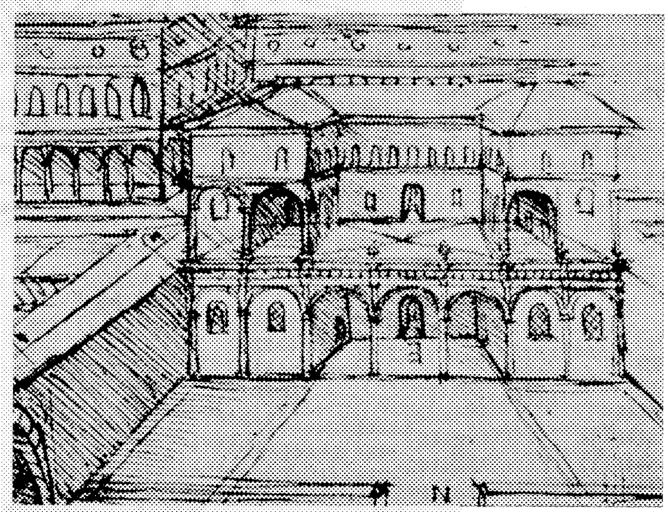
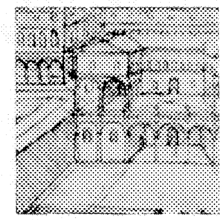
# DRAWING BASICS





# MULTIVIEW DRAWINGS

Chapter 10



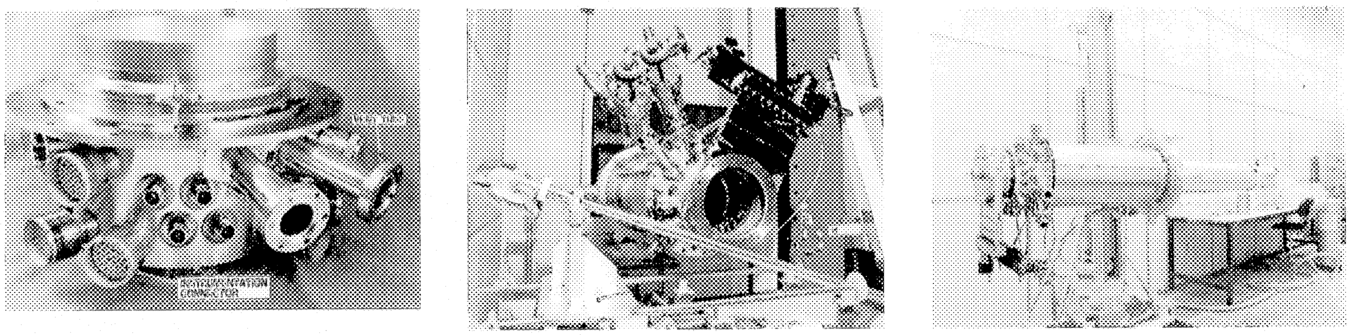
**LEARNING OBJECTIVES**

Upon completion of this chapter you will be able to:

1. Recognize the importance of orthographic projection for describing part features graphically.
2. Differentiate between first- and third-angle projection.
3. Identify the six standard views.
4. Demonstrate the ability to select a part's orientation and the number of views needed for complete feature description.
5. Produce multiview drawings demonstrating standard line precedence.
6. Understand partial, revolved, and enlarged views.
7. Describe methods for representing holes, fillets and rounds, tangent surfaces, runouts, and threads.
8. Integrate standard multiview projection methods into the CAD environment.

## 10.1 INTRODUCTION

**Multiview drawings** using orthographic projection are the primary means of graphic communication in engineering work. Drawings convey ideas, dimensions, shapes, and procedures for the manufacture of a part or the construction of a project. All manufactured parts require the creation of a database to document and produce the item. This is true for manually drawn and for CAD-generated parts. In some cases such, as the Dodge Neon, no paper drawings are output. But even in that case, though design and manufacturing relied on the CAD database to produce the car, every part was modeled and documented using engineering standards. In most cases the production process will require an actual drawing created manually on paper, or more typically created on a CAD/CAM/CAE system and plotted on paper. ANSI-standard views and dimensioning are used on all engineering drawings regardless of the design method. The gold-plated Top Hat Gravity Probe subassembly in Figure 10.1(a) required a number of separate detail drawings. The precision tools—jigs, fixtures, etc.—shown in Figure



(a) (b) (c)

FIGURE 10.1 Top Hat of Gravity Probe

10.1(b) and (c) also required modeling and detailing of each part before the Top Hat subassembly could be assembled with mating parts and other subassemblies.

**Orthographic projection** is a way to describe a part's shape and dimensions completely with two or more views, which are projected at 90° to each other. All engineering drawings are completed via this method of projection. The finished drawing is then reproduced and sent to the shop for manufacture or to the job site for construction.

With the widespread availability of computer-aided drafting and design (CAD), computers (Fig. 10.2) are now used to design and engineer most of the projects that formerly were drawn by hand. Regardless of whether you work on a 3D or a 2D CAD system, you must still understand the theory of orthographic projection and be able to apply it to the creation of multiview drawings. 2D CAD systems employ the same general techniques involved in drawing manually.

The two primary methods for demonstrating orthographic projection are the **natural method** and the **glass box method.** The natural method is typical of mechanical engineering (Fig. 10.3) and other engineering fields. The glass box method, used for descriptive geometry and in teaching orthographic projection, requires you to imagine that the part's points, lines, planes, etc. are enclosed in a transparent "box" (Fig. 10.4). Views of the part are established on their corresponding glass box surfaces by means of perpendicular projectors originating at each point of the part and extending to the related box surface. The box is hinged so that it can be unfolded onto one flat plane (the paper).

When the top, front, and side views are involved, each view has something in common with the other two views. The front view of a part shows the height and width, the top view the depth and width, and the side view the depth and height. Therefore, the width dimension will vertically align the top and front views, and the height dimension will horizontally align the front and side views. This method requires that the part be viewed perpendicular to each of its three primary surfaces, changing the position of the observer for each view.

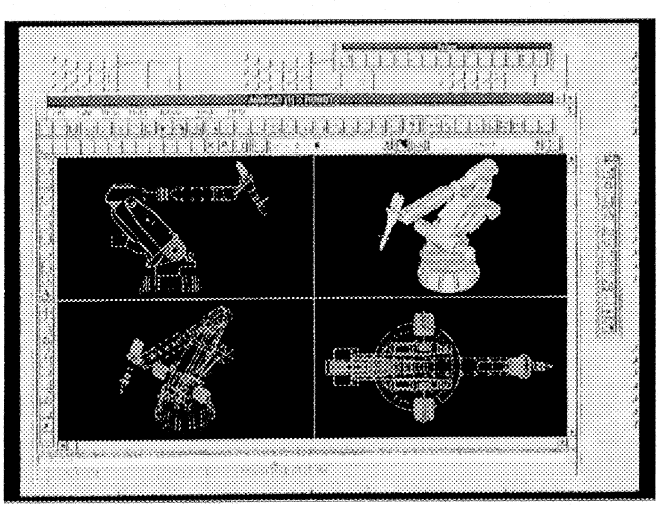


FIGURE 10.2 Displaying Views on a CAD System

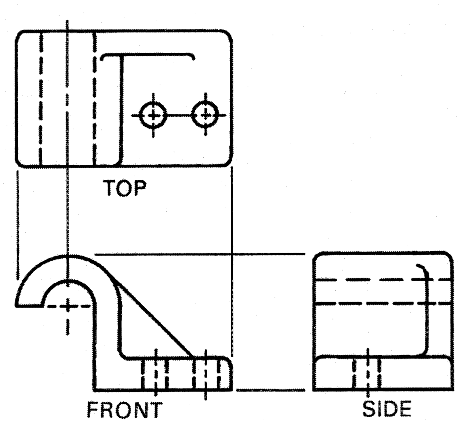
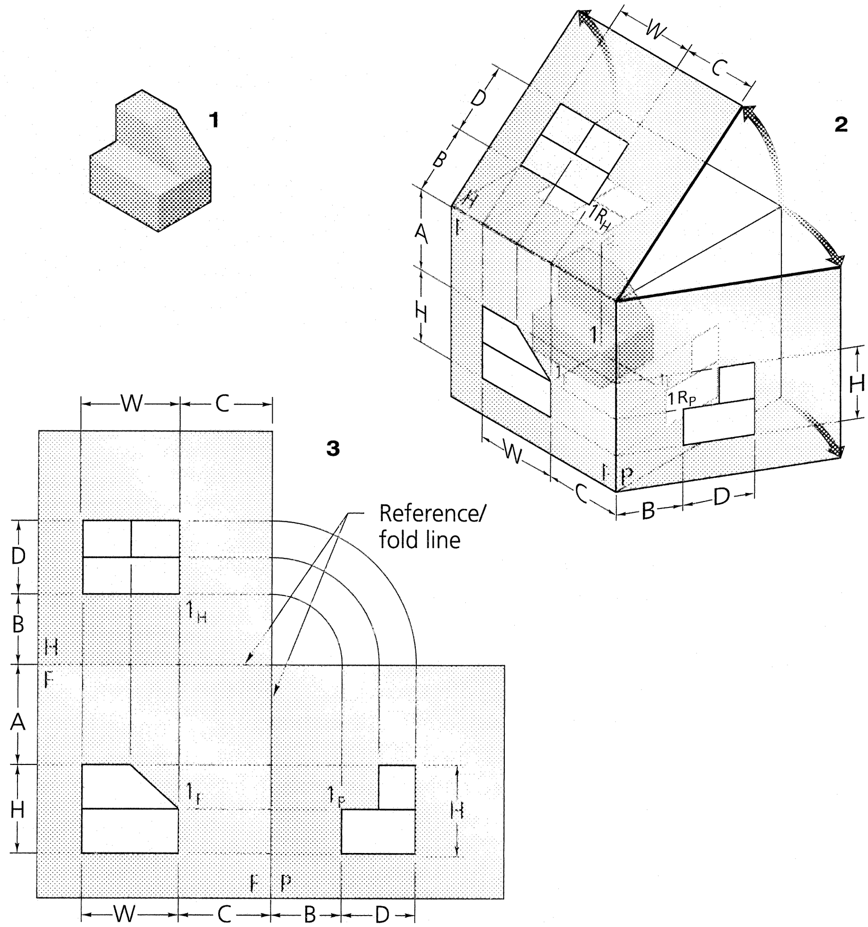



FIGURE 10.3 Three-View Drawing

FIGURE 10.4 Multiview Drawing




## 10.2 ORTHOGRAPHIC PROJECTION

**Orthographic projection** may be defined as a system of drawing composed of images formed by *projectors* extended from a part perpendicular to the desired planes of projection. The figure outlined on one of the projection planes, called an **orthographic view,** shows the true size and shape of a surface parallel to the projection plane (area ABCD with hole in Fig. 10.5). If an area is not parallel to the plane, the view of the area will be foreshortened (area BCEF in Fig. 10.5).

The glass box method of projection for a part is illustrated in its closed (folded) position and open (unfolded) position in Figure 10.6. The part has been theoretically enclosed in the transparent box. The following concepts are used throughout this chapter and the text.

*Lines*

**A = Vertical lines of sight**

**B = Horizontal lines of sight**

**C = Projection lines**

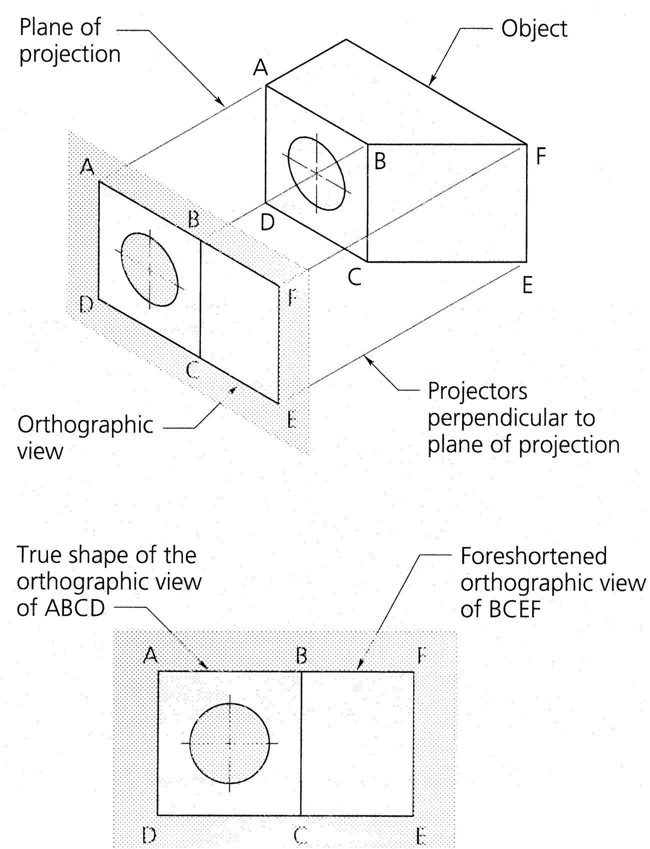



FIGURE 10.5 Third-Angle Orthographic Projection

*Dimensions*
**D = Depth**
**H = Height**
**W = Width**

*Image Planes (Principal Projection Planes)*
**F = Front (frontal plane)**
**H = Top (horizontal plane)**
**P = Side (profile plane)**

### 10.2.1 Line of Sight

When a part is projected onto an image plane, it creates a **"view"** of that part. *The lines of sight represent the direction from which the part is viewed* (Fig. 10.6). The vertical lines of sight (A) and horizontal lines of sight (B) are assumed to originate at infinity. The **line of sight** is always perpendicular to the image (projection) plane, represented by the surfaces of the glass box (top, front, and right side). **Projection lines** (C) connect the same point on the image plane from view to view, always at right angles to the adjacent view.

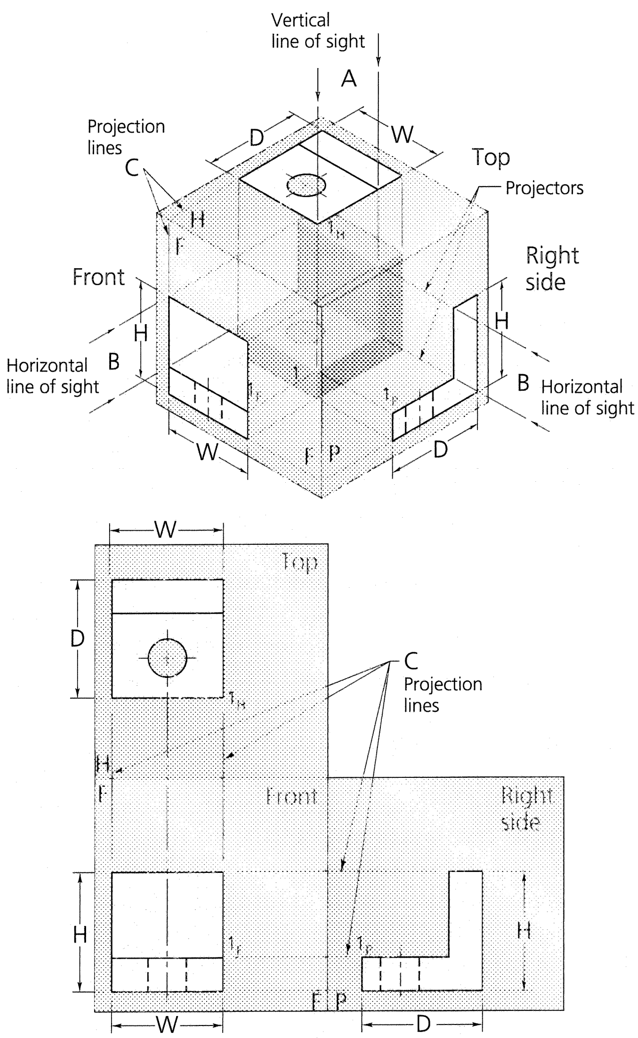



FIGURE 10.6 Orthographic Projection of a Part

A point is projected onto the image plane where its line of sight pierces that image plane. In Figure 10.6, point 1, which represents a corner of the part, has been projected onto the three primary image planes. Where it intersects the horizontal plane (top image plane), it is identified as $1_H$. Where it intersects the frontal plane (front image plane), it is identified as $1_F$. Where it intersects the profile plane (right side image plane), it is labeled $1_P$. The multiview drawing in Figure 10.6 shows the position of the unfolded image planes, which now lie in the same plane as the paper.

In Figure 10.7(a), the line of sight for each view is shown. These lines of sight establish the direction of viewing that the observer will take when completing the view. Figure 10.7(b) shows the three views properly aligned. In Figures 10.7(c), (d), and (e), the top, front, and side views are broken apart and analyzed separately. All points on each surface of the part are projected onto their corresponding image plane (view). The view of the part is created where these projectors pierce the image plane.

## 10.3 THE SIX PRINCIPAL VIEWS

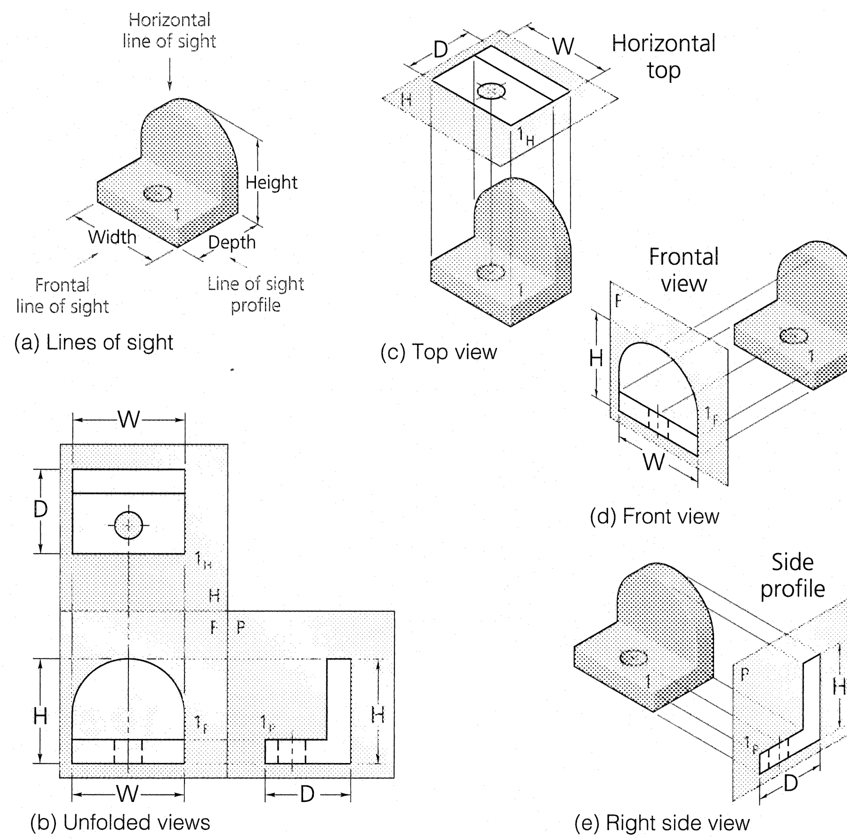
When the glass box is opened, its six sides are revolved outward so that they lie in the plane of the paper. Except for the back plane, all are hinged to the front plane. The back plane is normally revolved from the left side view, but it can also be hinged to the right side view. Each image plane, before it is revolved around its hinged fold line (reference line), is perpendicular to its adjacent image plane and parallel to the image plane across from it.

A **fold line** is the line of intersection between any hinged (adjacent) image planes. The left side, front, right side, and back are all **elevation views.** Each is vertical. In these views, the height dimension, elevation, and top and bottom of the view can be determined and dimensioned. The top and bottom planes are in the horizontal plane. The depth dimension, width dimension, and front and back are established in these two horizontal planes.

In most cases, the top, front, and right sides are required. These are sometimes referred to as the horizontal plane, H (top); frontal plane, F (front); and profile plane, P (side). These planes are the three **principal projection planes,** or **views.**

In Figure 10.8(a), the glass box is shown pictorially before it is revolved. The top, front, and bottom are in line vertically, and the left side, front, right side, and back are aligned horizontally. An exception to this alignment is when the glass box is revolved around the top (horizontal) view. This rotation is advantageous when the part has much greater depth than height.

When using directions to establish the location of a point, a line, etc., the top and bottom are shown in the frontal plane; the terms "above" and "below" can also describe directions in this plane. The horizontal view can help to determine if a point is "in front of" or "in back of" a




IGURE 10.7 Line of Sight

articular starting point or fold line. To locate a point to the ght or left of a fold line or an established point, the frontal r horizontal plane can be used. In the profile plane, the top, ottom, front, and back can be determined. In Figure 0.8(b) and (c) a part is shown in each of the six standard iews (and pictorially enclosed in a glass box). The six andard views are shown in first-angle and third-angle rojection.

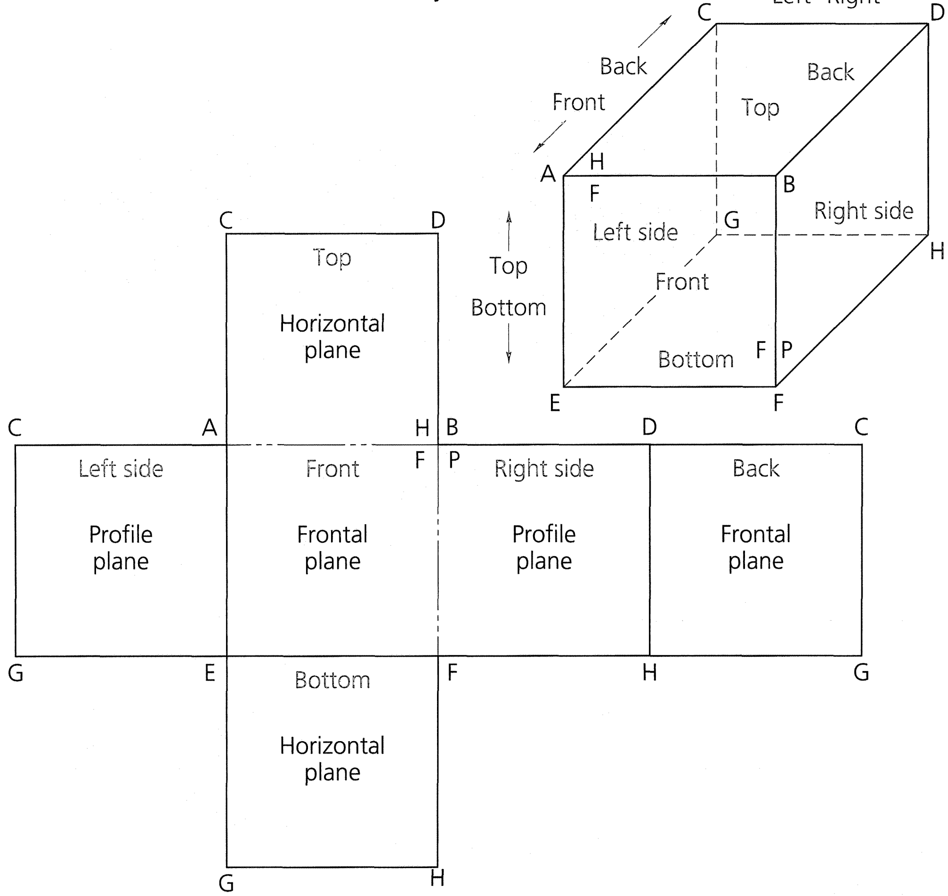
## 10.4 FIRST- AND THIRD-ANGLE PROJECTION

wo types of orthographic projection are employed in idustry throughout the world: **first-angle** and **third-angle rojections.** The six principal views of a part, or the glass ox, have been presented in the type of orthographic rojection known as *third-angle orthographic projection.* This orm of projection is used throughout the United States and anada and is the primary form of projection in all of American industry. In third-angle projection, the line of sight goes through the image plane to the part. To obtain views of the part, you must assume that the part is projected back (along the lines of sight) to the image plane. Projection lines serve to show this projection from the part to where they intersect the image plane. Figure 10.8(c) illustrates third-angle projection and the normal procedure for unfolding the glass box.

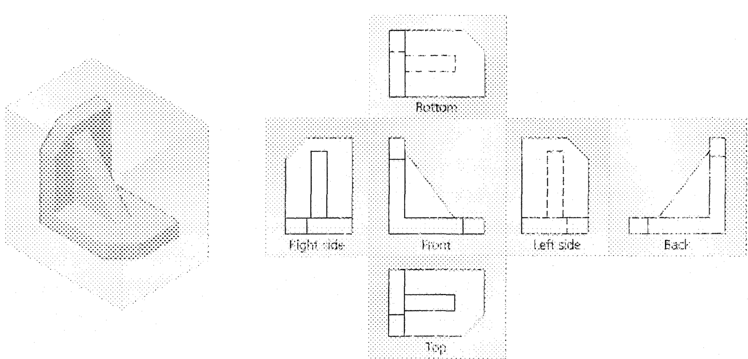
**First-angle orthographic projection** is common in most countries apart from the United States and Canada [Fig. 10.8(b)]. In this form of projection, the part is assumed to be in front of the image plane. Each view is formed by projecting through the part and onto the image plane. Figure 10.9 compares first- and third-angle projection.

In Figure 10.10(a), the four quadrants and their corresponding angles of projection are shown. A simple part is placed in the first quadrant in Figure 10.10(b). This is the quadrant used in first-angle projection. In Figure 10.10(c), the same part is placed in the third quadrant, as would be appropriate for third-angle projection. The glass box is added and the quadrants removed in Figure 10.10(e). Here, the part resides inside the glass box and is ready for projection. Figure 10.10(f) illustrates how the top, front,

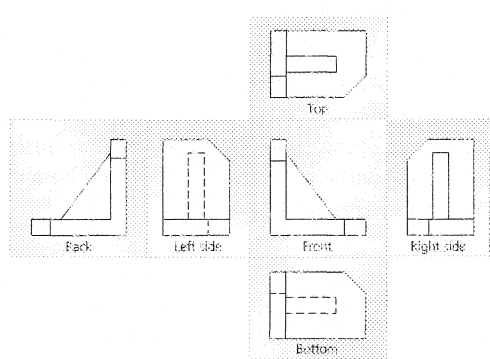
**FIGURE 10.8 Standard Views and Projection**




(a) Six standard views




(b) First-angle projection




(c) Third-angle projection

and side views are projected onto the glass box. The **six standard views** are established by the six directions of sight [Fig. 10.10(g)]: top, front, right side, left side, rear, and bottom. We have begun to unfold the glass box (with its corresponding projections of each of the six sides) in Figure 10.10(h). The unfolded position of the glass box is shown in Figure 10.10(i). This is the true projection of all six sides

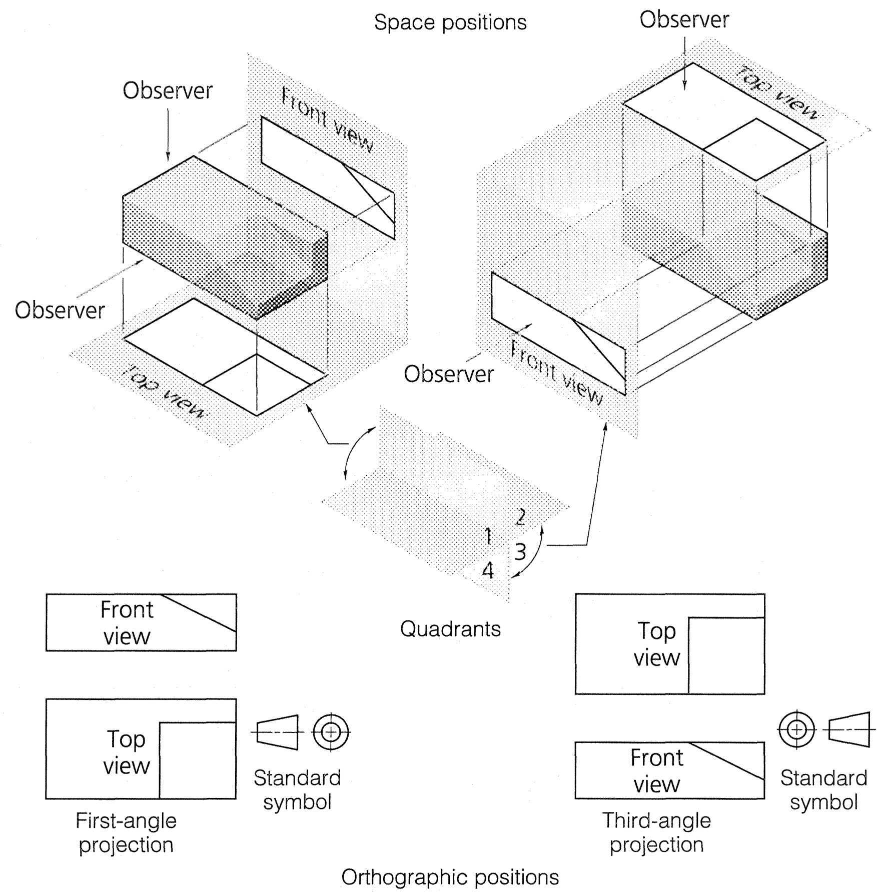



FIGURE 10.9 **First- and Third-Angle Projection**

using third-angle projection. The first-angle projection of this same part is shown in Figure 10.10(j). The part's left side view is drawn on the right side of the part. The top view is placed below the front view; the bottom view is placed above the front view.

The internationally recognized **ISO projection symbols** for first- and third-angle projections are shown on drawings as in Figure 10.9. Identifying symbols are required on drawings so that they can be understood and interchanged internationally. The symbol is normally placed to the left of the title block, as in Figure 10.11. This text uses third-angle projection exclusively.

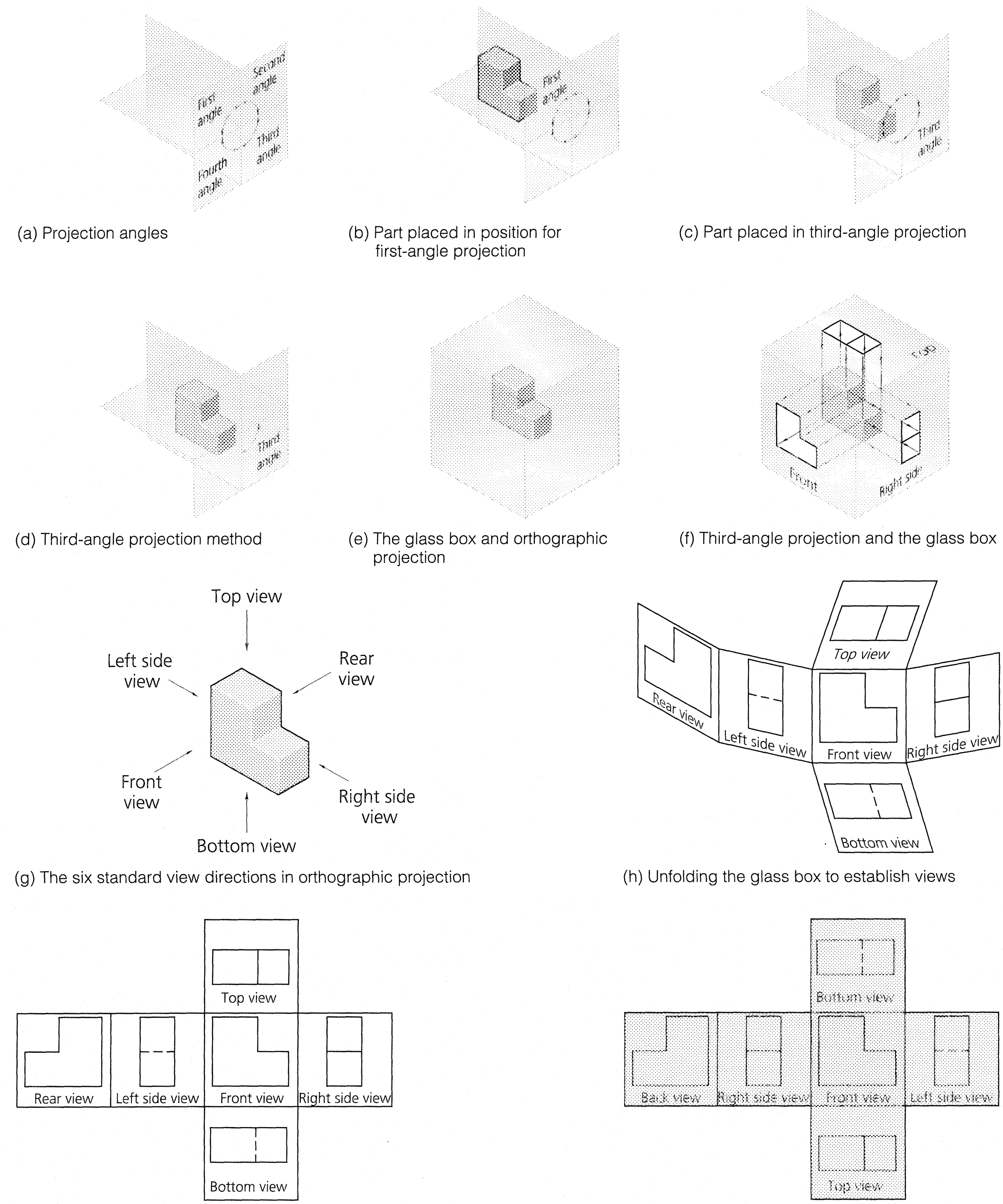
## 10.5 MULTIVIEW DRAWINGS

**Multiview drawings** represent the shape of a part through two or more views. These views, together with the necessary notes and dimensions, are sufficient for fabrication of the part without further information concerning its shape. Consideration is given to the choice and number of views so that all surfaces are shown in their true shape and with a minimum of confusion.

Four basic types of drawings are found in engineering work; one-, two-, three-, and multiple-view. The choice of how many views are needed is determined by the shape and complexity of the part. One view can be sufficient to describe many types of parts. You must draw as many views as are necessary to describe the part completely. The four types of drawings are:

*One-view drawings* [Fig. 10.12(a) and (b)] Two adjacent views are normally considered the minimum requirement to describe a three-dimensional part. However, the third dimension of some parts (washers, shafts, bushings, spacers, sheet metal parts, etc.) may be specified by a note giving the thickness or dimensions for the diameter.

*Two-view drawings* [Fig. 10.13(a) and (b)] Many parts may be adequately described by showing only two views. These views must be aligned in any standard position that will clearly illustrate the part. In Figure 10.13(a), the side view was necessary to describe and dimension the part.




(a) Projection angles

(b) Part placed in position for first-angle projection

(c) Part placed in third-angle projection

(d) Third-angle projection method

(e) The glass box and orthographic projection

(f) Third-angle projection and the glass box

(g) The six standard view directions in orthographic projection

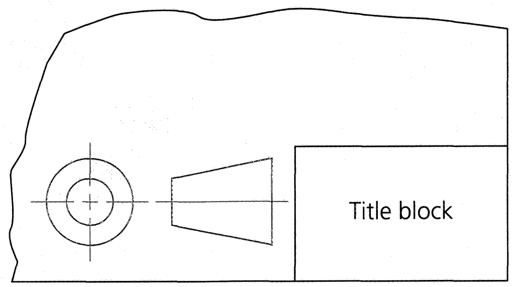
(h) Unfolding the glass box to establish views

(i) Third-angle projection and the six standard views

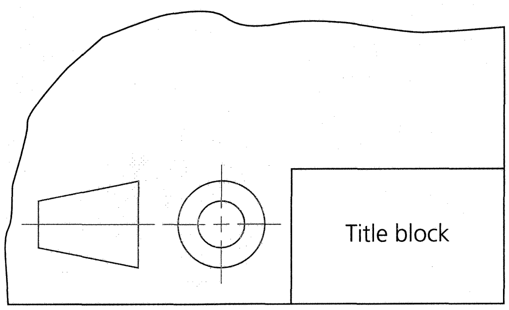
(j) First-angle projection and the six standard views

**FIGURE 10.10 Projection Theory**

*Three-view drawings* [Fig. 10.14(a) and (b)] Most drawings consist of front, top, and side views arranged in their standard positions. Any three adjacent views that best suit the shape of the part may be drawn. Each view of the part shows features that could not be graphically described in any of the other views. The holes show in the top and the front views, and the slot and angled surfaces show in the right side view.




(a) Third-angle projection symbols




(b) First-angle projection symbols

**FIGURE 10.11 Projection Symbols**

*Multiple-view and auxiliary-view drawings* (Fig. 10.15) When a part cannot be defined graphically with one, two, or three views, a multiple-view drawing may be required. The part shown here required four views to describe its configuration properly.

### 10.5.1 Choice and Orientation of Views on a Drawing

The first step in any drawing is deciding which views of the part should be drawn and dimensioned. Because dimensioning is not covered in this chapter, it will be somewhat difficult to estimate the space and view needs of a part. Alternate positions of views may be made to conserve space or to position dimensions, but they must be properly oriented to each other. For example, the right or left side might be placed adjacent to, and in alignment, with the top view. The rear view is sometimes placed in alignment with, and to the right of, the right side view. Under certain conditions, it may be impractical to place views in the normal aligned positions or even on the same sheet. Before starting the drawing, you must analyze the configuration of the part and its view requirements. The proper decisions at this stage will reduce drawing time, provide a clearer and more concise arrangement of views, and reduce the cost of the final drawing.

A part is normally shown in a **natural** or **assembled position.** The minimum number of views necessary to describe the part is established first. Views are selected that will show the fewest hidden lines and yet convey maximum clarity. In general, since the part will be mounted or sit on a surface, the top view is obvious. The choice of top view may also be determined by the machining process and its complexity.

The front view should normally be the longest orientation of the part. In Figure 10.16, the part requires three views. It could not have been adequately described without all three views. The top view choice is obvious. The front view is the longest orientation, and the right side view is necessary to describe the **V**-shaped cut.

### 10.5.2 Relationship of Views on a Drawing

The *relationship of views* on a drawing is determined by the choice of part orientation. In Figure 10.17(a), the six standard view directions of the part are labeled. In Figure 10.17(b), the views are laid out using third-angle projection. The placement and orientation of the top view determine that the front view will be the longest orientation, or principal shape, of the part. In Figure 10.17(c), the same part is shown slightly differently, but not incorrectly. Here, the part has been turned so that the front view will not show the part's longest orientation. In fact, the side views show the longest orientation [Fig. 10.17(d)]. Although this orientation is not incorrect, it is less acceptable than Figure 10.17(b). The longest orientation should be the front view so that the predominant dimension will be the width.

### 10.5.3 Spacing Views

After the number of views is determined, the next step is to establish the paper format size based on the part's size, the scale to be used, and the detailing requirements. Remember, the drawing must have space for views, dimensions, and notes.

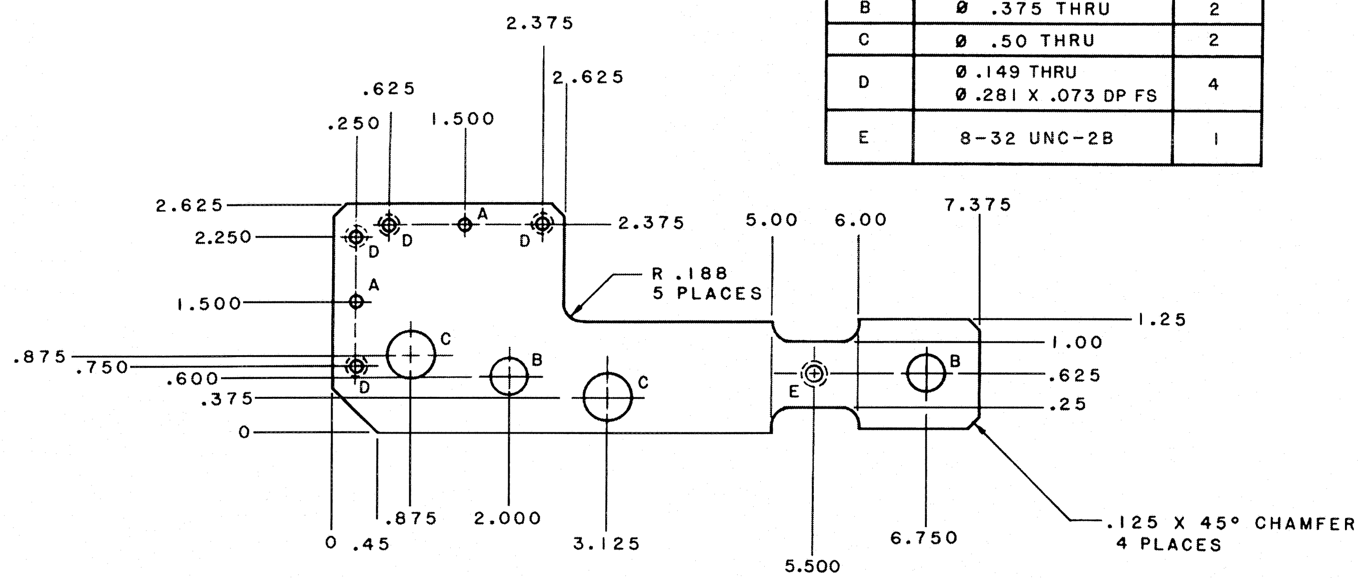
A simple method to determine roughly the sheet size is to add the dimensions of the part—width plus depth (if a side view is required), which gives the total width of the views. Add extra space for separation of the views and for a margin at each border. The height requirements of the drawing can be determined by adding the height of the part to its depth. Then add some space for the separation of the views and for the margin at the top and bottom borders. The drawing format—A, B, C, D, E, or larger—is determined by these dimensions and company/school practice.

In Figure 10.18, the part has been laid out on the sheet using the preceding formula. The height, the depth, the distance between the lower border and the front view (A), the space between the front and top views (B), and the space between the top view and the border (C) were added together to establish the vertical requirements of the drawing. The width, the depth, the space between the left border and the front view (D), the space between the front and the right side views (B), and the space between the right side

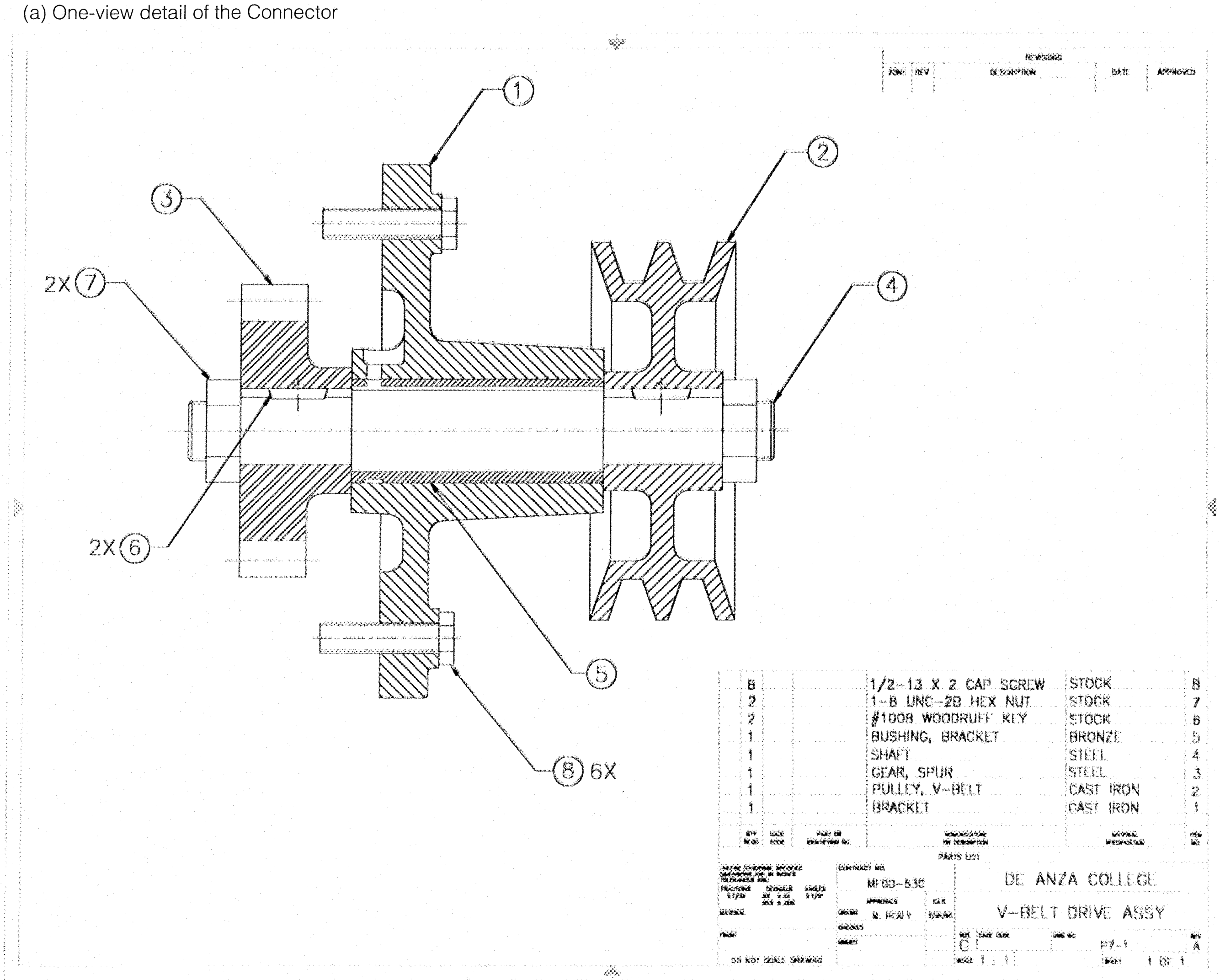
FIGURE 10.12 One-View Drawings

.25 ALY ALUM ANODIZE BLACK

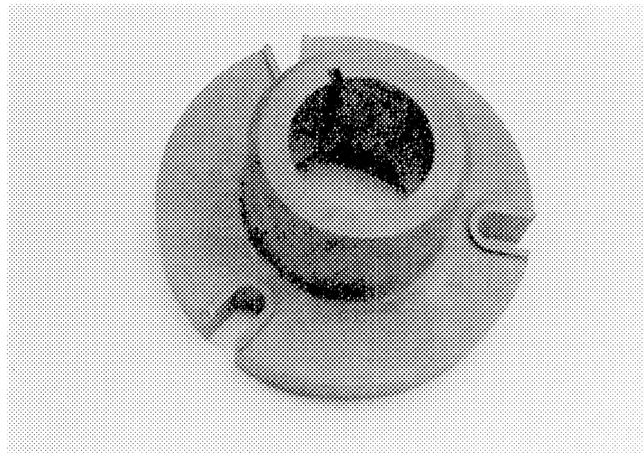
| HOLE | DESCRIPTION | QTY |
|---|---|---|
| A | Ø .125 THRU | 2 |
| B | Ø .375 THRU | 2 |
| C | Ø .50 THRU | 2 |
| D | Ø .149 THRU<br>Ø .281 X .073 DP FS | 4 |
| E | 8-32 UNC-2B | 1 |

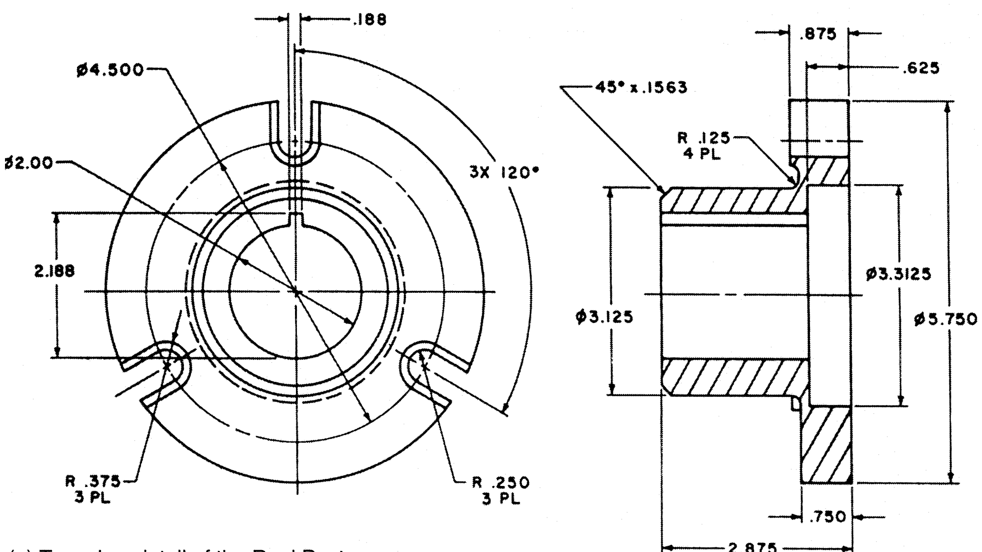



(a) One-view detail of the Connector

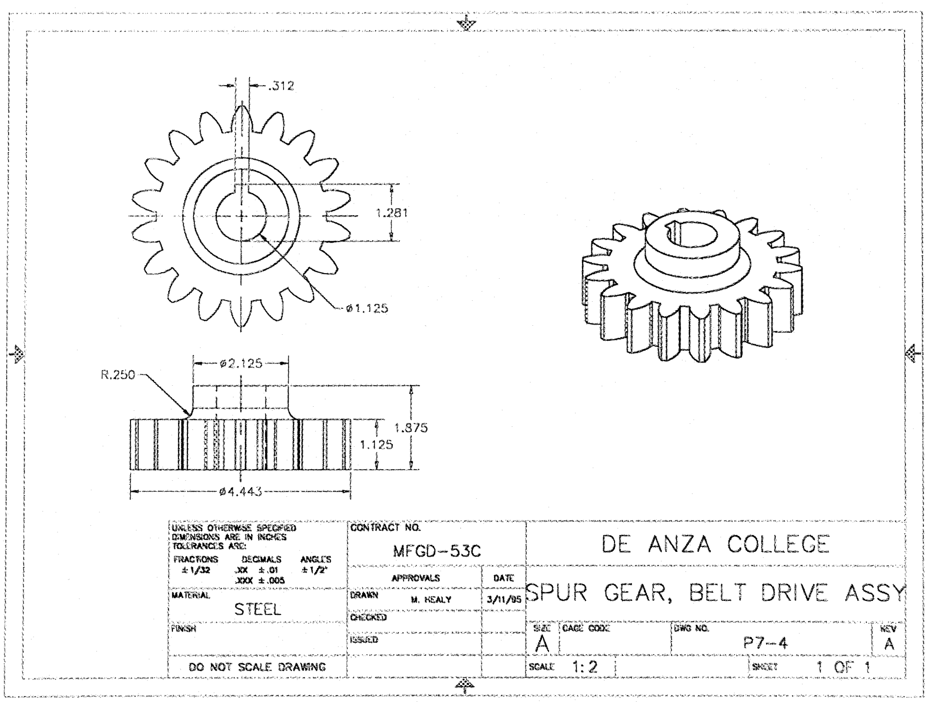



(b) One-view assembly drawing of V-Belt Drive drawn with a CAD system



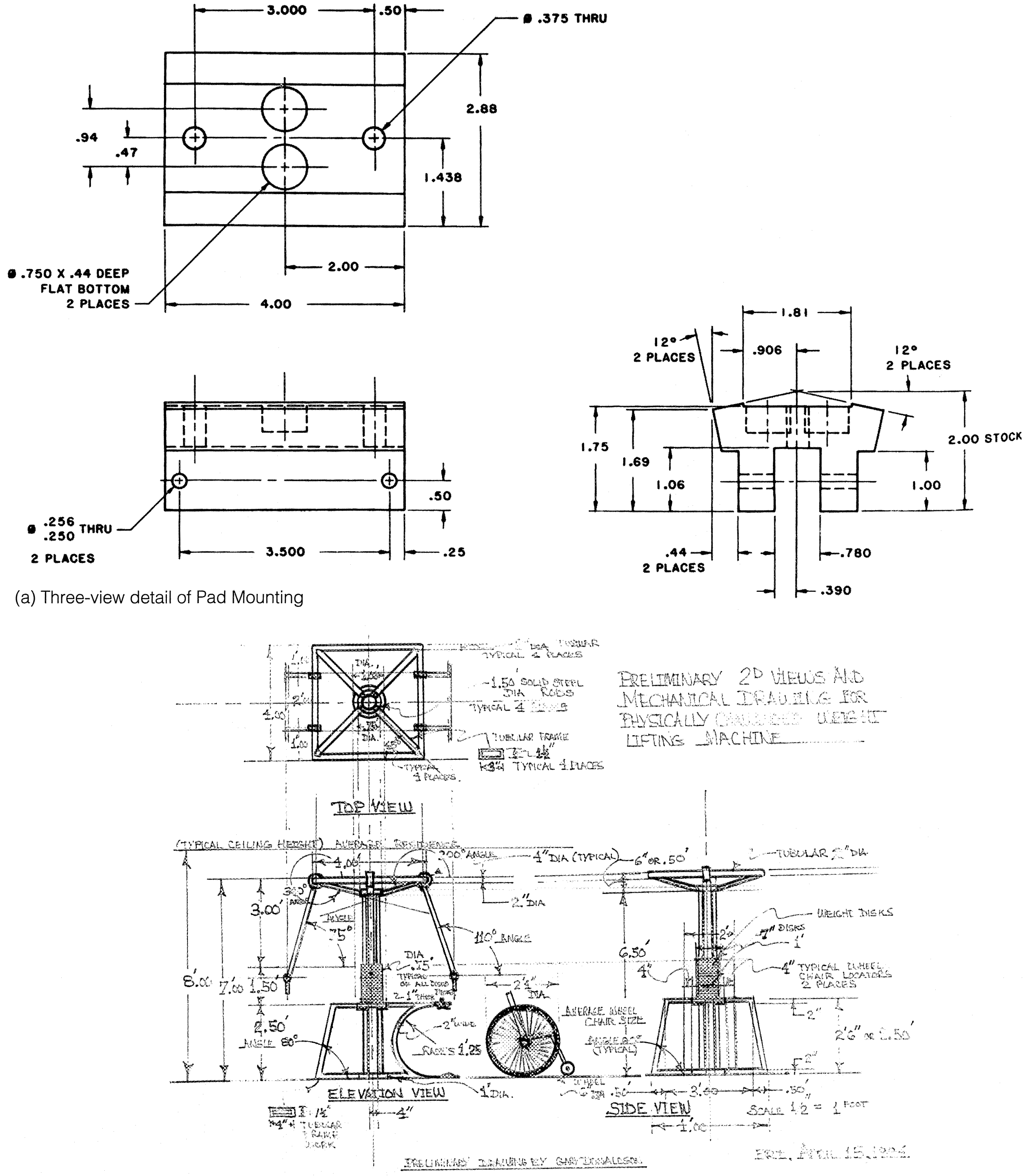



(a) Two-view detail of the Reel Post




ɔ) Two-view detail and model of Gear

ꞮGURE 10.13 Two-View Drawings




(a) Three-view detail of Pad Mounting

(b) Three-view assembly design drawing of physically challenged weight machine

**FIGURE 10.14 Three-View Drawings**

view and the right border (E) were added to establish the horizontal requirements. Remember, dimensions A, B, C, D, and E were determined by the space required for dimensioning.

How much space is needed between the views is usually determined by the number of dimensions that will be placed in this area. In Figure 10.19, the shaded portion of the drawing shows the space between the top and front views and between the front and side views. Some texts suggest that these areas always be equal. However, this will not always be the case. If a number of dimensions must be placed between the top and front views, this area should be greater than that between the front and side views (unless, of course, a number of dimensions are also needed there).

The drawing is laid out by *blocking in* the views with

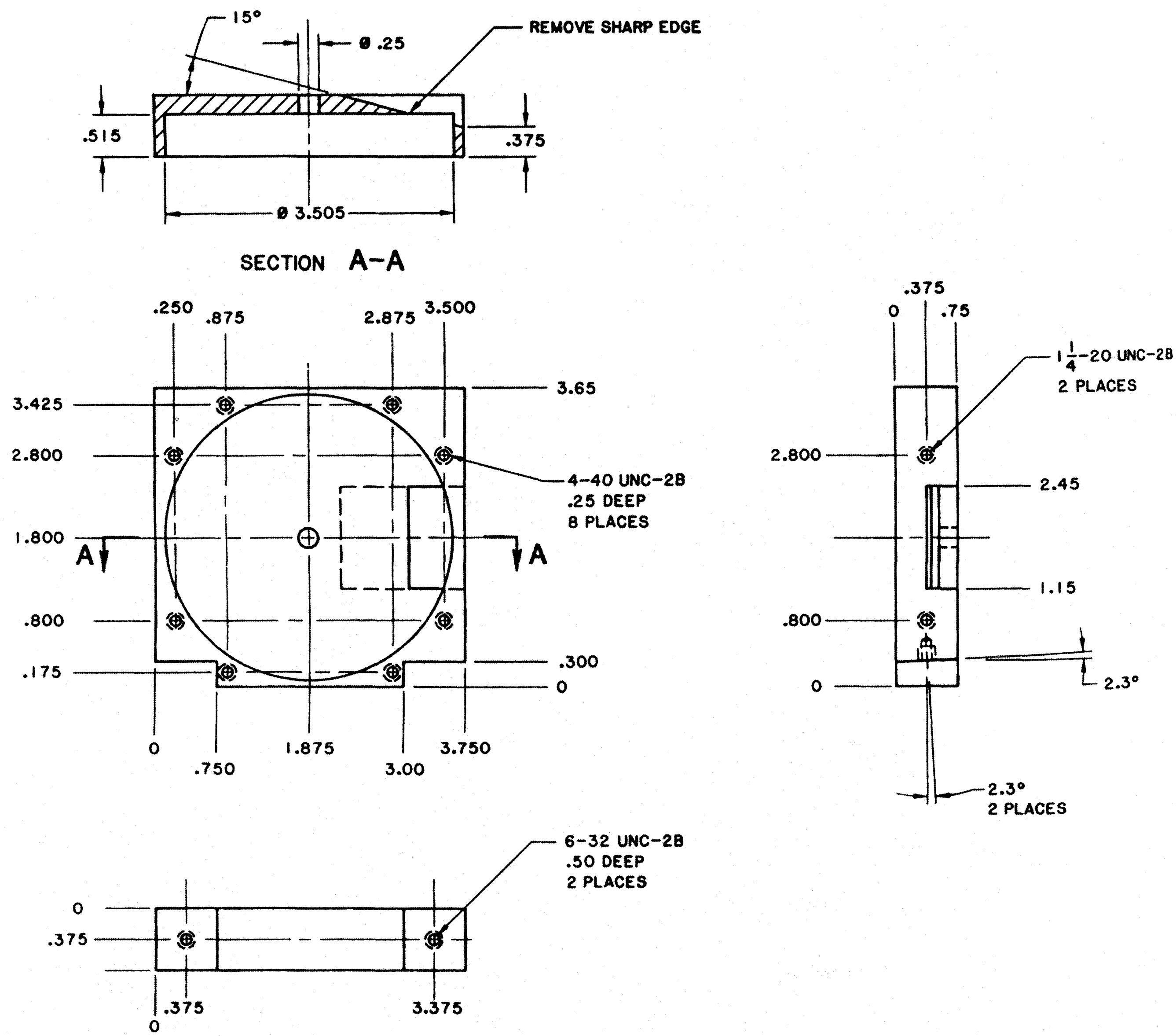
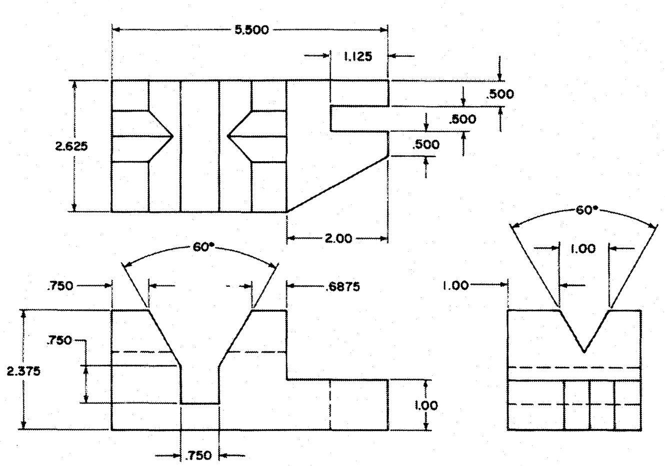



FIGURE 10.15 Top, Front, Back, and Side Views of the Interface Bracket



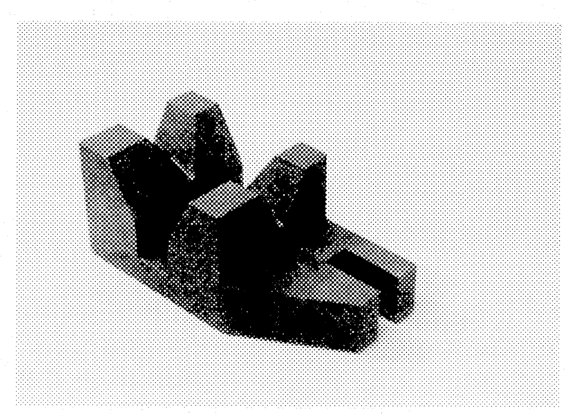
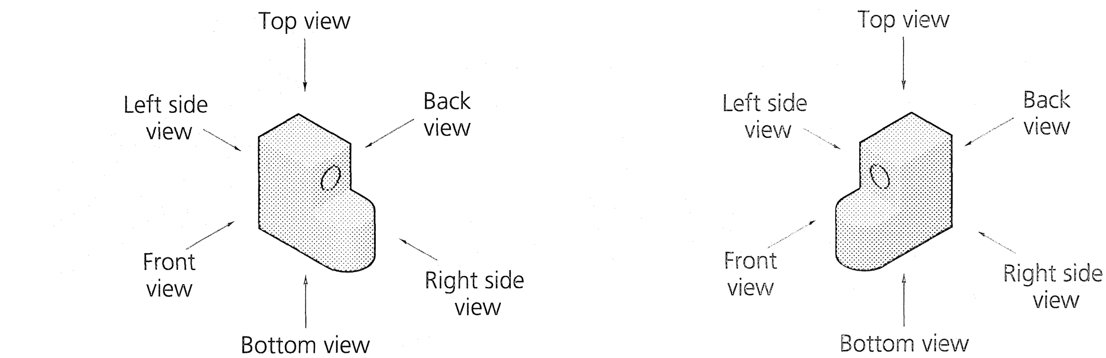
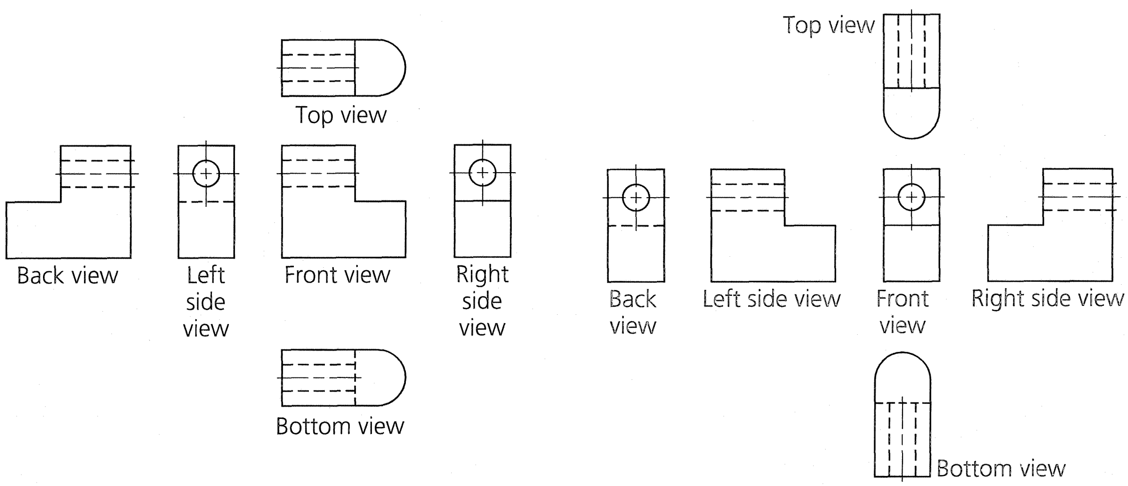



FIGURE 10.16 Three-View Detail of the Base Angle




(a) Six standard views of a part

(c) Alternative arrangement of a part in space




(b) Six standard views of a part using third-angle projection

(d) Alternative arrangement views of a part

**FIGURE 10.17 Views of a Part**

construction lines. At this stage of the drawing, changes are easily made in the spacing of the views and the general layout. After the construction lines are drawn, the circles and radii are darkened. Each part requires careful individual consideration. There are no hard-and-fast rules for drawing layouts. After some experience, you will understand intuitively a part's space requirements and adapt the drawing accordingly.

### 10.5.4 Related and Adjacent Views

Regardless of whether the drawing to be constructed is to have two, three, or more views, the basics of construction and projection are the same. Two adjoining orthographic views aligned by projection lines are considered **adjacent views.** Two views adjacent to the same intermediate view are called **related views.** Each view shares one dimension with a related

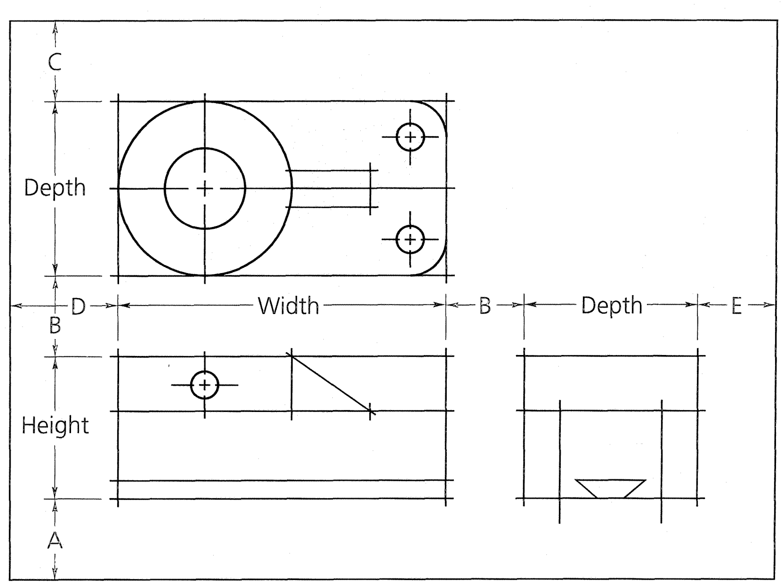
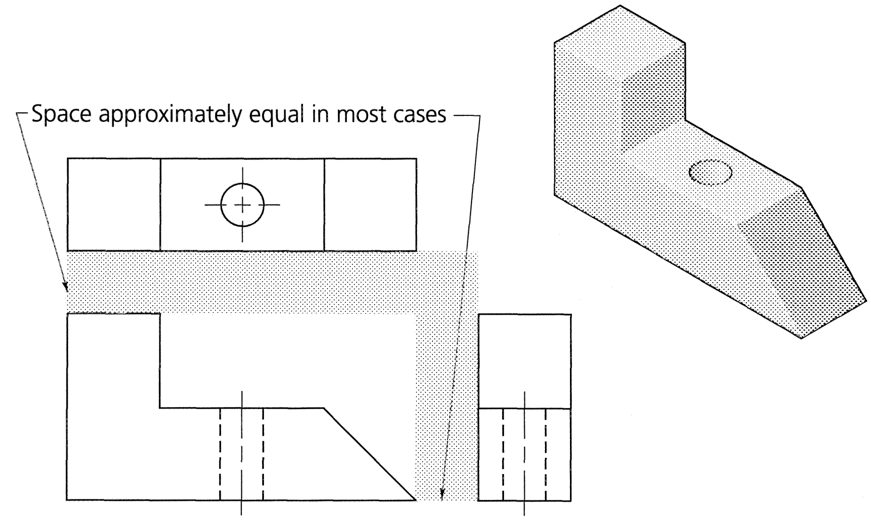



**FIGURE 10.18 Laying Out a Drawing**

**FIGURE 10.19 Spacing Views on a Drawing**




view and another dimension with an adjacent view (Fig. 10.20). The top and front views share one dimension—the width. The front and side views share the height dimension. The top and front views are therefore adjacent views, as are the front and side views. The top and side views share the depth dimension and are considered related views.

## 10.5.5 Drawing Order

Whether the project is a one-, two-, three-, or multiview project, the same sequence of construction will generally be applied. The order in which you do your work determines the efficiency and quality of the finished drawing. Figure 10.21 provides a series of steps in the construction of a drawing.

1. Figure 10.21(a) shows a pictorial view of the part. Based on the part's overall dimensions, establish the sheet size and format via the technique previously described. The scale and dimensioning requirements also have to be determined at this time. The number of views depends on the part's configuration and complexity and the dimensioning requirements. Sketching the possible view requirements and alternatives helps establish a well-planned drawing that requires fewer alterations at a later stage.
2. Based on the part's overall dimensions, lay out the principal dimensions to establish the three views. Use the scale to measure and establish the dimensions with small construction lines, as shown in Figure 10.21(b). Since dimensions are shared with adjacent views, it is necessary to scale only once for each of the three major dimensions. The width can be established in the top view and projected to the front view. The height can be established in the front view and projected to the side. Because the front view can be used for both the adjacent views—side and top—some designers prefer to draw the front view's outline first.
3. Using construction lines, connect the measured points to establish the outline of the part [Fig. 10.21(c)]. Since unneeded construction lines require erasing before darkening, draw only those construction lines that are necessary.
4. At this step [Fig. 10.21(d)], you need to use your scale to measure all secondary details of the part and establish them on the drawing. Measure from the existing principal lines. This step is also done with construction lines.
5. Draw all secondary features of the part [Fig. 10.21(e)]. To avoid more measuring, project features to adjacent views where possible.
6. Establish the centerlines and curved features of the part using construction lines [Fig. 10.21(f)]. The part's fillets and circles require centerlines for their construction. All curved features are drawn with the aid of a template or compass. On projects where the primary shape of the part is curved or where there are prominent circular features, this would be step 3 or 4. Do not darken the drawing yet. Check the drawing thoroughly before going on to the next step. Mistakes caught at this stage of the project, where there are no finalized (darkened) lines, are easily corrected.
7. It is easier to match a straight line to a curve than a curve to a straight line, so circles, arcs, and fillets are the first features darkened on a drawing [Fig. 10.21(g)].

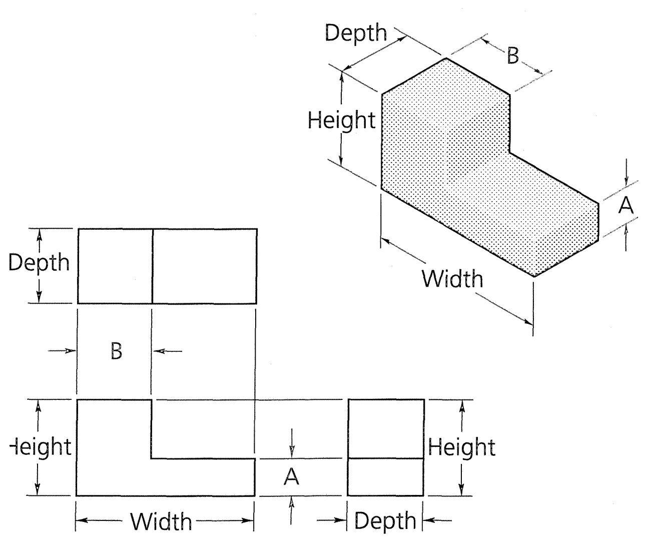
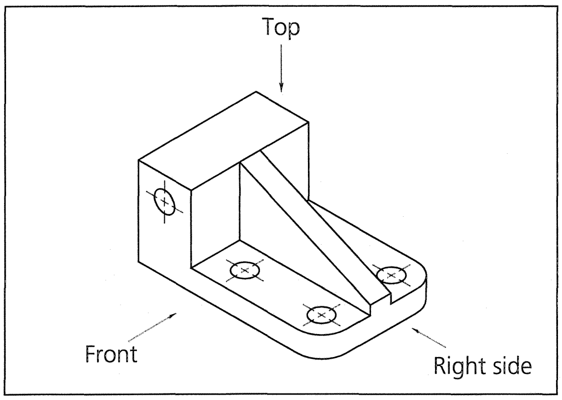
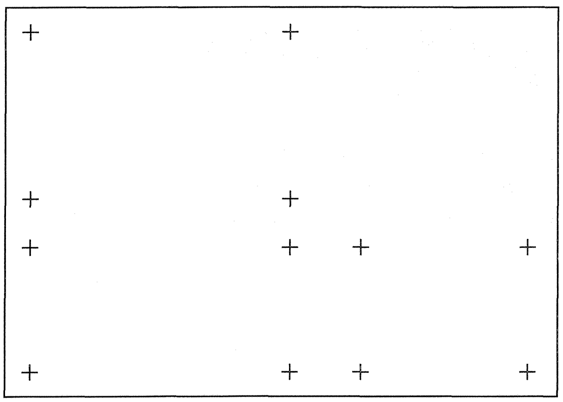


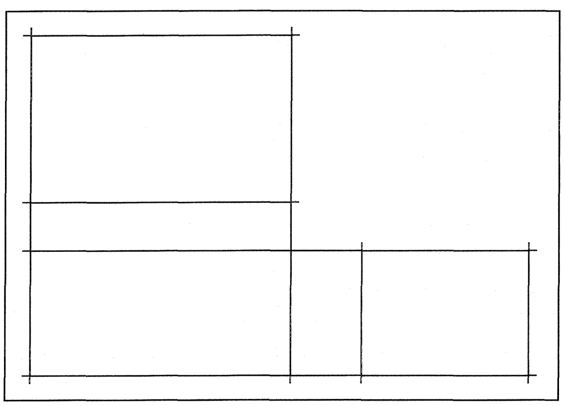

**FIGURE 10.20 Height, Width, Depth, and Dimensions of a Part**



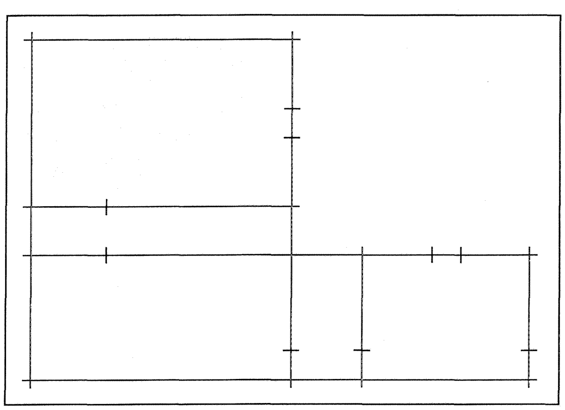

(a) Isometric view of a part



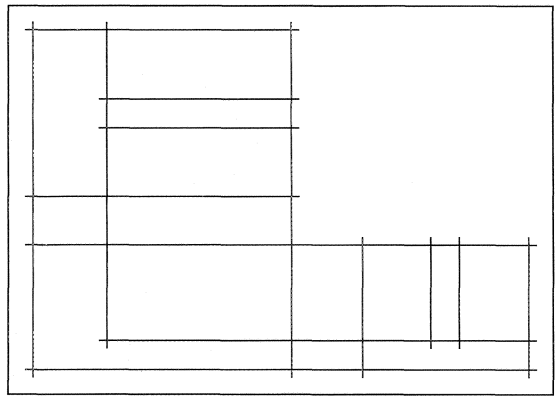
(b) Establish the overall dimensions of the part using a scale and space appropriately



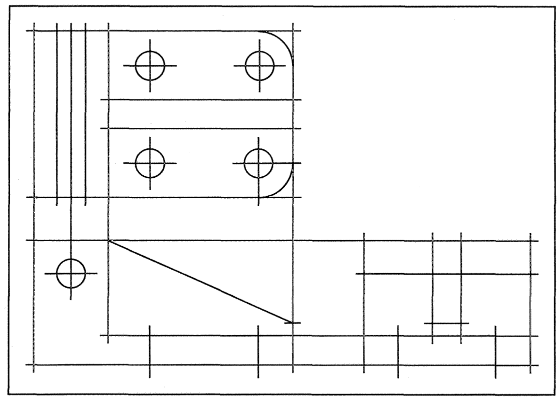
(c) Block-in the part using construction lines



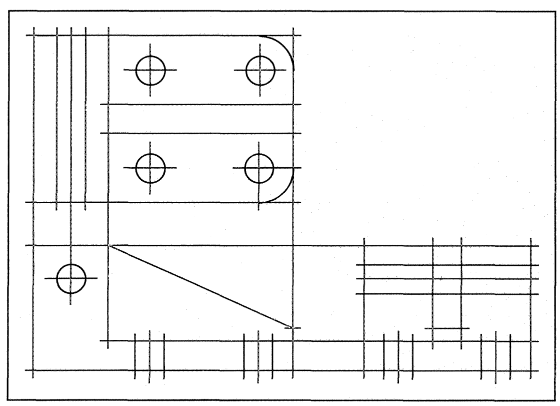
(d) Establish all the major features of the part



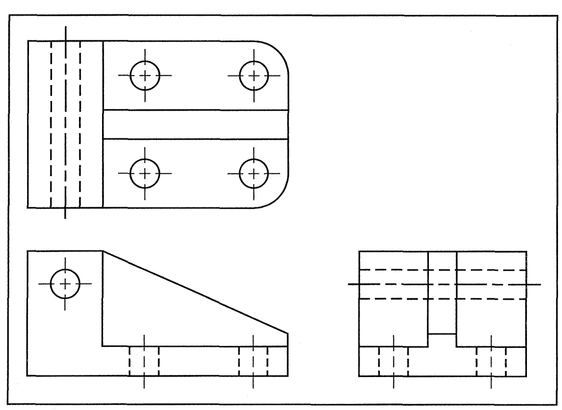
(e) Block-in the secondary features using construction lines



(f) Establish all holes and draw circles with construction lines using a compass or template



(g) Darken arcs and circles



(h) Darken drawing and remove construction lines

**FIGURE 10.21 Steps in the Construction of a Drawing**

8. The remaining lines can now be darkened [Fig. 10.21(h)]. Care should be exercised in matching the line thickness of the curves and the straight lines. Erase all construction lines still showing after all lines are darkened. (You could also erase extra constructions *before* darkening in the drawing. Try both ways to see which works best for you.)

After the drawing is complete, check it thoroughly. Fill in the title block as a last step. Since dimensioning is not discussed here, that step has *not* been included in the description.

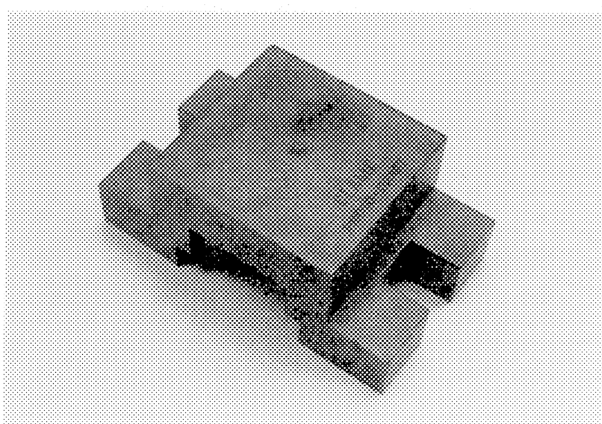
### 10.5.6 Alternative Selection of Views for a Drawing

Before discussing the construction of a drawing with three or more views, it is important to understand the selection of views. A part must be analyzed carefully before starting the drawing. During this step, the proper view selection and the number of views must be determined.

### 10.5.7 Models for View Description and Reading a Drawing

Learning to visualize a part's views can be aided by the use of models. Plastic, metal, wood, clay, or soap models enable you to position the part so that each of its views is readily observable (Fig. 10.22). By simply turning the model you can view the top, front, side, or any other aspect of the part.

Sketching the part pictorially aids in understanding each of its views. Normally, isometric or oblique sketching paper, with preprinted grid lines, is used to "block out" the part before it is drawn in orthographic projection (see Chapter 9). The sketch-modeling process helps you clearly define the part. Sometimes hidden edges, surfaces, or other parts of its geometry are discovered or clarified. Even with CAD, sketching is an important part of the design process.



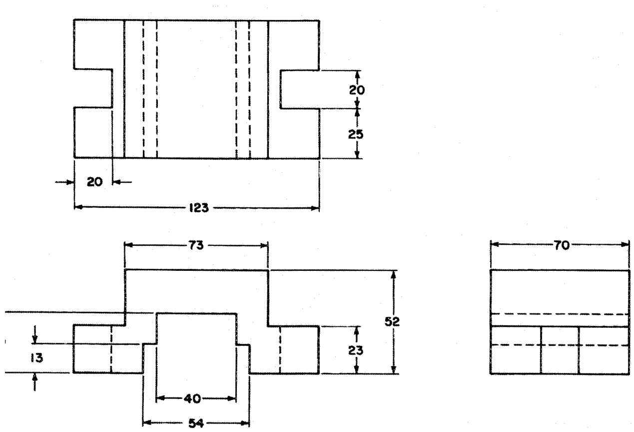



FIGURE 10.22 Three-View Detail of the Clamp

## 10.6 VIEW PROJECTION METHODS

The four separate ways to project the third view of a part are the **miter method**, the **radius method**, the **divider method**, and the **scale method.** The miter method helps in learning how to project the third view and in understanding the relationship of the top and side views. The miter method, along with the radius method, becomes less useful when the shape of the part is not simple or uncomplicated. Almost all industry drawings are completed by using the scale and the dividers to establish depth dimensions in the third view or simply by reading (understanding) the third view.

### 10.6.1 Miter Lines for Transferring Depth Dimensions

The **miter line method** is a simple and straightforward procedure for establishing the depth dimensions of a three-dimensional part. After the front and top views (or the front and side views) are drawn, construction of the third view can begin. The **miter line** is drawn as a construction line. A 45° line is drawn from the upper right-hand corner of the front view of the part, which is the intersection of the fold lines (Fig. 10.23). The upper edge line of the part, in the top

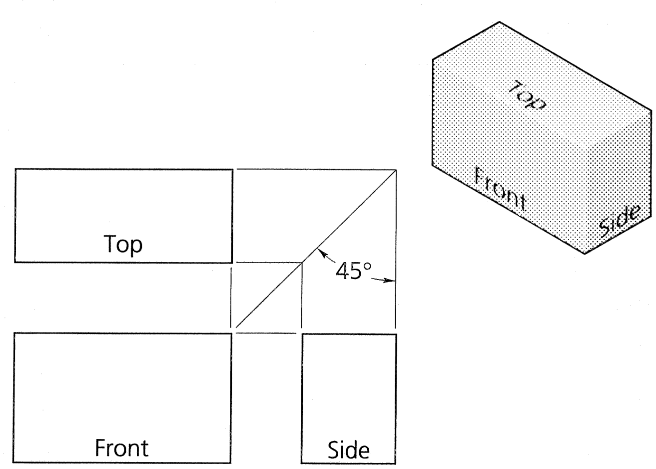
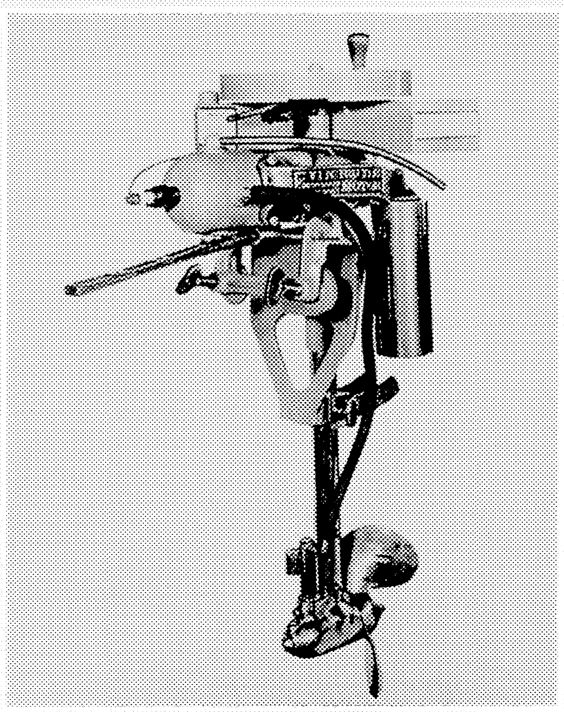



FIGURE 10.23 Establishing the Depth of a Part Using the Miter Line Method

## *Focus On . . .*

# EVINRUDE

Who would have imagined that the son of an immigrant farmer, with only a third-grade education, would be responsible for the hours of pleasure experienced by people who fish and boat? Ole Evinrude was born in Norway and came to America with his parents to farm in Wisconsin. He wasn't a very good farmer, preferring to channel his energies into work on mechanical devices. At sixteen, he built his first project, a sailboat. He used this project to secure a job as a machinist in Madison. After several jobs in Chicago and Pittsburgh, he settled in Milwaukee, working as a pattern maker. In his spare time, he "tinkered" with his idea of constructing a standard engine for the increasingly popular horseless carriage. The U.S. government became interested in this concept and contracted with him to produce fifty engines. As a result, he opened his own company.



Evinrude's first motor.

The idea for the outboard motor was the result of being embarrassed during a summer picnic. His future wife asked him to row across the lake to get ice cream. On the return trip, the wind became so gusty that he was unable to row fast enough to keep the ice cream from melting. Ole was a large, strong man and was embarrassed over his inability to control the boat. The following Monday he began work on his outboard motor.

Evinrude introduced his 1.5-hp motor in 1907. It has remained essentially unchanged to this day. It has a horizontal cylinder with a vertical crankshaft, employing power direction changes with gears in a submerged lower unit. Ole was only thirty-two when he formed Evinrude Motor Company to produce the outboard motors.

The company was sold in 1914. Later, another company, Evinrude Light Twin Outboard (ELTO), produced the first practical twin-cylinder outboard. In it, many heavy engine parts were replaced with aluminum. Also, exhaust gases were passed through the propeller hub.

Evinrude died in 1934. A few years ago his original 1909 outboard motor was dedicated as a National Historic Mechanical Engineering Landmark. It was the first consumer product to be so recognized. Another Wisconsinite, Harry Armenius Miller of Menomonie, is credited with inventing the first motorboat and motorcycle, although he never patented them.

No doubt Evinrude spent many hours sketching his ideas. To manufacture those motors, many working and assembly drawings were also produced. Evinrude had the genius to take an idea in his mind and turn it into a valuable product. This is not so different from what we try to do today.


A modern outboard motor.

view, is then extended until it intersects the miter line. The intersection point helps establish the outside edge of the side view by drawing a vertical construction line through it. Since it is adjacent to the side view, the height of the part is projected from the front view. Other depth dimensions can now be extended to the miter line from the top view and

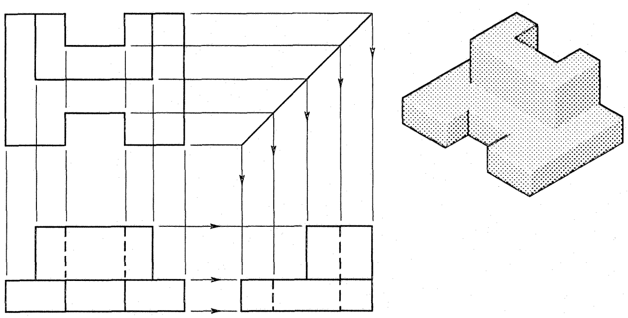


FIGURE 10.24 Miter Line Method of Projecting the Depth of the Third View

then to the side view. The drawing of the part in Figure 10.24 illustrates how each of the depth dimensions has been extended from the top view to the miter line and projected downward to establish the right side view. Height dimensions are projected directly from the front view. Miter lines and projection lines are erased after the view is completed.

### 10.6.2 Radius Method for Determining Depth

The **radius method** is shown in Figure 10.25. The upper right-hand corner of the front view is used to swing arcs R1 and R2 (90°) so as to establish the depth of the side view. In this method, as in the miter line method, the spacing between the front and top views and the front and side views is the same. Each feature in the top view is transferred to the side view via radii. Of course, the process could be reversed to transfer features from the side view to the top view, as is the case when the side view is drawn first. All radius lines and construction lines must be erased after the view is completed.

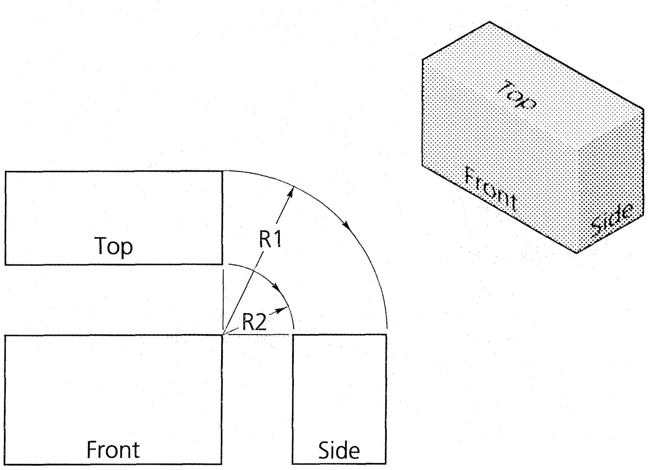



FIGURE 10.25 Radius Method of Projecting the Depth of the Third View

### 10.6.3 Divider Method for Establishing the Depth Dimension

Since the **divider method** (Fig. 10.26) is quick and accurate, it is used for descriptive geometry problems and for engineering drawings. This method allows the placement of the third view at any distance from its adjacent projection. In other words, the front and side view spacing need not be the same as the spacing between the front and top views. Since the spacing between the views is determined by the part's complexity and the dimensions required to detail the part, this will most likely be the case for most projects.

Dividers are used to establish all depth dimensions in the third view. Unlike the miter and radius methods, this method does not require that you erase construction lines after transferring the depth dimensions. In general, the miter and radius methods are limited to instructional drawings when learning to draw. A combination of dividers and scale measurements is the normal procedure to draw the third view. If the front and side views were drawn first, the "third" view could be the top view.

### 10.6.4 Scale Method for Transferring Depth Dimensions

The **scale method** to establish the depth dimension is common in industry. The scale is used to measure the depth dimension of the part in the top or side view (whichever view was constructed first). Depth dimensions then help establish the third view. You could utilize the dimensions of the part to construct each view (with the scale) without transferring dimensions. Though this method is acceptable,

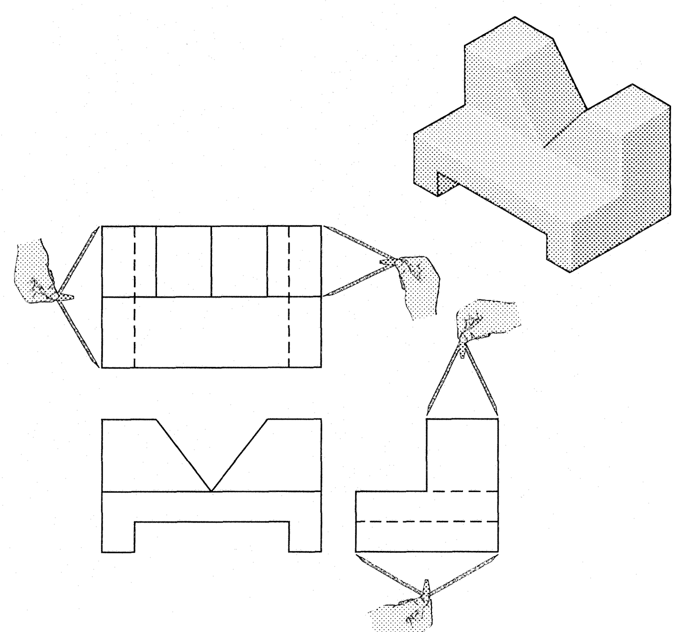


FIGURE 10.26 Transferring the Depth Dimension Using Dividers

it does require the repetitious use of the scale and takes longer. Measurements established once can normally be projected from adjacent views or transferred by dividers from related views.

Which method to employ depends on the configuration of the part and the views required. A minimum amount of scaling in each view will increase efficiency and speed.

### 10.6.5 Precedence of Lines on a Drawing

Views of a part will show its edges, surfaces, centerlines, and other features. A surface in one view will show as an edge in its adjacent view and as a surface in its related view. Since each view has so many features, they will at times interfere with one another. In other words, some features will coincide. Because showing all features in every view would only confuse the drawing, an order of importance, or **precedence of lines,** has been established for engineering drawings. The most important lines are drawn and the less important are left off the drawing. Figure 10.27 shows the proper precedence of lines on a drawing.

All outside edges of a part (boundary lines), in a particular view, will be drawn as **visible lines** and have precedence over all other lines. Visible edges are solid lines and always have precedence over hidden lines (dashed). **Dashed lines** represent hidden edge lines of the part and, therefore, have precedence over **centerlines** (which do not really exist as aspects of the part's geometry; they represent the center of curved features, e.g., circles and arcs). **Dimension lines** and **extension lines** should be positioned so as to avoid coinciding with visible and hidden lines.

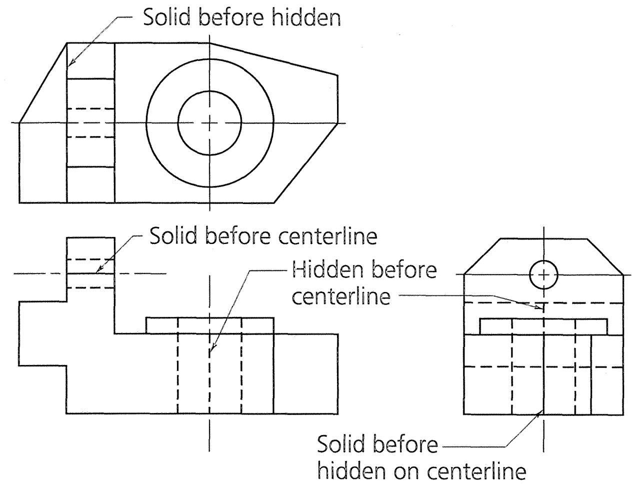



FIGURE 10.27 Precedence of Lines on a Drawing

*Order of Precedence of Lines on a Drawing*

1. Visible lines (solid)
2. Hidden lines (dashed)
3. Cutting-plane lines or centerlines (depending on importance)
4. Break lines (solid)
5. Extension lines and dimension lines (solid, thin)
6. Section lines (crosshatch)

### 10.6.6 Interpreting Multiview Drawings

Labeling a part's features with numbers or letters may help develop understanding and visualization of three-dimensional parts. This may also help in constructing views of complicated shapes. In Figure 10.28, each edge line of the

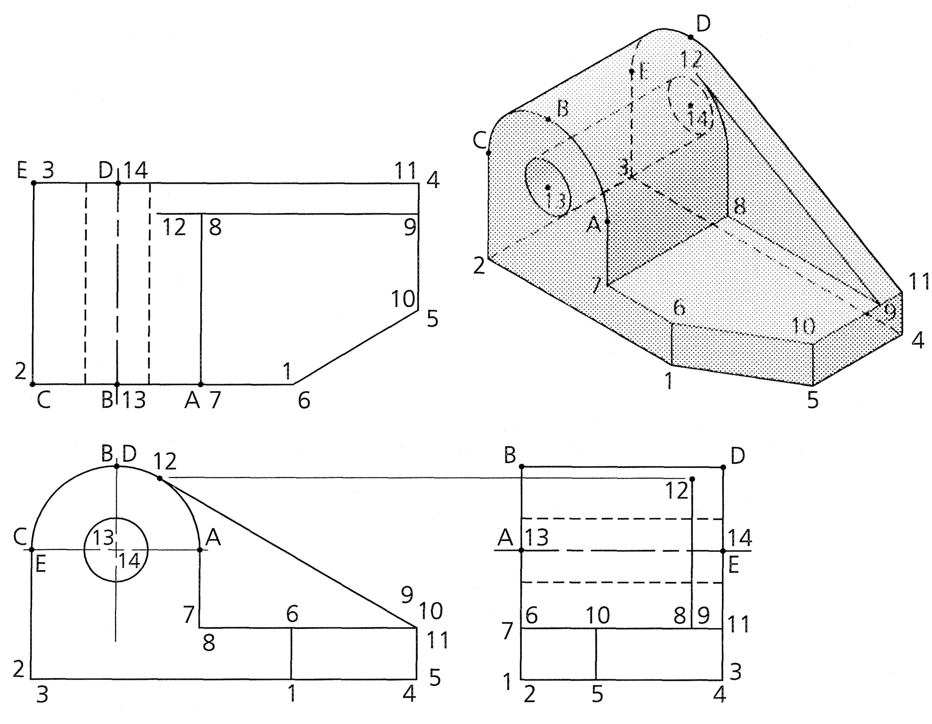



FIGURE 10.28 Labeling Points on a Part to Establish Features in Views

part, where it meets another edge line, has been identified with a number or a letter. This method is also used for completing descriptive geometry problems. Notice that the ends of curved features are identified with letters, and the ends of straight-line features are labeled with numbers. Each line can be seen in every view as either *true length, foreshortened,* or a *point.* Most lines, except for the angled lines 1-5, 6-10, and 9-12, will show as two numbered ends in two views and as a point view (coincident ends) in another view. Line 13-14 is the centerline for the hole and for the curved surface. Projecting views (and individual features) of the part becomes a matter of locating points from view to view.

### 10.6.7 Projection Lines for Views

**Projection lines** are thin, lightly drawn construction lines that "project" features between adjacent views. Such lines are erased after the views are complete and before darkening. The lines eliminate the need to measure and scale every aspect of a view. Elements that are already established in one view can easily be extended (projected) to the adjacent view. As an example, in Figure 10.29 the front view has been drawn first. Since the front view is adjacent to both the top and side views, it can be used to establish those views by projection. The top view is constructed with projectors extended from the front view to establish its width dimensions. The depth dimensions for the top view are constructed with scale measurements. Since it shares all height and elevation dimensions, the side view can be projected from the front view. The depth dimensions must be established by one of the four methods described previously.

Most parts are too complicated to draw only one view at a time. Edges and features in one view may need to be drawn in the adjacent view first and then located by projection. A majority of the time, you will construct aspects of each view that are easily identified and then project those features to the adjacent view, working back and forth until the drawing is complete.

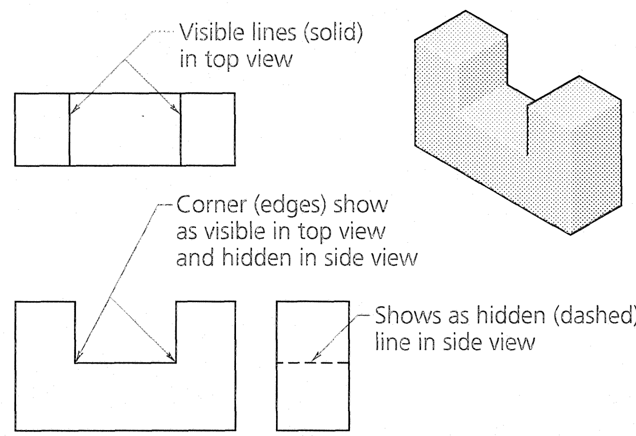



FIGURE 10.30 **Solid (Visible) and Dashed (Hidden) Lines of a Part**

### 10.6.8 Hidden Lines in Views

Since every feature of a part is seen in each view as an edge or a surface, many aspects of the part may be viewed as "*hidden*" features (Fig. 10.29). Features that lie behind other features of a part are still represented. To show the part's features, both hidden and visible, different line symbols are required. All features (edge lines, surfaces, and intersecting surfaces) that cannot be seen directly as visible lines in a particular view will be drawn with **hidden lines.**

In Figure 10.30, the use of visible (solid) and hidden (dashed) lines is shown. Visible lines in the top view of this part show as visible edges and corners in the front view and as hidden lines in the side view. When constructing dashed and solid lines, the following drafting conventions for spacing must be maintained.

*Spacing Conventions for Constructing Dashed and Solid Lines*

- Do not leave a gap between a hidden (dashed) line and a visible (solid) line that meet (Fig. 10.31).
- When a hidden line crosses a solid line, leave a gap (Fig. 10.32).
- When a hidden line continues as a visible line after crossing a visible line, leave a gap (Fig. 10.33).

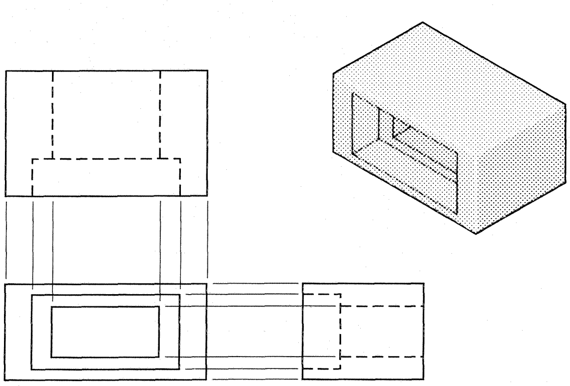


FIGURE 10.29 **Projecting Hidden Features of a Part**

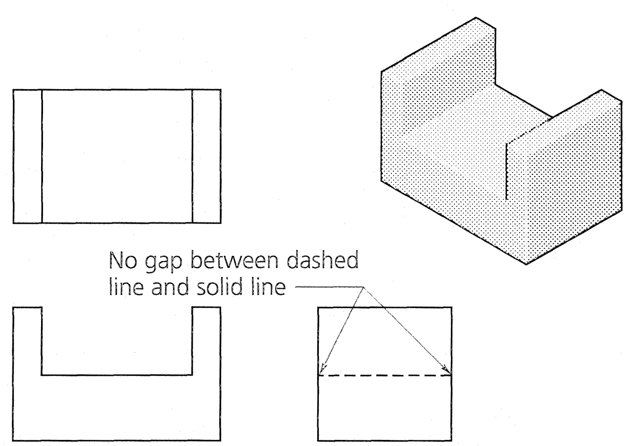



FIGURE 10.31 **Drawing Dashed (Hidden) Lines**

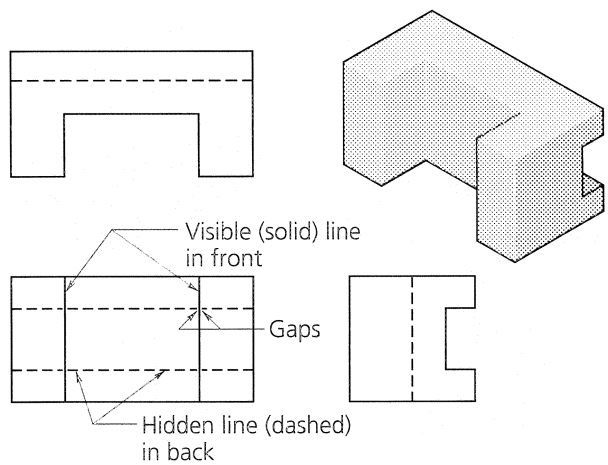



FIGURE 10.32 **Visible and Hidden Lines on a Drawing**

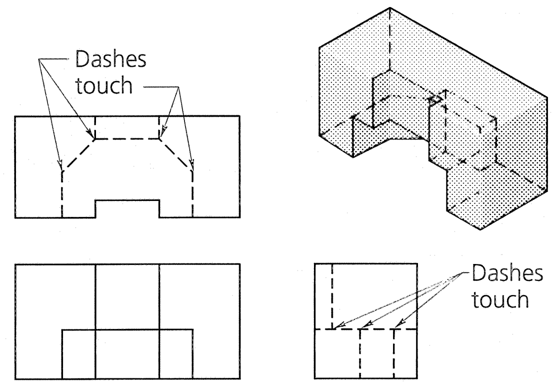



FIGURE 10.34 **Dashed Lines and Drawing Conventions**

- Hidden lines that meet other hidden lines should not have gaps between them. In other words, the dashes will touch (Fig. 10.34). Hidden lines that establish corners always touch.
- When a hidden line (or arc) meets a visible line (or arc) and is tangent to that line, leave a gap.
- When hidden lines cross, draw the one that lies in front of the other as continuous and through a space (between dashes) in the one behind it.

### 10.6.9 Curved Lines in Views

All curved features of a part are shown in each view. In most cases, a curved feature shows as a curved line or surface in only one view and as an edge line (straight) in its adjacent projection. The most common type of **curved feature** is the circle. Arcs (and fillets) also occur widely on parts. Circles, arcs, and fillets are really one end of a curved surface. **Curved surfaces** make up much of a typical machined part. A hole is really a cylindrical surface. Connected arcs and fillets are also portions of cylinders. Holes are formed by drills and other rotating tools. Parts that are made up of curved surfaces such as spheres, cylinders, and conical shapes are normally machined on lathes or other turning devices.

Since an internal curved surface (hole) and an external curved surface (normally a cylinder) are both curved surfaces, they are drawn the same way. In Figure 10.35, the part has both internal and external curved surfaces. The holes and the cylinder both show as curves in the top view and as straight edge lines in the front and side views. The hole shows as hidden features in these views and the cylindrical surface as visible lines. The outside arcs of the part also show as visible edge lines in the front and side views. The part in Figure 10.36 shows holes, arcs, and cylindrical surfaces. Here the miter line method is used to project the third view

### 10.6.10 Use of Centerlines in Views

Curved features are normally established, located in space, and dimensioned with the aid of a centerline. Except for outside arcs, **centerlines** are required in all views of curved features (Figures 10.35 and 10.36). Except for fillets and

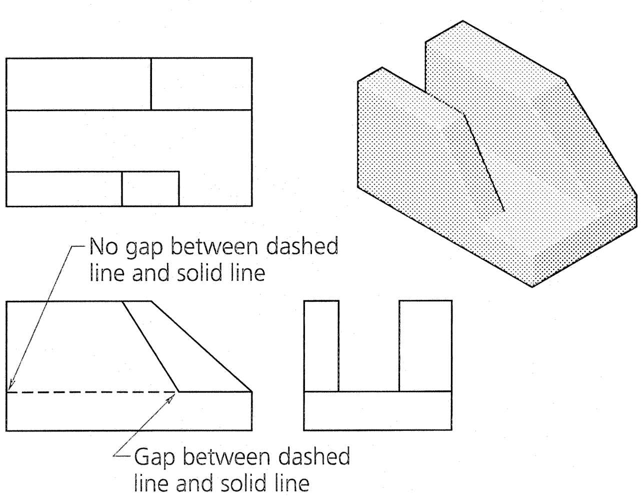



FIGURE 10.33 **Drawing Dashed Lines**

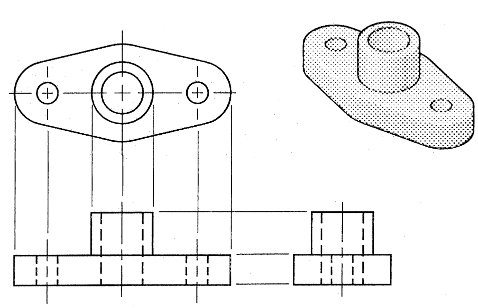


FIGURE 10.35 **Curved Features in Views**

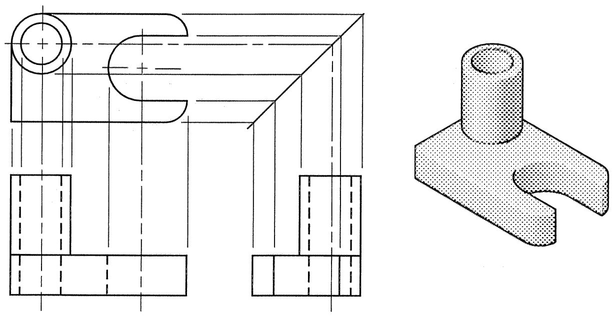


FIGURE 10.36 **Miter Line Method for Projecting Curved Features**

rounds, all curves require centerlines to establish their curved features. Centerlines for the end view of curved features are drawn as perpendicular crossing lines with short dashes at the center and as single centerlines (long dash, short dash) in adjacent views. Centerlines do not really exist as a feature of the part—they are not edge or surface lines. Therefore, they are drawn to extend slightly beyond the boundaries of the part or curved feature. They do *not* take precedence over visible or hidden lines.

Centerlines also appear on drawings where the part is *symmetrical about a centerline.* Cones, spheres, and other curved shapes require centerlines. When looking into the curve's end view, centerlines establish the center point of the curved feature; when shown in adjacent views, they represent the axis line of the curved surface.

**You May Complete Exercises 10.1 Through 10.4 at This Time**

### 10.6.11 Parallel Lines on Parts

When lines are *parallel* in all three views, they will show as parallel in all views of the part. If the lines are shown from an end view, they appear as points (point view). *Parallelism* can easily be seen in the pictorial view of the part. In Figure 10.37 the part has an angled surface that does not show as a true shape in any of the three principal views. This *oblique surface* is shown by edge lines 1-2 and 3-4 (or you could say lines 2-4 and 1-3). The top, front, and right side views show that each of these edge lines is parallel to the other in every view (including the pictorial view of the part).

## 10.7 DRAFTING CONVENTIONS AND SPECIAL VIEWS

A variety of **drafting conventions** and procedures and **design conventions** and procedures have been devised to draw projects concisely, clearly, and quickly. A number of conventions are covered here, including partial views, enlarged views, and revolved views.

The need for complete views with all hidden lines shown

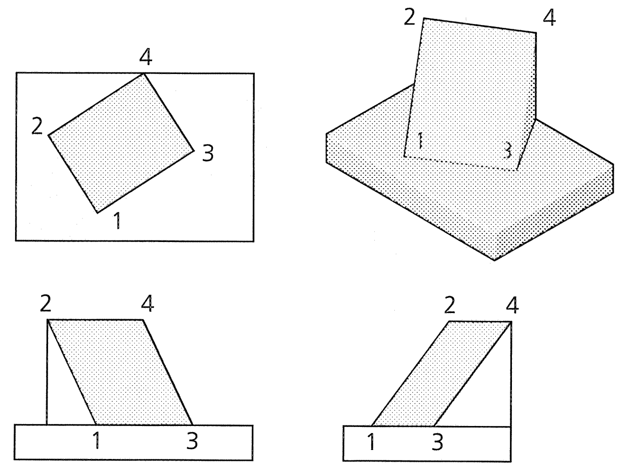



FIGURE 10.37 **Parallel Lines on Parts**

would take too much costly time and create drawings that were less usable than those with only the necessary lines shown. **Partial views** are one convention for solving this problem. Complicated, cluttered portions of drawings need to be shown in larger, clearer representations; therefore the use of **enlarged views** was established. **Rotated (revolved) views** came into practice to describe portions of a part that were projected as oblique surfaces and actually confused the drawing rather than clarifying it. Each of these methods was developed and standardized over a number of years.

### 10.7.1 Partial Views

As long as the geometry of a part is adequately described in another view, a partial view may be used. A **partial view** is a view where the dominant features, shape, and outline of the part are shown without the extra clutter of unneeded hidden lines. In Figure 10.38, the part has different shapes on each end. Since the top view would be very similar to the front view, it has been eliminated. And since the right and the left side views show only the visible lines of the corresponding end, they are partial views. These views do not show the hidden features of the opposite end, which

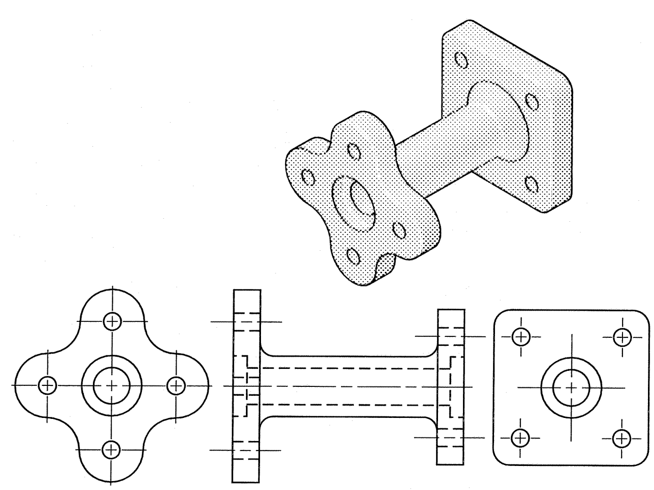


FIGURE 10.38 **Partial Views**

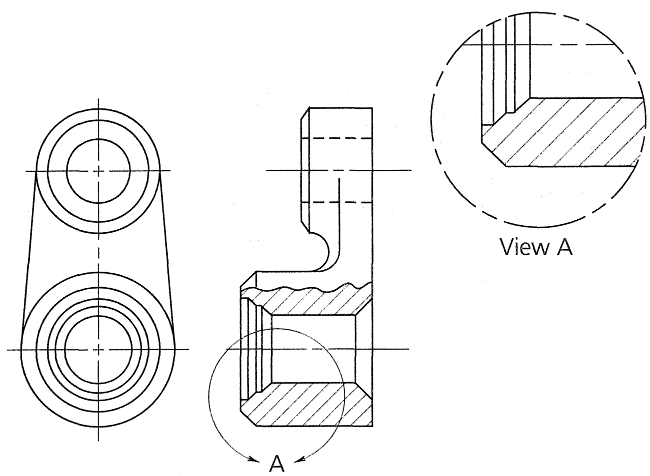



FIGURE 10.39 Enlarged Views

would add nothing to the drawing.

Hidden features on a partial view should include only those directly behind the visible shapes. In Figure 10.38, the cylinder's outside diameter (OD) lies directly behind the counterbored hole on each base plate. Therefore, since visible lines take precedence over hidden lines, this feature does not show on the drawing. The two side views have no hidden lines. On parts where the hidden feature will not appear in another view, the feature must be included on the partial view.

### 10.7.2 Enlarged Views

**Enlarged views** increase the size of a crowded or complicated area of a part. Many times this procedure is necessary to provide sufficient space for dimensions. In Figure 10.39, **VIEW A** is the enlarged portion of the part. The interior and exterior chamfers are now clearly visible. The area to be enlarged is circled with a phantom line, and the **view-letter designation** is positioned as in Figure 10.39. The enlarged view is identified on the drawing by placing the view-letter designation under the view (in the case of Fig. 10.39, **VIEW A**).

### 10.7.3 Revolved Views

**Rotated (revolved) views** are utilized where a true projection of the part would only confuse the reader. Figure 10.40a shows an example of a part that is better described with a rotated view. The detail of this part requires two views to describe its geometry and place dimensions adequately. If a *true projection* had been used, the front view would have been confusing and complicated. The clevis portion of the arm was rotated parallel to the front view and projected as a normal (true shape) view. This procedure saved considerable drawing time and is less misleading. The Gravity Probe Tilt Stand assembly shown in Figure 10.40b shows how the probe was rotated vertically in the right side view instead of using a true projection from the front view, where the probe is tilted.

### 10.7.4 Surfaces and Edges on Multiple-View Drawings

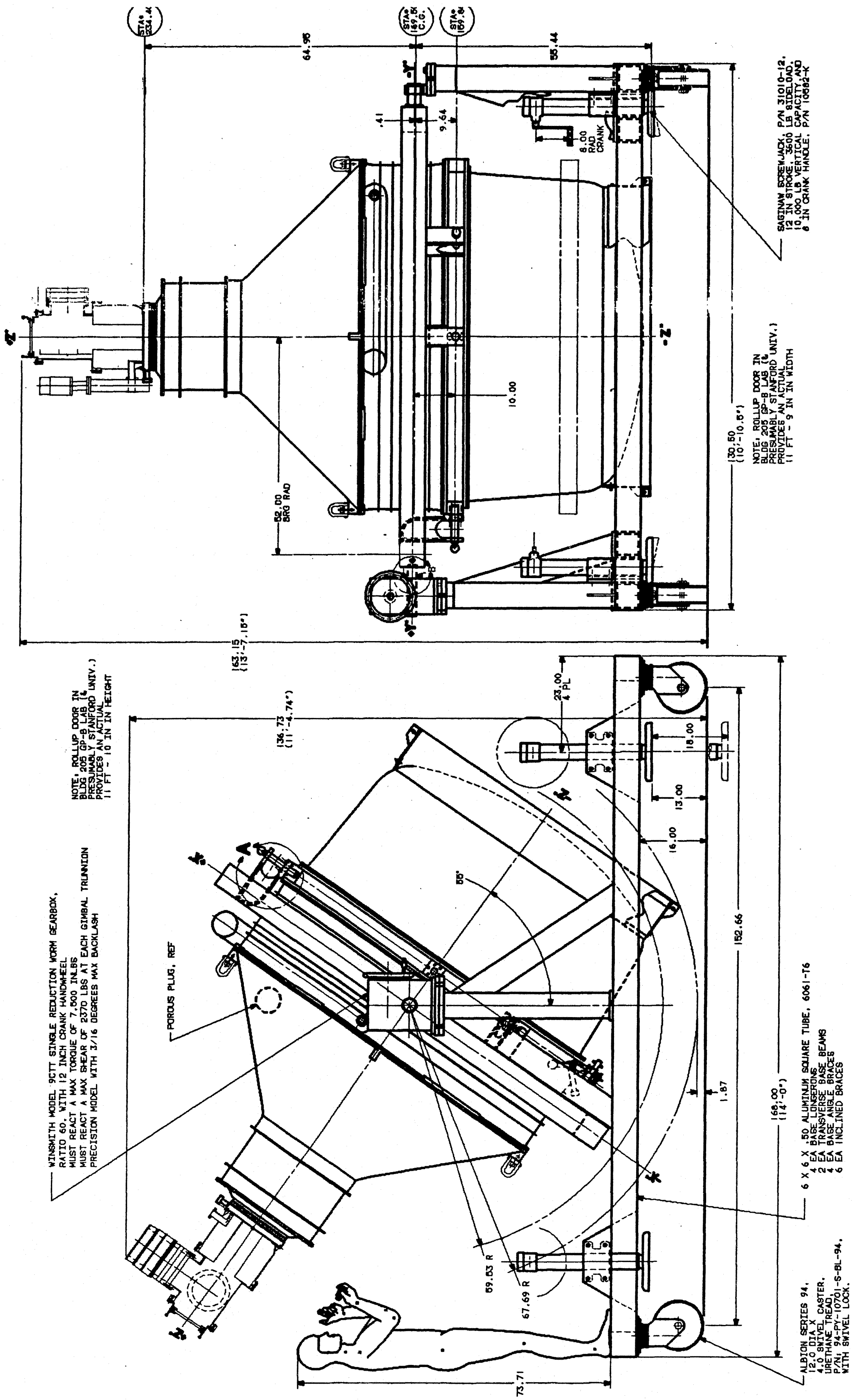
To understand orthographic projection, you must begin to see parts as simple shapes, edges, lines, and points. Surfaces are created by combining lines. The surfaces can be combinations of straight lines or straight and curved lines. Surfaces, or *areas* as they are sometimes called, show **true shape/size (TS)** when they are parallel to the plane of projection and show as **edges (EV)** when they are perpendicular to the plane of projection. A plane that appears true shape/size in a view is called a **normal surface.** The view is a normal view of a plane. The adjacent projection (view) of the plane shows as an edge (edge view).

Curved surfaces show as curved edges in views where they are perpendicular to the viewing plane, and as plane shapes with straight sides in views where they are parallel to the viewing plane. When three surfaces come together, they meet at a corner (point). Most parts can be defined by establishing their corners (points in space). Figure 10.41 provides examples of each condition. The pictorial view in the upper right provides a 3D model of the part. The part is composed of planar surfaces and curved surfaces. The hole shows as circular only in the side view. It appears as an edge in the front and top views. Notice that the circular surface of the projected hole shows as a rectangle in the front and top views. The same is true of the vertical curved surface on which the hole appears. All planar surfaces of the part show as true shape or as edges in their adjacent views.

### 10.7.5 Reading a Drawing

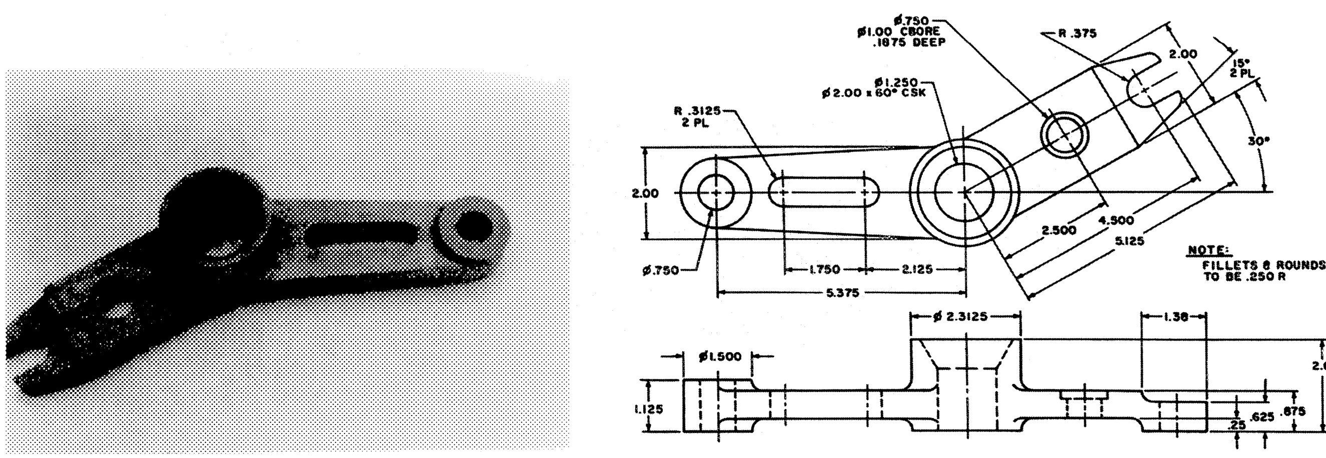
We have already said that a drawing is "*read*," not scaled. This does not mean that you read it aloud. "*Reading*" is what a designer or engineer does mentally to understand and then interpret the drawing. Here are the mental steps required to read a drawing.

1. Study the total drawing by scanning all views and dimensions.
2. Visualize the shape of the part by orientating yourself as the observer for each view.
3. Reduce the part to simple geometric shapes, e.g., planes, circles, surfaces, and other common features.
4. Study each view and feature as it corresponds to its adjacent and related projection. The depth, for instance, can be studied in the top view and related side view. Adjacent views can be studied to establish the true shape of a surface and its edge view projection.
5. If necessary, sketch a simple 3D pictorial of the part to clarify the general configuration and details.



(a) Angle frame detail

FIGURE 10.40a Rotated/Revolved Views




(b) Drawing of angle frame

FIGURE 10.40b Rotated/Revolved Views—*Continued*

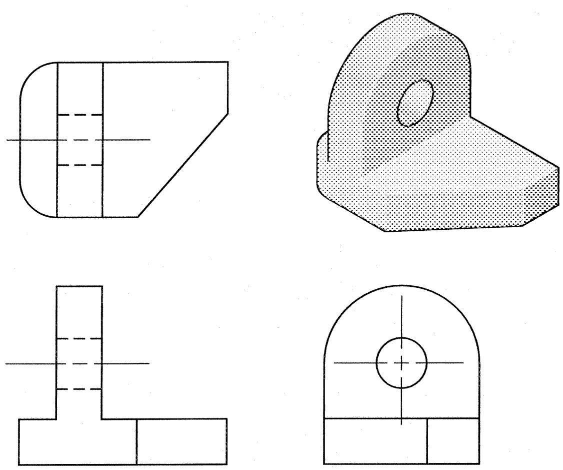


FIGURE 10.41 Surfaces on a Part

6. Note each hole, tangent area, curved feature, and other special contour that distinguishes the part.

Assuming that the pictorial view of the part (right side orientation) in Figure 10.42 is not provided, read the part.

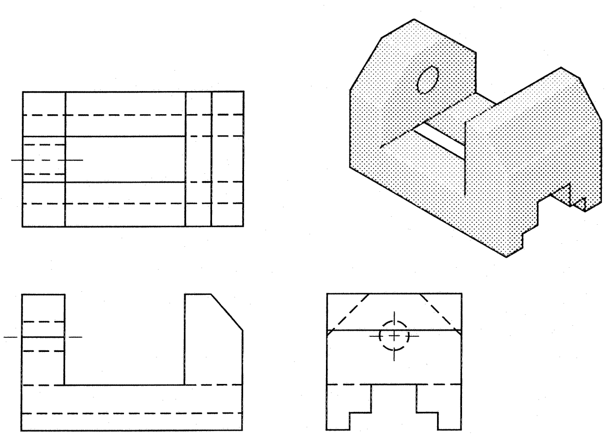


FIGURE 10.42 Three Views and Pictorial Illustration of the Guide

Notice that three views were required to represent the part's geometry adequately. Most of the part's features can be seen in the front and side views. The top view adds little to our understanding of the drawing but does show that the slot extends through the part. The front view shows the angled cut (its edge view). This is the only surface that is not normal and, therefore, does not appear true shape on the drawing in any view. The hole is described in the side view. Since the hole is hidden, only the portion of the part on the far side is penetrated. The side view also shows that the slot extends the entire length of the part. A pictorial sketch would help in reading this project.

## 10.8 Visualization and Shape Description

To read a drawing, the reader needs some skill at **visualization.** *Visualization is the process of converting a 2D drawing into a 3D image and being able to understand the part as it exists in three-dimensional space.* This skill is not innate for everyone, but can be developed, in most cases, through the study of a variety of drawings, parts, and models.

Upon entering an engineering field that requires the use of drawings, you must be able to understand both the 3D and the 2D illustrations of a part and its representative drawing. Visualizing is a skill that will be necessary for both situations.

### 10.8.1 Areas on Adjacent and Related Views of a Drawing

Visualization helps in examining a part by comparing surfaces and edges on adjacent and related views. When studying adjacent areas, remember that *adjacent areas cannot lie in the same plane.* If they did, they would not exist; they would not have a boundary between them.

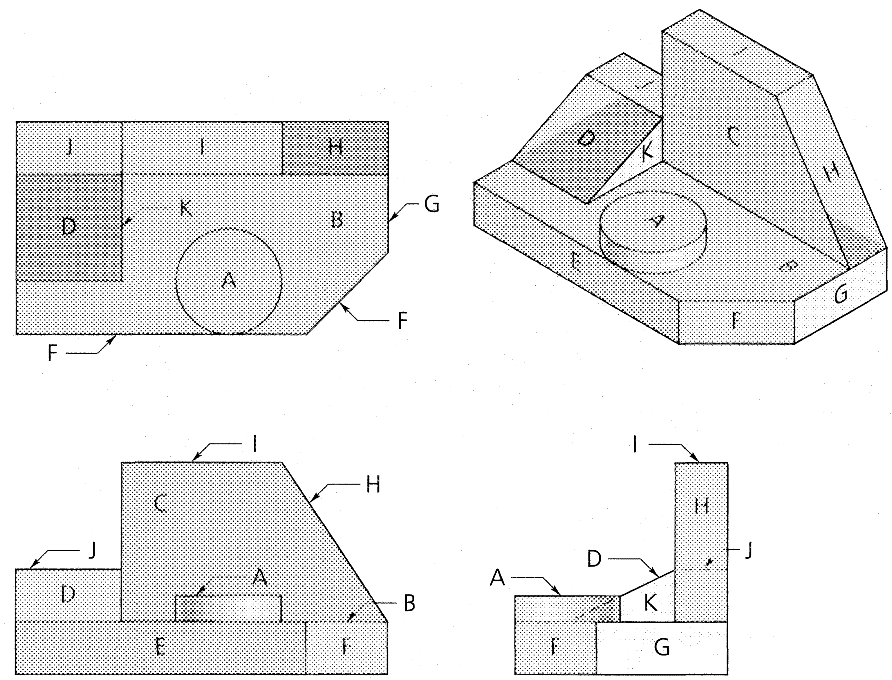



FIGURE 10.43 Related Surfaces and Edges

**Adjacent areas** can be studied in Figure 10.43. The three principal views are labeled in each projection and on the pictorial view. In each view, a surface or an edge is labeled.

- Surface A is shown true shape in the top view and as an edge in the front and side projections.
- Surface B is also true shape in the top view and, therefore, an edge view in the front and side views of the drawing.
- Surface C is true shape in the front view. Can you find it in the top and side views? It will show as an edge view in each. If you cannot find it in the top and side projections, the pictorial view will locate surface C. Remember, if a surface is true shape in the top view, it shows as an edge in the other two views (front and side).
- Surface D is an angled surface. Its slant angle can be seen in the side view, where it shows as an edge. The front and top projections of surface D are not true shape.
- Surface E is along the front of the part and is true shape in the front view. It shows as an edge in the top view and the side view.
- Surface F is at an angle and does not show in any view as true length. The top view shows this surface as an edge view, and its angle to the part can be measured from the edge view of surface E. The side and front views of surface F show as foreshortened (not true shape).
- Surface G forms the right side of the part and shows as an edge in the top view and as true shape in the front and side views.
- Surface H is an inclined surface, and its slant angle can be measured in the front view as the angle it makes with surface B. Surface H is an edge in this view and is shown foreshortened in the other two projections.
- Surface I is the top, or highest, surface on the part and shows as an edge in the front view, true shape in the top view, and as an edge in the side view. If a surface appears as an edge in the front view, it will also be an edge in the side projection; it will be true shape only in the top view.
- Surface J is parallel to surface I. Therefore, it also is true shape in the top view and an edge in the other two projections.
- Surface K is true shape in the side view and an edge in the top and front views. Surfaces G and K are the only labeled surfaces that are true shape in the side view.

In addition to seeing the true shape and the edge views of a surface, it is important to develop a sense of how each surface relates to another surface. Surface C, for instance, is parallel to surface E and perpendicular to surfaces I and B. Surface D is at an angle to surface B and surface C. Surface G is parallel to surface K and perpendicular to surfaces B and E. Being aware of *parallelism, perpendicularity,* and *angularity* are important aspects of visualizing a 3D part and reading its 2D representation—its drawing.

## 10.8.2 Visualizing Similar Shapes of Surfaces

Here is a simple rule of projection: *In an adjacent view, an area will project as a similar shape or as an edge.* Adjacent projections of a normal surface project as edges. Related views of a surface project as similar shapes. In Figure 10.44, the drawing of the part shows that the angled surface is a *similar shape* in the side and top views. It shows as an edge in the front view. Even though the top and side views show the surface as distorted, their outlines appear as similar shapes. The angled surface has the same number of sides in each view where it does not appear as an edge. Thus, the preceding rule has a corollary: *The area shapes will have the*

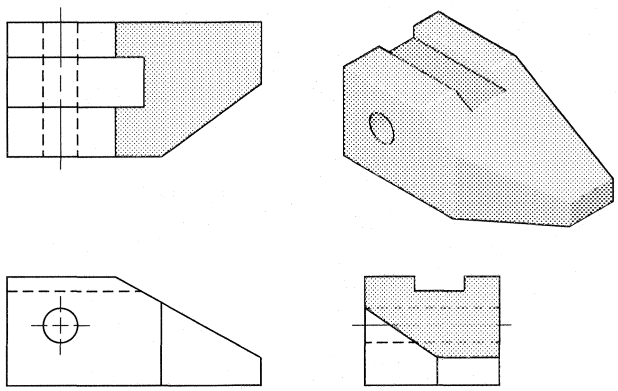


FIGURE 10.44 Angled Surfaces and Edge Views

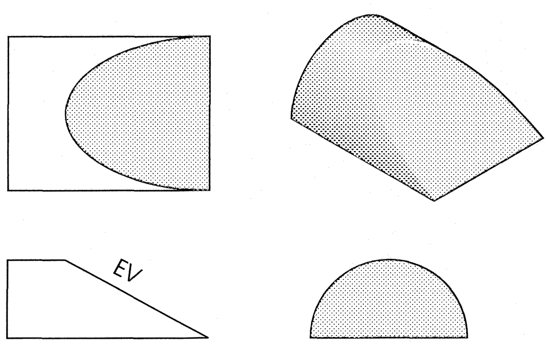



FIGURE 10.45 Elliptical Surface

*same number of sides, and the sides of the areas are connected in the same sequence.*

Curved shapes may distort in related views, but they also maintain similar shapes, as in Figure 10.45. The top and side views show similar-shaped views of the angled surface. The front view shows the surface as an edge (EV).

### 10.8.3 True Shape or Normal Surfaces of a Part

Much has already been said about normal views and true shapes of surfaces. *Surfaces that are parallel to a plane of projection are normal surfaces.* In other words, they will show as **true shape,** and each line, arc, circle, or other form that lies on this surface, or is parallel to it, will be true shape and **true length/size.** Figure 10.46 demonstrates this rule. The true-shape surfaces (normal surfaces) are labeled in each of the three views of the part. The surfaces that are not normal to the projection plane are **inclined surfaces** and do not project as true shape in any given view on this drawing.

### 10.8.4 Edge Views and Edge Lines of a Surface

A surface projects as an edge in a view where the plane of projection is perpendicular to the surface. A line that shows

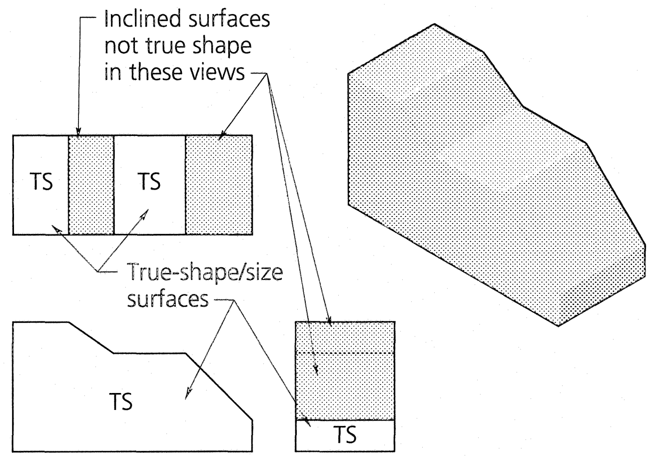



FIGURE 10.46 Inclined Surfaces and True-Shape Surfaces

as a **point view** is a normal edge; that is, it is perpendicular to the projection plane.

**Edge lines** are always shown on views where the surfaces they represent are perpendicular to the adjacent view. In Figure 10.47, the front view of the part shows two perpendicular surfaces that will project as edge lines in the top view. The surface that is at a slight angle and blends with its mating surfaces is not represented with an edge line in the top view. The same convention applies in the right side view and the left side view.

### 10.8.5 Angles on Multiview Drawings

In Figure 10.48, the part has two **angled surfaces.** The **true angle** of these surfaces is shown in the side view of the part, where they show as edge lines that lie normal to the view. Angles can be measured only in views where they are in a normal plane. The front and top views show the angled surfaces as if they were rectangular and true shape; their inclination cannot be read in these views. Without the side view, the part's configuration could not be determined.

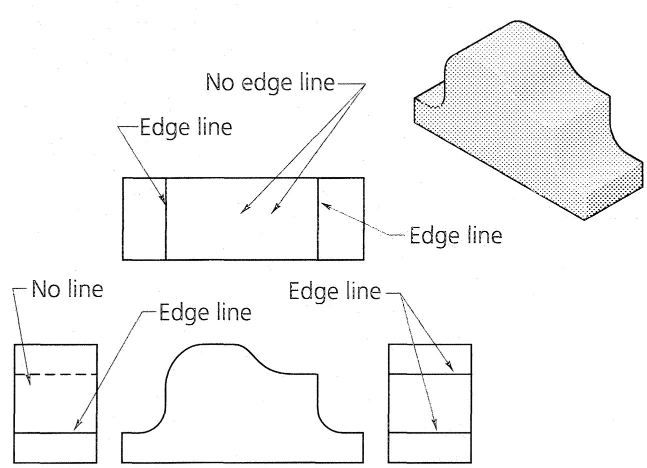



FIGURE 10.47 Curved Surfaces and Edge Lines

## *Applying Parametric Design . . .*

# VIEWS FROM PARAMETRIC MODELS

**Views** created by a CAD/CAM system are exactly like views constructed manually by a designer on paper. The same rules of projection are applied, the only difference is that you merely command the system to create the views as needed (see Fig. A). The original 3D part database must be completed before views are established. The first view established is the general view showing the parametric model (Fig. B).

A wide variety of views can be derived automatically from the parametric model. One of the most common is projection views. The system automatically creates projection views by looking to the left, to the right, above, and below the picked view location (see Fig. C) to determine the orientation of a projection view (see Fig. D). When conflicting view orientations are found by the system, you are prompted to select the view to be the parent view. A view will then be constructed from the selected view.

At the time they are created, projection, auxiliary, detailed, and revolved views have the same representation and explosion offsets, if any, as their parent views. From that time onward, each view can be simplified and restored and have its explosion distance modified without affecting the parent view. The only exceptions are detailed views, which will always be displayed with the same explosion distances and geometry as their parent views.

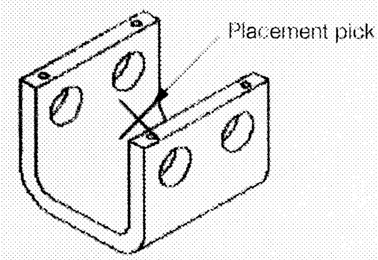



FIGURE B The General View (Default Orientation—First View Added to a Drawing)

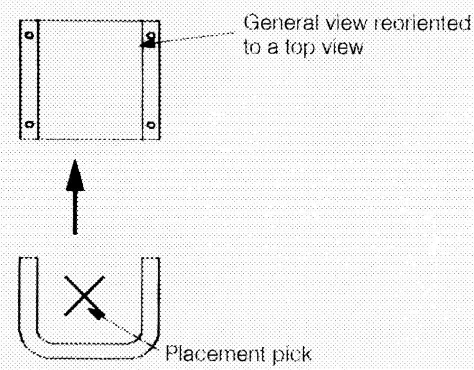



FIGURE C Projecting the Front View

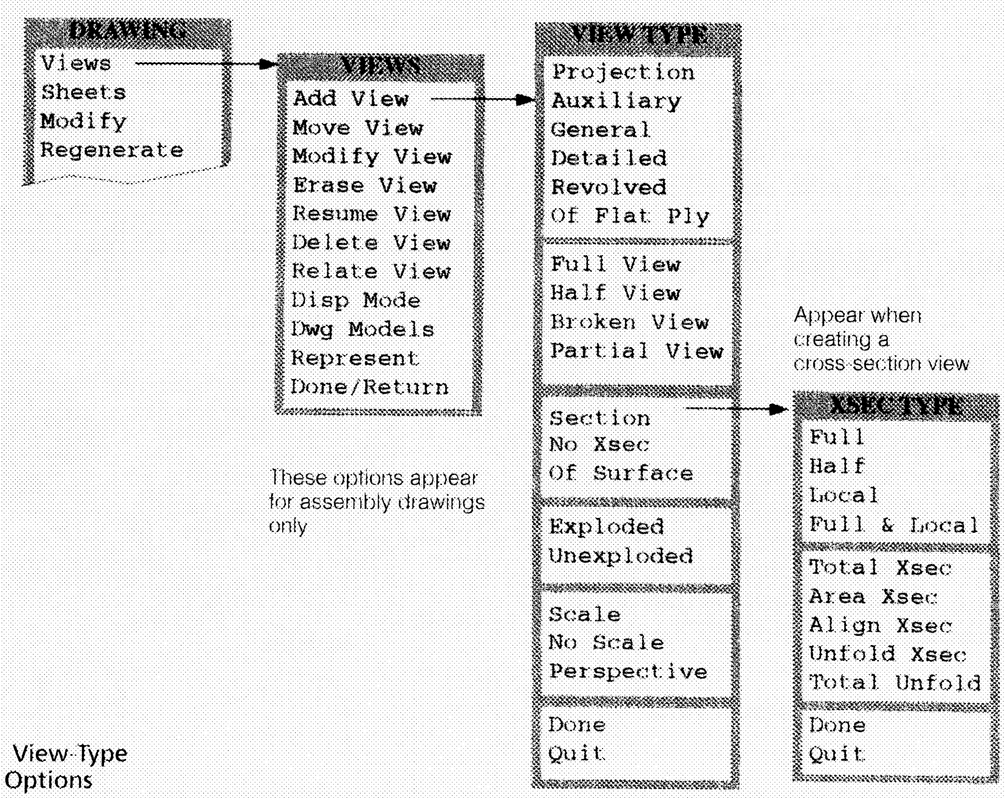



FIGURE A View-Type Command Options

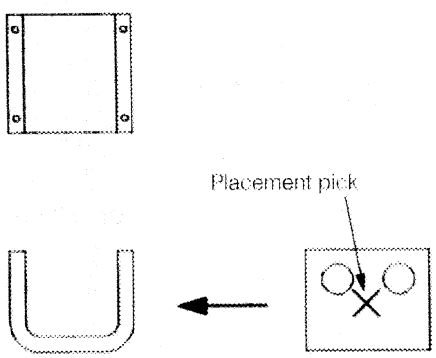



FIGURE D **Projecting the Right Side View**

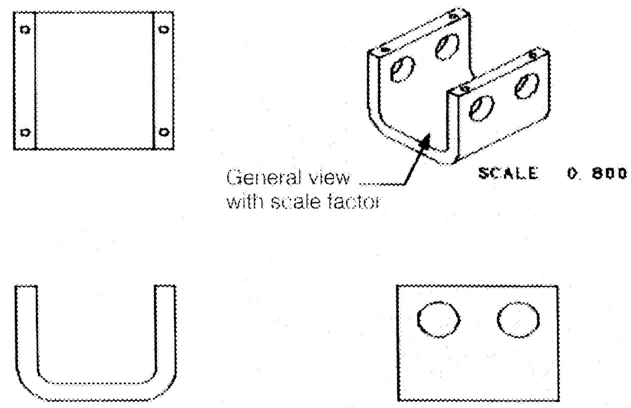



FIGURE F **Adding a General View to the Drawing**

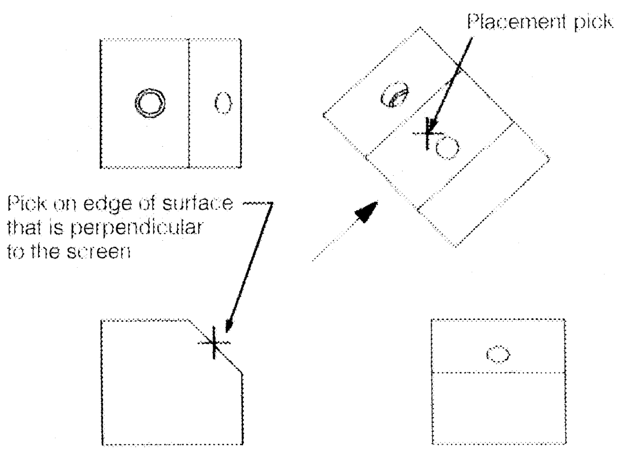



FIGURE E **Projecting an Auxiliary View**

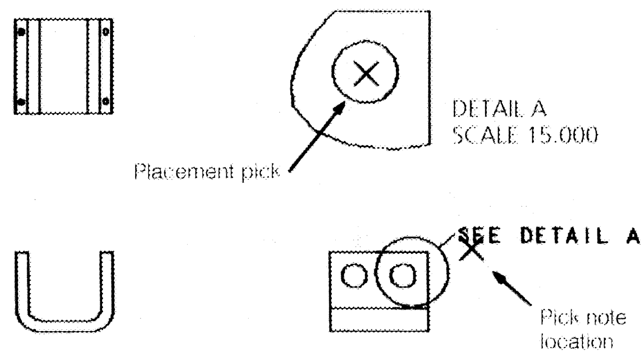



FIGURE G **Adding a Detail View**

### Available View Types

**Auxiliary** Develops one view from another view by projecting the geometry at right angles to a selected surface or along an axis. The surface selected from the parent view must be perpendicular to the plane of the screen (Fig. E).

**Projection** Develops one view from another view by projecting the geometry along a horizontal or vertical direction of viewing (orthographic projection). The projection type is specified by you in the drawing setup file, and may be based on third-angle (default) rules or first-angle rules (see Fig. F).

**General** Creates a view with no particular orientation or relationship to other views in the drawing. The model will first be oriented in the default view orientation established by you (Fig F).

**Detailed** Details a portion of the model appearing in another view. Its orientation is the same as the view from which it is created, but its scale may be different so that the portion of the model being detailed can be visualized better (Fig. G).

**Revolved** Creates a planar-area cross section from an existing view; the section is revolved 90° around the cutting-plane projection and offset along its length. A revolved view may be full or partial, exploded or unexploded (Fig. H).

### View Options That Affect How Much of the Model Is Visible in the View

**Full View** Shows the model in its entirety.

**Half View** Removes a portion of the model from the view on one side of a cutting plane.

**Broken View** Removes a portion of the model from between two selected points, and closes the remaining two portions together within a specified distance.

**Partial View** Displays a portion of the model in a view within a closed boundary. The geometry appearing within the boundary is displayed; the geometry outside of it is removed (Fig. D).

### Options That Determine If the View Is of a Single Surface or Has a Cross Section

**Section** Displays an existing cross section of the view if the view orientation is such that the cross-section plane is parallel to the screen.

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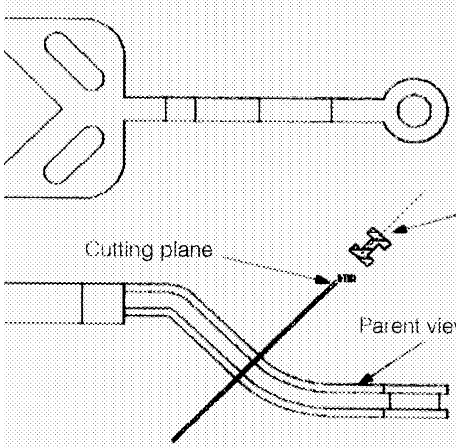



IGURE H A Revolved-Area Cross-Section View

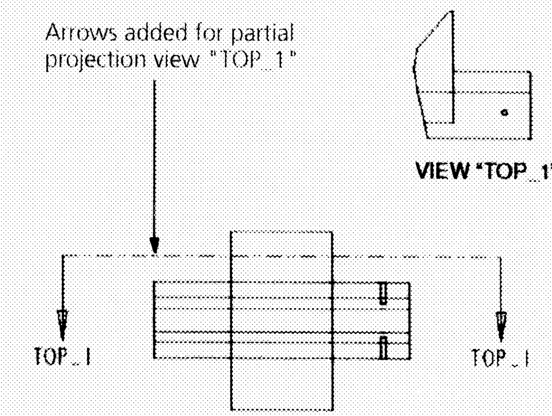



FIGURE I Partial View

**lo Xsec** Indicates that no cross section is to be displayed in the iew.

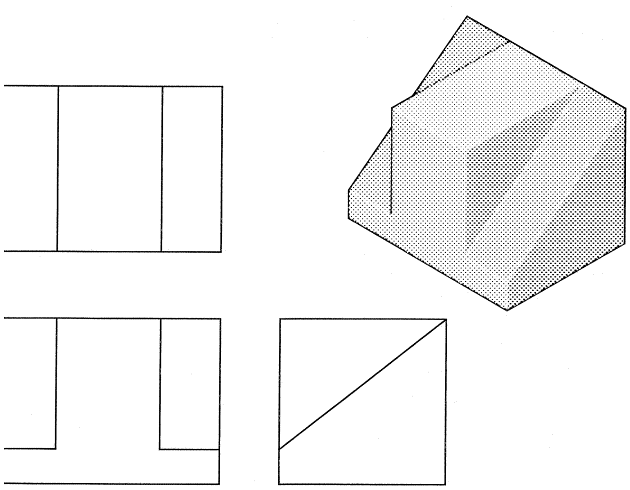
**)f Surface** Displays a selected surface of a model in the view. he single surface view can be of any view type except detailed.

**)ptions That Determine If the View Is Scaled**

**cale** Allows you to create a view with an individual scale shown under it. When the view is being created, the system will prompt you for the scale value. This value can be modified later. General and detailed views can be scaled.

**No Scale** A view will be scaled automatically using a predefined scale value that will appear in the lower left corner of the screen as "SCALE."

**Perspective** Creates a perspective general view.



GURE 10.48 Inclined Surfaces on Parts

## 0.8.6 Inclined Surfaces of a Part

n **inclined surface** shows as an edge in one view and as reshortened in the adjacent view. The edge view of the clined surface shows the *true angle* of the surface. Figure ).49 has three inclined surfaces. The angles that surfaces A ıd C make with the horizontal plane is shown in the front ew, where each appears as an edge line. The true angle of surface B can be measured in the side view, where it appears as an edge line. The other views of surface B show as *foreshortened.* The amount of foreshortening depends on the *angle of the inclination:* The greater the angle of incline to a view, the more the surface is foreshortened.

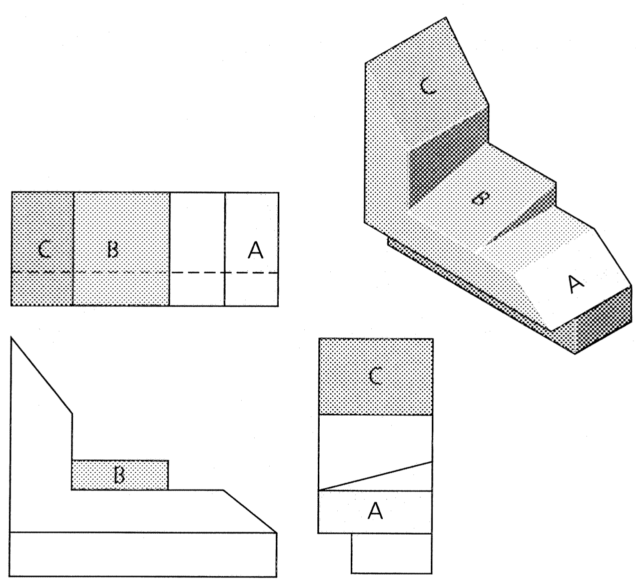



FIGURE 10.49 Inclined Surfaces in Adjacent and Related Views

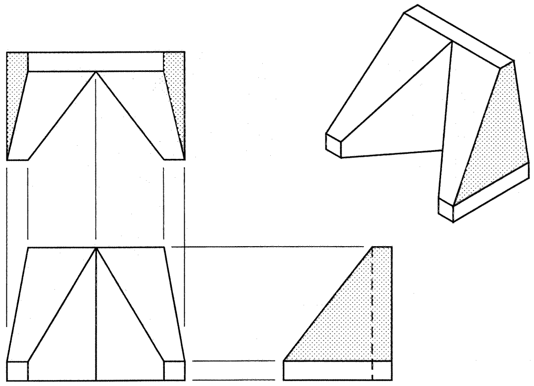


FIGURE 10.50 Inclined Surfaces

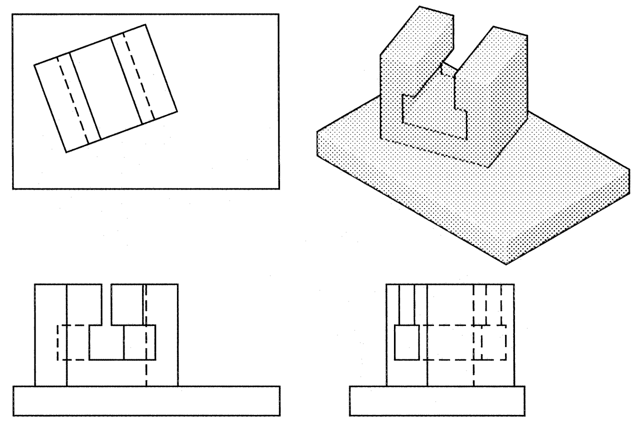


FIGURE 10.52 Distorted View of Surfaces

The part shown in Figure 10.50 has a number of angled surfaces, each represented by different shading. Each view shows the angle of two surfaces. The V-shaped cut in the top view shows two edge lines of surfaces that appear foreshortened in the front (and side) view. The angled surface on the front of the part is seen in the side view as an edge line making a true angle with the part's base. The front view shows the edge lines of the two angled sides of the part.

## 10.8.7 Edge Views of Inclined Surfaces

As was stated in the last section, the **edge view** of an inclined surface shows in a view where it forms a true angle in a normal plane. The adjacent and related views of the inclined surface always appear foreshortened (they never appear as true shape or larger than the plane itself). This is seen in Figure 10.51. The part has two angled surfaces: one inclined to the horizontal projection plane (top view), the other inclined to the frontal projection plane (front view). The first inclined surface appears as an edge in the front view, and its true angle with the horizontal plane (its base) can be measured here. The second inclined surface shows as an edge line in the side view (hidden line) and as foreshortened in the top and front views. The angle it makes with the frontal plane (and the horizontal-base plane) can be measured in only the side view.

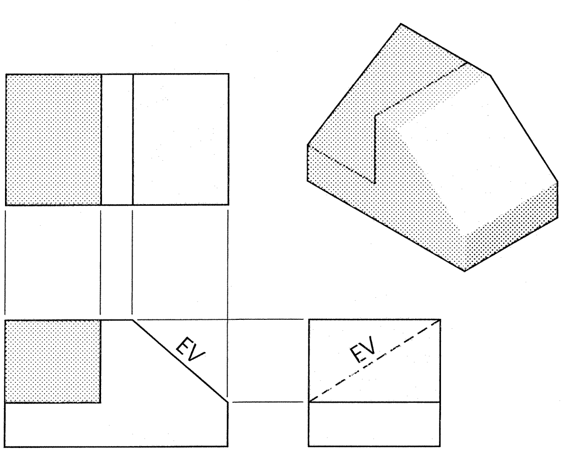



FIGURE 10.51 Edge Views and Inclined Surfaces

Since many of the surfaces of the part in Figure 10.52 are at an angle to the standard projection planes, it is an example of a drawing that does not adequately describe its features. When this happens, an auxiliary view showing the angled surface as true shape is necessary. The surfaces are at an angle to the frontal projection plane, the front view, and the profile projection plane, side view. Nowhere do the vertical surfaces of the part's upper portion show as true shape.

## 10.8.8 Oblique Surfaces

**Oblique surfaces** are inclined to all three principal planes of projection, which makes each view of the surface appear foreshortened (distorted). Since the oblique plane cannot appear as an edge line, each view of it always displays the same number of sides and has a similar shape. Figure 10.53 is an example of a part with an oblique surface. Since it is three-sided, each view of the surface will have three sides and each view shows the plane distorted.

The true shape of an oblique plane cannot be seen in any of the principal projection planes. To establish a true-shape view of an oblique surface, a secondary auxiliary view must be projected (auxiliary views are discussed in Chapter 12).

In Figure 10.53, the oblique surface is labeled and shaded. The surface is formed by the removal of the front corner of the part. In Figure 10.54, the part has two oblique surfaces. The intersecting line formed by the two oblique surfaces shows as true length in the side view. This line is inclined to the base of the part; but since it shows as true length in one of the three principal planes of projection, it is not an oblique line. *An oblique line is inclined to all three principal planes of projection.*

## 10.8.9 Curved and Cylindrical Surfaces

**Curved features,** such as **cylindrical, conical,** and **spherical** shapes, are displayed on drawings as shown in Figures 10.55, 10.56, and 10.57. Cylindrical shapes, as in Figure

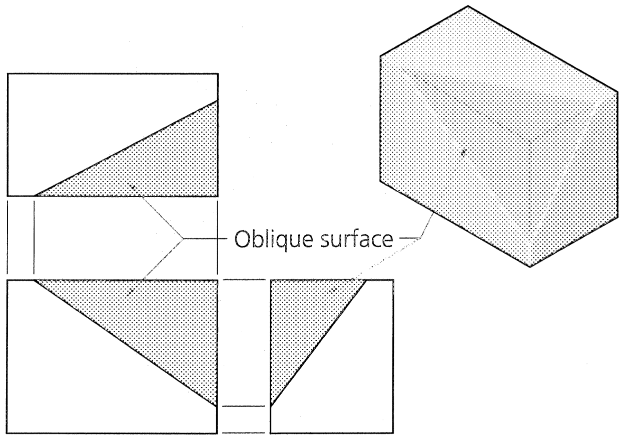



FIGURE 10.53 Oblique Surfaces in Related and Adjacent Views

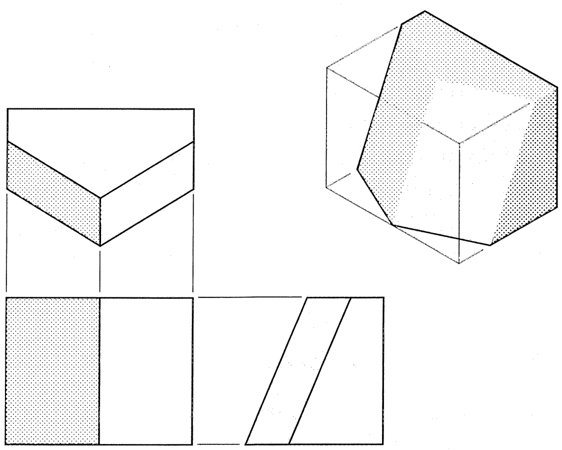
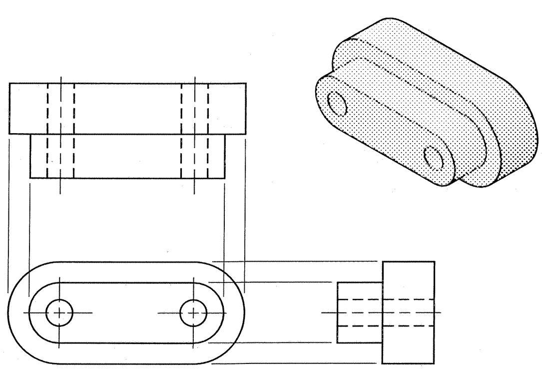


FIGURE 10.54 Oblique Surfaces

.0.55, show as true-shape curves in views that are perpen-licular to their surface. The front view of this part shows the rue-shape/size curve of the cylindrical surface. The side and op views are parallel to the curved surface. Therefore, in hese views, the cylindrical shape appears as a rectangle.

In Figure 10.56, the part has a number of cylindrical urfaces. The side view of the part shows the true shape and



IGURE 10.55 Cylindrical Features

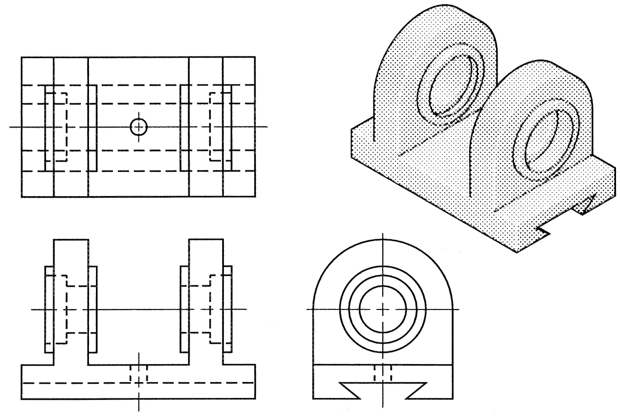


FIGURE 10.56 Curved Surfaces on Drawings

size of the curves, whereas the top and front views display only the edges of the curved surfaces. Without the side view, the drawing could not have been accurately read; the curved features would not have been apparent. For parts with curved features, always provide at least one view where the curve appears true shape.

Figure 10.57 displays the three types of curved surfaces. The **cylindrical surface** shows as a circle in one view and as a rectangle in the other two views. The **conical surface** appears as a circle in one view also, but its other two views show the surface as a triangle. The **spherical surface** shows as a circle in all three views, as would a ball when viewed from any direction.

In both Figures 10.56 and 10.57, the pictorial view of the part provided in the upper right of the illustration is a CAD-modeled true 3D wireframe model of the part, as are many of the examples in the text. Wireframe models are displayed with all edge lines. True visibility is difficult to establish without some experience.

### You May Complete Exercises 10.5 Through 10.8 at This Time

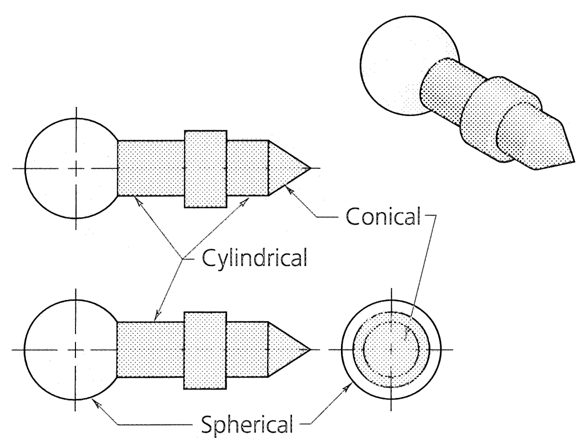



FIGURE 10.57 Representing Cylindrical, Conical, and Spherical Features

## 10.8.10 Intersection of Curved Surfaces

Where two cylindrical surfaces meet, a line of intersection must be determined. With a 3D CAD system, the line of intersection is determined automatically with either an "intersection of surfaces" or "union of solids" command. The system displays the surfaces and calculates their common line (intersection line). When the line of intersection is derived manually, it must be plotted or represented according to established drafting conventions. Three conditions are possible:

- The two curved surfaces have the same diameter.
- The two curved surfaces have different diameters.
- One of the two curved surfaces is so small that it would be a waste of time to plot the line of intersection.

In Figure 10.58, the small-diameter cylindrical surface, which intersects the vertical cylinder, does not show a distinct-enough line of intersection. Therefore, it is accepted conventional practice to show the intersection as a straight line or to use an ellipse template and show a small curved intersection line. The right side of the intersecting cylinders shows a cylindrical surface large enough to be plotted. The miter line method can be used, or transferring the points with dividers will suffice. Points are established on the curve of the cylinder in the top view, either randomly or evenly spaced, as shown here. The points are projected to the side view first. The side and the top views of each point are then projected to the front view as shown. *The intersection of related projection lines locates a point on the line of intersection.* The points are connected with an irregular curve.

## 10.8.11 Plotting Elliptical Curves

**Elliptical shapes** are created by the intersection of planes and curved surfaces. In Figure 10.59, the curve is formed by the intersection of the curved surface and a flat plane surface (not shown). The resulting shape is a surface that is elliptical on one end and a straight line on the other. This inclined surface does not appear as true shape in any of the given three principal planes of projection. To establish the line of intersection in the top view (curved edge), the side view of the cylindrical surface has a series of points located along it. The greater the number of points, the greater the accuracy of the plotted curve. Each point on the curve is projected to the front view. The points are then transferred to the top view via the miter line method or with the aid of dividers. The intersection of related projection lines and transferred distances establishes points along the line of intersection. Connecting the points with a smooth curve completes the view.

## 10.8.12 Space Curves

Irregular-shaped surfaces (**space curves**) must be plotted (Figure 10.60). The curved surface of this part was cut by an inclined plane (not shown). The true shape of the inclined surface does not appear in any of the three views. To plot the resulting intersection, establish a number of points along the curve in the top view, where the curve's edge line is shown. The more points that are plotted, the greater the accuracy of the curve. Each point is projected to the front view. The points are then projected to the side view from the front view. Lastly, the points are transferred to the side view from the top view. The resulting series of points in the side view is connected using an irregular curve to establish a smooth curve.

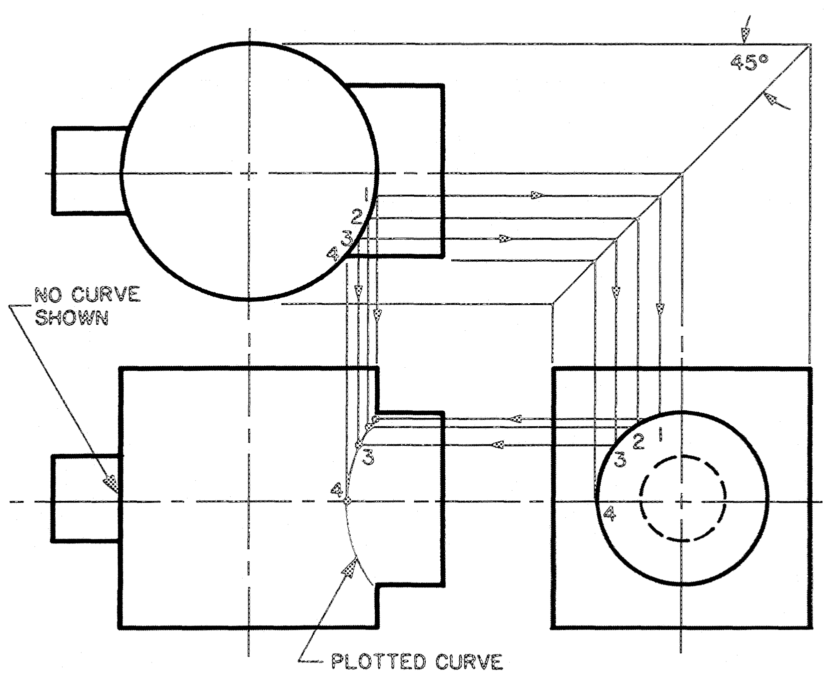



FIGURE 10.58 Intersection of Dissimilar-Size Cylinders

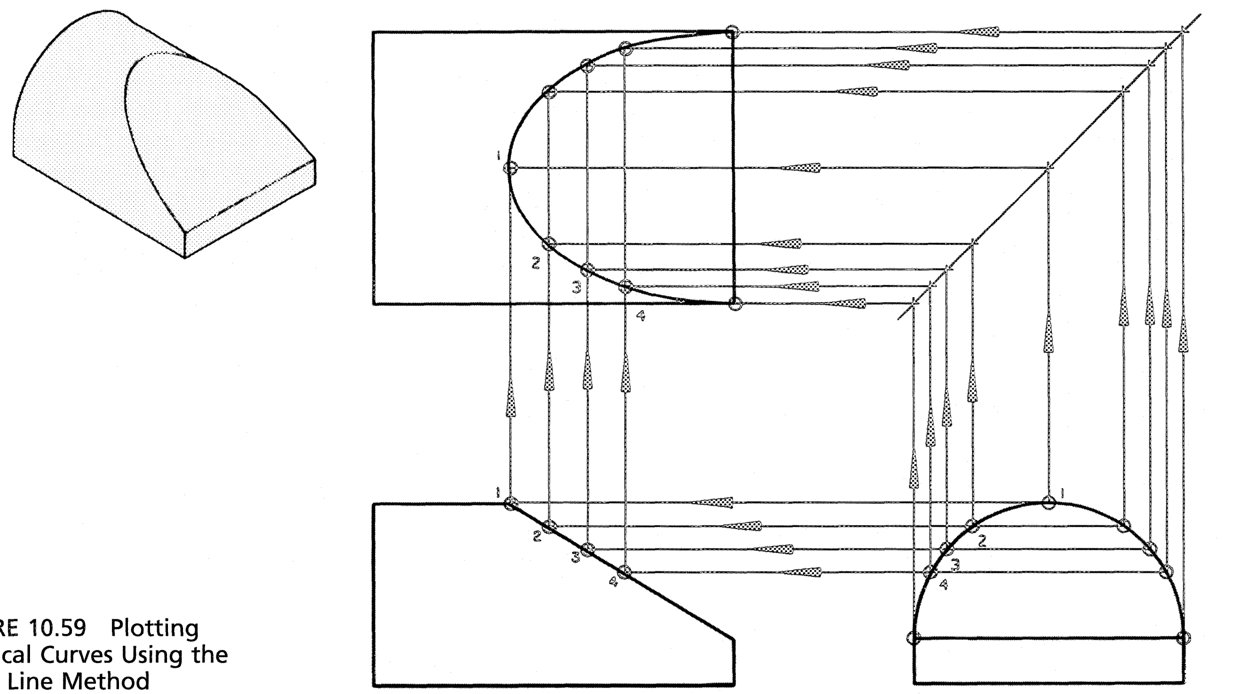


FIGURE 10.59 Plotting Elliptical Curves Using the Miter Line Method

### 10.8.13 Hole Representation

The part in Figure 10.61 has a number of curved features, including a through-hole and a counterbored hole. The diameter of the hole (.8125) is given for the two holes that are aligned. The counterbored hole has a diameter of .5625 for the thru-hole and a counterbore diameter of .875 to a depth of .250. A machinist reading this drawing would be able to choose the proper equipment to accomplish these machined features. In most cases, the type of hole is no longer noted. The machinist determines whether to use a drill, a reamer, or a boring tool. The decision depends on the hole size, the material of the part, and the tolerance requirements. *A hole is always defined by its diameter, never by its*

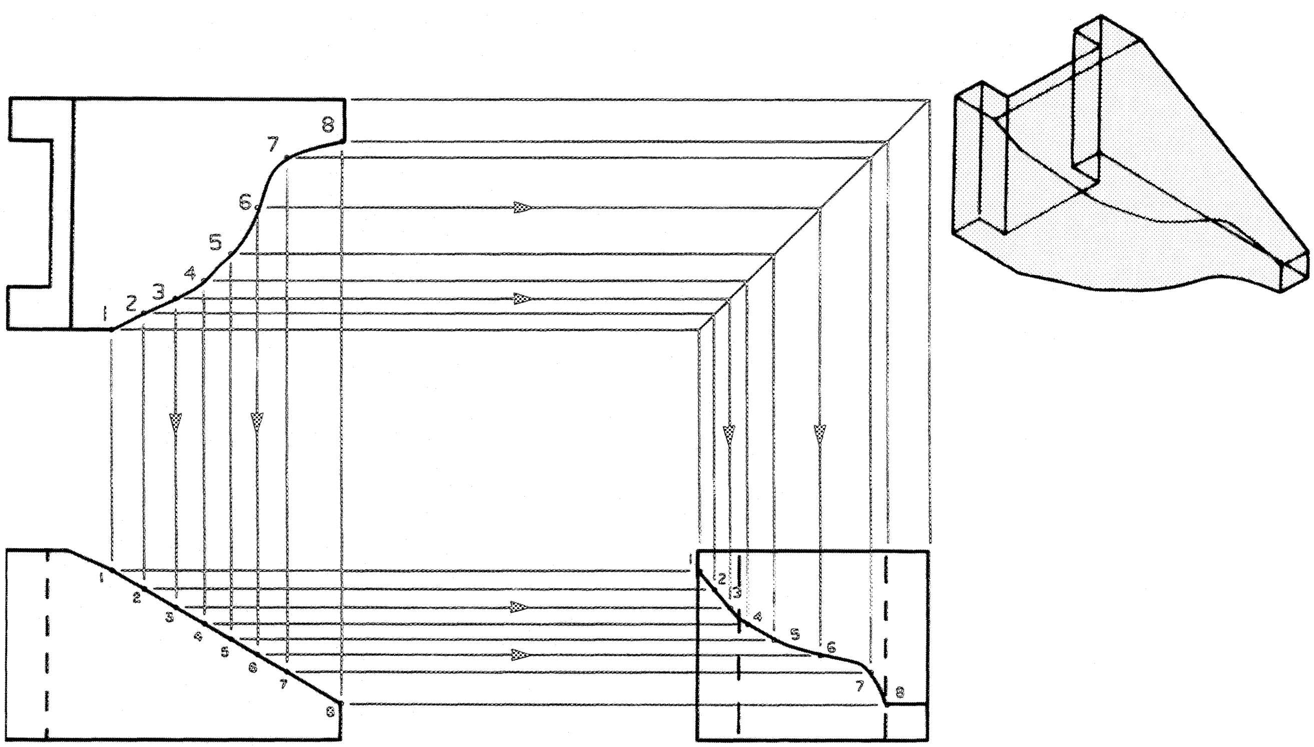


FIGURE 10.60 Plotting Space Curves Using the Miter Line Method

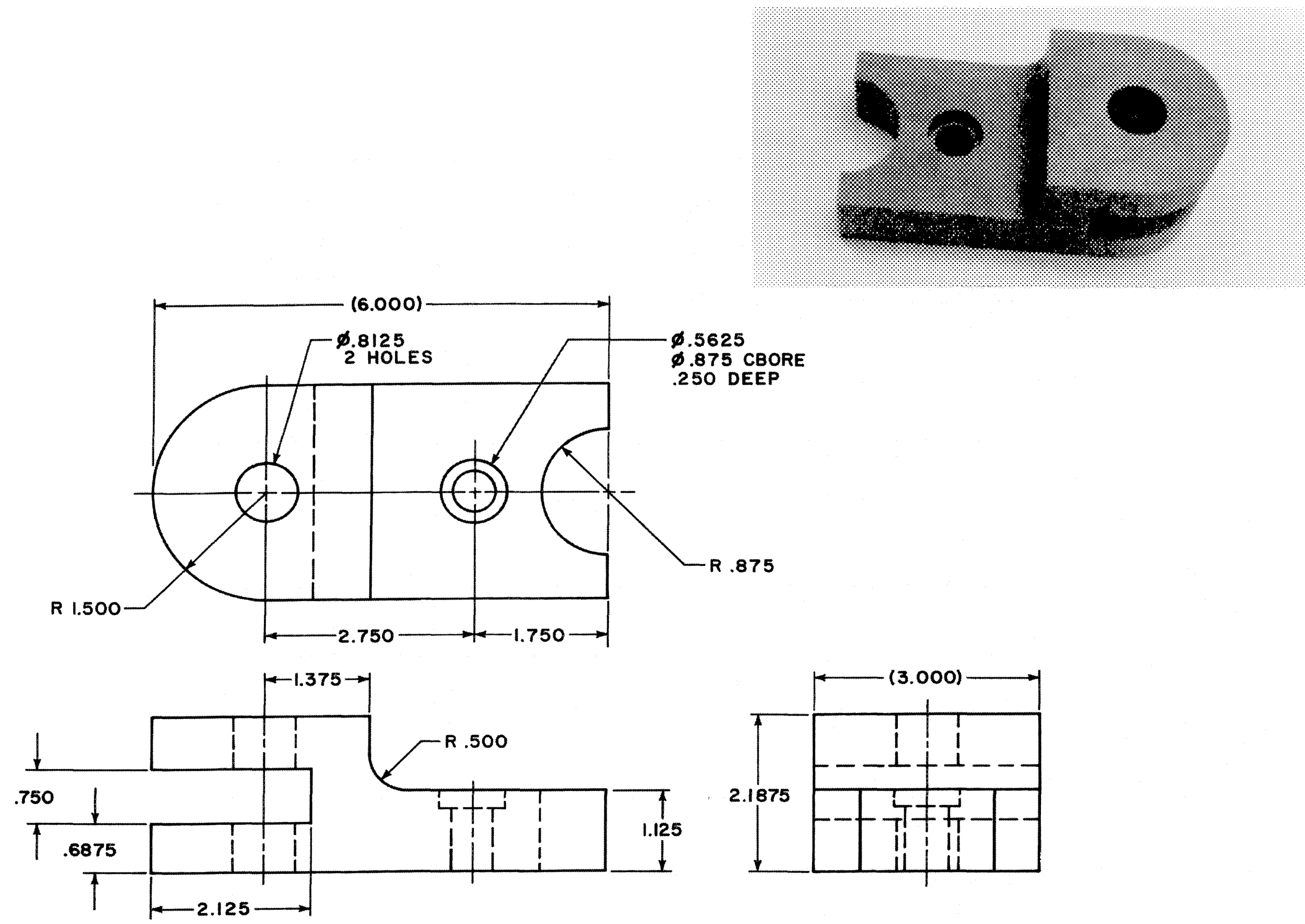



FIGURE 10.61 Detail of Breaker

*radius.* Drills, reamers, bores, and other hole machining tools are described by their diameter, not their radius value.

Figure 10.62 provides a detailed demonstration of how holes should and should not be represented on drawings. Since each situation and type of hole will be encountered repeatedly, this illustration should be studied carefully. The simplest hole callout provides the diameter symbol and the diameter value as in the DRILL OR REAM callout. Unless the depth is given, the hole depth is understood to be through the part. When holes will completely penetrate the part, they are sometimes noted on drawings with the word THRU (instead of "through"). Notice the difference between the CORRECT REPRESENTATION and the INCORRECT REPRESENTATION for each type of hole.

A hole that does not go through the part is called a **blind hole.** In Figure 10.62 it is shown in the depth view as two lines that represent the edges of the hole diameter, plus a centerline. A centerline is required for both blind holes and through-holes in every view in which they are shown. The bottom of the hole is a conical point. The conical shape is formed by the drill tip and, for convenience, is drawn at 30°. The depth of the blind hole is represented by the end of the cylindrical portion of the hole. The depth value is noted in the dimension under the diameter

Holes are either blind holes or through-holes. In Figure 10.62, five are depicted as thru-holes. If they were blind holes, the drill depth would be stated under the diameter callout in the dimension. The following hole types are found on machined parts throughout industry.

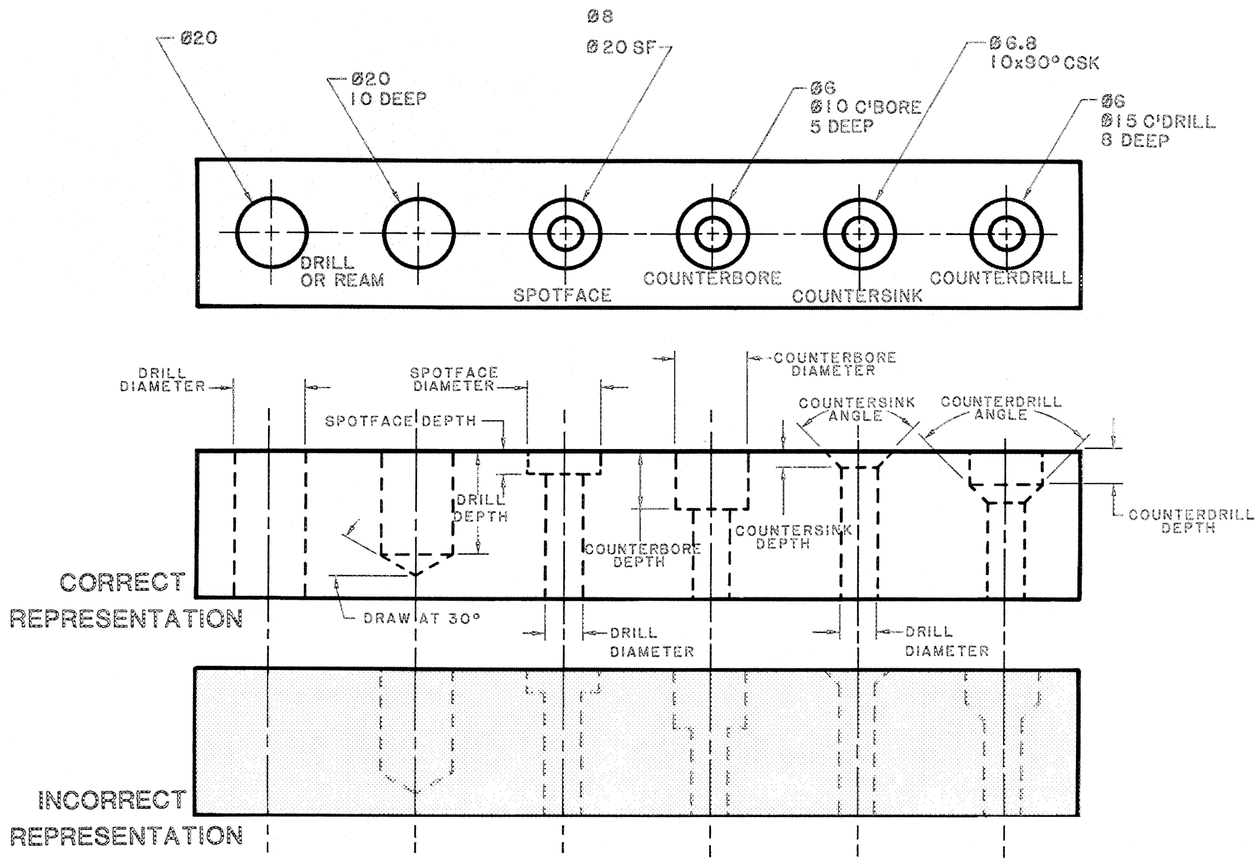
*Common Hole Types on Machined Parts*

*Spotface* A hole that has been drilled to the required depth and whose upper part is enlarged. The depth of the spotface is sometimes not noted. The spotface depth is drawn, depending on the part, at .0625 (1.5 mm) to .125 (3 mm). Spotfacing cleans up the surface around the hole so that a bolt head or other item may rest flush with the surface.

*Counterbore* Similar to a spotface, except the enlarged hole has a specific depth. The counterbore depth is specified in the callout dimension under the counterbore diameter.

*Countersink* A hole that has been enlarged conically to a specified diameter and depth The conical angle is drawn at 90° for simplicity.

*Counterdrill* A countersink and a counterbore combined. The transition between the two diameters is a conical surface formed by the angle of the Moors tip. Countrdrills are




IGURE 10.62 **Types and Representation of Holes on Drawings**

pecified by their diameter and depth. The angle of the ounterdrill is shown in the adjacent view.

## .0.8.14 Fillets and Rounds

**:astings** are rough parts that are usually machined along one r more of their surfaces. A casting will have curved interections between mating surfaces. Castings cannot be formed ccurately without these curved corners. Perfectly sharp corers are not possible with the casting process. Drawings of ıachined castings require the representation of these surfaces nd their intersections. Two rough interior surfaces intersect nd form a rounded corner called a **fillet.** Two rough exterior ırfaces meet and form a corner called a **round.** The part in igure 10.63 has a variety of rounds and fillets.

When two intersecting surfaces meet and one is mahined, the corner becomes a sharp edge. If both surfaces re machined, the corner is also shown as a sharp edge. ounds will show only when both of the mating exterior ırfaces are unmachined (rough or cast surfaces). The ıaterial removed during machining is determined by the art's casting dimensions and the machining dimensions. ɔmetimes separate drawings are used. A **casting drawing** is one for the foundry, and a machine drawing is completed ɪr the machine shop (see Chapter 14).

As a design requirement, fillets and rounds serve to :duce the possibility of failure of a joint. Sharp points are ɔssible points of fracture. Most fillets are determined by the

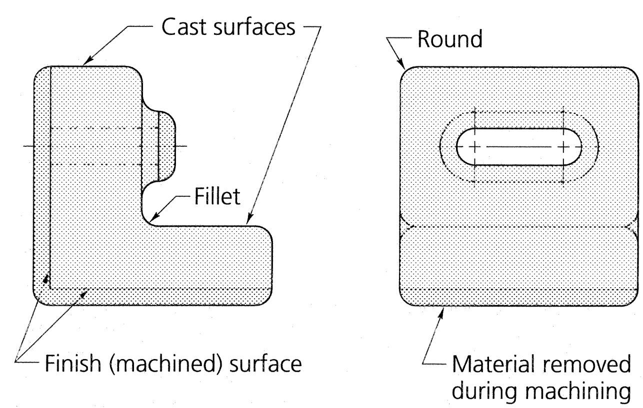



FIGURE 10.63 **Fillets, Rounds, and Castings**

foundry to meet the design requirements, the methods of casting, and the thickness of the part. In many cases, the selection of the fillet diameter is left to the pattern maker.

## 10.8.15 Tangent Surfaces

When a curved and a plane surface are tangent, a *point of tangency* may be required. In Figure 10.64, the cylindrical surfaces are connected by plane surfaces along the sides of the part. Since the cylindrical ends are different diameters, the tangent points of the cylinders and the planes will not fall along the centerline in the front view. The back surface is flush with the two diameters; tangent points A therefore fall

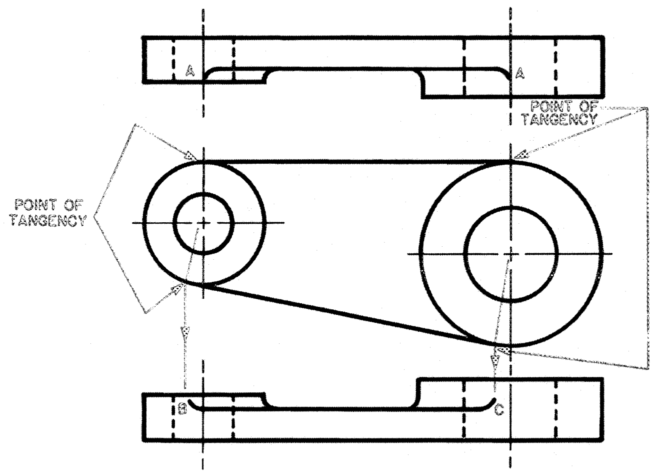



FIGURE 10.64 Runouts and Points of Tangency on Drawings

along the centerline. Because the circles are staggered and of different diameters, the front view of the tangent points does not fall along the centerline. Tangent points B and C are determined by drawing construction lines perpendicular to the front edge and through the center of each cylindrical surface in the view where the diameter shows true shape (top view here). The intersection of this line and each circle's circumference determines the points of tangency (B and C).

### 10.8.16 Runouts and Edge Representation

After the point of tangency between a plane surface and a cylindrical surface has been determined, the runout can be drawn. **Runouts** are curves at the point of tangency. If the part is a casting, the runout will be a fillet at the tangent point, as in Figure 10.64. Points B and C are the points of tangency of the surface intersections, but they are also the transition points of the cast surfaces. Therefore, the fillet must be drawn as shown. The radius of the fillet establishes the runout; it is normally constructed with a template. Only 45° (one-eighth) of the curve need be drawn for most situations.

**You May Complete Exercises 10.9 Through 10.12 at This Time**

## 10.9 OPPOSITE-HAND PARTS

There are many industrial applications for parts that are the exact opposite of one another. These are called **opposite-hand parts** or **right-hand** and **left-hand parts.** In most cases, only one drawing is needed to describe both parts. To visualize a right-hand and a left-hand part, take an existing drawing (one from the text will do) and hold it up to a mirror. The reflection in the mirror shows the opposite hand of the part. If a right-hand part was used, the mirror shows the left-hand projection. Of course, to see a simple example of right-hand and left-hand parts, just look at your own hands.

Examples of industrial applications of right-hand and left-hand parts are numerous. A car has many opposite-hand parts, both in the engine and on the body of the automobile. When viewing parts, care must be taken not to confuse right-hand and left-hand parts with parts that are the same but just happen to be installed on both sides of an assembly. For instance, a car's fenders and doors are obviously right-hand and left-hand parts. But headlights, wheels, hubcaps, and headrests are not.

Right-hand and left-hand parts are required in many circumstances. If a project requires a right-hand and a left-hand part, it is accepted practice to draw only one of the parts and to note on the drawing:

**NOTE: RIGHT-HAND AND LEFT-HAND PART REQUIRED.
RH PART SHOWN.**

In general, if there are any differences between the two parts, it is normal practice to draw both. If the differences are minor, such as a hole size or the addition of a hole, then these differences can sometimes be established with a note or with a callout. The following example for the diameter dimension for a hole shows this situation:

.500 DIA THRU
LH PART ONLY

When both left-hand and right-hand parts must be drawn, you can save time and energy by tracing the completed side (or making a copy on an office copier), turning it over, and using it to draw the opposite side. A light table allows you to see through the paper to view the reversed drawing that is to be traced.

### 10.9.1 2D and 3D CAD Mirroring Commands for Opposite-Hand Parts

A CAD system will eliminate the need to draw the opposite-hand part. The **MIRROR** command displays the mirror-image view of the part (or selected geometry of a part). Even a 2D system can project the opposite hand of one view of the part. The choice of mirrored views depends on the complexity of the part. The view with the most complex geometry should be mirrored.

A 3D CAD system will have the advantage of projecting a true 3D model of the part's opposite hand, as shown in Figure 10.65. In this illustration, the right-hand and left-hand projections of the part are shown. The part has been mirrored about a plane (shown as a line in the lower illustration and as a plane in the 3D projection). After one hand of the part has been modeled on the system, it is a simple matter to give one command to establish the opposite-hand part.

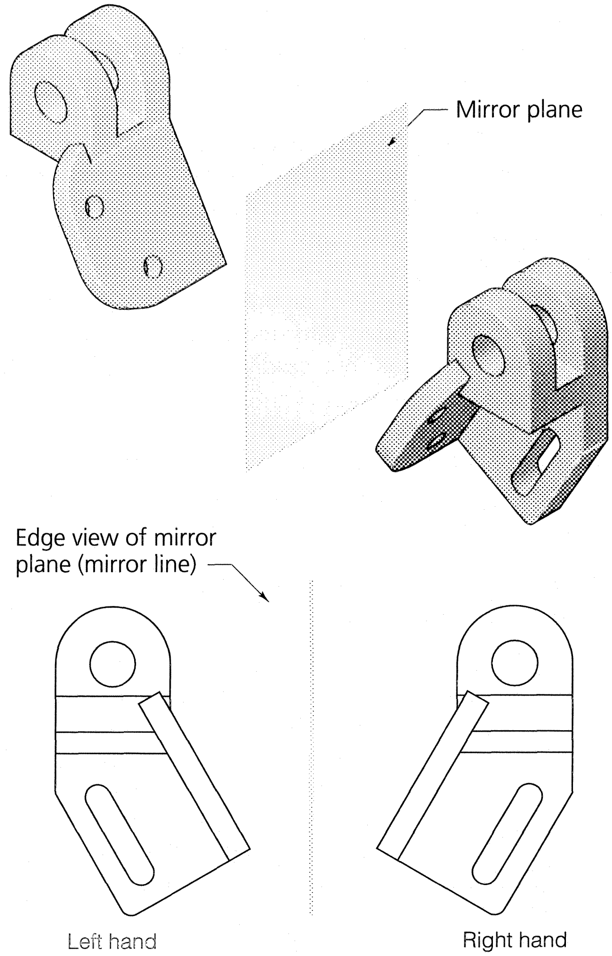



**FIGURE 10.65 Using the MIRROR Command to Create Opposite-Hand Parts**

If using AutoCAD, the **MIRROR** command is given as follows:

`Command:` MIRROR
`Select objects:` use a window (or select each object)
`Pick first corner:` Pick first corner of window
`Pick second corner:` Pick second point of window (enclose the whole part)
`Select objects:` <RETURN>
`First point of mirror line:` Pick point on mirror line
`Second point:` Pick any point above or below—near mirror line
`Delete old objects?` <N> <RETURN>

Because of the speed and simplicity of creating the opposite-hand part, a CAD system for generating the second drawing of the part is a practical alternative to just noting the need for an opposite-hand part on the drawing.

## 10.10 VIEW CONSTRUCTION USING CAD

Views are constructed on a 2D CAD system in much the same way as with manually drawn projects, so all of the preceding descriptions for constructing views are valid. 3D CAD systems, on the other hand, create true three-dimensional models of the part. Because of this, the construction process is very different.

In general, every 3D system requires one or more standard views (or **VIEWPORTS**) when modeling. Since the part can be rotated in 3D space, you need to display only one "view" (the top, normally). As the construction progresses, the model geometry is rotated into other orientations to model the complete part. Afterward, you can request the system to display additional views of the part for dimensioning. In Figure 10.66, the 3D system is displaying the completed model in three standard views and in a rotated view.

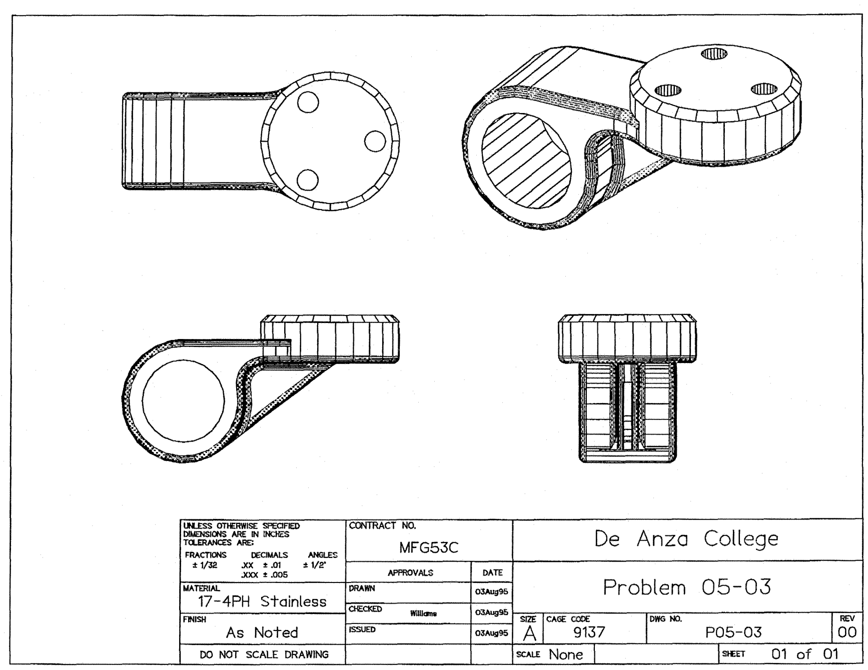



**FIGURE 10.66 Views on a CAD System**

It is not the purpose of this text to explain in detail the process of 3D modeling, but you should understand the differences in establishing views via this procedure. Most CAD systems have six or seven standard views, along with an infinite number of user-defined views. Six of the predefined views are the same as the six principal views. The seventh (when available) is a standard isometric (or rotated) view. Figure 10.67 shows the seven views: (1) top, (2) front, (3) right side, (4) bottom, (5) left side, (6) back/rear, and (7) isometric.

A part was modeled and is shown in a rotated 3D position in Figure 10.68(a). Since you need to show the model in accepted standard orthographic views to place the dimensions, a number of views must be established. The top is displayed in Figure 10.68(b). The front view is then displayed in Figure 10.68(c), and the drawing's right side view is defined in (d).

Regardless of the design method (manual, 2D CAD, or 3D CAD), knowledge and understanding of orthographic projection to create multiview drawings is essential.

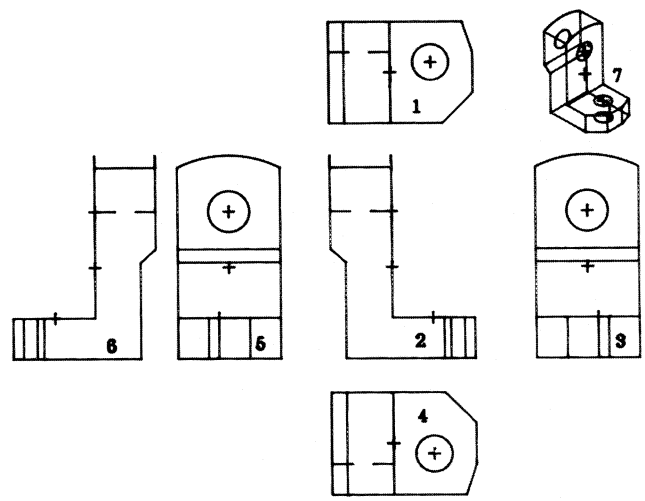
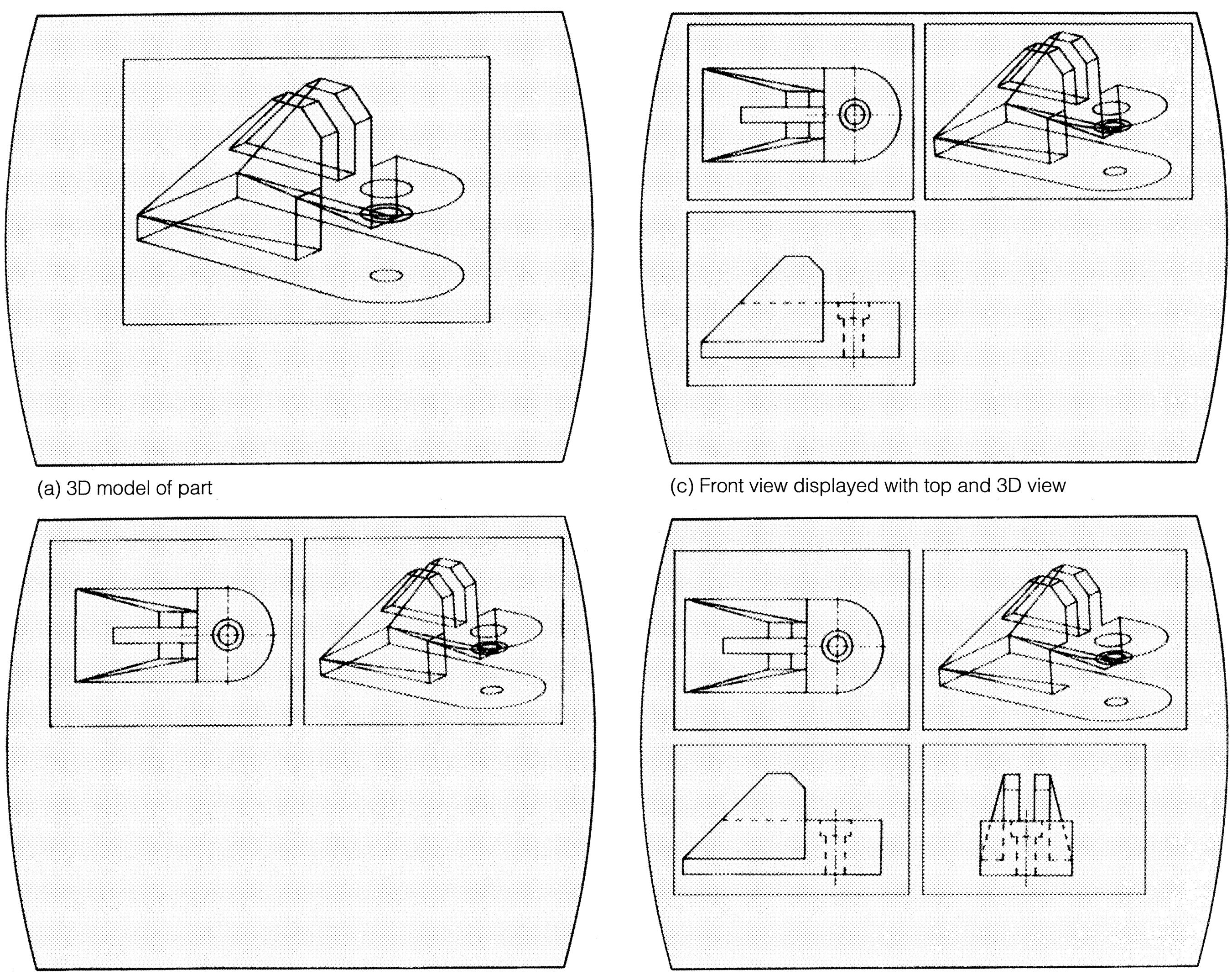



FIGURE 10.67 Seven Predefined Views on a 3D CAD System

**You May Complete Exercises 10.13 Through 10.16 at This Time**



(a) 3D model of part

(c) Front view displayed with top and 3D view

(b) Top view of part displayed along with 3D view

(d) Right side view displayed with top, front, and 3D view

FIGURE 10.68 Orthographic Views

## QUIZ

### True or False

1. Partial projections of views help to save space and paper.
2. Centerlines, phantom lines, dimension lines, and leader lines are all drawn with the same thickness.
3. Centerlines take precedence over hidden lines.
4. The glass box method of projection is used for most drawings.
5. Adjacent and related views are the same.
6. Parallel lines are parallel in all views.
7. Most foreign countries use third-angle projection for their engineering drawings.
8. All orthographic projection is right-angle projection.

### Fill in the Blanks

9. __________ view drawings are normally limited to thin, flat, or __________ round parts.
10. When the object is relatively simple, a __________ line is used to project the third view.
11. Dimensions can be transferred from the top to the side view using __________ lines, the __________ method, or __________ .
12. __________ are considered to be a series of __________ in space having __________ but not __________ .
13. __________ __________ are used to show round features of a part on drawings.
14. **MIRROR** commands are useful in creating __________ __________ and __________ __________ parts.
15. __________ lines always take precedence over hidden lines.
16. A __________ is a specific location in space.

### Answer the Following

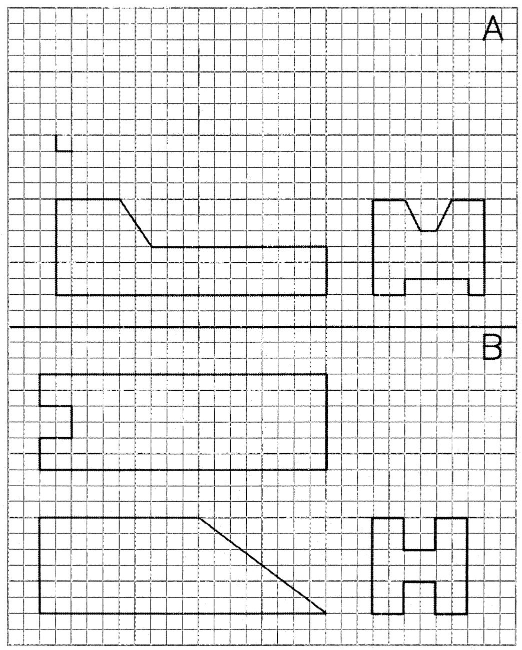
17. What is a fold line, and how is it used?
18. What are the six standard views? How do they relate to the use of 3D CAD?
19. What is the difference between the glass box method and the natural method?
20. What is the image plane for projection?
21. Describe adjacent and related views.
22. Explain the difference between first- and third-angle projection.
23. What determines the spacing and choice of views for a drawing?
24. Describe the ISO projection symbol and its use.

## EXERCISES

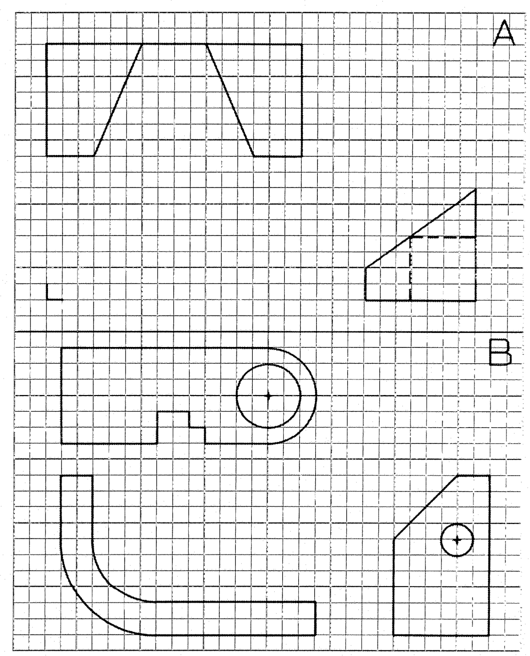
Exercises may be assigned as sketching, instrument, or CAD projects. Transfer the given information to an "A"-size sheet of .25 in. grid paper. Complete all views and ***solve for proper visibility, including centerlines, object lines, and hidden lines.*** Exercises that are not assigned by the instructor can be sketched in the text to provide practice and enhance understanding of the preceding instructional material.

### After Reading the Chapter Through Section 10.6.10, You May Complete the Following Exercises

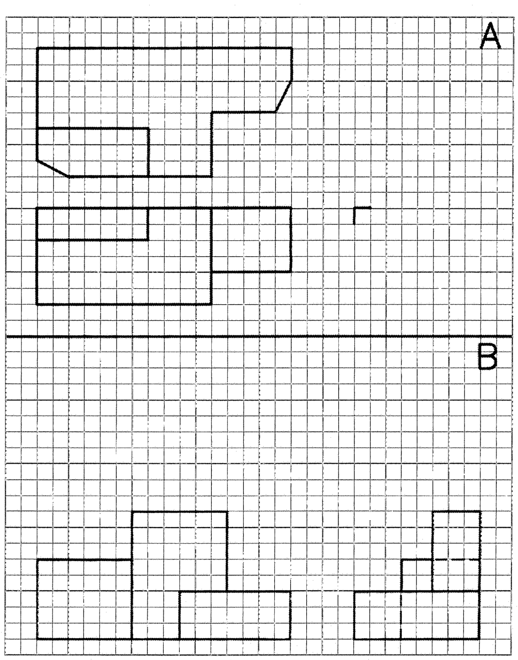
*Exercise 10.1 Through Exercise 10.4* Complete each of the given views and the third view, if required.



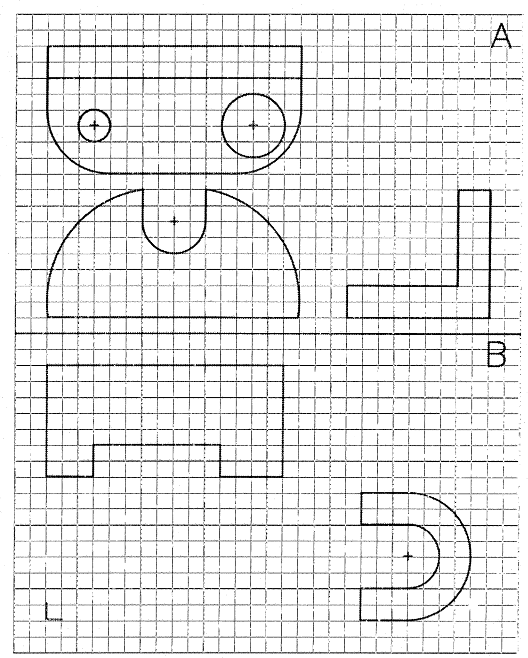
EXERCISE 10.1



EXERCISE 10.3



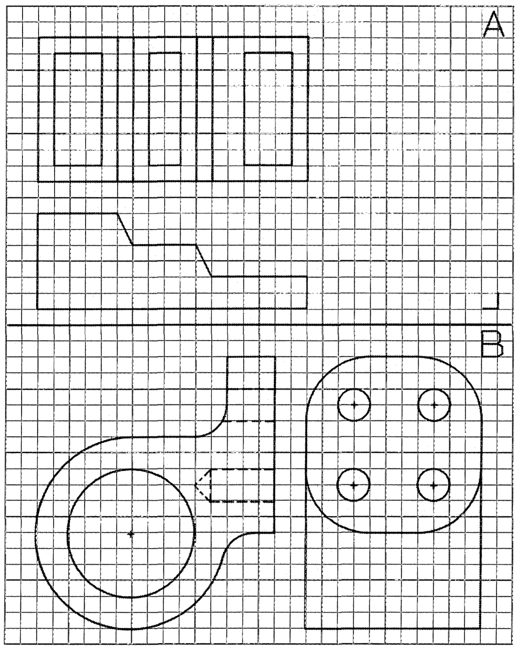
EXERCISE 10.2



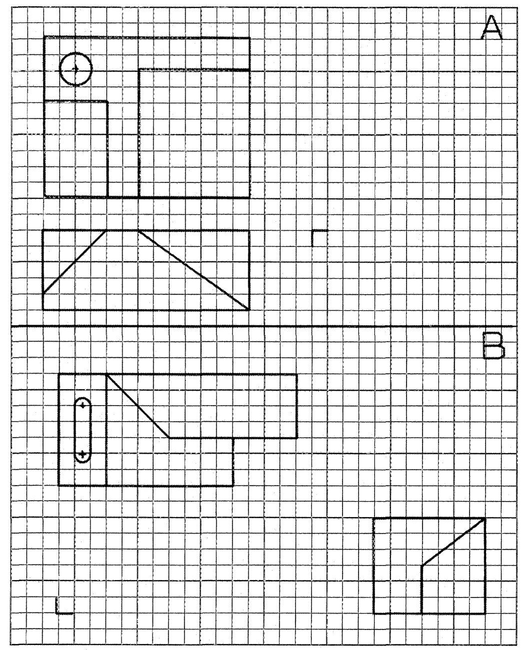
EXERCISE 10.4

**After Reading the Chapter Through Section 10.8.9, You May Complete the Following Exercises**

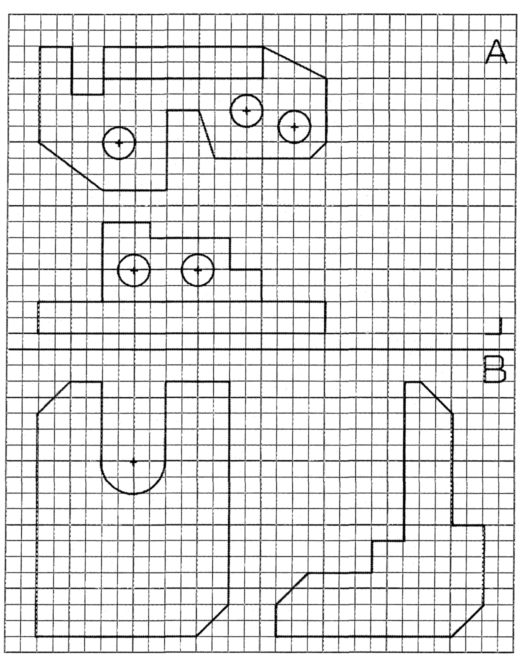
*Exercise 10.5 Through Exercise 10.8* Complete each of the given views and the third view, if required.



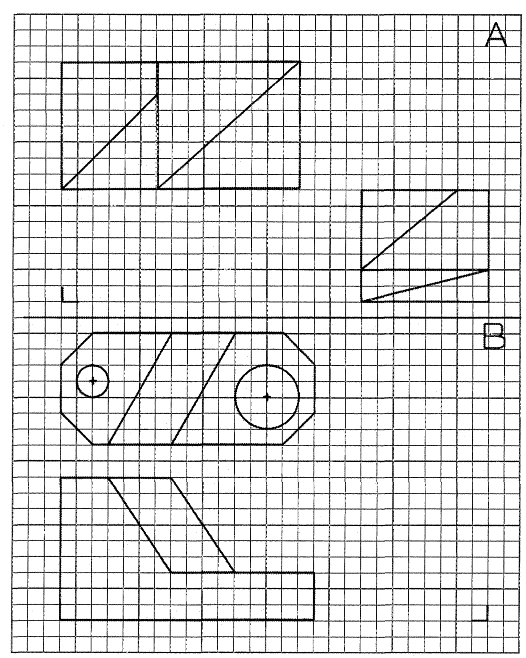

EXERCISE 10.5




EXERCISE 10.7



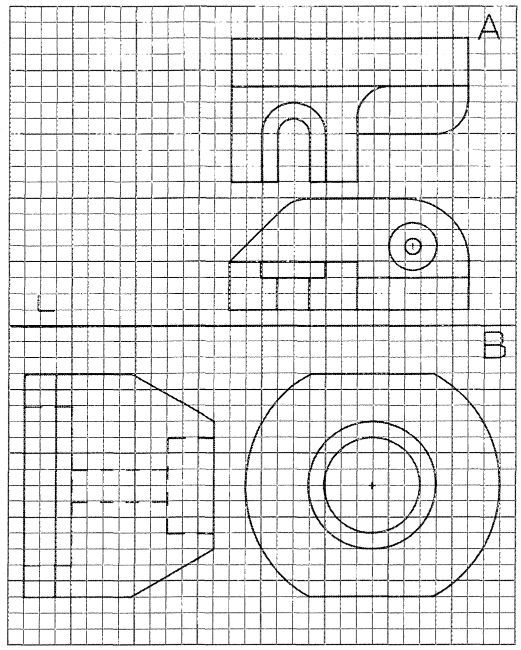

:XERCISE 10.6



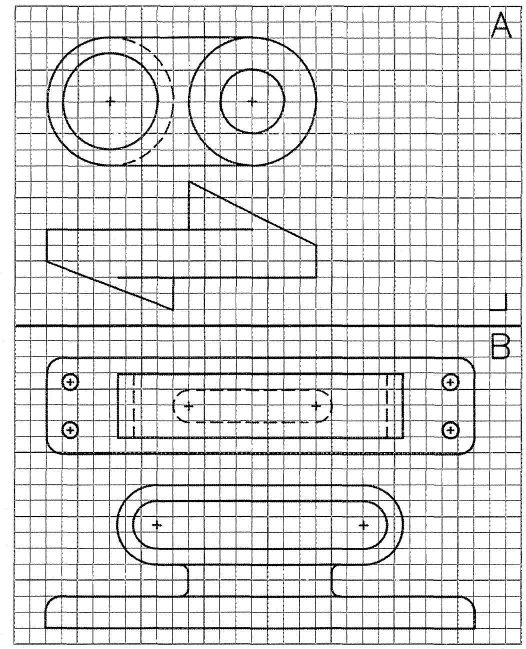

EXERCISE 10.8

**After Reading the Chapter Through Section 10.8.16, You May Complete the Following Exercises**

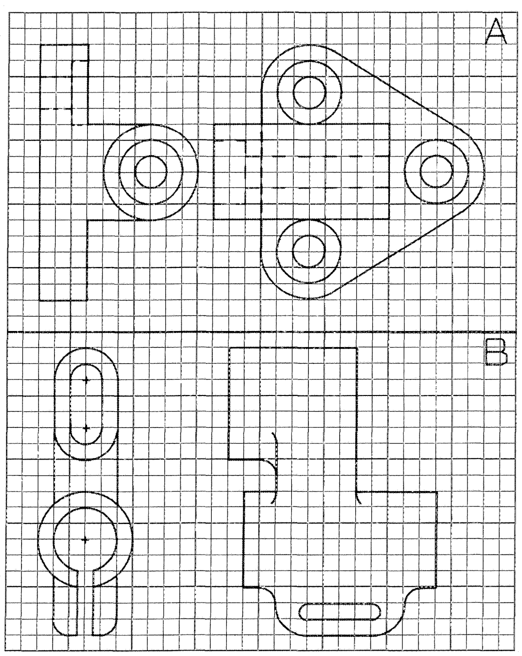
*Exercise 10.9 Through Exercise 10.12* Complete each of the given views and the third view, if required.



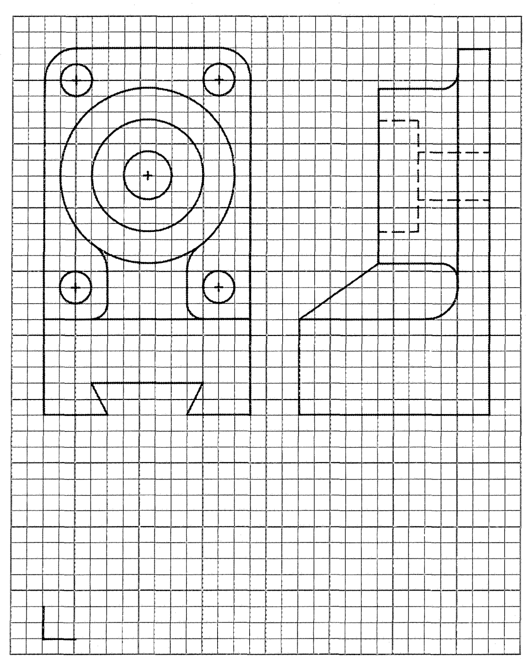

EXERCISE 10.9




EXERCISE 10.11



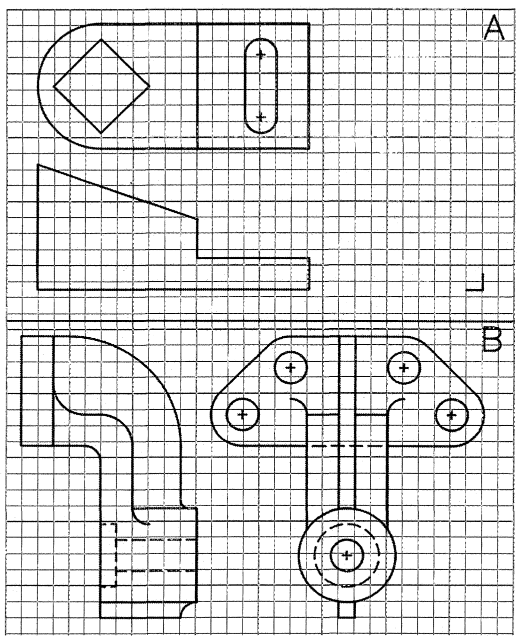

EXERCISE 10.10



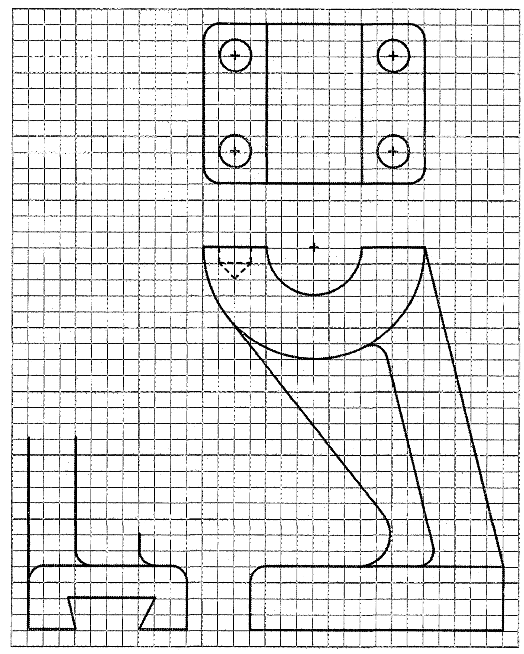
EXERCISE 10.12

**After Reading the Chapter Through Section 10.10, You May Complete the Following Exercises**

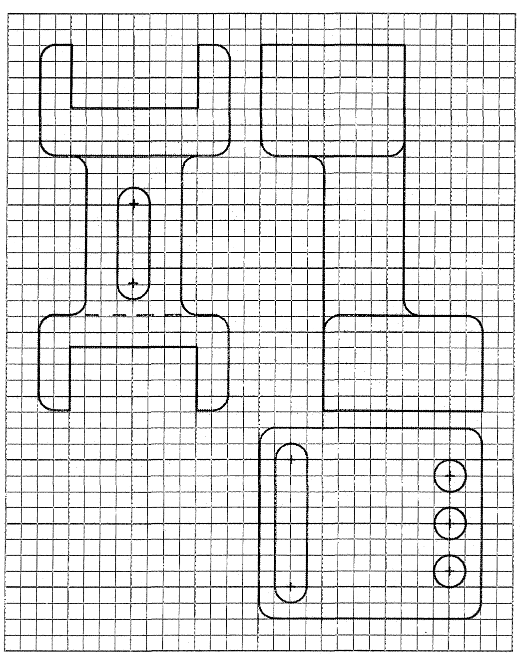
*Exercise 10.13 Through Exercise 10.16* Complete each of the given views and the third view, if required.



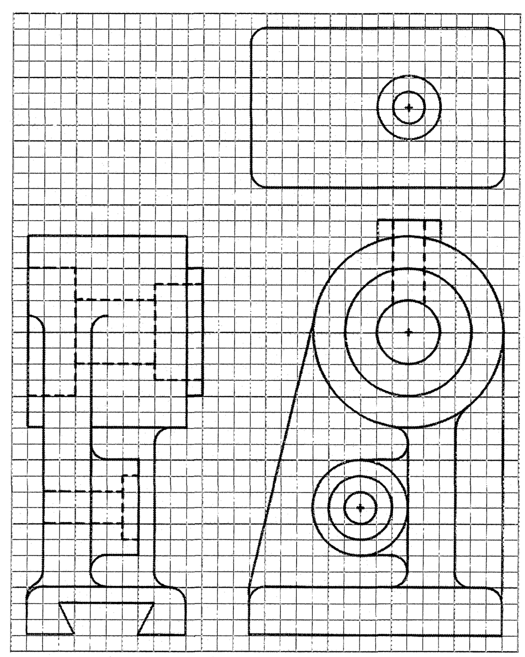

EXERCISE 10.13



EXERCISE 10.15



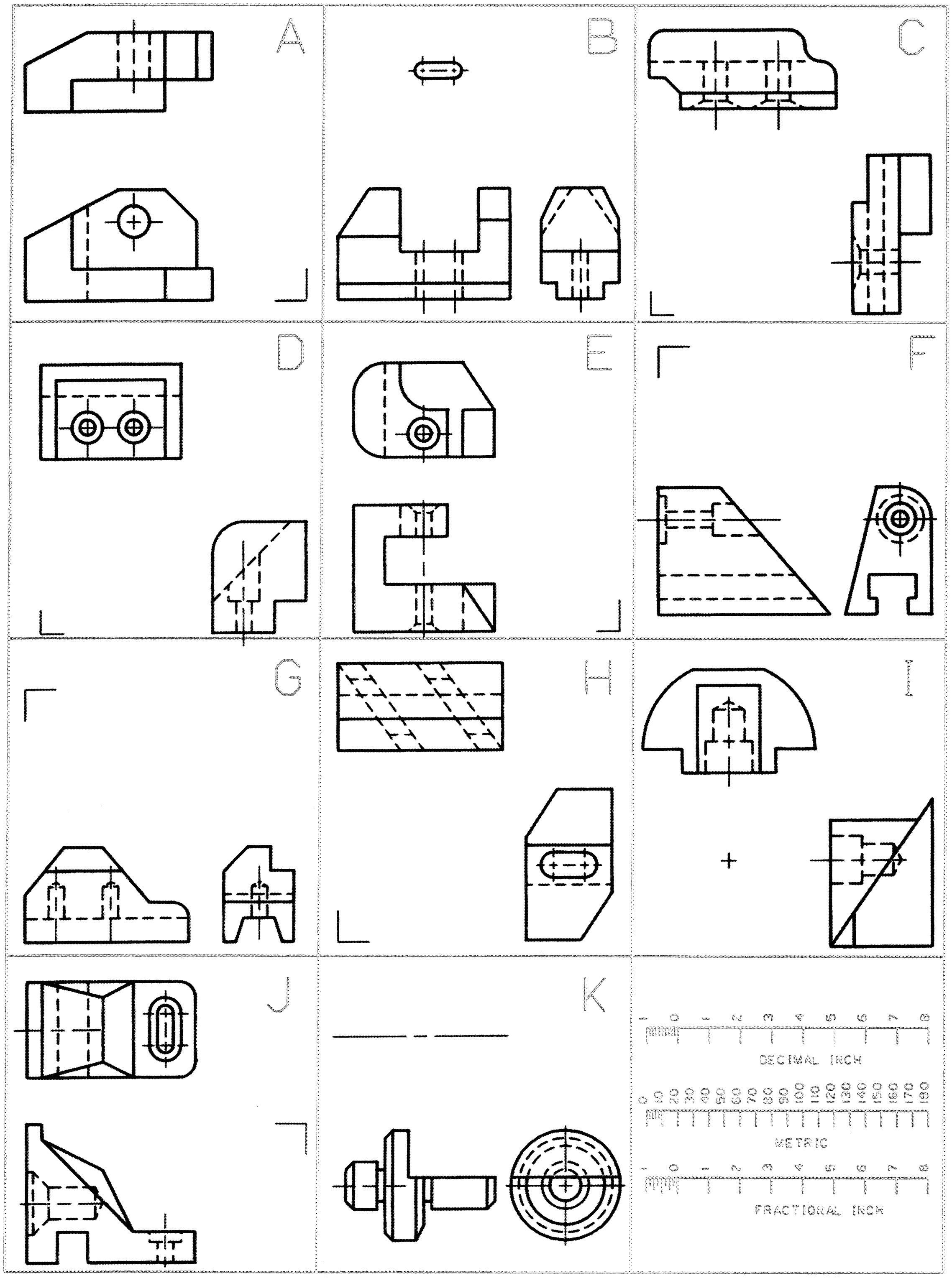
EXERCISE 10.14



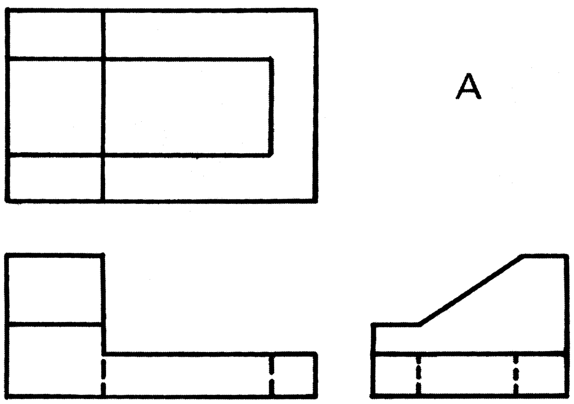
EXERCISE 10.16

## PROBLEMS

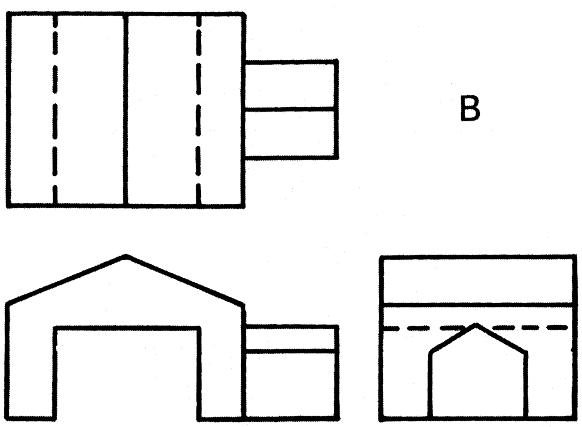
**Problems 10.1(A) Through (K)** Complete each of the problems on an "A"- or "B"-size sheet, as required. Use one of the three scales provided in the lower left corner of the page. Use dividers to take measurements from the drawing and set off on one of the scales to establish the parts dimensions. Round off dimensions where necessary. Solve for the missing view in each problem. All projects will have three views.



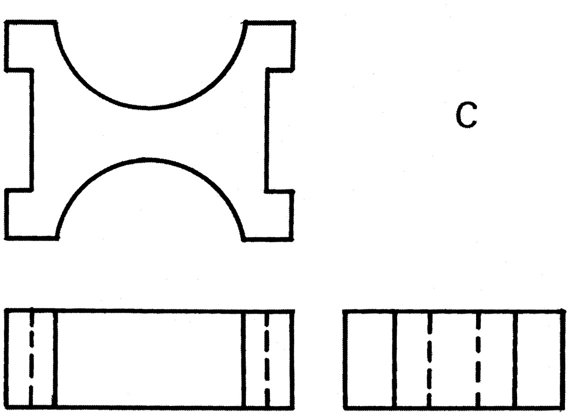
**Problems 10.2(A) Through (G)** Use the same directions as for Problem 10.1. In these problems some of *the given views are incomplete,* though the outline of each of the three views is given. Complete the views as needed.



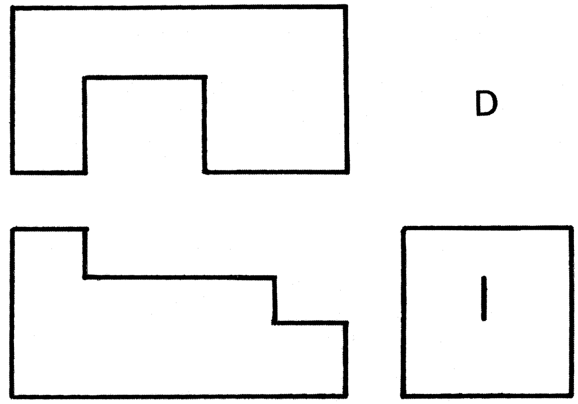
A



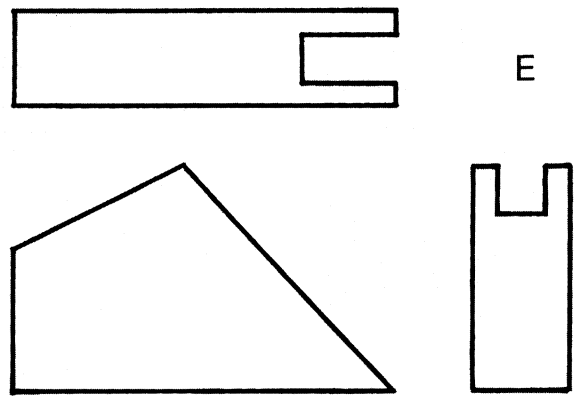
B



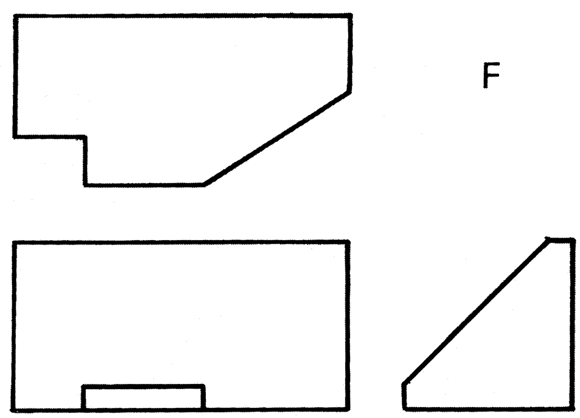
C



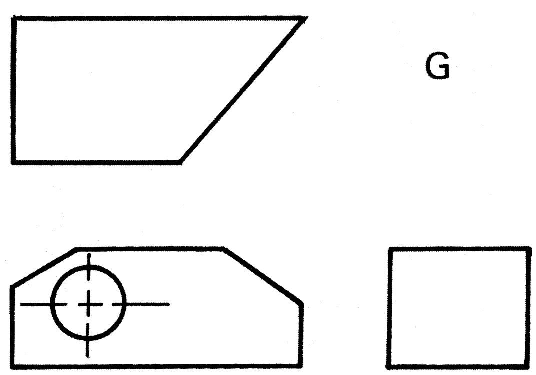
D



E

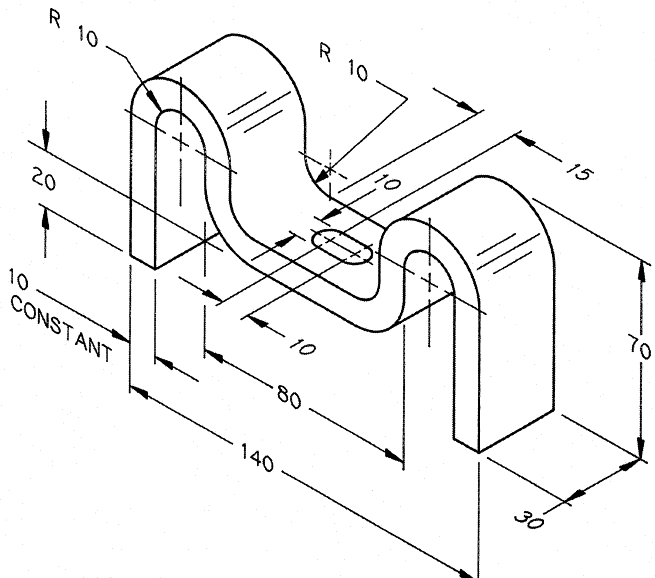


F

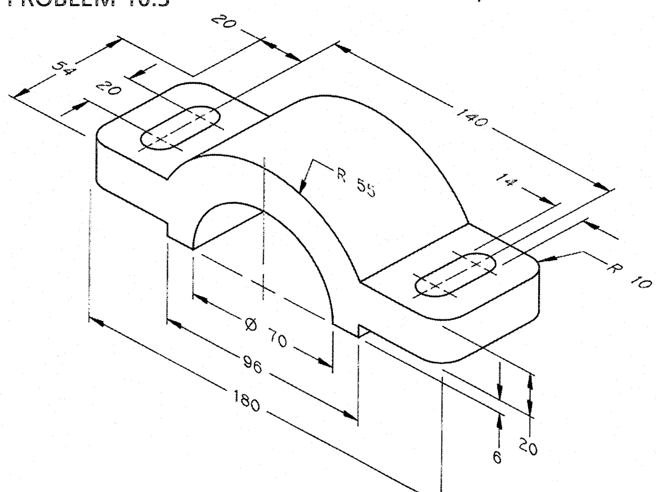


G

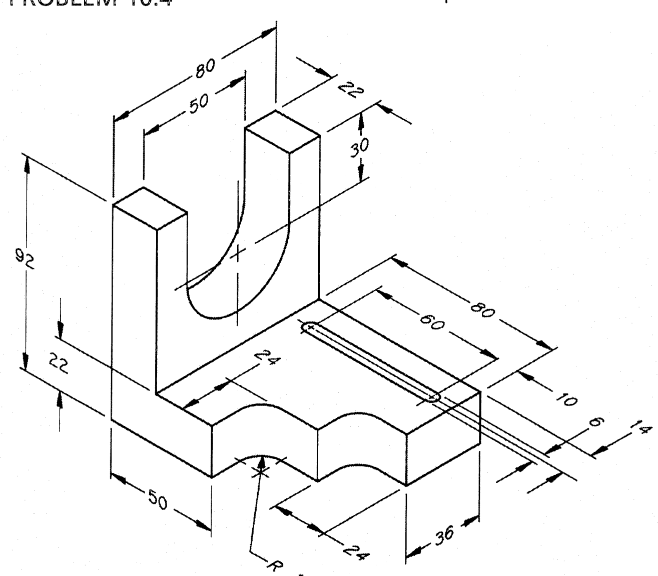
**Problems 10.3 Through 10.8** Draw enough views to describe the part graphically. These projects can be used later for dimensioning projects after completing Chapter 15. Because of this, leave sufficient spacing between views to accommodate dimensions and notes.



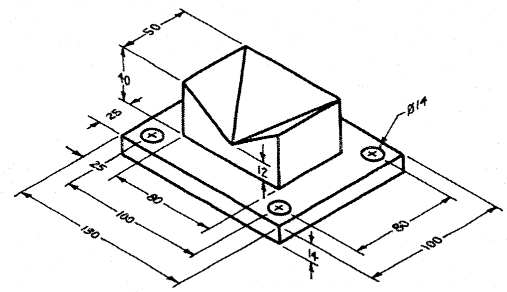

PROBLEM 10.3



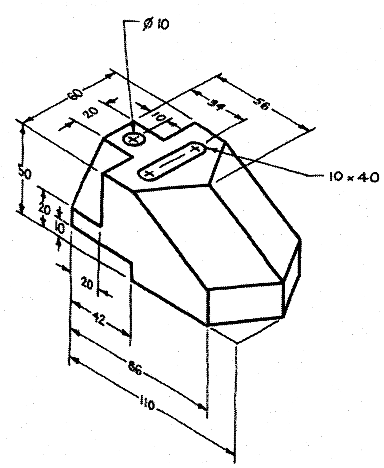

PROBLEM 10.4



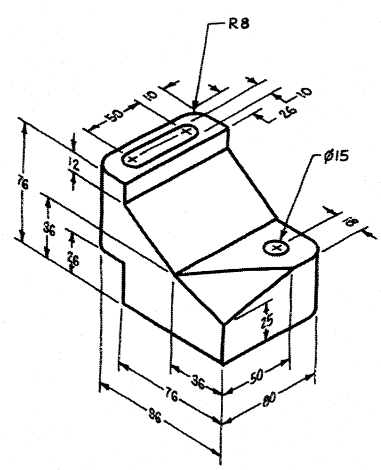

PROBLEM 10.5




PROBLEM 10.6

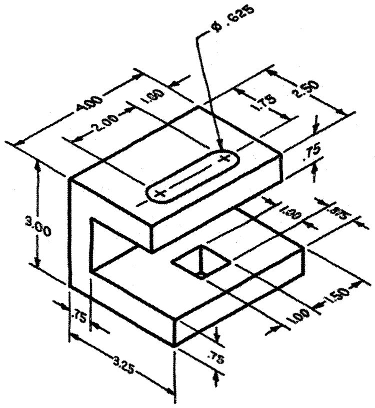



PROBLEM 10.7

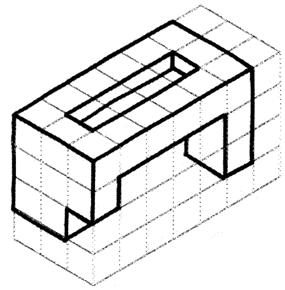



PROBLEM 10.8

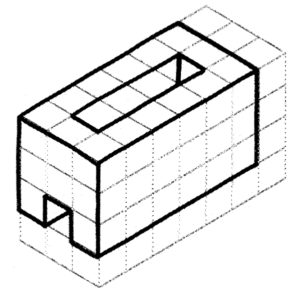
**Problems 10.9 Through 10.26** Draw three views for each of the given problems. Use an "A"-size sheet for each project. Establish all dimensions through grid squares equal to 1.00 in. or 20 mm, as assigned by the instructor.



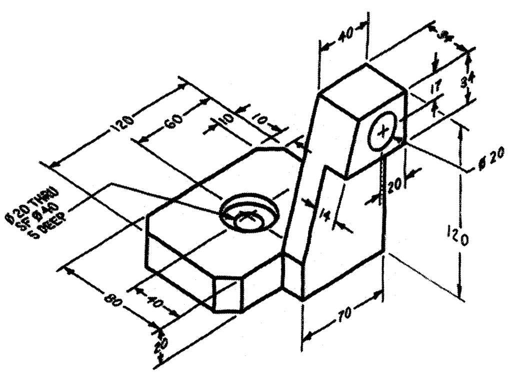

PROBLEM 10.9



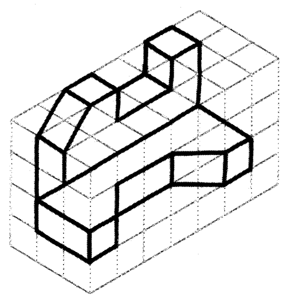
PROBLEM 10.15



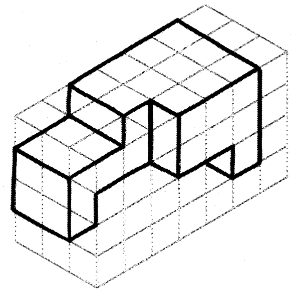
PROBLEM 10.16



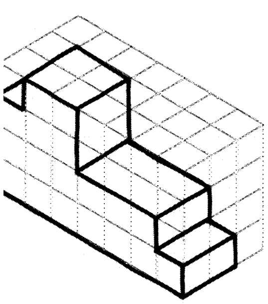

PROBLEM 10.10



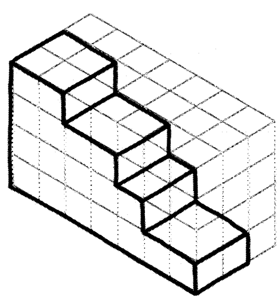
PROBLEM 10.17



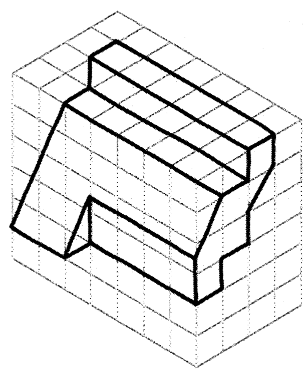
PROBLEM 10.18



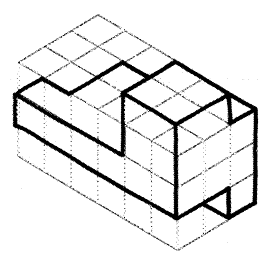
PROBLEM 10.11



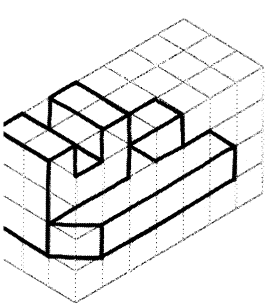
PROBLEM 10.12



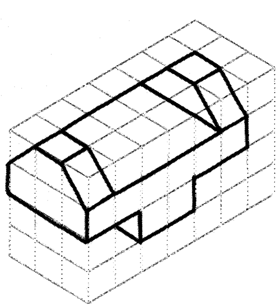
PROBLEM 10.19



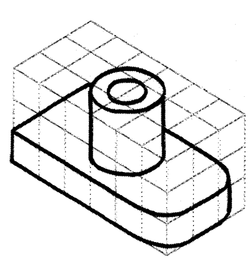
PROBLEM 10.20



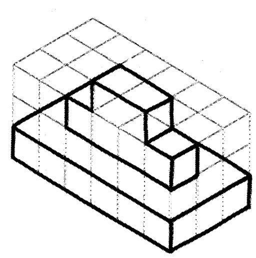
PROBLEM 10.13



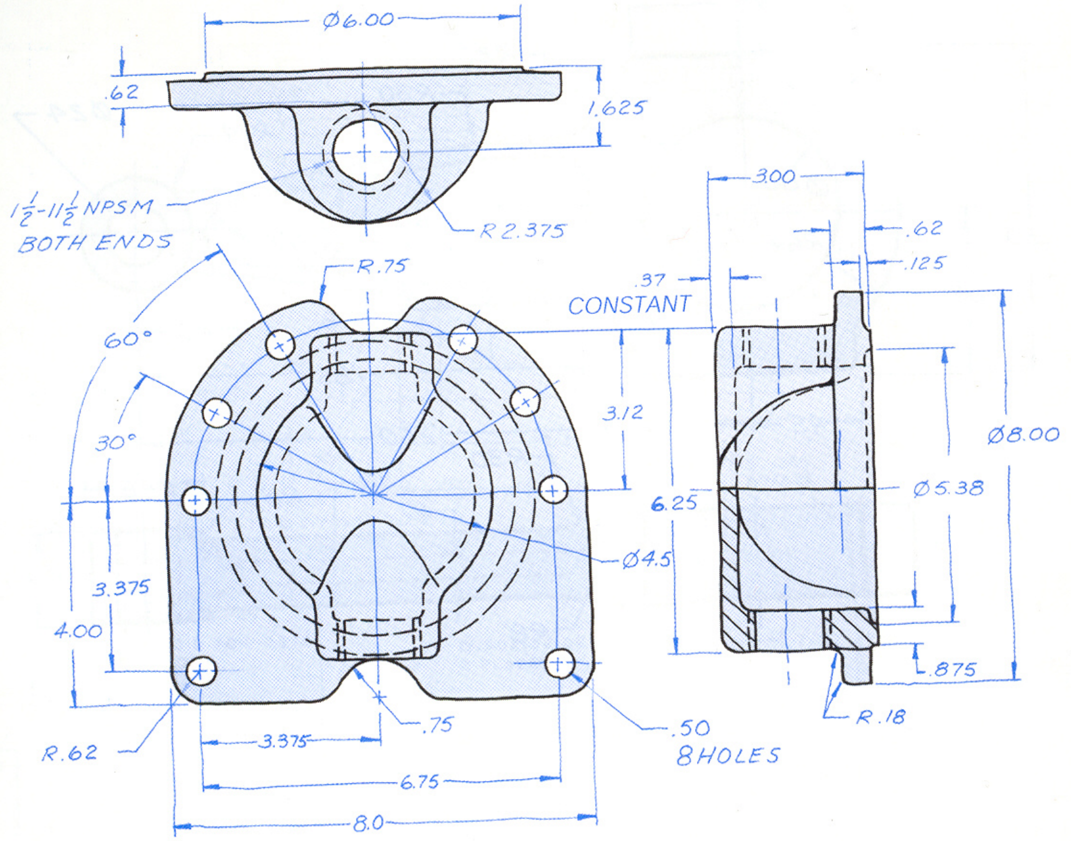
PROBLEM 10.14



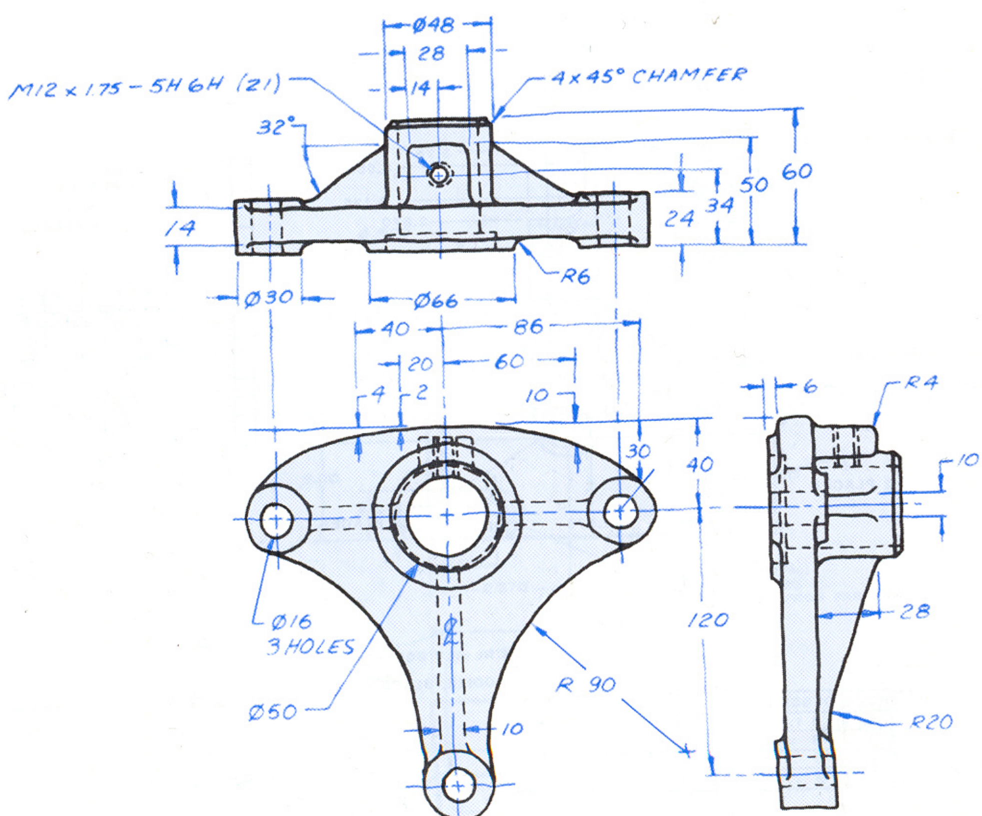
PROBLEM 10.21



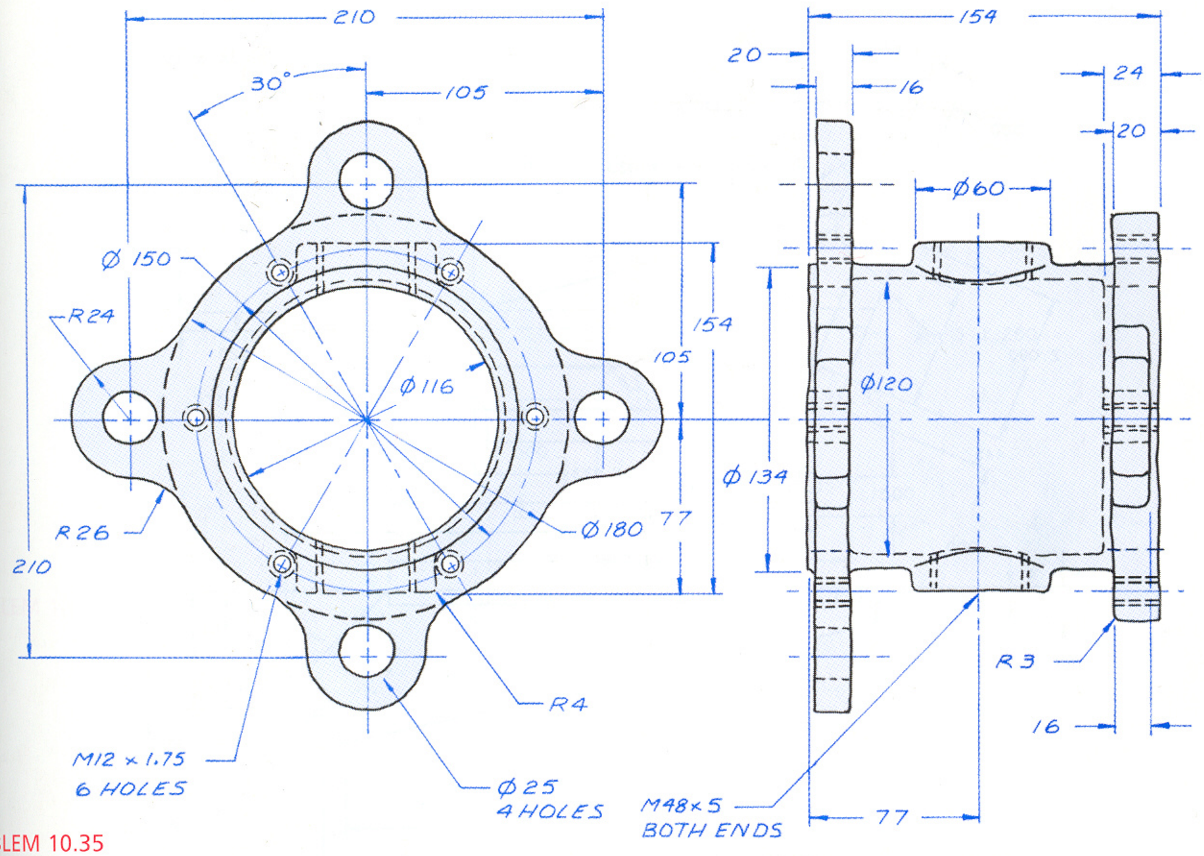
PROBLEM 10.22



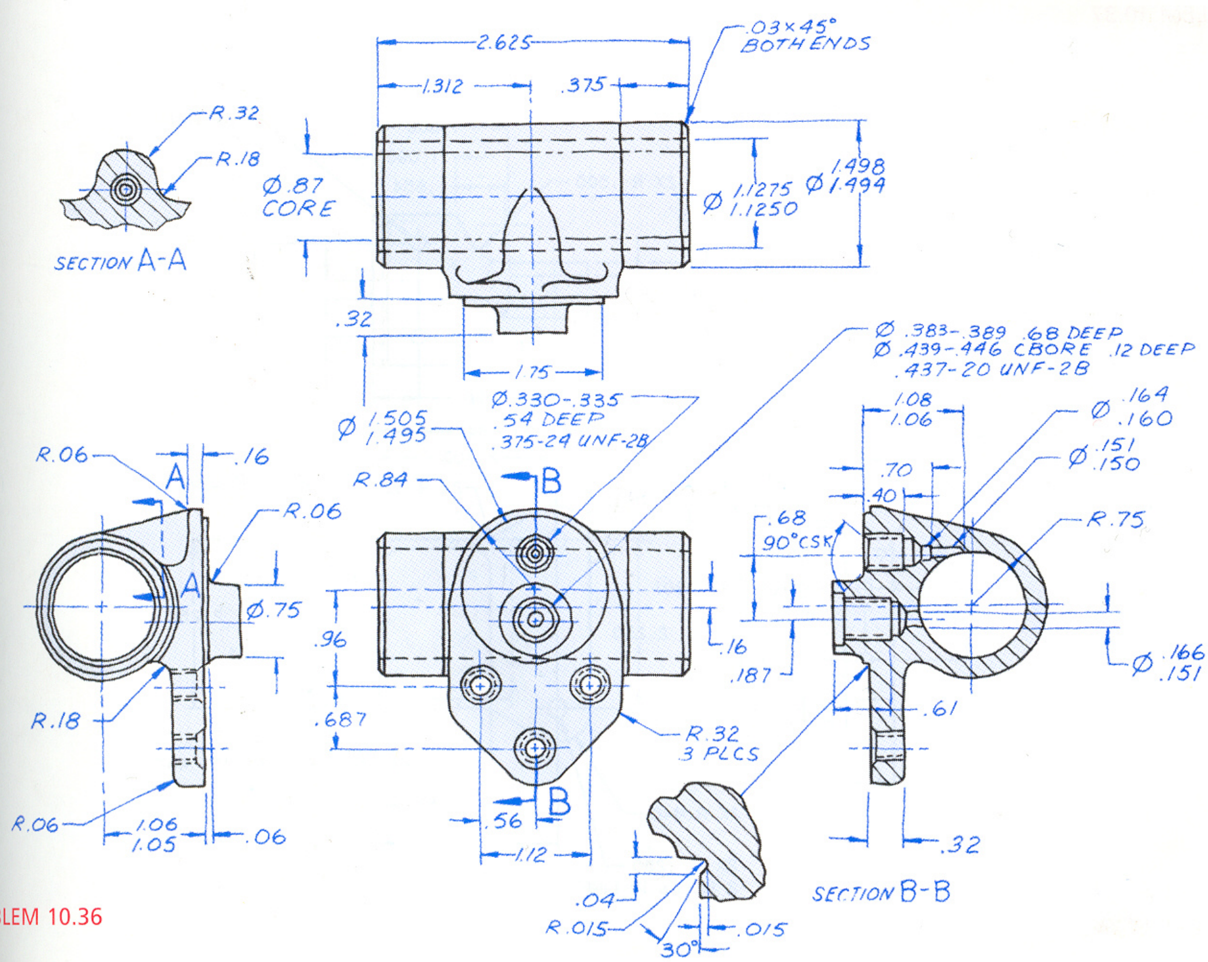

PROBLEM 10.33



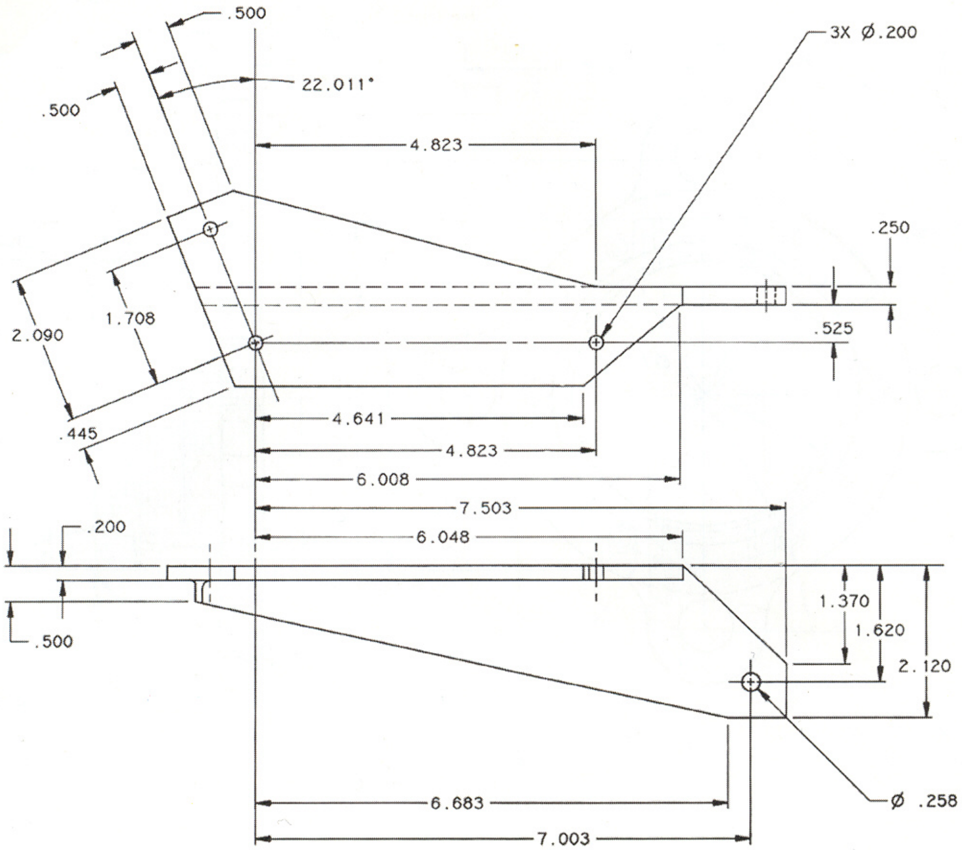

PROBLEM 10.34




PROBLEM 10.35




PROBLEM 10.36

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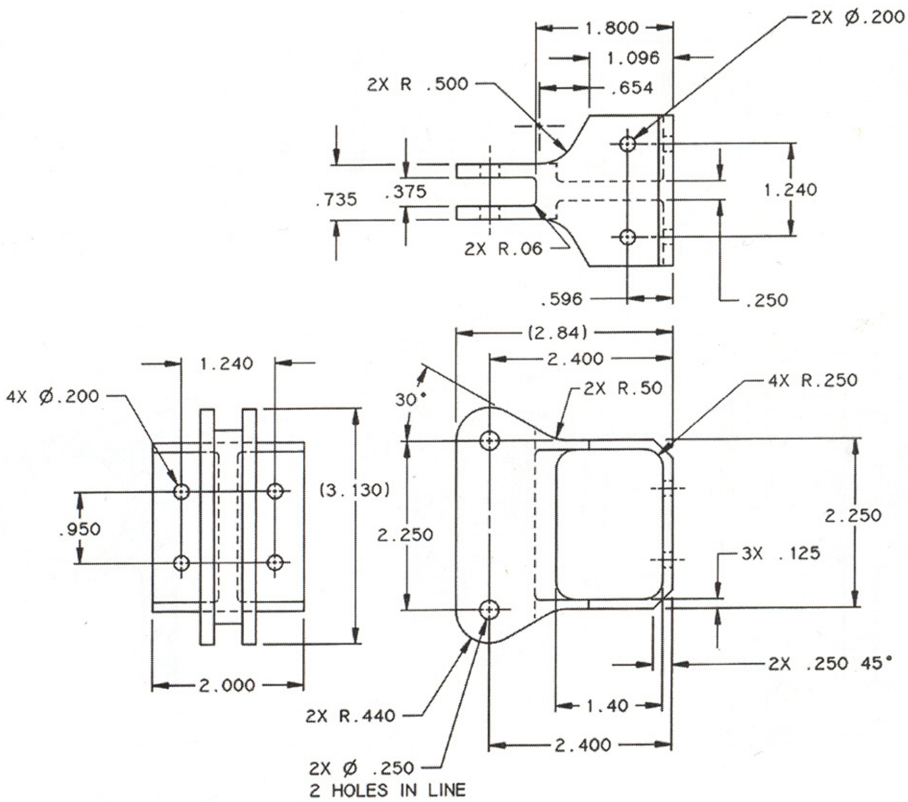
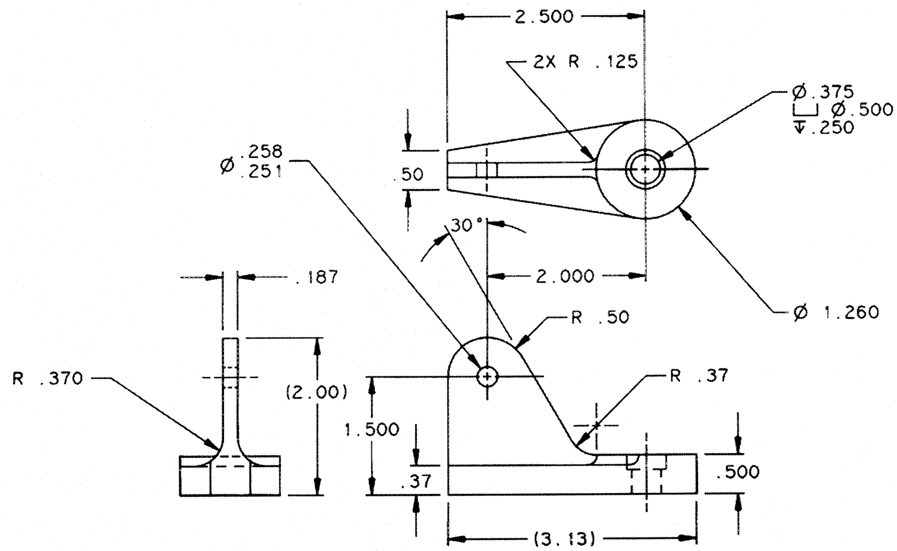



PROBLEM 10.38




PROBLEM 10.39