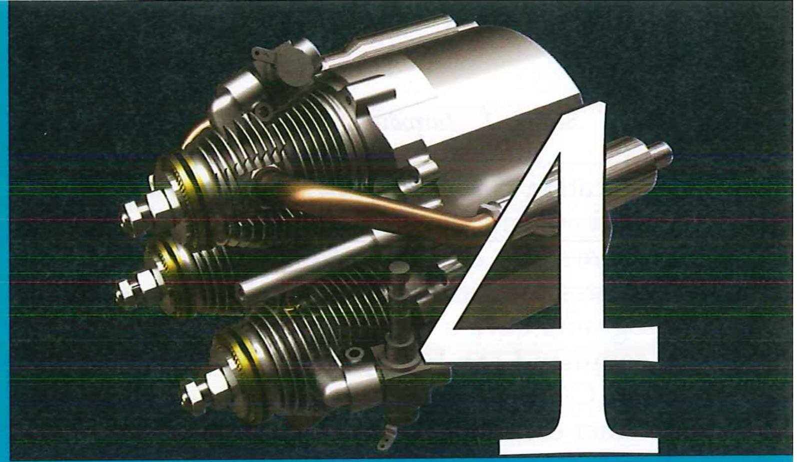
CAD Commands and Functions



Learning Objectives

After studying this chapter, you will be able to:

- List several general categories of commands used in popular CAD programs.
- Explain how points and objects are located using a coordinate system.
- Explain the use of linear, angular, and leader dimensioning.
- Identify and describe drawing aids.
- Discuss the purposes of colors, linetypes, and layers.
- Explain layer naming conventions as related to CAD drawings.
- Describe 3D drawing.
- Explain rendering.
- Explain animation.

Technical Terms

3D modeling
Absolute coordinates
Animation
Attribute
Blocks
Cartesian coordinate system
Command line
Commands
Coordinate systems
Dimensioning commands
Display control commands
Display grid
Drawing aids
Drawing commands
Editing commands
Ellipse
Fillet
File
File management commands
Grid snap
Inquiry commands
Isometric drawing
Layer
Object snap
Origin
Ortho
Polar coordinates
Pull-down menus
Regular polygon
Relative coordinates
Rendering
Round
Snap
Solid modeling
Spline
Surface modeling
Symbol library
Template
Toolbars
User coordinate system (UCS)
Wireframe
World coordinate system
X axis
XY drawing plane
Y axis
Z axis

Computer-aided drafting and design (CAD) is a powerful tool. However, just as with any tool, you have to know how to use it. ***Commands*** are the instructions you provide to CAD software to achieve the end result. There are several general groups of commands that are common to most CAD software. These groups are file management commands, drawing commands, editing commands, display control commands, dimensioning commands, and drawing aid commands. Examples of these commands and other types of CAD functions are discussed in this chapter. It is important to understand that each CAD package may have slightly different names for the commands and functions discussed here. This may be confusing, but there are many similarities between various products.

Just as command names vary among CAD software, the method of command entry can vary as well. Even within a particular program, there may be more than one way to enter a given command. For example, a command may be selected from a pull-down menu. ***Pull-down menus*** appear at the top of Windows-based software. The **File** menu found in a Windows application is a pull-down menu. In addition to pull-down menus, many CAD programs have ***toolbars*** that contain buttons. Picking a button activates a particular command. Also, some CAD programs have a ***command line***. This is where a command can be typed to activate it. Finally, some CAD programs support the use of a tablet to enter commands with a puck. The method in which a command is activated does not change the basic function of the command.

File Management Commands

Each time you use a computer, you are working with files. ***File management commands*** allow you to begin, save, and open drawings. A ***file*** is a collection of related data. When working with a CAD program, you create, save, open, and otherwise manipulate drawing files. Common commands used to manage files in a CAD program include the **New**, **Save**, and **Open** commands.

After starting a CAD program, you typically have a choice between beginning a new drawing or opening an existing drawing. The **New** command is used to start a new drawing file. This command typically gives you the option of starting a new drawing from "scratch" or from a template. A ***template*** is a drawing file with preconfigured user settings. A template typically contains settings based on a certain sheet size, title block format, or drafting discipline. Many of the functions discussed in this chapter can be set in a template file. The template file is then saved for future use. Drafters normally have template files for a variety of applications.

While working on a drawing, you will want to save information as it is added. The **Save** command is used to save a drawing. If you are saving a drawing file for the first time, you will be asked to specify a file name. You must also specify where to save the file. CAD files are typically saved to folders on a local or network drive, but there may be cases when you want to save to portable media.

It is important to save your work frequently. Every 10 to 15 minutes is recommended. This ensures that your drawing remains intact in the event of a power failure or system crash.

The **Open** command is used to access a drawing that has been previously saved. You will frequently need to recall a drawing file for continued work. During a typical drawing session, you may find it necessary to have several drawings open at the same time. Remember to save your drawings again after making any updates to preserve your work.

Coordinate Systems

Each line, circle, arc, or other object you add to a drawing is located by certain points. A line is defined by its two endpoints. A circle is defined by its center point and a point along the circumference. A square is located by its four corner points. To precisely locate points for objects, all CAD programs use standard point location systems called ***coordinate systems***.

The most common type of coordinate system in a CAD program is the ***Cartesian coordinate system***. Points are located in this system using three coordinate axes—the X axis, Y axis, and Z axis. The X and Y axes are used in two-dimensional drafting, **Figure 4-1A**. (The Z axis is used in three-dimensional drafting and is not shown in this example.) In 2D drafting, the ***X axis*** represents the horizontal axis. The ***Y axis*** represents the vertical axis. The intersection of these axes is the ***origin***. In 2D drafting, the location of any point can be determined by measuring the distance, in units of measurement, from the origin along the X and Y axes. Point locations are designated by X and Y coordinates and are represented as (*X*,*Y*). The units of measurement for coordinates may refer to inches, feet, or metric units, such as millimeters.

Coordinates specified for absolute point locations can be positive or negative (depending on their location in relation to the origin) and are known as ***absolute coordinates***. As shown in **Figure 4-1A**, when used for basic 2D drafting applications, the Cartesian coordinate system is divided into four quadrants and points are measured in relation to the origin (0,0). A point located in the upper-right quadrant has a positive X coordinate value and a positive Y coordinate value. A point located in the lower-right quadrant has a positive X coordinate value and a negative Y coordinate value. Points

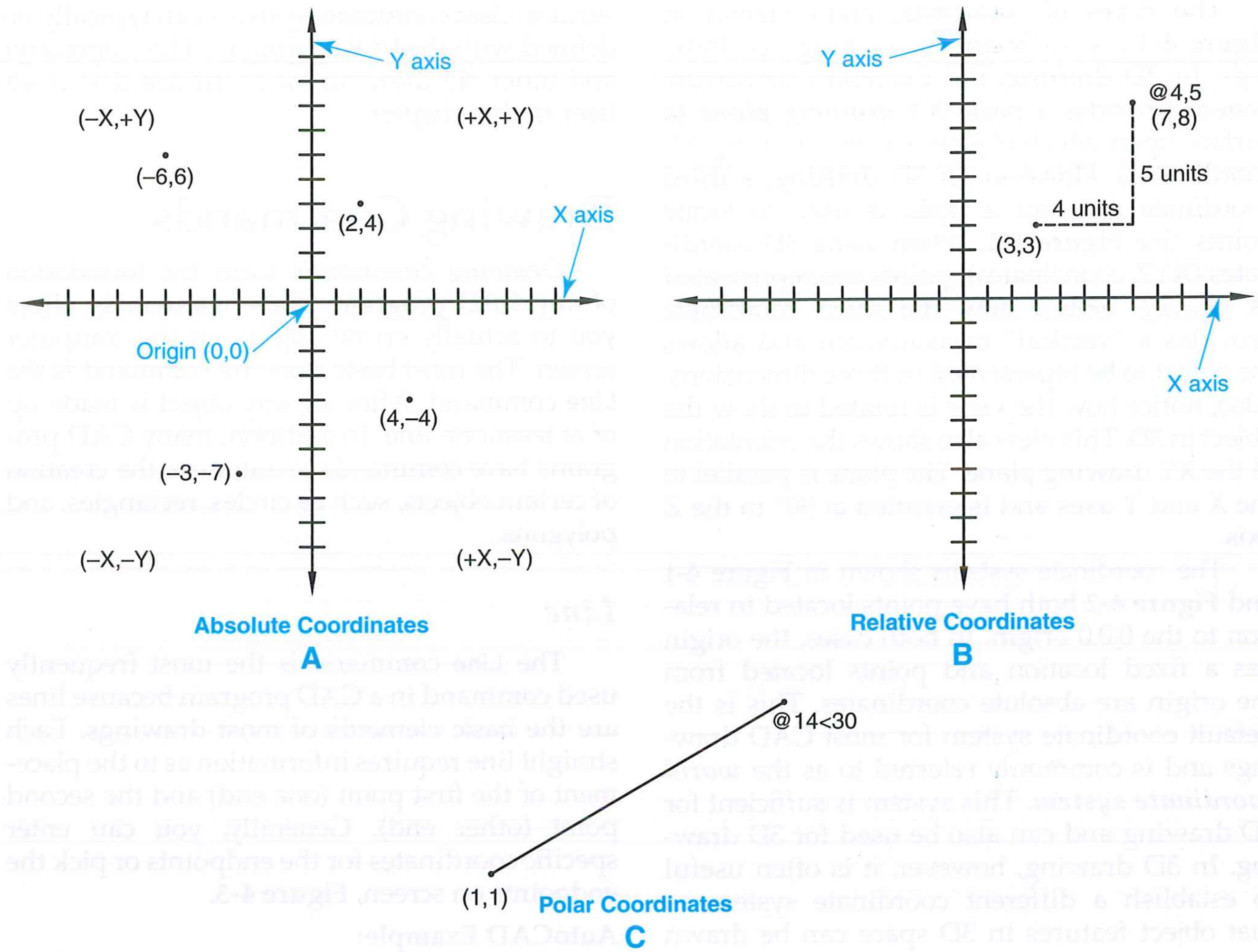


Figure 4-1. Methods of two-dimensional coordinate entry in the Cartesian coordinate system. A—Absolute coordinates are located in relation to the origin along the horizontal and vertical axes (XY axes). Coordinates can have positive or negative values and are designated as (*X*,*Y*). B—Relative coordinates are located in relation to a previous point. C—Polar coordinates are located by specifying a distance and angle relative to a given point.

located in the two left quadrants have negative X coordinate values and positive or negative Y coordinate values. Specifying point locations with absolute coordinates is the most basic way to locate points.

Coordinates for point locations can also be entered as relative coordinates, **Figure 4-1B**. ***Relative coordinates*** define a location from a previous point. The @ symbol is typically used to designate a relative coordinate entry. For example, suppose you have located the first point of a line at the coordinate (3,3) shown in **Figure 4-1B**. Entering the relative coordinate @4,5 would place the endpoint of the line four units to the right and five units above the previous point at the coordinate (7,8).

When it is necessary to locate points relative to given points at specific angles (such as the endpoints of an inclined line), polar coordinates are useful. ***Polar coordinates*** are relative coordinates that define a location at a given distance and angle from a fixed point (most typically a previous point). See **Figure 4-1C**. The coordinate entry format @*distance*<*angle* is normally used to specify polar coordinates. In this type of coordinate entry, angular values are typically measured from 0° horizontal. For example, in **Figure 4-1C**, after locating the first point of a line at the coordinate (1,1), entering the polar coordinate @14<30 locates the endpoint 14 units away from the first point at an angle of 30° counterclockwise in the XY plane.

The types of coordinate entry shown in **Figure 4-1** are sufficient for making 2D drawings. In 2D drafting, the Cartesian coordinate system provides a basic ***XY drawing plane*** (a surface upon which objects are drawn using XY coordinates). However, in 3D drafting, a third coordinate axis, the ***Z axis,*** is used to locate points. See **Figure 4-2**. When using 3D coordinates (XYZ coordinates), points are represented as (X,Y,Z). Notice how the third coordinate provides a "vertical" measurement and allows the object to be represented in three dimensions. Also, notice how the view is rotated to show the object in 3D. This view also shows the orientation of the XY drawing plane. The plane is parallel to the X and Y axes and is oriented at 90° to the Z axis.

The coordinate systems shown in **Figure 4-1** and **Figure 4-2** both have points located in relation to the 0,0,0 origin. In both cases, the origin has a fixed location and points located from the origin are absolute coordinates. This is the default coordinate system for most CAD drawings and is commonly referred to as the ***world coordinate system.*** This system is sufficient for 2D drawing and can also be used for 3D drawing. In 3D drawing, however, it is often useful to establish a different coordinate system so that object features in 3D space can be drawn more easily. A user-defined coordinate system is known as a ***user coordinate system (UCS).*** A UCS can be established at any origin and orientation so that points can be located in relation to a point in space, such as the corner of an object surface. User coordinate systems can typically be defined with the **UCS** command. This command and other 3D drawing functions are discussed later in this chapter.

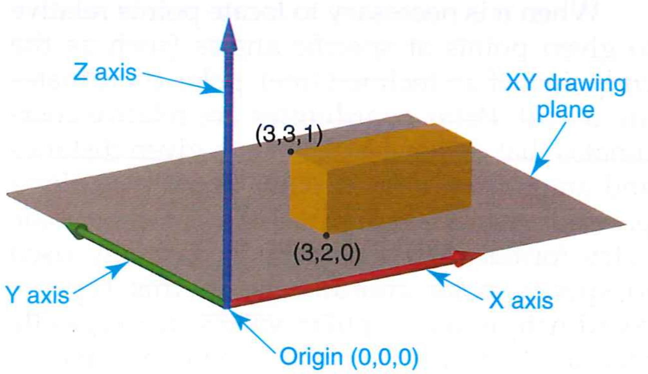


Figure 4-2. A third coordinate axis (the Z axis) is used in 3D drawing. The point locations making up the 3D object shown have XYZ coordinate values. Note the relationship of the coordinates to the origin and the XY drawing plane.

Drawing Commands

Drawing commands form the foundation of any CAD program. These commands allow you to actually create objects on the computer screen. The most basic drawing command is the **Line** command. After all, any object is made up of at least one line. In addition, many CAD programs have commands to automate the creation of certain objects, such as circles, rectangles, and polygons.

Line

The **Line** command is the most frequently used command in a CAD program because lines are the basic elements of most drawings. Each straight line requires information as to the placement of the first point (one end) and the second point (other end). Generally, you can enter specific coordinates for the endpoints or pick the endpoints on screen, **Figure 4-3**.

AutoCAD Example:

Command: **line**↵
Specify first point: **3,5**↵ *(or pick a point on screen)*
Specify next point or [Undo]: **6,4**↵ *(or pick a point on screen)*
Specify next point or [Undo]: ↵
Command:

Double Line

Some CAD packages provide a **Double Line** command, although it may not have this name. This command is useful in creating grooves on parts and in similar applications where parallel

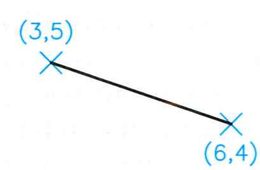


Figure 4-3. A line consists of two endpoints and a segment.

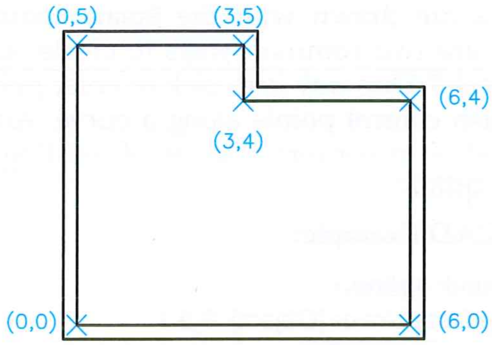


Figure 4-4. A double line can be used to quickly create walls.

lines are required. The **Double Line** command is especially useful in architectural drafting for drawing walls on a floor plan, **Figure 4-4**. Most CAD programs allow you to set the distance between the double lines. In addition, some programs allow you to control how the corners and intersections are formed.

AutoCAD Example:

Command: **mline**↵
Current settings: Justification = Top, Scale = 1.00, Style = STANDARD
Specify start point or [Justification/Scale/STyle]: **0,0**↵ *(or pick a point on screen)*
Specify next point: **6,0**↵ *(or pick a point on screen)*
Specify next point or [Undo]: **6,4**↵ *(or pick a point on screen)*
Specify next point or [Close/Undo]: **3,4**↵ *(or pick a point on screen)*
Specify next point or [Close/Undo]: **3,5**↵ *(or pick a point on screen)*
Specify next point or [Close/Undo]: **0,5**↵ *(or pick a point on screen)*
Specify next point or [Close/Undo]: **close**↵ *(or pick a point on screen)*
Command:

Point

Points define exact coordinate locations. In addition to serving as coordinates for lines and other entities, points can also be created as objects in most CAD programs. Points are helpful as a reference for making constructions and placing other objects. They can typically be created with the **Point** command. After entering the command, you can enter coordinates or pick a location on screen.

Most CAD programs provide different visibility modes for displaying points on screen. For example, you can display points as small crosses or boxes.

AutoCAD Example:

Command: **point**↵
Current point modes: PDMODE = 0 PDSIZE = 0.0000
Specify a point: **3,0**↵ *(or pick a point on screen)*
Command:

Circle

The **Circle** command automates the creation of a circle object. Instead of drawing several small straight-line segments to approximate a circle, this command draws an object based on the mathematical definition of a circle, **Figure 4-5**. Most CAD software allows you to select from several common methods of defining a circle. These methods include:

- Center and radius.
- Center and diameter.
- Three points on the circle.
- Two points on the circle.
- Radius and two lines or two circles to which the circle should be tangent.

AutoCAD Example:

Command: **circle**↵
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **0,0**↵ *(or pick a center point on screen)*
Specify radius of circle or [Diameter]: **diameter**↵
Specify diameter of circle: **4**↵ *(or pick a point on the circle on screen)*
Command:

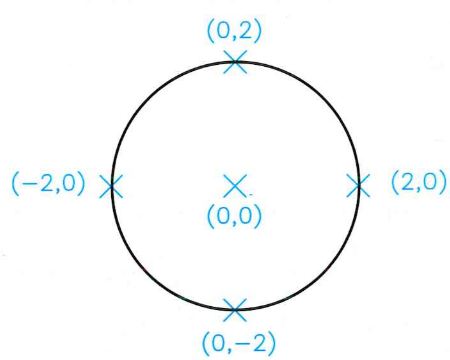


Figure 4-5. There are several ways to define a circle.

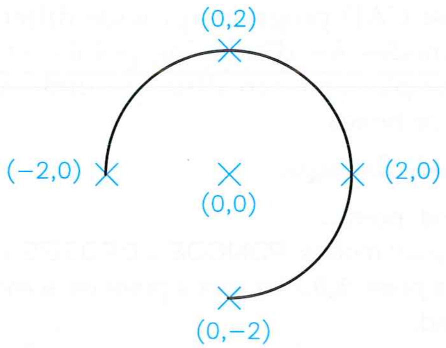


Figure 4-6. There are several ways to define an arc.

Arc

An arc is a portion of a circle. Just as the **Circle** command automates the creation of a circle, the **Arc** command automates the creation of an arc, **Figure 4-6**. Most CAD software allows you to select from several methods of defining an arc. Examples include:

- Three points on the arc.
- Starting point, center, and endpoint.
- Starting point, center, and included angle.
- Starting point, center, and length of chord.
- Starting point, endpoint, and radius.
- Starting point, endpoint, and included angle.
- Starting point, endpoint, and a starting direction.

AutoCAD Example:

Command: **arc**↵
Specify start point of arc or [Center]: **0,–2**↵ *(or pick a point on screen)*
Specify second point of arc or [Center/End]: **0,2**↵ *(or pick a point on screen)*
Specify end point of arc: **–2,0**↵ *(or pick a point on screen)*
Command:

Spline

A ***spline*** is a smooth curve that passes through a series of points. Usually, the points can be edited to change the "fit" of the curve after creating the spline. This provides greater accuracy for approximating irregular curves and other shapes that are difficult to draw as arcs. Splines are drawn with the **Spline** command. There are two common ways to create splines, **Figure 4-7**. One way is to pick or enter points to establish control points along a curve. Another method is to convert a series of existing lines into a spline.

AutoCAD Example:

Command: **spline**↵
Specify first point or [Object]: **2,3**↵
Specify next point: **5,4**↵
Specify next point or [Close/Fit tolerance] <start tangent>: **8,3**↵
Specify next point or [Close/Fit tolerance] <start tangent>: ↵
Specify start tangent: ↵ *(or pick a point to specify the beginning direction of the curve)*
Specify end tangent: ↵ *(or pick a point to specify the ending direction of the curve)*
Command:

Ellipse

An ***ellipse*** is a closed circular object with an oval shape. The arcs making up the shape

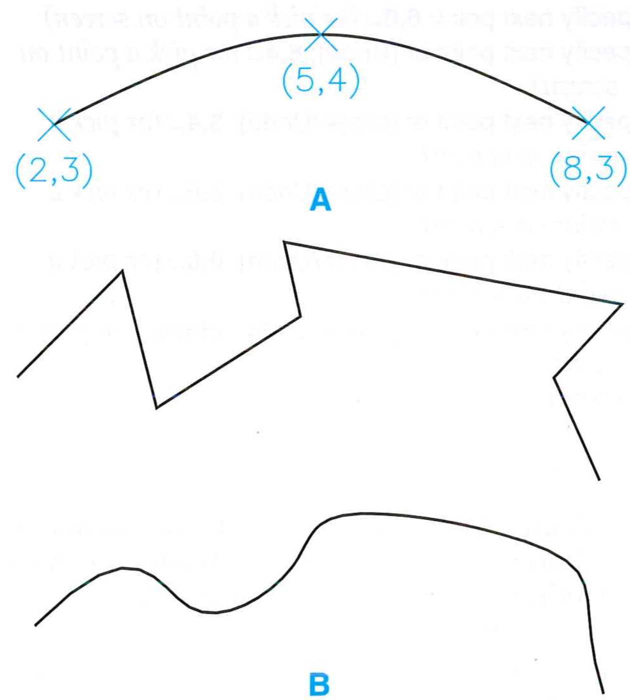


Figure 4-7. Creating splines. A—Picking points to establish control points for the fit of the curve. B—Creating a spline from connected lines.

are defined by the intersection of a major axis and minor axis. The axes intersect at the center point of the object and divide the ellipse into four quadrants. The **Ellipse** command draws the shape automatically based on points specified for the major and minor axis endpoints.

Ellipses can be drawn by several methods. One method is to locate the two axes by selecting two endpoints of one axis and one endpoint of the other axis, **Figure 4-8**. Another method is to locate the ellipse's center, and then specify one endpoint of each axis. A third method is to pick the ellipse's major axis endpoints and then enter a rotation angle.

AutoCAD Example:

Command: **ellipse↵**
Specify axis endpoint of ellipse or [Arc/Center]: **4,4↵**
Specify other endpoint of axis: **8,4↵**
Specify distance to other axis or [Rotation]: **6,5↵**
Command:

Rectangle

A square or rectangle can be drawn using the **Line** command. However, the **Rectangle** command automates the process of creating a square or rectangle, **Figure 4-9**. Most CAD software provides at least two methods for constructing a rectangle. These are specifying the width and height of the rectangle or specifying opposite corners of the rectangle.

AutoCAD Example:

Command: **rectangle↵**
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **1,5↵** *(or pick a point on screen)*
Specify other corner point or [Area/Dimensions/Rotation]: **6,3↵** *(or pick a point on screen)*
Command:

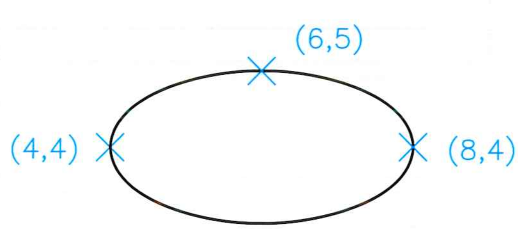


Figure 4-8. There are several ways to create ellipses. In this example, points are picked to identify the major and minor axes.

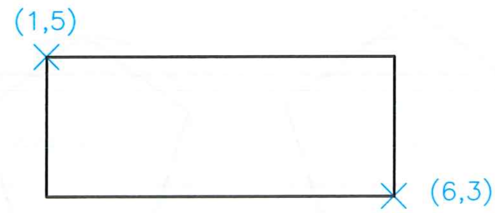


Figure 4-9. You can draw a rectangle by specifying opposite corners.

Polygon

The **Polygon** command automates the construction of a regular polygon. A ***regular polygon*** is an object with sides of equal length and included angles. The **Polygon** command can create an object with three or more sides. A common approach used by many CAD programs is to either inscribe the polygon within a circle or circumscribe it about a circle, **Figure 4-10**. The information required in these instances includes the radius of the circle, method desired, and number of sides for the polygon. Another method available in some CAD programs is to define the end points of one side of the polygon. The software generates the remaining sides to create a regular polygon.

AutoCAD Example:

Command: **polygon↵**
Enter number of sides <4>: **5↵**
Specify center of polygon or [Edge]: **1,5↵** *(or pick a point on screen)*
Enter an option [Inscribed in circle/Circumscribed about circle] <I>: **c↵**
Specify radius of circle: **2↵**
Command: ↵
POLYGON Enter number of sides <5>: **5↵**
Specify center of polygon or [Edge]: **6,5↵** *(or pick a point on screen)*
Enter an option [Inscribed in circle/Circumscribed about circle] <C>: **i↵**
Specify radius of circle: **2↵**
Command:

Text

You can add text to a drawing using the **Text** command. This is important for placing notes, specifications, and other information on a

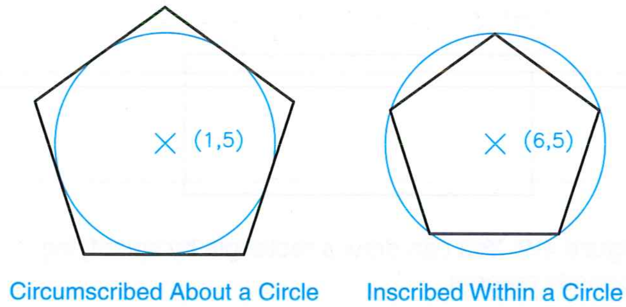


Figure 4-10. A polygon can be circumscribed (left) or inscribed (right).

drawing, **Figure 4-11**. Most CAD packages provide several standard text fonts to choose from. Text generally can be stretched, compressed, obliqued, or mirrored. Placement can be justified left, right, or centered. Text can also be placed at angles.

AutoCAD Example:

Command: **mtext**↵
Current text style: "Standard" Text height: 0.2500
Specify first corner: **2,3**↵ *(or pick a point on screen)*
Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: **9,5**↵ *(or pick a point on screen)*
(enter the text in the text boundary that appears and then pick the OK button)
Command:

Hatch

Hatching is a fundamental part of drafting. In both mechanical and architectural drafting, hatching is used in section views to show cutaway parts and to represent specific materials, **Figure 4-12**. Hatching is also used on pictorial drawings to represent surface texture or other features.

The **Hatch** command is used to hatch an area of a drawing. Areas to be hatched are selected with the pointing device and elements within the boundary can be excluded, if desired. Most CAD software includes several standard hatch patterns for use with the command. Some programs also provide other types of fill patterns, such as color gradients. In addition, most CAD software allows you to add more patterns and define your own.

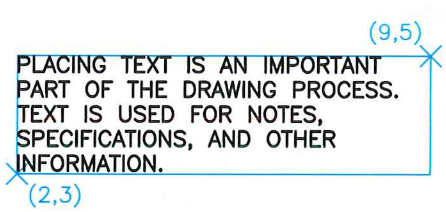


Figure 4-11. Text can be easily added to a drawing.

AutoCAD Example:

Command: **hatch**↵
(In the **Hatch and Gradient** *dialog box, select a pattern. Then, select the* **Add: Pick points** *or* **Add: Select objects** *button. When the dialog box is temporarily hidden, select internal points or pick objects to hatch. Then, press* [Enter] *to redisplay the dialog box. Pick the* **OK** *button to apply the hatch.)*
Command:

Editing and Inquiry Commands

Editing commands allow you to modify drawings. ***Inquiry commands*** are designed to list the database records for selected objects and calculate distances, areas, and perimeters. Common editing and inquiry commands described in this section include: **Erase**, **Undo**, **Move**, **Copy**, **Mirror**, **Rotate**, **Fillet**, **Chamfer**, **Trim**, **Extend**, **Array**, **Scale**, **List**, **Distance**, and **Area**.

Erase

The **Erase** command permanently removes selected objects from the drawing. Many CAD programs provide a "select" option in the command that allows you to select the objects to erase. Also, some programs provide a "last" option that erases the last object drawn.

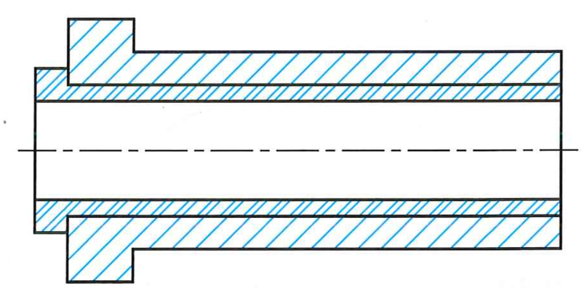


Figure 4-12. Hatch patterns can be used to represent different materials in a section view.

AutoCAD Example:

Command: **erase**↵
Select objects: **last**↵
1 found
Select objects: ↵ *(or pick other objects on screen)*
Command:

Undo

The **Undo** command reverses the last command. If the last command was **Erase**, the objects that were deleted are restored. You can sequentially step back through previous commands, but you cannot "jump" a command in the sequence. Certain limits are usually applied to this command.

AutoCAD Example:

Command: **erase**↵
Select objects: **last**↵
1 found
Select objects: ↵
(the last object drawn is erased)
Command: **undo**↵
Current settings: Auto = On, Control = All, Combine = Yes
Enter the number of operations to undo or [Auto/Control/BEgin/End/Mark/Back] <1>: ↵
ERASE
(the erased object is restored)
Command:

Move

The **Move** command allows one or more objects to be moved from the present location to a new one without changing their orientation or size. Generally, you must pick a starting point and a destination point. Relative displacement is often used for this operation. With relative displacement, you pick any starting point. Then, you specify a displacement from that point in terms of units, or units and an angle.

AutoCAD Example:

Command: **move**↵
Select objects: *(pick any number of objects using the cursor)*
Select objects: ↵
Specify base point or [Displacement] <Displacement>: *(pick any point on screen)*
Specify second point or <use first point as displacement>: **@2,3** *(the @ symbol specifies relative displacement; the object will be moved 2 units on the X axis and 3 units on the Y axis)*
Command:

Copy

The **Copy** command usually functions in much the same way as the **Move** command. However, it is used to place copies of the selected objects at the specified location without altering the original objects. Many CAD programs offer a "multiple" option with this command. This option is sometimes the default option and allows multiple copies of the selected objects to be placed in sequence.

AutoCAD Example:

Command: **copy**↵
Select objects: *(select the objects to copy)*
Select objects: ↵
Specify base point or [Displacement] <Displacement>: *(enter coordinates or pick a point to use as the first point of displacement)*
Specify second point or <use first point as displacement>: *(enter coordinates or pick a second point of displacement for the first copy)*
Specify second point or [Exit/Undo] <Exit>: *(enter coordinates or pick a second point of displacement for the second copy)*
Specify second point or [Exit/Undo] <Exit>: *(enter coordinates or pick a second point of displacement for the third copy)*
Specify second point or [Exit/Undo] <Exit>: ↵
Command:

Mirror

The **Mirror** command draws a mirror image of an existing object about a centerline. This command is especially useful when creating symmetrical objects, **Figure 4-13**. The **Mirror** command in most CAD programs allows you to either keep or delete the original object during the operation. The mirror line can generally be designated.

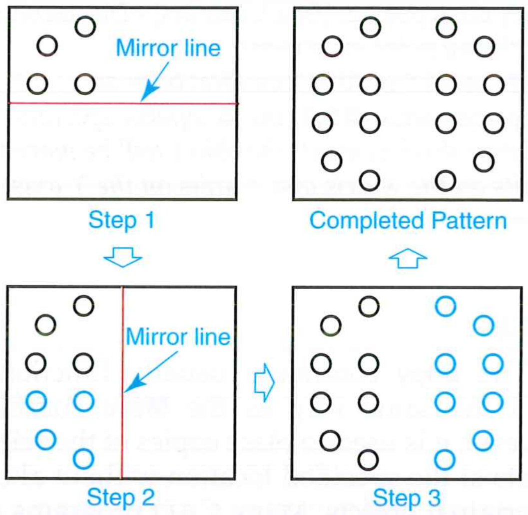


Figure 4-13. The hole pattern shown was created using mirror operations. The pattern was first mirrored vertically, then the original and the mirrored copy were mirrored horizontally. The mirrored copies are shown in color.

AutoCAD Example:

Command: **mirror**↵
Select objects: *(select the objects to mirror)*
Select objects: ↵
Specify first point of mirror line: *(enter coordinates or pick an endpoint of the line about which to reflect the objects)*
Specify second point of mirror line: *(enter coordinates or pick the second endpoint of the line about which to reflect the objects)*
Erase source objects? [Yes/No] <N>: **n**↵
Command:

Rotate

The **Rotate** command is used to alter the orientation of objects on the drawing. Typically, you must specify a center for the rotation. This command is perhaps one of the most used editing commands.

AutoCAD Example:

Command: **rotate**↵
Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0
Select objects: *(pick the objects to rotate)*
Select objects: ↵
Specify base point: *(enter coordinates or pick a point about which to rotate the objects)*
Specify rotation angle or [Copy/Reference]: <0>: *(enter an angle or drag the cursor to the desired rotation)*
Command:

Scale

The size of existing objects can be changed using the **Scale** command. When using the **Scale** command, most CAD programs require you to specify a base point for the operation. This point is generally on the object, often the center of the object or a reference corner.

In CAD programs with parametric modeling capability, you can change the base size parameter, or any other parameter, of the object without using the **Scale** command. For example, you can scale a ∅5 circle up by 50% by simply changing its diameter to 7.5 without using the **Scale** command.

AutoCAD Example:

Command: **scale**↵
Select objects: *(pick the objects to scale)*
Select objects: ↵
Specify base point: *(enter coordinates or select a point about which the objects will be scaled)*
Specify scale factor or [Copy/Reference] <1.0000>: **1.5**↵
Command:

Fillet

A ***fillet*** is a smoothly fitted internal arc of a specified radius between two lines, arcs, or circles. A ***round*** is just like a fillet, except it is an exterior arc, **Figure 4-14**. Most manufactured parts, including those for architectural applications, have some fillets or rounds. The **Fillet** command is used to place fillets and rounds onto the drawing. After drawing the curve, the command trims the original objects to perfectly meet the curve.

AutoCAD Example:

Command: **fillet**↵
Current settings: Mode = TRIM, Radius = 0.2500
Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **radius**↵
Specify fillet radius <0.2500>: **.50**↵

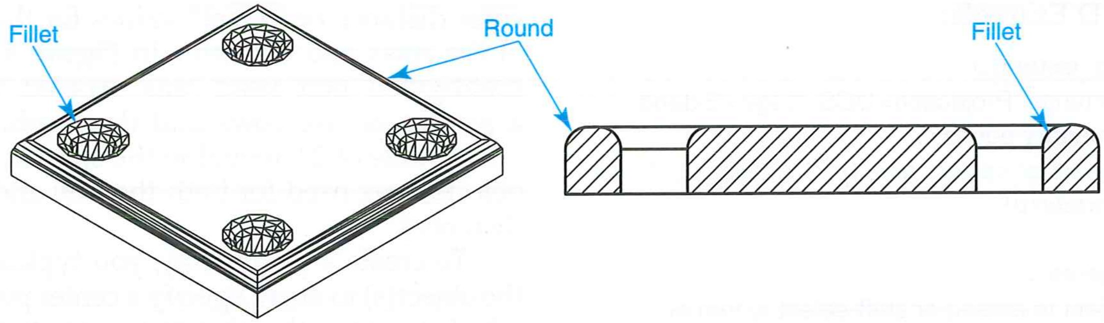


Figure 4-14. Fillets and rounds on a drawing.

Select first object or [Undo/Polyline/Radius/Trim/
Multiple]: *(select one of the two objects between which the fillet or round is to be placed)*
Select second object or shift-select to apply corner:
(select the second of the two objects between which the fillet or round is to be placed)
Command:

Chamfer

The **Chamfer** command is very similar to the **Fillet** command. However, instead of a curve, a straight line is placed between the chamfered lines. Just as with the **Fillet** command, the original lines are trimmed to meet the straight line (chamfer). Depending on the CAD program, this command may require that the two objects to be chamfered are lines, not arc segments.

AutoCAD Example:

Command: **chamfer**↵
(TRIM mode) Current chamfer Dist1 = 0.5000,
Dist2 = 0.5000
Select first line or [Undo/Polyline/Distance/Angle/
Trim/mEthod/Multiple]: **distance**↵
Specify first chamfer distance <0.5000>: **.25**↵
Specify second chamfer distance <0.2500>: ↵
Select first line or [Undo/Polyline/Distance/Angle/
Trim/mEthod/Multiple]: *(pick the first line to chamfer)*
Select second line or shift-select to apply corner: *(pick the second line to chamfer)*
Command:

Trim

The **Trim** command is used to shorten a line, arc, or other object to its intersection with an existing object. The object that establishes the edge you are trimming to is called a *cutting edge*. The cutting edge is defined by one or more objects in the drawing. Some CAD programs allow you to trim objects without specifying a cutting edge. In this case, the nearest intersection is used for the trim operation.

Most CAD programs place limitations on which types of objects can be trimmed. In addition, there are usually only certain types of objects that can be used as boundary edges.

AutoCAD Example:

Command: **trim**↵
Current settings: Projection=UCS, Edge=Extend
Select cutting edges...
Select objects or <select all>: *(pick a cutting edge)*
1 found
Select objects: ↵
Select object to trim or shift-select to extend or
[Fence/Crossing/Project/Edge/eRase/Undo]:
(select the object to trim)
Select object to trim or shift-select to extend or
[Fence/Crossing/Project/Edge/eRase/Undo]: ↵
Command:

Extend

Extending an object lengthens the object to end precisely at an edge called a *boundary edge*. The **Extend** command sequence is similar to the **Trim** command sequence. The boundary edge is defined by one or more objects in the drawing. There are usually limitations on which types of objects can be extended or used as boundary edges.

AutoCAD Example:

Command: **extend**↵
Current settings: Projection=UCS, Edge=Extend
Select boundary edges...
Select objects or <select all>: *(pick the objects to use as a boundary)*
1 found
Select objects: ↵
Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]: *(select the objects to extend to the boundary)*
Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]: ↵
Command:

Array

The **Array** command is essentially a copy function. It makes multiple copies of selected objects in a rectangular or circular (polar) pattern. See **Figure 4-15**. CAD programs that have 3D drawing capability typically have an option of the **Array** command to create arrays in 3D.

To create a rectangular array, you typically select the object(s) to array, specify the number of rows, specify the number of columns, and then enter distance or "offset" values for the spacing of the rows and columns. In **Figure 4-15A**, the highlighted bolt head was arrayed to create a pattern of two rows and three columns. An offset value of 2.0 (equal to the spacing between objects) was used for both the row and column distances.

To create a polar array, you typically select the object(s) to array, specify a center point about which to array the object(s), enter the number of objects in the array, and enter an angular rotation value. In **Figure 4-15B**, the highlighted circle was arrayed about the center point of the part in a 360° pattern, with a total of 12 objects specified.

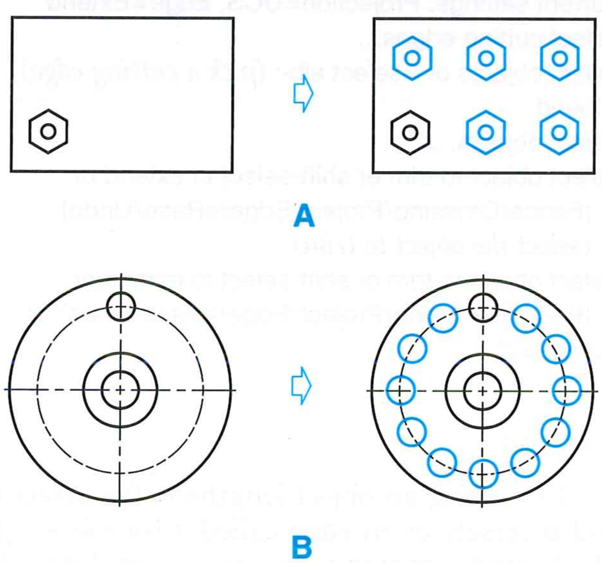


Figure 4-15. Creating rectangular and polar arrays. A—A pattern of bolt heads drawn as a rectangular array. The arrayed objects are shown in color. B—The bolt holes in this part were drawn as a polar array. The arrayed objects are shown in color.

AutoCAD Example:

Command: **-array**↵ *(If you enter the command without the hyphen, the array settings are made in a dialog box.)*
Select objects: *(pick the objects to array)*
Select objects: ↵
Enter the type of array [Rectangular/Polar] <P>: **r**↵
Enter the number of rows (---) <1>: **2**↵
Enter the number of columns (|||) <1>: **3**↵
Enter the distance between rows or specify unit cell (---): **2**↵
Specify the distance between columns (|||): **2**↵
Command: **-array**↵
Select objects: *(pick the objects to array)*
Select objects: ↵
Enter the type of array [Rectangular/Polar] <R>: **p**↵
Specify center point of array or [Base]: *(pick a point about which the objects will be arrayed)*
Enter the number of items in the array: **12**↵
Specify the angle to fill (+=ccw, −=cw) <360>: ↵
Rotate arrayed objects? [Yes/No] <Y>: **n**↵
Command:

List/Properties

The **List** and **Properties** commands show data related to an object. For example, the properties for a line may include the coordinates of the endpoints, length, angle from start point, and change in X and Y coordinates from the start point. These commands can be useful in determining the type of object, which layer it is drawn on, and the color and linetype settings of the object.

AutoCAD Example:

Command: **list**↵
Select objects: 1 found
Select objects: ↵
(The text window that appears lists the properties of the selected object.)
Command:

Distance

The **Distance** command measures the distance and angle between two points. The result is displayed in drawing units. This command is very useful in determining lengths, angles, and distances on a drawing without actually placing dimensions.

AutoCAD Example:

Command: **dist**↵
Specify first point: *(pick the first endpoint of the distance to measure)*
Specify second point: *(pick the second endpoint of the distance to measure)*
Distance = 9.1788, Angle in XY Plane = 29, Angle from XY Plane = 0
Delta X = 8.0000, Delta Y = 4.5000, Delta Z = 0.0000
Command:

Area

The **Area** command is used to calculate the area of an enclosed space. Often, you can select a closed object or simply pick points on an imaginary boundary. Most CAD programs allow you to remove islands, or internal areas, **Figure 4-16**. The **Area** command has many applications in technical drafting, such as calculating the area of a surface to determine the weight of an object, or calculating the square footage of a house.

AutoCAD Example:

Command: **area**↵
Specify first corner point or [Object/Add/Subtract]: **add**↵
Specify first corner point or [Object/Subtract]: *(pick the first point of the area, as shown in Figure 4-16)*
Specify next corner point or press ENTER for total (ADD mode): *(pick the next point of the area, as shown in Figure 4-16)*
Specify next corner point or press ENTER for total (ADD mode): *(pick the next point of the area, as shown in Figure 4-16)*

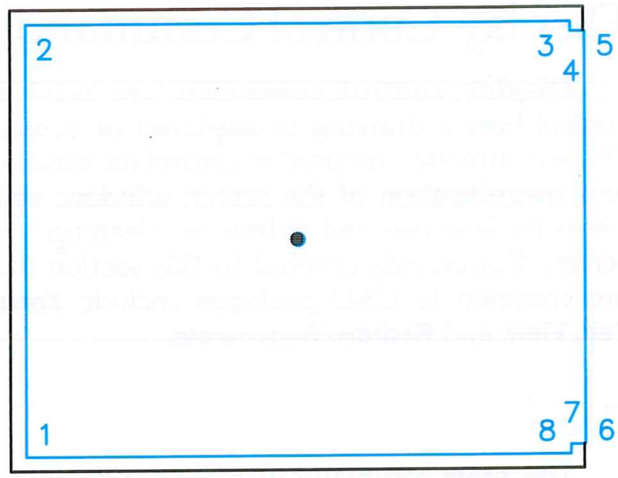


Figure 4-16. The **Area** command can be used to calculate how many square feet of tile are required for this garage floor. The surface to be covered in tile is outlined in color. Notice the drain that will be removed from the calculation.

Specify next corner point or press ENTER for total (ADD mode): *(pick the next point of the area, as shown in Figure 4-16)*
Specify next corner point or press ENTER for total (ADD mode): *(pick the next point of the area, as shown in Figure 4-16)*
Specify next corner point or press ENTER for total (ADD mode): *(pick the next point of the area, as shown in Figure 4-16)*
Specify next corner point or press ENTER for total (ADD mode): *(pick the next point of the area, as shown in Figure 4-16)*
Specify next corner point or press ENTER for total (ADD mode): *(pick the last point of the area, as shown in Figure 4-16)*
Specify next corner point or press ENTER for total (ADD mode): ↵
Area = 657.3750, Perimeter = 103.5000
Total area = 657.3750
Specify first corner point or [Object/Subtract]: **subtract**↵
Specify first corner point or [Object/Add]: **object**↵
(SUBTRACT mode) Select objects: *(select the internal circle shown in Figure 4-16)*
Area = 1.7671, Circumference = 4.7124
Total area = 655.6079
(SUBTRACT mode) Select objects: ↵
Specify first corner point or [Object/Add]: ↵
Command:

Display Control Commands

Display control commands are used to control how a drawing is displayed on screen. These commands are used to control the position and magnification of the screen window, save views for later use, and redraw or "clean up" the screen. Commands covered in this section that are common to CAD packages include **Zoom**, **Pan**, **View**, and **Redraw**/**Regenerate**.

Zoom

The **Zoom** command increases or decreases the magnification factor, which results in a change in the apparent size of objects on screen. However, the actual size of the objects does not change. You can think of this as using the zoom feature on a video camera or set of binoculars. **Zoom** may be the most-used display control command. Generally, the **Zoom** command has several options that may include zooming to the drawing limits or extents, dynamically zooming, and zooming by a magnification factor.

AutoCAD Example:

Command: **zoom**↵
Specify corner of window, enter a scale factor (nX or nXP), or [All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: **.5**↵
(the magnification factor is reduced by 50%)
Command: ↵
ZOOM Specify corner of window, enter a scale factor (nX or nXP), or [All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: **previous**↵
(the previous magnification factor is restored)
Command:

Pan

The **Pan** command moves the drawing in the display window from one location to another. It does not change the magnification factor. If you think of the drawing as being on a sheet of paper behind the screen, panning is moving the sheet so a different part of the drawing can be seen, **Figure 4-17**. The **Pan** command is useful when you have a magnification factor that you like, but there are objects that are "off" the screen.

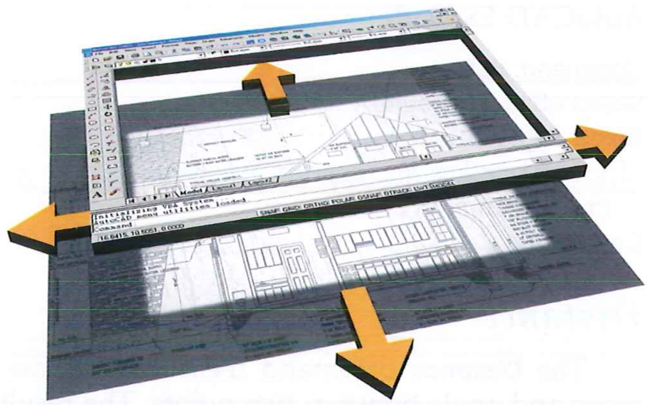


Figure 4-17. You can think of panning as moving a drawing sheet around underneath the CAD drawing screen. Only the portion of the drawing directly "below" the drawing area will be visible. (Eric K. Augspurger; print courtesy of SoftPlan Systems, Inc.)

AutoCAD Example:

Command: **pan**↵
Press ESC or ENTER to exit, or right-click to display shortcut menu.
(This is AutoCAD's "realtime" pan function; pick, hold, and drag to pan the drawing; then press [Enter] *or* [Esc] *to end the command.)*
Command:

View

When constant switching back and forth between views and magnification factors on a large drawing is required, the **View** command can be used to speed the process. This command allows you to save a "snapshot" of the current drawing display. The "snapshot" includes the view and the magnification factor. You can then save the view and quickly recall it later. This can be much faster than zooming and panning to return to the desired view.

AutoCAD Example:

(Pan and zoom the drawing so the desired view is displayed.)
Command: **view**↵
(The **View Manager** *dialog box is displayed; pick the* **New...** *button and enter a name in the* **New View** *dialog box that is displayed. Then close both dialog boxes.)*
Command:

Redraw/Regenerate

The **Redraw** command "cleans up" the display by removing marker blips, etc. Some commands automatically redraw the screen, as when a grid is removed or visible layers are changed. However, sometimes it is useful to request a redraw when other operations are being performed. The **Regenerate** command forces the program to recalculate the objects in the entire drawing and redraw the screen. This operation takes longer than **Redraw**, especially on large or complex drawings.

AutoCAD Example:

Command: **regen**↵
Regenerating model.
Command:

Dimensioning Commands

One of the advantages of using CAD is automated dimensioning. In almost all drafting applications, the drawing must be dimensioned to show lengths, distances, and angles between features on the objects (parts). There are five basic types of ***dimensioning commands***. These are **Linear**, **Angular**, **Diameter**, **Radius**, and **Leader**, **Figure 4-18.**

A linear dimension measures a straight line distance. The distance may be horizontal, vertical, or at an angle. Typically, you have several choices on how the dimension text is placed. The text may be aligned with the dimension lines, always horizontal on the drawing, or placed at a specified angle. In architectural drafting, dimension text for a linear dimension is never perpendicular to the dimension line.

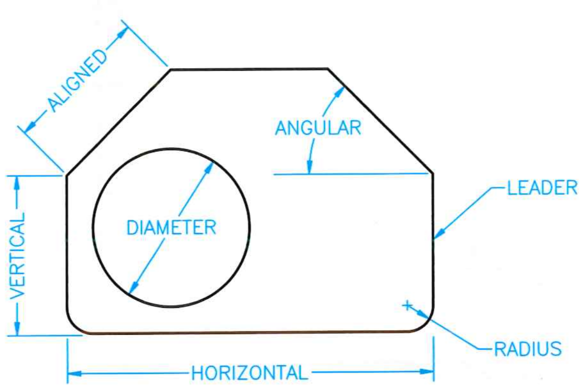


Figure 4-18. Common types of dimensions that appear on a CAD drawing.

An angular dimension measures the angle between two nonparallel lines. The lines can be actual objects or imaginary lines between an origin and two endpoints. Typically, you have the same options for text placement as with linear dimensions.

Diameter and radius dimensions are very similar. A diameter dimension measures the distance across a circle through its center. A radius dimension measures the distance from the center of an arc to a point on that arc. A radius dimension can also be used for a circle, but it is not typically used in this manner.

A leader is used to provide a specific or local note. A leader consists of an arrowhead (in some form), a leader line, and the note. Often, an optional shoulder is placed on the end of the leader before the note.

AutoCAD Example:

Command: **dim**↵
Dim: **horizontal**↵
Specify first extension line origin or <select object>: *(pick the first endpoint of the horizontal distance)*
Specify second extension line origin: *(pick the second endpoint of the horizontal distance)*
Specify dimension line location or [Mtext/Text/Angle]: *(drag the dimension to the correct location)*
Enter dimension text <15.500>: *(enter a value for the dimension text or press* [Enter] *to accept the default actual distance)*
Dim: **vertical**↵
Specify first extension line origin or <select object>: *(pick the first endpoint of the vertical distance)*
Specify second extension line origin: *(pick the second endpoint of the vertical distance)*
Specify dimension line location or [Mtext/Text/Angle]: *(drag the dimension to the correct location)*
Enter dimension text <6.000>: *(enter a value for the dimension text or press* [Enter] *to accept the default actual distance)*
Dim: *(press* [Esc] *to exit dimension mode)*
Command:

Drawing Aids

Drawing aids are designed to speed up the drawing process and, at the same time, maintain accuracy. Most CAD packages provide several different drawing aids. These can range from a display grid or viewport ruler to various forms of snap. The features discussed in this section include grid, snap, and ortho.

Grid

A ***display grid*** is a visual guideline in the viewport, much like the lines on graph paper. How the grid appears when displayed depends on which CAD program you are using. For example, AutoCAD uses dots to show the grid. See **Figure 4-19**. In most CAD programs with a grid function, you can change the density, or spacing, of the grid.

Some CAD programs also have rulers that can be displayed along the horizontal and vertical edge of the drawing screen. The display of these rulers is often controlled by a single command. However, the display may also be part of an **Options** or **Settings** command, depending on the CAD program.

AutoCAD Example:

Command: **grid**↵
Specify grid spacing(X) or [ON/OFF/Snap/Major/
aDaptive/Limits/Follow/Aspect] <0.5000>: **.25**↵
Command: **grid**↵
Specify grid spacing(X) or [ON/OFF/Snap/Major/
aDaptive/Limits/Follow/Aspect] <0.2500>: **off**↵
Command:

Snap

Snap is a function that allows the cursor to "grab on to" certain locations on the screen. There are two basic types of snap. These are grid snap and object snap. A ***grid snap*** uses an invisible grid, much like the visible grid produced by the **Grid** command. When grid snap is turned on, the cursor "jumps" to the closest snap grid point. In most CAD programs, it is impossible to select a location that is not one of the snap grid points when grid snap is on. Just as with a grid, you can typically set the snap grid density or spacing.

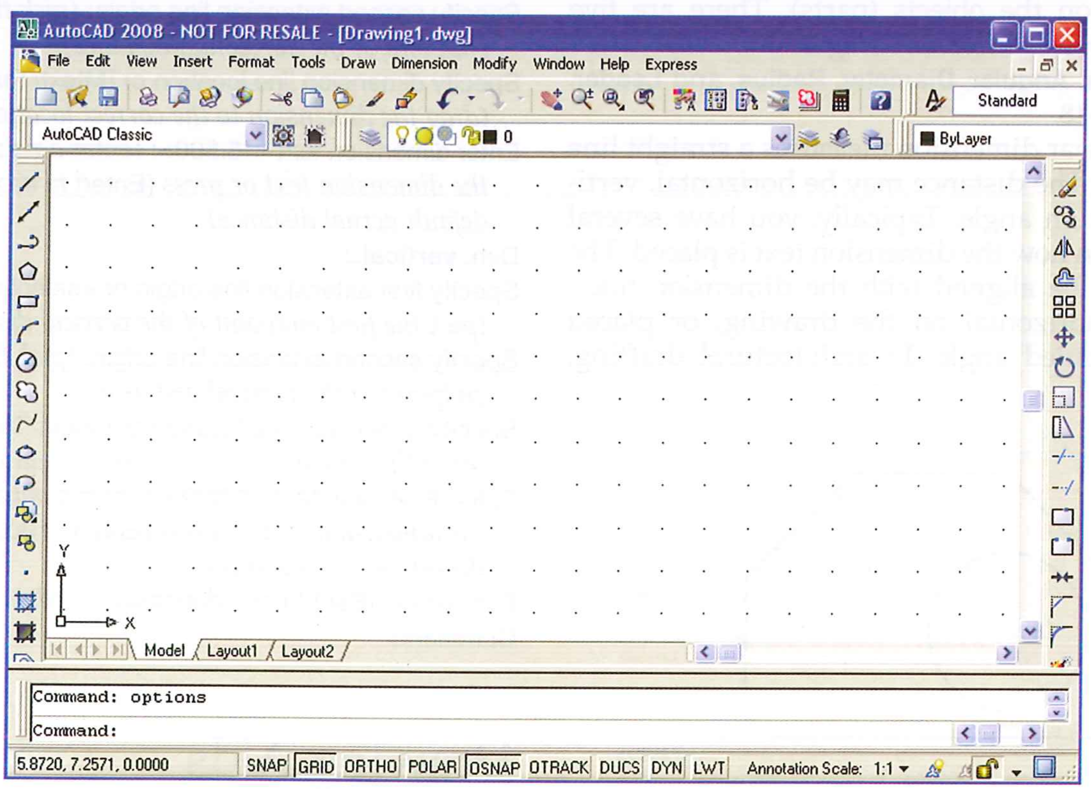


Figure 4-19. AutoCAD shows its grid as a matrix of dots.

An ***object snap*** allows the cursor to "jump" to certain locations on existing objects. Most CAD programs have several different object snaps. These can include Endpoint, Center, Midpoint, Perpendicular, Tangent, and Intersection, as well as many others. Depending on the CAD program you are using, there may be additional object snaps available. Generally, you can turn on the object snaps that you want to use while another command is active. For example, suppose you have a line already drawn and you want to draw another line from its exact midpoint. You can enter the **Line** command, temporarily set the midpoint object snap, and pick the first endpoint of the second line at the midpoint of the first line.

Object snaps provide a very quick way of accurately connecting to existing objects. They are likely the most important feature of CAD software to ensure accuracy. Think twice before buying CAD software without object snaps.

AutoCAD Example:

Command: **line↵**
Specify first point: **mid↵**
of *(move the cursor close to the line from which to select the midpoint; when the snap cursor is displayed, pick to set the endpoint of the new line)*
Specify next point or [Undo]: *(move the cursor to the second endpoint of the new line and pick)*
Specify next point or [Undo]: ↵
Command:

Ortho

Ortho is a drawing mode used to ensure that all lines and traces drawn using a pointing device are orthogonal (vertical or horizontal) with respect to the current drawing plane. Ortho is useful in drawing "square" lines that will be later extended or trimmed to meet other objects. Ortho is activated with the **Ortho** command. Ortho must be turned off to draw a line at an angle unless coordinates are manually entered.

AutoCAD Example:

Command: **ortho↵**
Enter mode [ON/OFF] <OFF>: **on↵**
(Lines can now only be drawn horizontally and vertically at 90° angles unless coordinates are entered.)
Command:

Layers

One of the fundamental tools in any good CAD program is the ability to draw on and manage layers. A ***layer*** is a virtual piece of paper on which CAD objects are placed. All objects on all layers, or sheets of paper, are visible on top of each other. If you are familiar with traditional (manual) drafting, you can think of layers as vellum overlays. Layers can be turned on and off, resulting in the display of only those objects needed. For example, layers are especially useful in checking the alignment of mating parts in an assembly drawing.

Most CAD programs have a **Layer** command that allows you to create and manage layers. Generally, you can assign a unique layer name and color to each layer. You can also use the **Layer** command to control the visibility of layers. In addition, some CAD programs allow you advanced control over layers. For example, some CAD programs allow you to prevent certain layers from printing.

Proper layer management is very important to effective CAD drawing. This is especially true when the drawing is jointly worked on by several drafters, designers, or engineers. In an effort to standardize layer use in industry, several organizations have attempted to develop layer naming/usage standards. There is not one universally accepted standard.

It is important to follow whatever standards your company, department, or client require. While you should always attempt to follow accepted industry standards, it is more important that everyone working on a project follow the same convention no matter what that convention may be. For example, you and several coworkers may adopt simple naming conventions for the types of drawings you produce and the layers that you are using. If these conventions adequately meet your needs, then adopt the conventions and make sure everyone follows them.

AutoCAD Example:

Command: **layer↵**
(In the **Layer Properties Manager** *dialog box that is displayed, pick the* **New Layer** *button to create a new layer, pick a color swatch to change a layer's display color, or make on-off and layer printing settings.)*
Command:

Colors and Linetypes

Another important "management" aspect of CAD programs is object display color. At a very simple level, object display colors help to visually catalog the objects in a drawing. For example, if all objects in a drawing are displayed in the same color, it can be hard to identify the individual features. On the other hand, if all dimension lines are displayed in red, object lines are displayed in green, and all symbols are displayed in yellow, at a glance anybody who is familiar with this color scheme can determine what is represented. Just as with layer names, it is important to adopt a color usage convention and make sure everybody follows it.

Often, object display colors are determined by the layer on which they are drawn. This is one reason that layer conventions are so important, as described in the previous section. However, most CAD programs allow you to "override" this setting and assign a specific display color to an object, regardless of which layer it is on. The command used to change an object display color can be **Change**, **Color**, **Properties**, or another command. The exact command will be determined by the CAD program you are using.

Managing the types of lines used on a drawing is also important. Fortunately, there is an almost universally accepted practice for representing lines. This practice is the Alphabet of Lines. The Alphabet of Lines is covered in detail in Chapter 2. You should always follow the Alphabet of Lines. Just because you use CAD to create a drawing does not remove your responsibility to follow this practice. Most CAD programs provide several linetypes that conform to the Alphabet of Lines. You can typically adjust the scale of each line (or all lines) so your particular application better conforms. Do not let the ease at which lines can be drawn "slide" into bad drafting habits. *Always follow the Alphabet of Lines.*

As with colors, most CAD programs allow you to assign linetypes to individual objects. You can also generally assign a linetype based on which layer the object is currently "residing" on. While the Alphabet of Lines provides clear direction on which linetypes to use, be sure to develop a convention based on *how* the linetypes are assigned. Choose to assign linetypes "by layer" or "by object," or some combination of the two, and then be sure everybody follows that convention.

Blocks and Attributes

Blocks are special objects that can best be thought of as symbols inserted into a drawing. Most CAD systems support blocks. In addition, in most CAD systems, the block function supports a feature called *attributes*. An ***attribute*** is text information saved with the block when it is inserted into a drawing. For example, you may create a block that consists of all lines you would normally draw to represent a case-molded window. In addition to creating the block, you assign attributes to the block describing the window size, style, and manufacturer. See **Figure 4-20A**.

A

Command: **INSERT**↵
(In the **INSERT** *dialog box, pick the block)*
Specify insertion point or
[Basepoint/Scale/Rotate]:
(specify an insertion point)
Enter attribute values
Enter window width <36">:↵
Type manufacturer name <GlazRus>:↵
Enter wood or vinyl <Wood>:↵
Command:

B

Figure 4-20. A—This window block contains attributes, which have values assigned. B—The AutoCAD command sequence for inserting the block and assigning attribute values.

When using attributes with a block, the attributes are typically assigned to the block when it is created. However, another feature supported by many CAD programs is to prompt the user for attributes when the block is inserted, **Figure 4-20B**. This allows a single block, or symbol, to serve for many different sizes, styles, manufacturers, etc., for similar applications. Using the window example above, you may draw a generic window and prompt the user for a size, style, and manufacturer when the block is inserted.

One of the biggest advantages of using blocks and attributes is the ability to automatically generate schedules. For example, if you insert all windows in a drawing as blocks with correctly defined attributes, some CAD systems can automatically generate a window schedule that can be used for design, estimating, and purchasing. With some CAD programs, this "generator" is within the program itself, while other CAD programs link this function to database software (such as Excel or Lotus 1-2-3).

In addition to the advantage of automated schedules, blocks save time. Once you have spent the time to create the symbol, it is saved to a symbol library. A ***symbol library*** is a collection of blocks that are typically related, such as fasteners, structural connections, or electrical symbols. Each block in the library can be inserted over and over. This saves the time of redrawing the symbol each time you need it. Blocks can save time even if attributes are not assigned to them.

AutoCAD Example:

Command: **attdef**↵

(In the **Attribute Definition** *dialog box, enter a name to appear on screen in the* **Tag:** *text field, a prompt to appear on the command line in the* **Prompt:** *field, and a default value in the* **Value:** *field; then, enter coordinates or check* **Specify On-screen** *to specify where the attribute will be inserted; finally, pick* **OK** *in the dialog box to create the attribute.)*

Command: **block**↵

(In the **Block Definition** *dialog box, name the block; pick the* **Select objects** *button and pick the objects to include in the block on screen, including the attribute; then, pick the* **Pick point** *button to specify an insertion base point for the block. Complete the block definition in the* **Write Block** *dialog box and pick the OK button.)*

Command: **insert**↵

(Select the block you created; when prompted, enter the attribute information.)

Command:

3D Drawing and Viewing Commands

When CAD programs were first developed, they were used to create 2D drawings. This was the natural progression from traditional (manual) drafting, which is strictly 2D on paper. As computers and CAD programs became more advanced, 3D capabilities were added. At first, these capabilities made it easier to draw 3D representations, such as isometrics and perspectives, but these representations are really 2D drawings. Eventually, "true" 3D modeling capabilities were added to CAD programs. These features allow you to design, model, analyze, and in some cases "premachine" a part all within the computer.

Isometric Drawing

An ***isometric drawing*** is a traditional 2D pictorial drawing. It shows a 3D representation of an object, but it is really only two-dimensional, **Figure 4-21**. If you could rotate the "paper" computer screen, there would be no part of the object behind the current drawing plane. Some CAD programs have drawing aids to help make isometric drawings. These drawing aids typically are a rotated grid, orthographic cursor, and snap representing the three isometric planes (top, left, right). The way in which these drawing aids are activated varies with the CAD program being used.

3D Modeling

A more realistic type of 3D drawing is called ***3D modeling***. This is "true" 3D where objects are created with a width, depth, and height. Unlike isometric drawing, if you rotate the screen "paper," you can see "behind" the object. See **Figure 4-22**. There are two general types of 3D models—surface and solid.

Surface modeling creates 3D objects by drawing a skin, often over a wireframe. A ***wireframe***

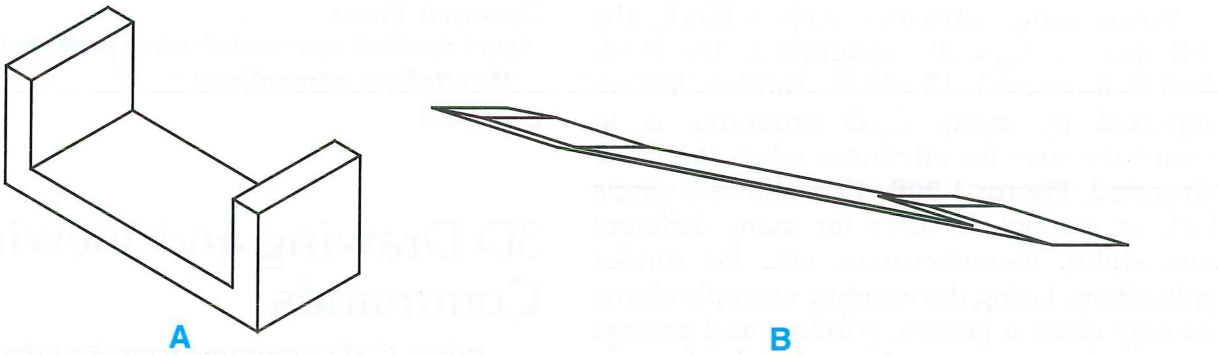


Figure 4-21. A—An isometric drawing of a mechanical part. This is a 2D isometric drawing that appears to show the object in three dimensions. B—When the isometric drawing is viewed from a different viewpoint, you can see that it is two-dimensional.

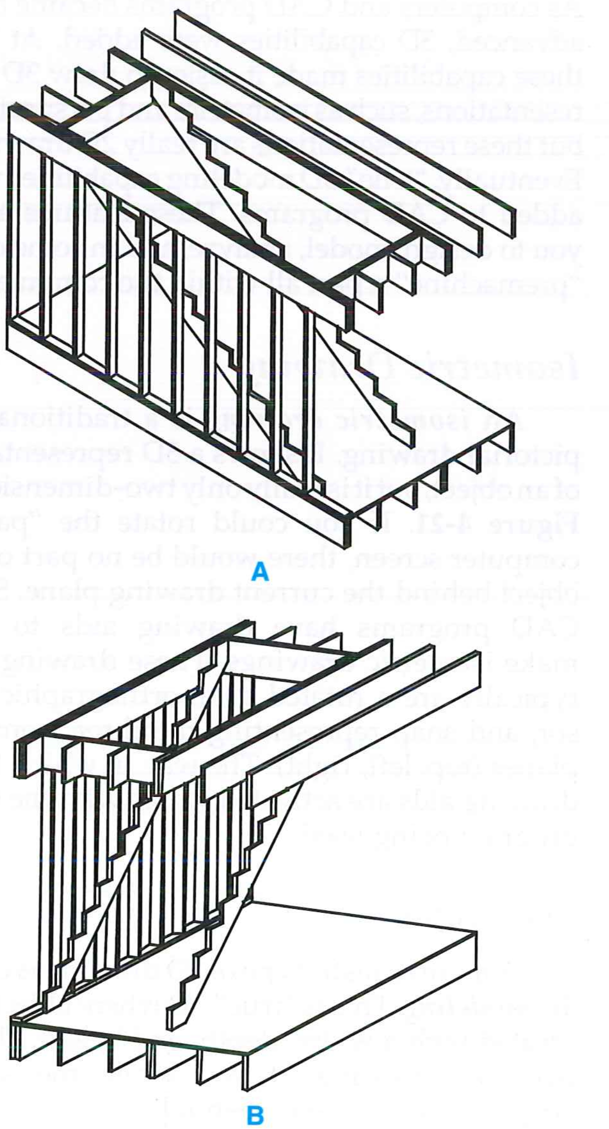


Figure 4-22. A—An isometric view of a 3D model. B—When the 3D model is viewed from a different viewpoint, you can see that it is truly three-dimensional. (Eric K. Augspurger)

is a group of lines that represent the edges of a 3D model. A wireframe does not have a "skin" and is less realistic than a surface model. Surface models are used for rendering and animation. However, surface modeling is not often used for engineering applications because a surface model does not have volume or mass properties.

Solid modeling creates 3D objects by generating a volume. If you think of surface modeling as blowing up a balloon to obtain a final shape, solid modeling is obtaining the final shape by filling it with water. A solid model can be analyzed for mass, volume, material properties, and many other types of data. Many CAD packages that can produce solid models also allow you to create cross sections, which is hard or impossible to do with a surface model. In addition, a solid model can be rendered and imported into many animation software packages.

3D Views

CAD software that is 3D-capable typically has a **Hide** command to remove lines that would normally be hidden in the current view. These are the lines that would be drawn as hidden lines in a 2D drawing. Hiding lines can help visualize the 3D model. The **Hide** command is used in **Figure 4-22**.

In addition to hiding lines, you also need to be able to see the objects from different angles. It would be nearly impossible to create a 3D model of any complexity only being able to see a top view, for example. It may seem as if each CAD program has its own unique way of displaying different 3D views. Some CAD programs have preset isometric views. Others allow you to dynamically change the view. Yet other programs have both of these options and more. However, the basic goal of all of these functions is the same. You need to "rotate" the point from which you are viewing the model to better see another part or feature on the object. The object in **Figure 4-22A** is shown from a preset isometric viewing point. In **Figure 4-22B**, however, a dynamic viewing command was used to rotate the viewpoint.

AutoCAD Example:

Command: **3dorbit**↵
Press ESC or ENTER to exit, or right-click to display shortcut-menu.
(Using the cursor, pick and drag to change the view; press [Esc] *to exit the command.)*
Command:

User Coordinate Systems

As discussed earlier in this chapter, user coordinate systems are very helpful in 3D drawing. A user coordinate system (UCS) allows the drafter to orient the current drawing plane so that it is parallel to a surface, such as an object surface or feature. The XYZ drawing axes remain oriented at 90° angles, but the drawing plane rotates to match the orientation desired. This is very useful when creating a 3D model, because the model may have many features that lie on different surfaces. Each time you need to construct features on a different surface, you can create a new UCS to establish a different drawing plane in 3D space.

In a typical CAD program, the **UCS** command is used to establish a user coordinate system. After entering the command, coordinates for the UCS origin are entered or picked and the orientation of the drawing axes is specified. Points are then located along the XYZ axes in relation to the new UCS origin. In most programs, a UCS icon is displayed in 3D drawing views to identify each coordinate axis and the origin location. This helps the user visualize the drawing orientation when creating objects and changing views.

An example of using a UCS in 3D modeling is shown in **Figure 4-23**. The solid model shown includes a large hole feature on an inclined surface. To simplify construction of this feature, a UCS is created to establish a drawing plane on the inclined surface. A solid cylinder is then drawn with its base parallel to the surface so that it can be used to "drill" the hole feature. The original coordinate system is shown in **Figure 4-23A**. In this system, the XY drawing plane is parallel to the base of the object. This system was used to construct the two holes in the base. Notice the orientation of the XYZ axes and the corner location of the origin. In **Figure 4-23B**, the UCS is moved to the inclined surface and the cylinder is drawn. The origin selected for the UCS is a corner on the inclined surface. The diameter and height of the cylinder are determined by the dimensions of the hole feature. In **Figure 4-23C**, the cylinder is moved "down" along the Z axis so that the base of the object establishes the depth of the drill. The completed model after removing material with the cylinder is shown in **Figure 4-23D**.

The **UCS** command normally provides options for rotating the UCS about one of the axes to a different orientation and aligning the UCS to an object face. In some programs, the UCS can also be dynamically moved to a temporary drawing location without changing the UCS definition. This allows you to quickly draw 3D objects on existing surfaces without changing the UCS each time.

AutoCAD Example:

Command: **ucs**↵
Current ucs name: **current**
Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>: *(enter coordinates or pick a point on screen, such as an object corner)*
Specify point on X-axis or <Accept>: *(enter coordinates, pick a point along the X axis, or press* [Enter] *to accept the orientation shown)*
Specify point on the XY plane or <Accept>: *(enter coordinates, pick a point along the Y axis, or press* [Enter] *to accept the orientation shown)*
Command:

3D Animation and Rendering Commands

As previously discussed, CAD programs that have 3D drawing capability generally have a command that allows you to create a

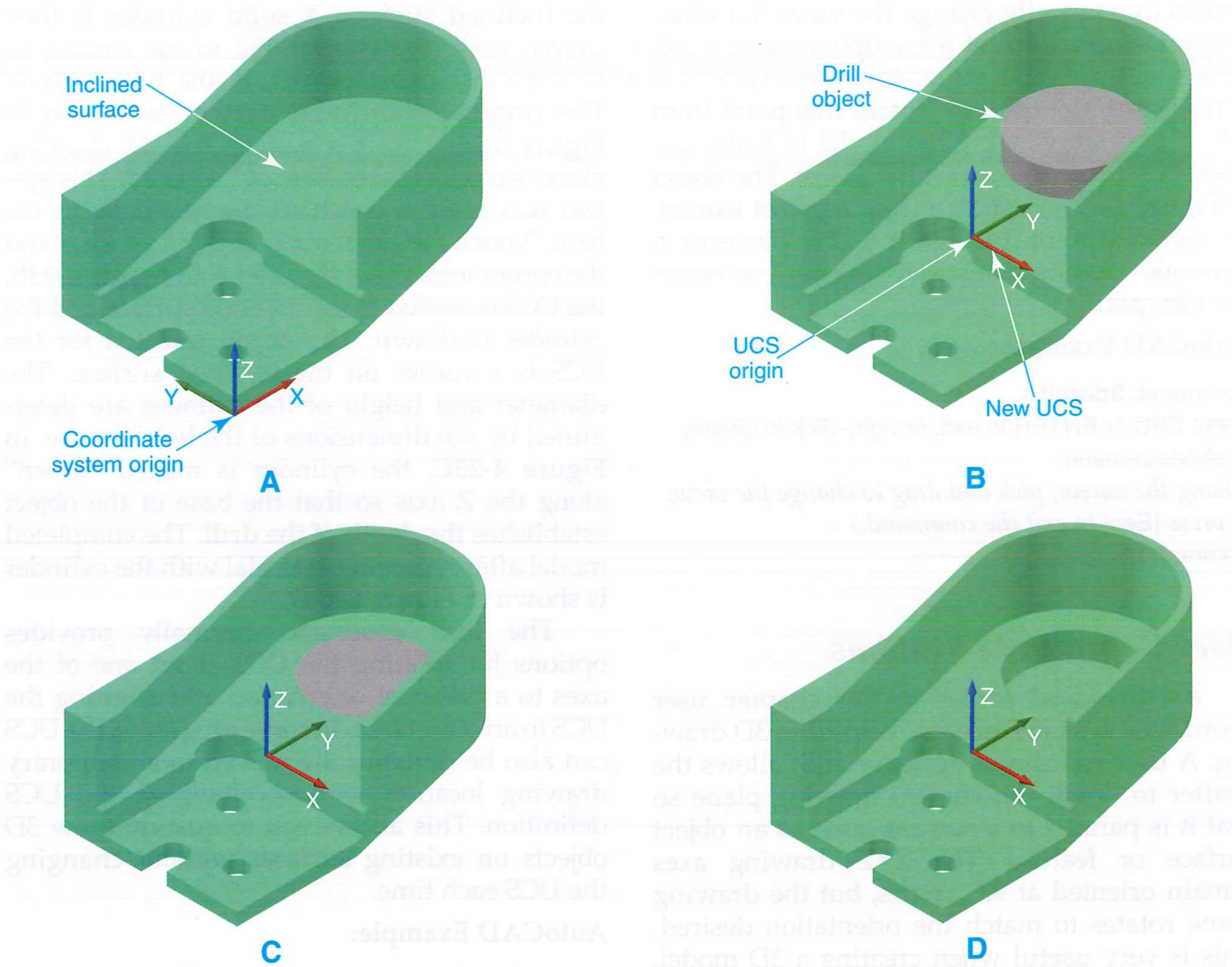


Figure 4-23. User coordinate systems are commonly used in 3D modeling. For this model, a UCS is established on the inclined surface to simplify construction of the large hole feature. A—The coordinate system shown is parallel to the base of the model and was used to construct the features in the base. B—A UCS is created parallel to the inclined surface to construct the cylinder (drill object). The origin specified for the UCS is a corner on the surface. Notice the direction of the XYZ axes. C—The drill object is moved down along the Z axis to the required depth for removing material. D—The completed model.

hidden-line-removed display. However, to create a more realistic representation of the 3D objects, most CAD systems have the ability to shade or color the model. This is called *rendering*. Rendering has traditionally been done by hand with paint, charcoal, chalk, pencils, and ink. However, just as the process of creating a drawing has been automated with CAD, so too has the process of rendering the drawing. Generally, there is a **Material** command used to define and apply surface textures to objects. There is also typically a **Render** command used to "color" the drawing. Many high-end CAD programs can produce very realistic renderings, given enough "drafting" time to properly set up lights and materials, **Figure 4-24**.

Some CAD programs have the ability to add movement to objects in the drawing to create an animation. An ***animation*** is a series of still images played sequentially at a very fast rate, such as 30 frames per second, **Figure 4-25**. There are very small differences between each frame and, when each frame is viewed quickly, the brain "mistakes" these differences as movement. Generally, there is an **Animate** command used to add movement to the objects and a **Render** command to render the animation.

Figure 4-24. Good planning and a lot of time were required to create this rendered model. (Bucyrus International)

Chapter Summary

Computer-aided drafting and design (CAD) is a powerful tool, but as with any tool, you have to know how to use it. Commands are the interactions you provide to CAD software to achieve end results. Commands are usually grouped according to function, such as file management, drawing, editing, inquiry, display control, and dimensioning.

Commands may be entered in several ways. In many cases, commands may be selected from pull-down menus. Many CAD programs also have toolbars that contain buttons for activating commands. In addition, some CAD programs have a command line. This is where a command can be typed to activate it.

Drawing commands form the foundation of any CAD program. These commands allow you to actually create objects on screen.

Editing and inquiry commands allow you to modify drawings, list the database records for

Figure 4-25. An animation is really a series of images played together at a fast rate. The slight differences between each image are interpreted by the brain as motion. There is not much difference between "neighboring" frames. However, over the length of the animation, you can see that the window is opening and closing. (Eric K. Augspurger)

selected objects, and calculate distances, areas, and perimeters.

Display control commands are used to control how a drawing is displayed on screen. These commands allow you to control the position and magnification of the screen window, save views for later use, and redraw or "clean up" the screen.

Dimensioning commands enable the drafter to automate the dimensioning process. They are used to draw dimensions showing lengths, distances, and angles between object features.

Drawing aids allow the drafter to speed up the drawing process, and at the same time, maintain accuracy. Typical drawing aids in a CAD program include grid, snap, and ortho.

Other features of a good CAD package include: layers, colors, linetypes, blocks, and attributes. A layer is a virtual piece of paper on which CAD objects are placed. Proper layer management is important to effective CAD drawing. Object display colors and linetypes should also be managed properly. Object colors help to visually catalog the objects in a drawing. Linetypes enable the drafter to apply the Alphabet of Lines. Blocks are special objects that can best be thought of as symbols inserted into the drawing. An attribute is text information saved with a block.

Most quality CAD packages also include 3D drawing and viewing commands. Some of the most common 3D-based commands are used for isometric drawing, 3D modeling and viewing, 3D animation, and rendering. These commands allow you to design, model, analyze, and in some cases "premachine" a part all within the computer.

Additional Resources

CAD Software Resources

Autodesk, Inc.
Developer of AutoCAD, Inventor, Revit Systems, 3ds max
www.autodesk.com

Auto-des-sys, Inc.
Developer of Form•Z
www.formz.com

Bentley Systems, Inc.
Developer of MicroStation
www.bentley.com

Cadalyst
www.cadalyst.com

Parametric Technology Corp.
Developer of Pro/ENGINEER, Pro/MECHANICAL, Pro/DESKTOP
www.ptc.com

SolidWorks Corporation
Developer of SolidWorks
www.solidworks.com

UGS Corp.
Developer of Solid Edge
www.solidedge.com

Review Questions

1. The instructions you provide to CAD software to achieve the end result are called _____.
2. _____ menus appear at the top of Windows-based software.
3. What type of commands allow you to begin, save, and open drawings?
4. A _____ is a drawing file with preconfigured user settings.
5. In the Cartesian coordinate system, the horizontal axis is the _____ axis and the vertical axis is the _____ axis.
6. _____ commands (such as the **Line** command) allow you to create objects on the computer screen.
7. Identify four common methods of defining a circle.
8. A(n) _____ is a portion of a circle.
9. A(n) _____ is a closed circular object with an oval shape.
10. A regular _____ is an object with sides of equal length and included angles.
11. You can add text to a drawing using the _____ command.

12. _____ is used in section views to show cutaway parts and to represent specific materials.
13. Which of the following is *not* a common editing or inquiry command?
    A. **Rotate**
    B. **Extend**
    C. **Dimension**
    D. **List**
14. The _____ command permanently removes selected objects from the drawing.
15. The _____ command reverses the last command.
16. The size of existing objects can be changed using the _____ command.
17. A _____ is just like a fillet, except it is an exterior arc.
18. Extending an object lengthens the object to end precisely at an edge called a _____ edge.
19. The **Array** command is essentially a _____ function.
20. The _____ command is used to calculate the area of an enclosed space.
21. Name three display control commands.
22. The **Zoom** command increases or decreases the _____ factor.
23. Name the five basic types of dimensioning commands.
24. A(n) _____ is used to provide a specific or local note.
25. Grid, snap, and ortho are examples of _____.
26. Name the two types of snap.
27. A(n) _____ is a virtual piece of paper on which CAD objects are placed.
28. _____ are special objects that can best be thought of as symbols inserted into a drawing.
29. One of the biggest advantages of using blocks and attributes is the ability to automatically generate _____.
30. An isometric drawing is a traditional 2D _____ drawing.
31. Surface modeling creates 3D objects by drawing a skin, often over a _____.
32. UCS is an abbreviation for _____.
33. Define *animation*.

Problems and Activities

1. Using the reference section of this text for samples, create a basic symbol library. Create one symbol library that includes symbols from a variety of applications, such as material section symbols, structural symbols, welding symbols, and electrical symbols.
2. Identify the 3D commands in your school's CAD software. Create a list of the command names and the functions they perform.
3. Obtain electronic examples of AEC renderings or animations. The Internet can be a great source for this, but be sure to download only those files labeled as "freeware" or "freely distribute." All others are copyrighted material.
4. Collect as many different examples of 3D computer-generated illustrations as you can. Search through books and magazines and bring them to class to share with your classmates. Classify each one as a wireframe, "true" 3D model, or animation.

Drawing Problems

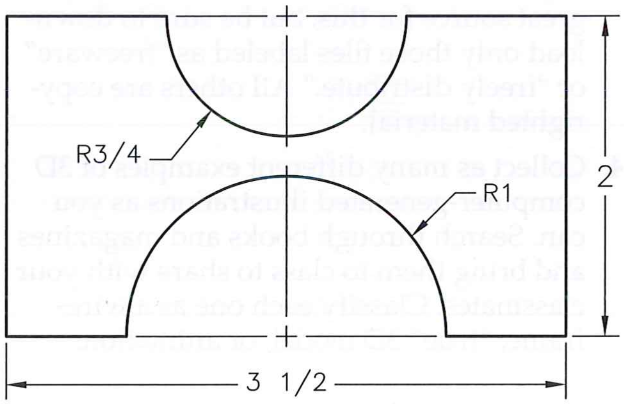
Using standard drafting practices and a CAD system, draw the following problems. The problems are classified as introductory, intermediate, and advanced. A drawing icon identifies the classification.

Draw the problems at an appropriate scale and use a proper sheet size for each drawing. Create layers as needed and assign linetypes based on standard drawing conventions. Use CAD commands and drawing aids to your advantage. Do not dimension these drawings.

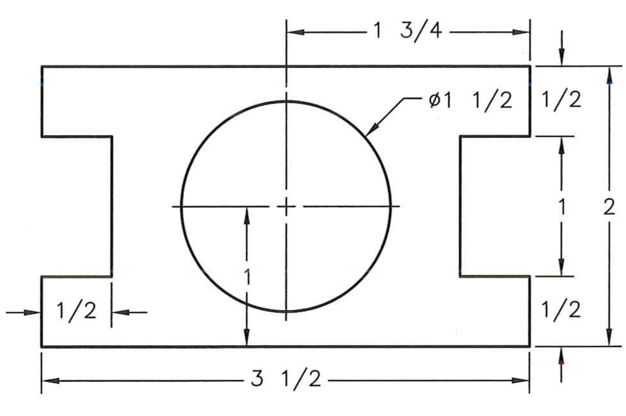
Save each drawing as a file named P4-(*problem number*). For example, save Problem 1 as P4-1.

Introductory

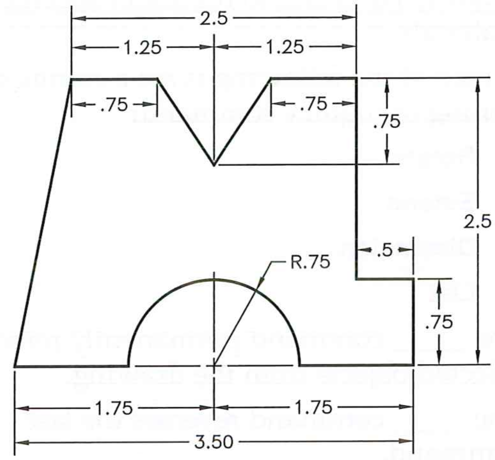
1. Template



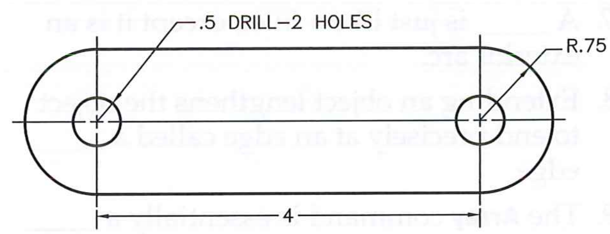
2. Hole Guide



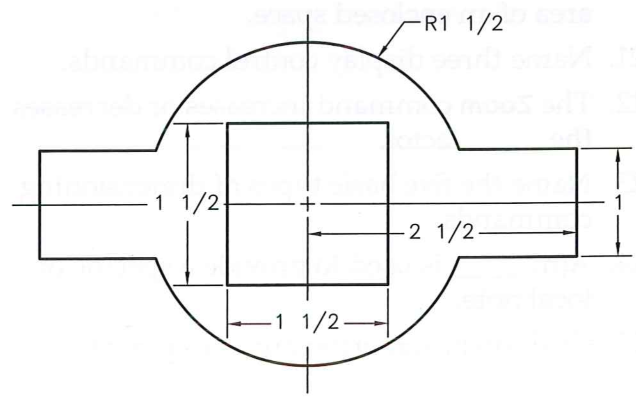
3. Cutting Guide



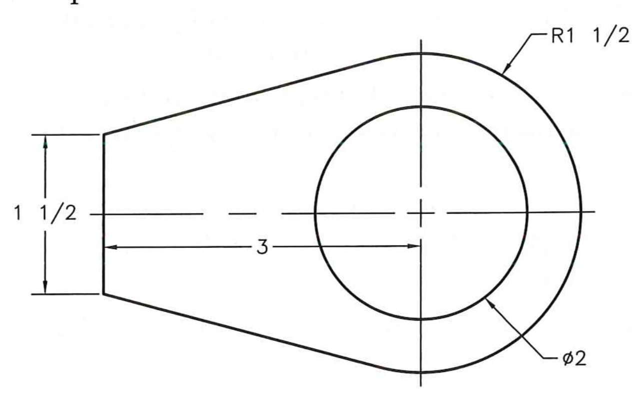
4. Clamp



5. Tab Lock

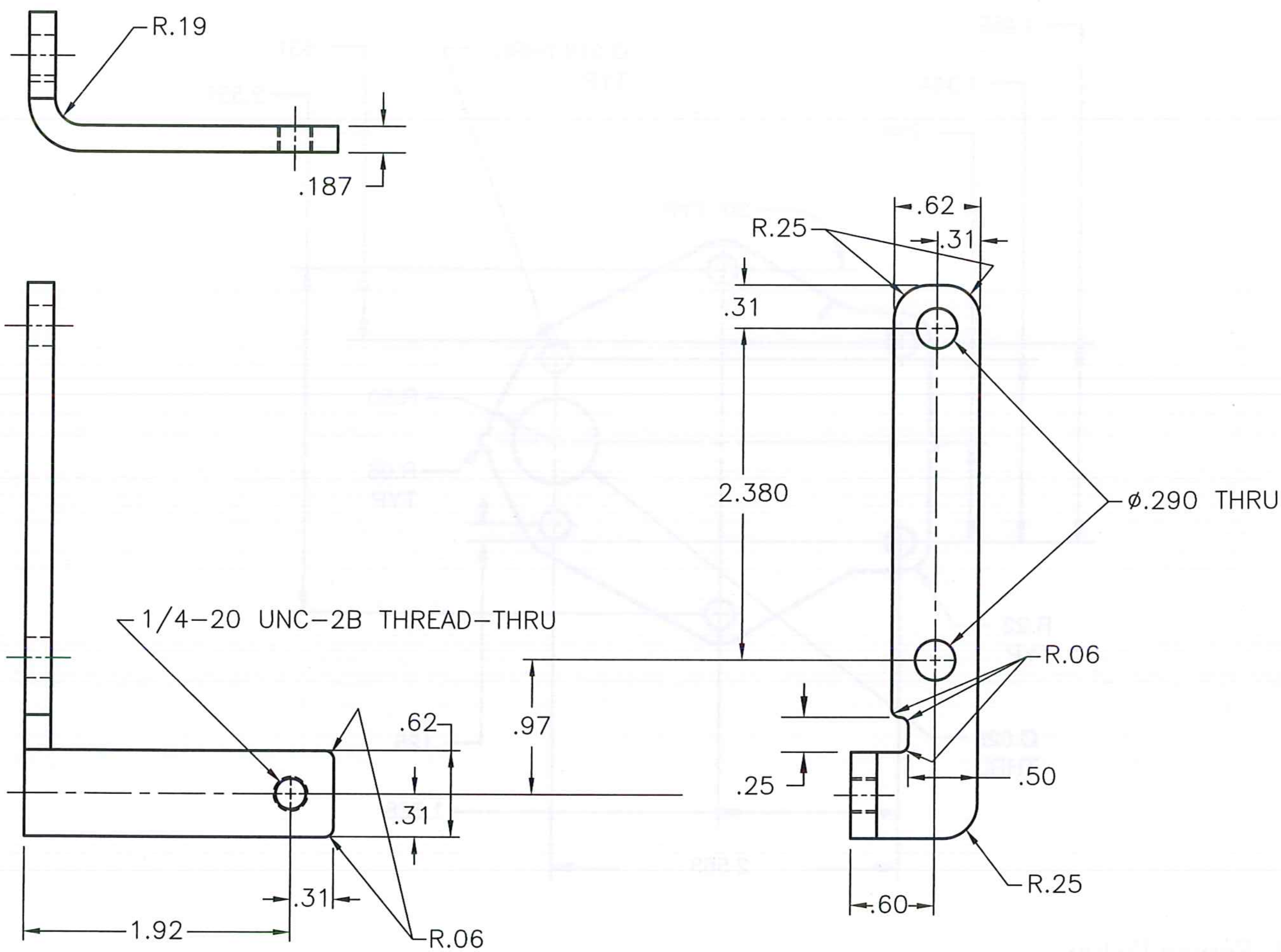


6. Spacer



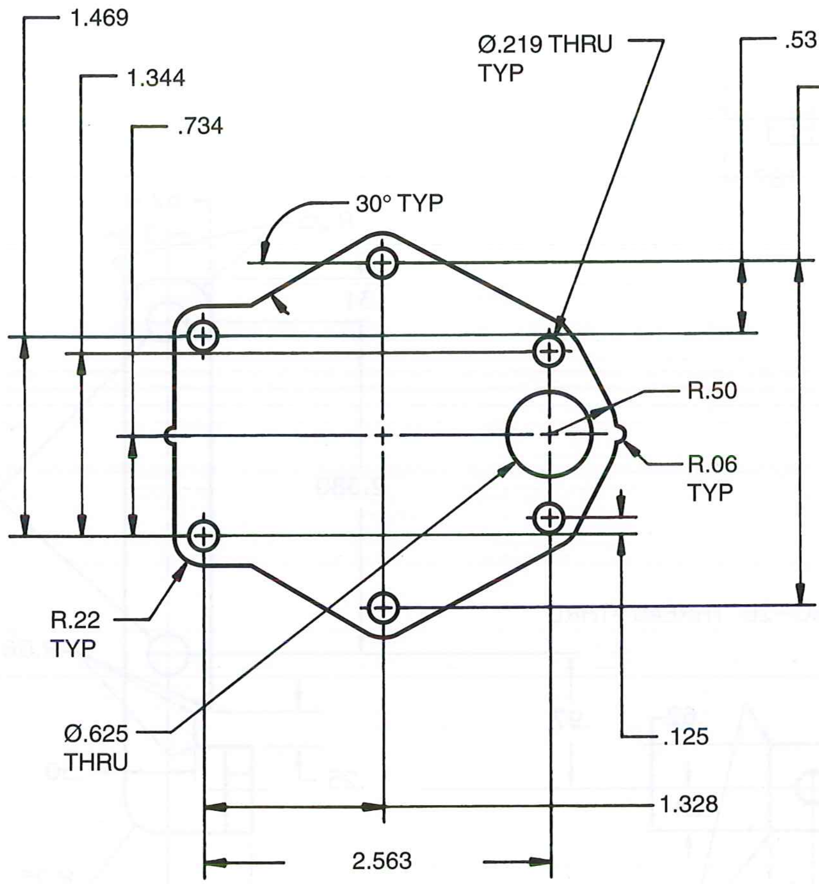
Intermediate

7. Bracket

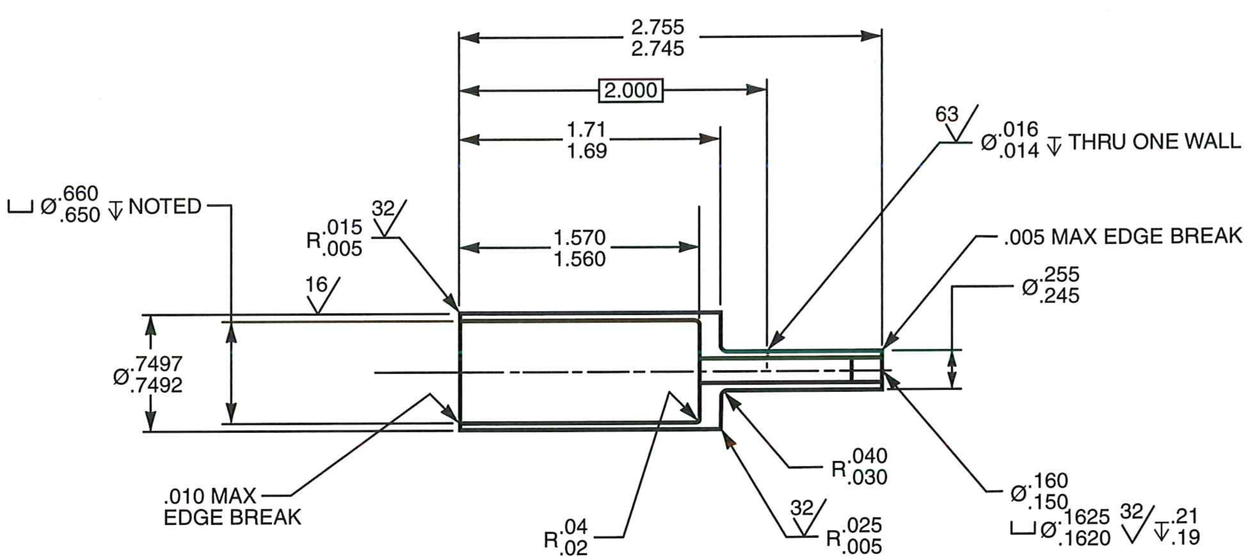


Intermediate

8. Diaphragm

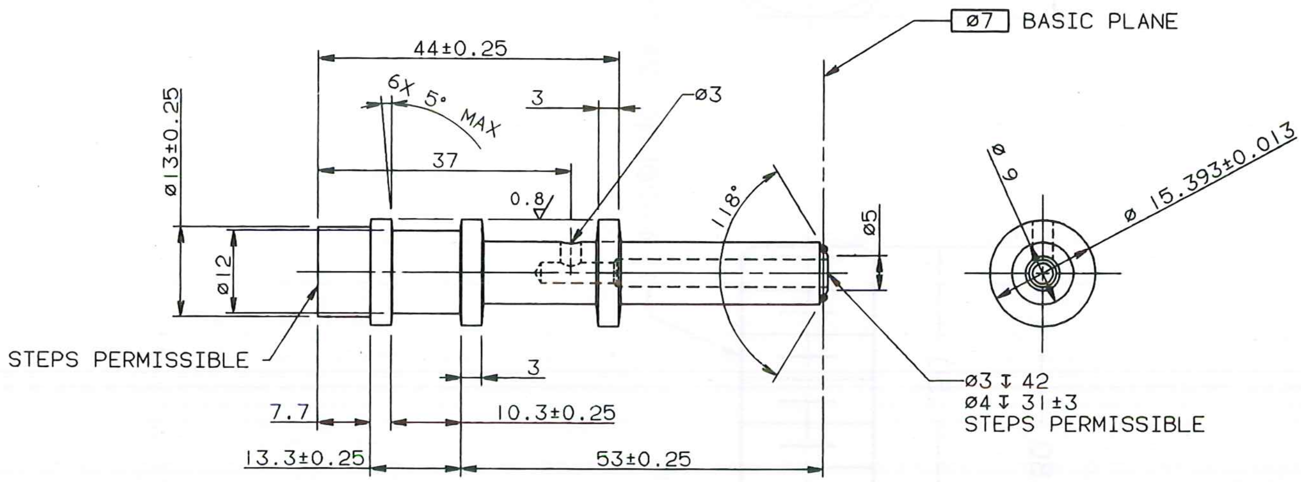


9. Return Piston

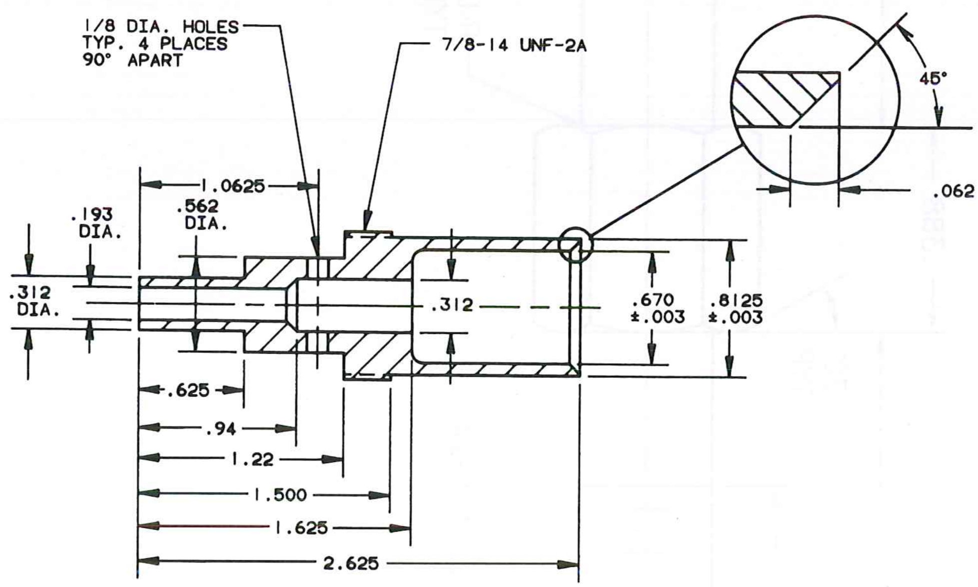


Advanced

10. Shuttle Valve



11. Torch Fitting Adapter



Advanced

12. Stud