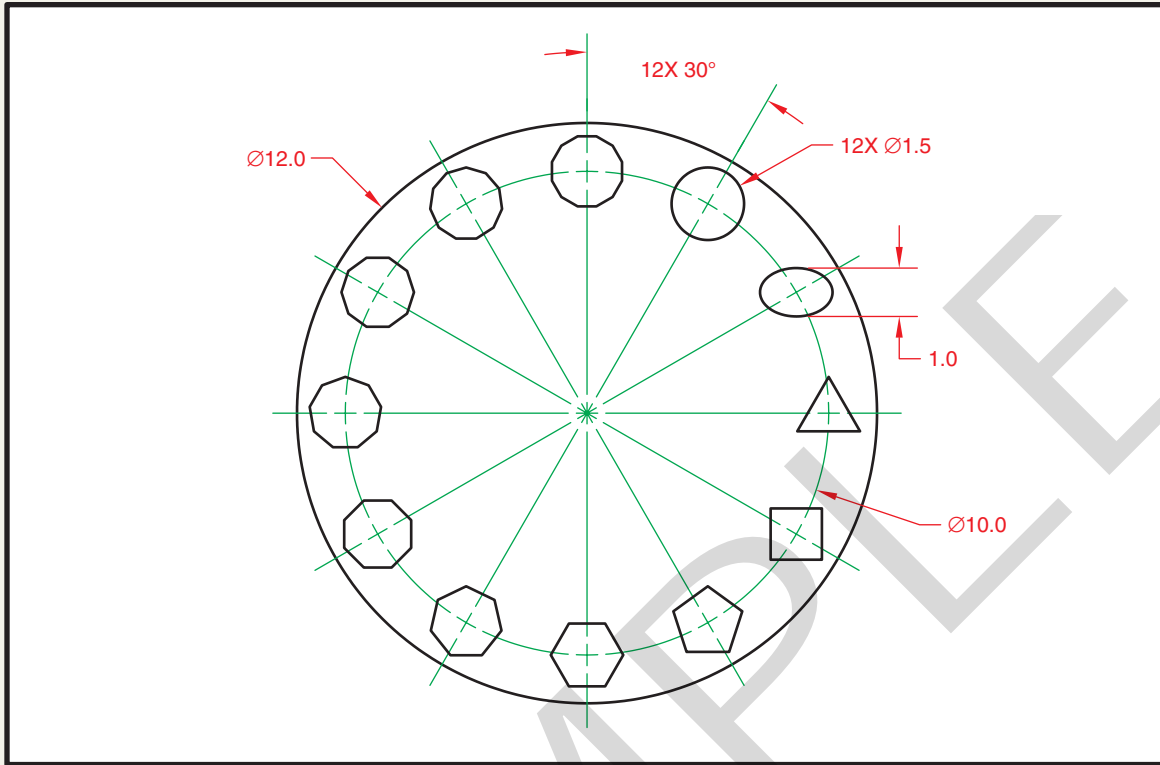

Problem 7-20. ★★★ Tape Dispenser. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
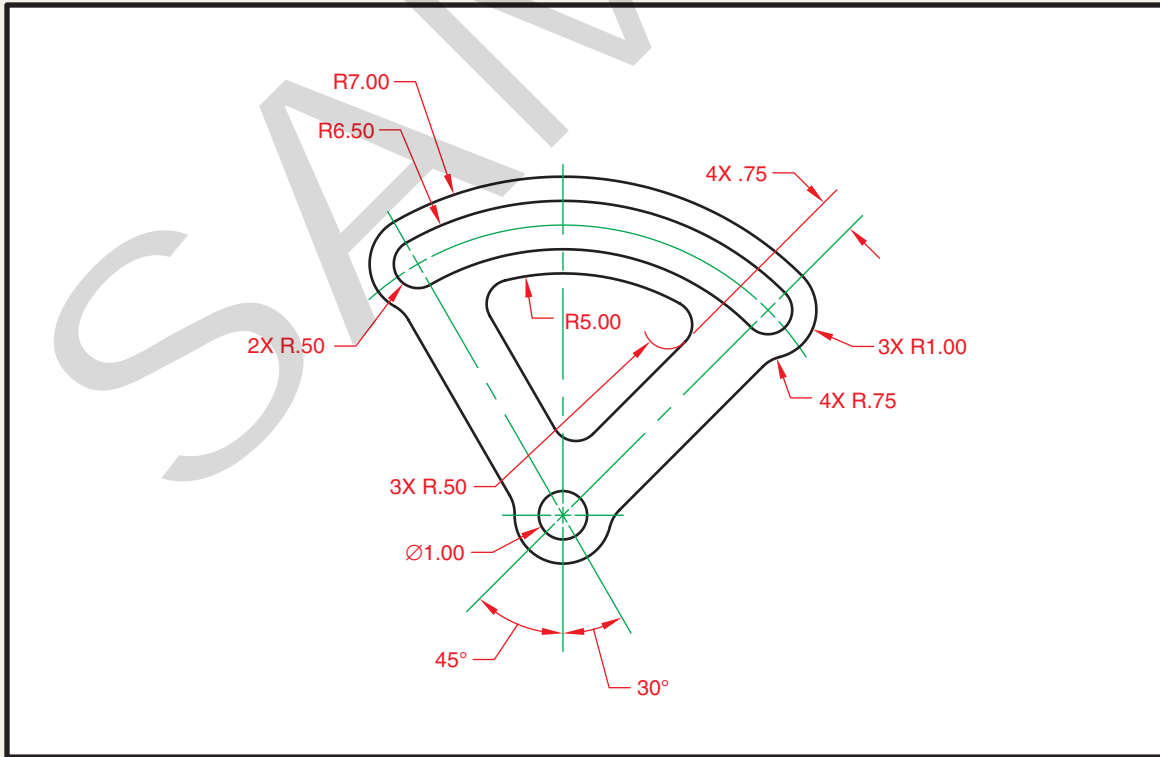
Problem 7-21. ★★★ Pan Gasket. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
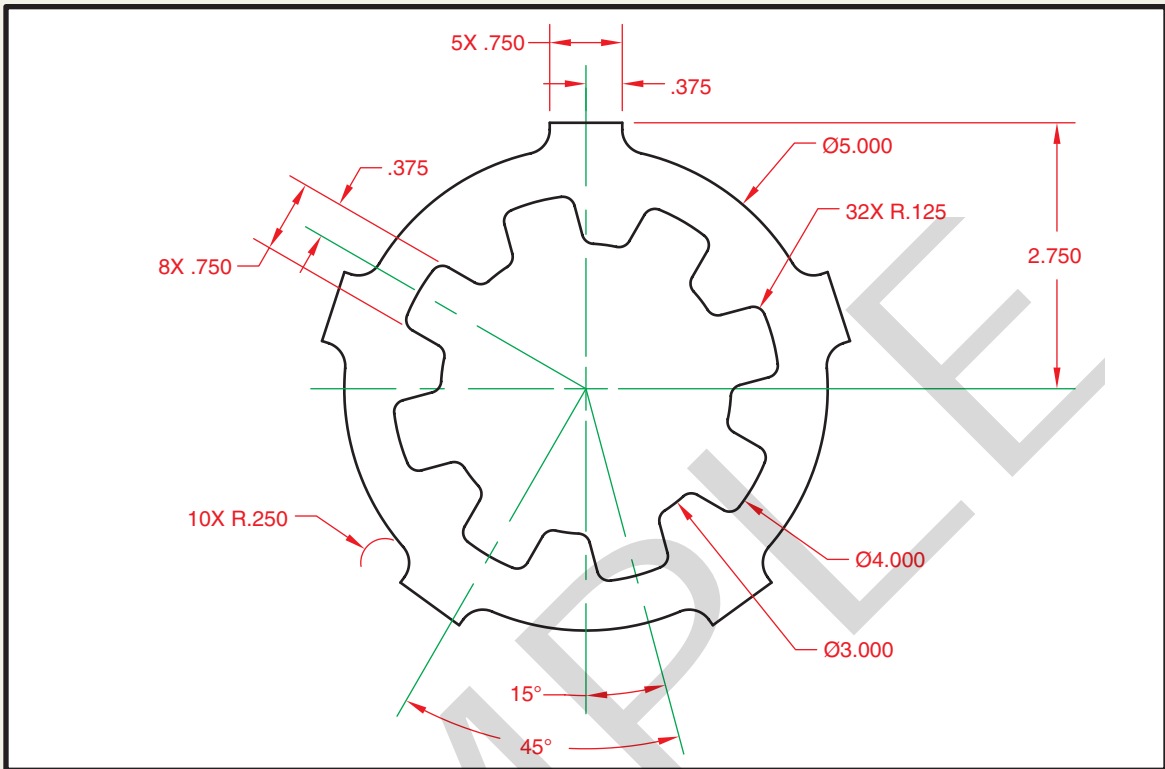

Problem 7-22. ★★★ Knob. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
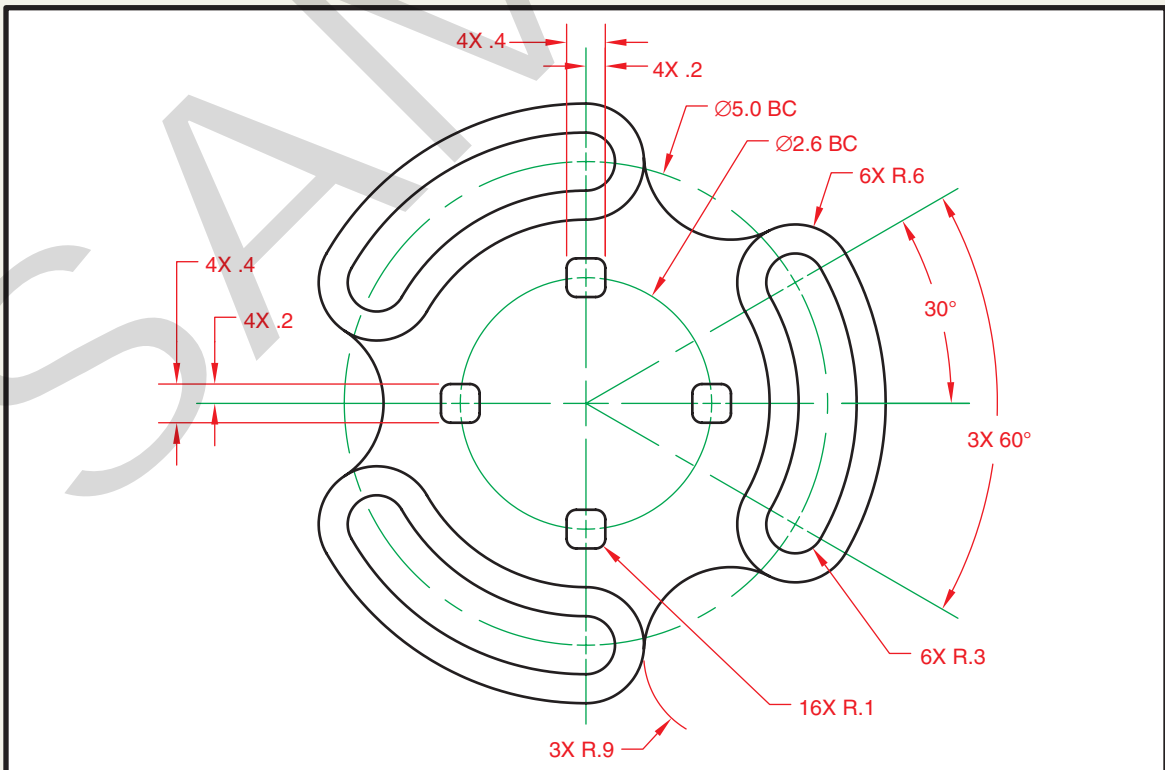
Problem 7-23. ★★★ Polygon Clock. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
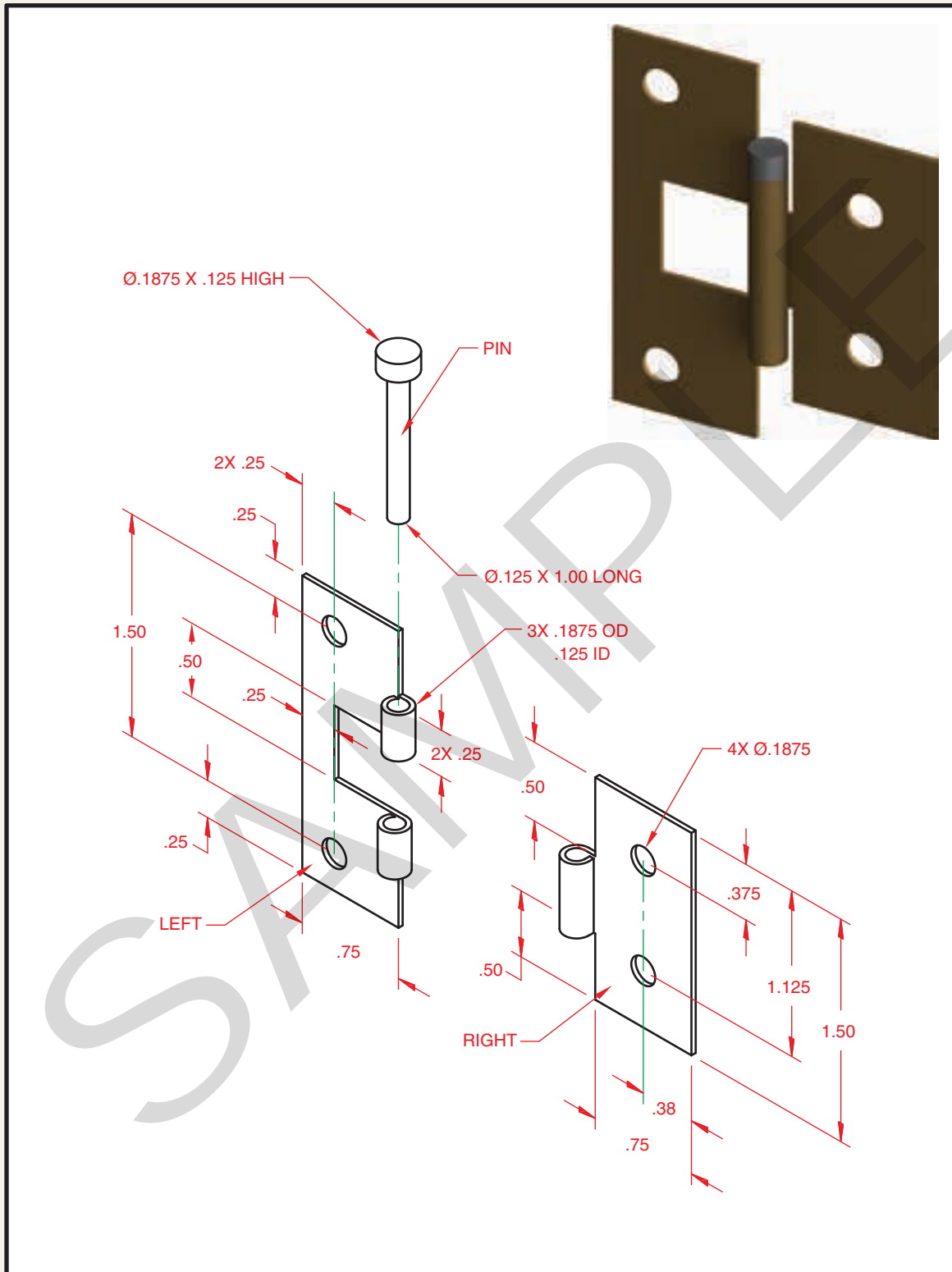

Problem 7-24. ★★★ Adjustment Plate. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.
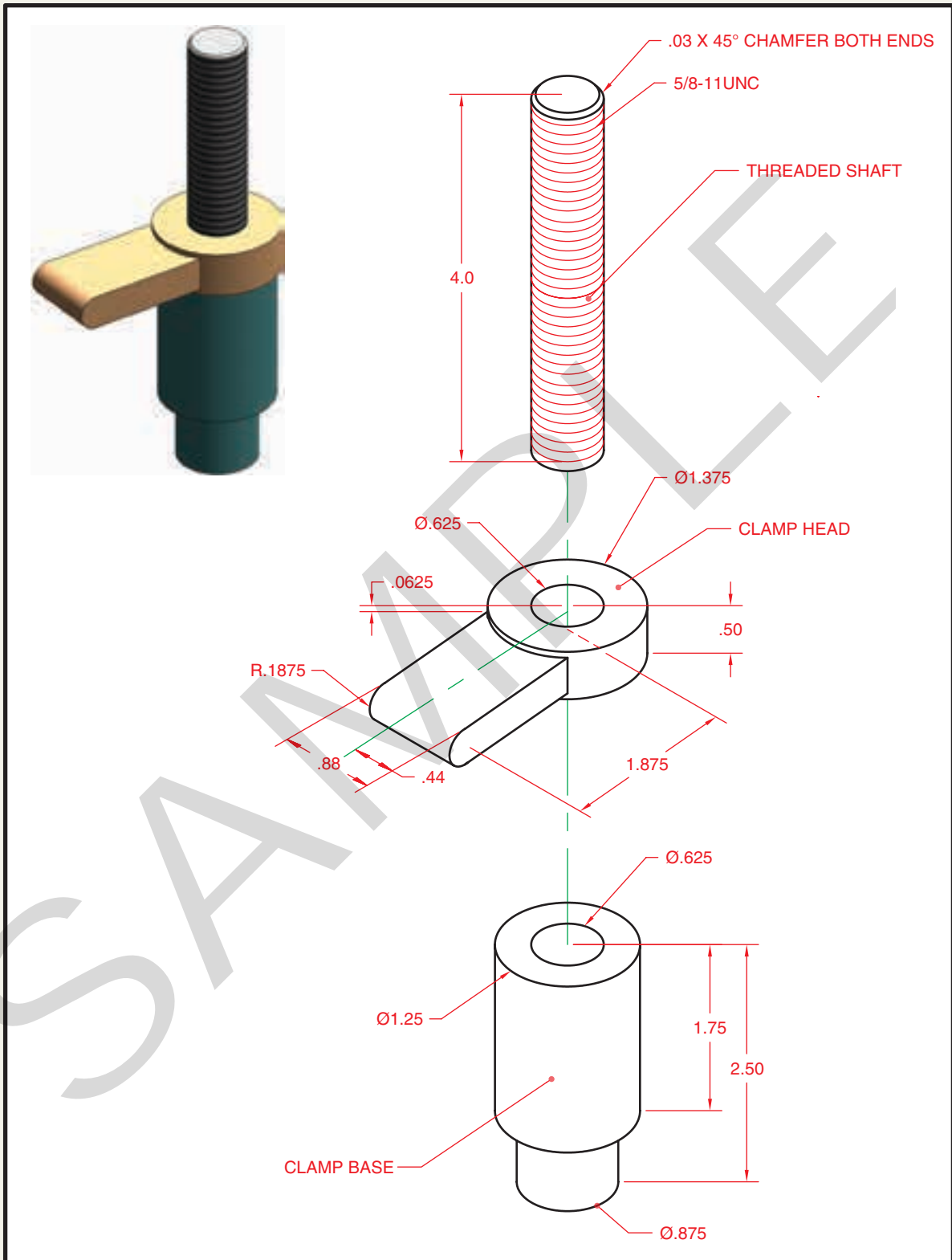
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Problem 7-25. ★★★ Cam Gasket. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.

Problem 7-26. ★★★ Fixture Plate. Draw a single view of the object as shown. Create layers as needed. Use the appropriate drawing commands. Do not dimension.

Ø.1875 X .125 HIGH

PIN

2X .25

.25

Ø.125 X 1.00 LONG

3X .1875 OD
.125 ID

1.50

.50

.25

2X .25

.50

.25

4X Ø.1875

LEFT

.75

.375

.50

1.125

RIGHT

1.50

.38

.75

Problem 7-27. ★★★ Hinge. Create the hinge assembly as a parametric model. Create the three parts shown using the dimensions provided. Then, create an assembly of the parts and mate them so that the left and right leaf sections align and the pin fastens them together. Create 2D drawings that show the appropriate views of the parts and assembly, including an exploded view.

.03 X 45° CHAMFER BOTH ENDS

5/8-11UNC

THREADED SHAFT

4.0

Ø1.375

Ø.625

CLAMP HEAD

.0625

.50

R.1875

1.875

.88

.44

Ø.625

Ø1.25

1.75

2.50

CLAMP BASE

Ø.875

Problem 7-28. ★★★ Machinist Fixture Clamp. Create the machinist fixture clamp assembly as a parametric model. Create the three parts shown using the dimensions provided. Then, create an assembly of the parts and mate them so that the clamp base, clamp head, and shaft align along the center axis. Create 2D drawings that show the appropriate views of the parts and assembly, including an exploded view.