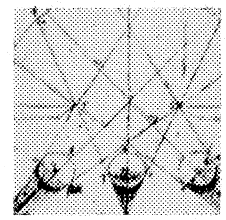
SKETCHING

Chapter 9



LEARNING OBJECTIVES

Upon completion of this chapter you will be able to:

1. Understand how sketches are used to transform design concepts into visual communication.
2. Recognize the importance of proper proportioning and thorough dimensioning.
3. Develop skills and techniques required to sketch engineering designs efficiently.
4. Demonstrate an understanding of multiview projection and selection of views.
5. Produce isometric, oblique, and multiview engineering sketches.

9.1 INTRODUCTION

Because sketching is one of the primary ways to communicate graphic ideas in the engineering community, the ability to sketch is essential and useful for all designers and engineers. A **sketch** is often employed to convey original design ideas. For example, it is not unusual to see members of a design team making freehand sketches to clarify the design of three-dimensional parts or to explore alternative configurations for an assembly. Figure 9.1 shows a manufacturing sketch depicting the process stages involved in machining and welding a neck cylinder. The design sketch of an assembly normally shows the general configuration of the part or assembly along with callouts, notes, and basic overall and setup dimensions.

Engineers and designers sketch preliminary ideas. Drafters or layout designers then refine those original ideas and requirements and produce drawings. There are three primary types of sketches: pictorial, multiview, and diagrammatic.

Sketching is usually done at any location that has a flat surface. The materials and tools used most often for sketching are paper (preferably grid paper), soft lead pencils, and an eraser. The grid on grid paper speeds the construction of any sketch. Sometimes it is necessary to evolve a layout through a series of sketches. Figure 9.2 shows the evolution of such a layout through the following stages: preliminary (rough) sketch (a), refined sketch (b), and the final CAD drawing (c).

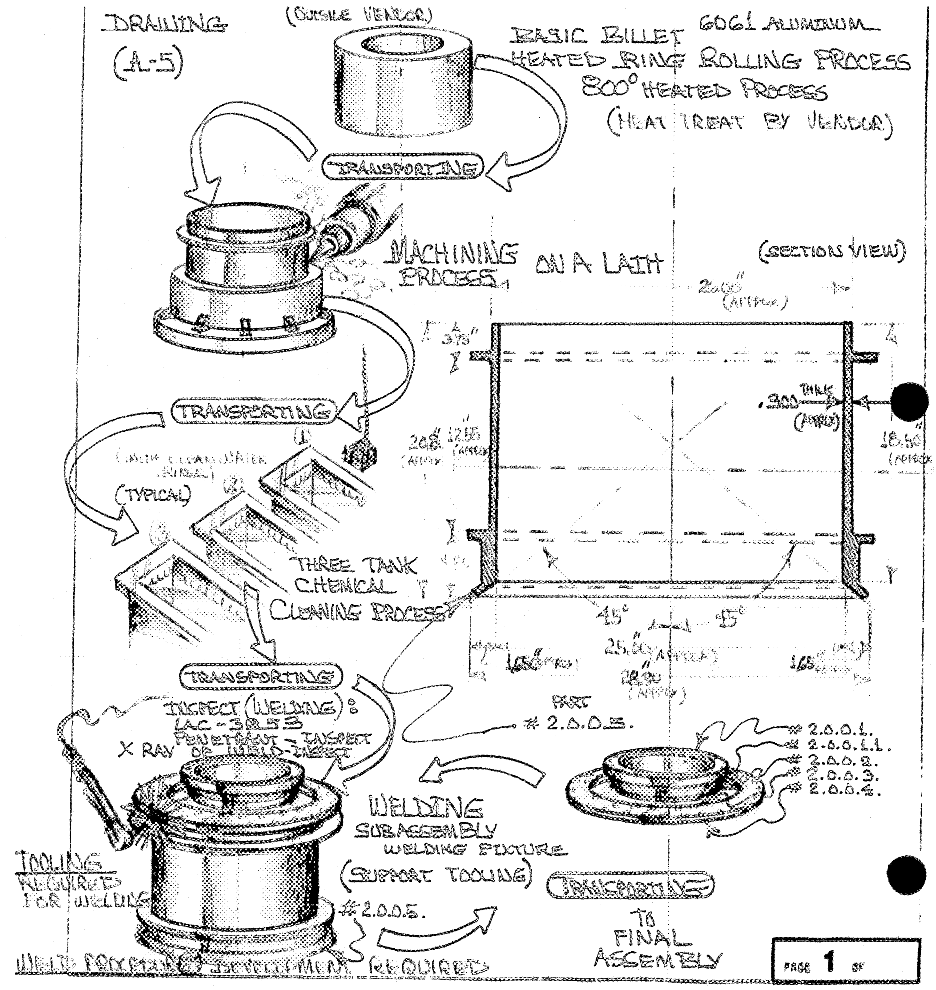


FIGURE 9.1 Sketch of Manufacturing Sequence

9.2 MATERIALS AND EQUIPMENT USED IN SKETCHING

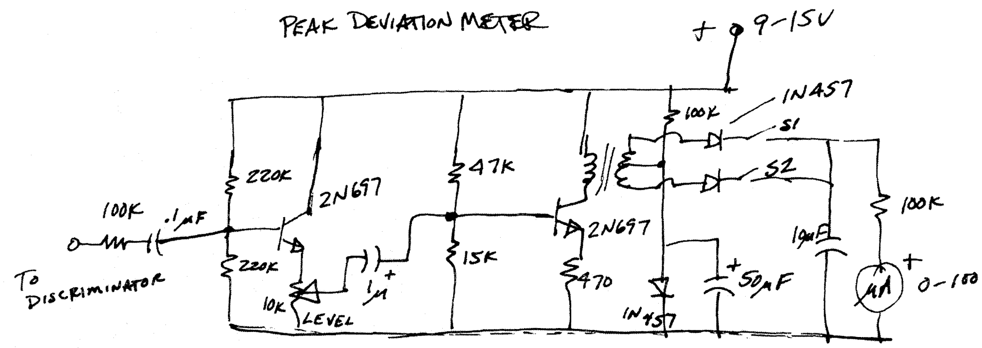
Only a few basic items and materials are required for sketching. Any sketch pad should be grid lined or have a crosshatched underlay grid sheet that can be placed beneath the transparent sketch paper. Nonreproducing grid sheets are also available. Isometric grid-lined paper is available for pictorial sketching. Posterboard grid formats include isometric, oblique, orthographic, and a variety of perspective formats. Figure 9.3 illustrates a simple part sketched on isometric grid paper. This grid paper has, in addition to the 30° receding lines in both directions, vertical and horizontal lines for multiview projection. Grid squares are very useful in maintaining the proper proportion for a part because it is easy to count grid squares while sketching.

9.2.1 Drawing Size When Sketching

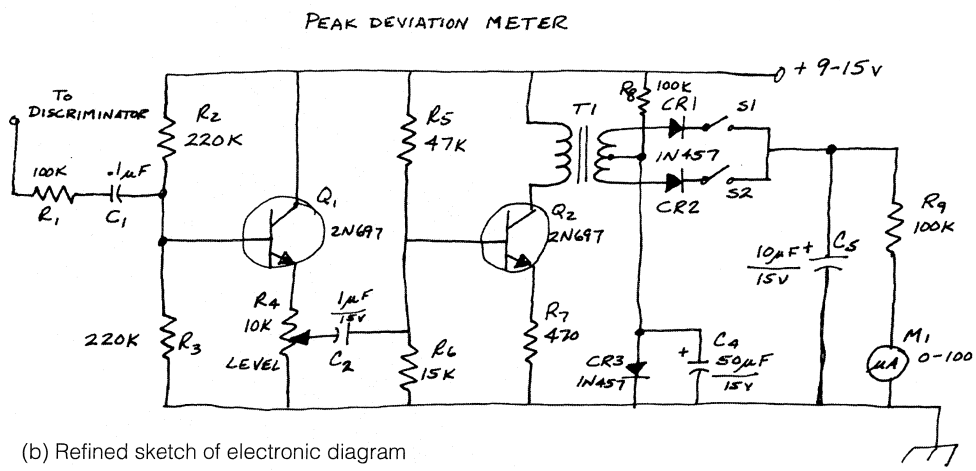
While the size of a particular sketch is usually unimportant, conveying the proper proportions of the part in a sketch is essential. Using grid paper helps to ensure that the sketch is in the proper proportion. Sketches are seldom drawn full size. However, sketches, like all other drawings, must be dimensioned. *Drawings of every type should not be measured, but "read."* **"Reading"** a drawing means you should be able to find every dimension for every feature of the part and use that information in later stages of the project. The one exception to the dimensioning rule is when the drawing is a diagram (e.g., as in Fig. 9.2). Diagrams tell a story and do not represent parts or objects. Two-dimensional diagrams can be measured and digitized when using a CAD system.

9.2.2 Line Types Used in Sketching

The line types and widths used in freehand sketches are the same as those used in instrument drawing (ANSI-standard line weights, types, and symbols). The line quality in a sketch



(a) Preliminary sketch of electronic diagram



(b) Refined sketch of electronic diagram

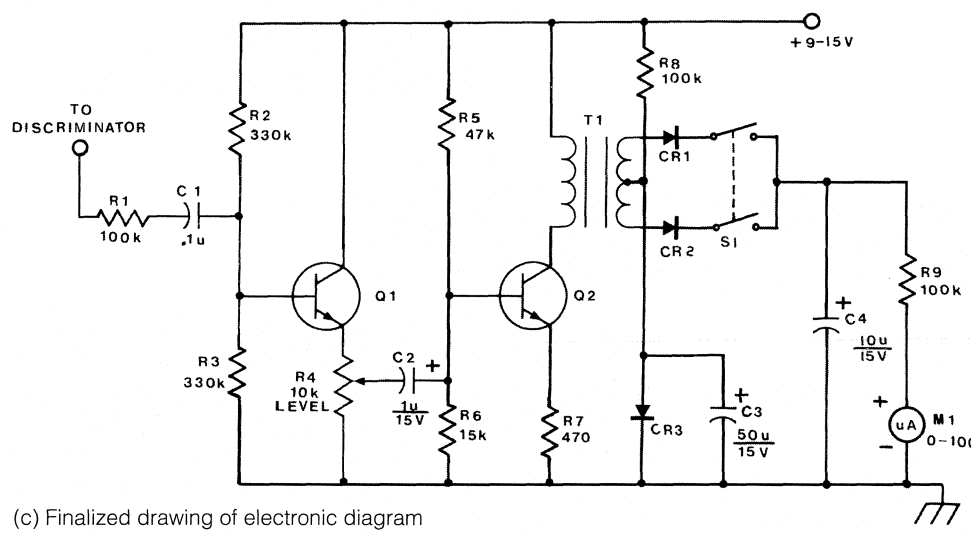


FIGURE 9.2 Sketching Steps (c) Finalized drawing of electronic diagram

is not perfect. Figure 9.4 shows the typical range of line types that may be encountered in sketches. Cutting-plane lines are the widest; object and hidden lines are medium thickness; extension, dimension, centerline, phantom, and section lines are thin lines. Lines should be equally black, as in instrument drawing. Construction lines are usually not removed. Lettering must be clear and easy to read.

Sketching skills are developed over a period of time through practice and effort, and should be cultivated throughout your career. Speed in sketching is not important while you are learning the basic techniques. However, later on, sketching with ease and speed will enhance your ability to communicate graphic ideas efficiently.

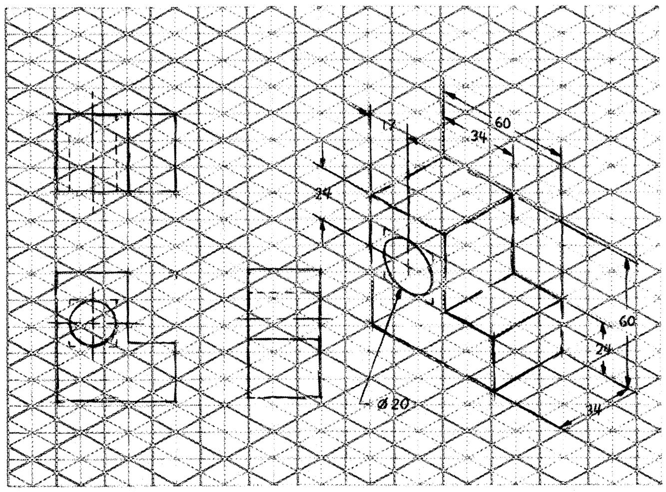


FIGURE 9.3 Grid Paper and Sketching

9.3 SKETCHING TECHNIQUES

When sketching, *hold your pencil at an angle to the paper.* As shown in Figure 9.5, 50° to 60° is recommended for straight lines, and 30° to 45° for circles and arcs. *Rotate your pencil while sketching,* to help maintain a conical point and reduce the time required for sharpening. You may prefer to sketch with fine-line mechanical pencils (0.7 to 0.9 mm, with H or HB lead).

Hold the pencil 1.5 to 2 in. (30 to 50 mm) from the tip, as shown in Figure 9.6. Some drafters prefer to hold the pencil in flat position, as demonstrated in Figure 9.7. Remember, it is not the intent of this text to change the way you hold your pencil. The information in this section is provided to help develop sketching skills. Left-handers may hold their pencils at different angles and orientations.

9.3.1 Sketching Horizontal Lines and Vertical Lines

Horizontal lines are drawn by locating the endpoints and connecting them with a line. Draw lines using construction lines first and, later, after the design is close to completion, go back and darken them. The pencil is moved from left to right (Fig. 9.8). Use short strokes, but try to avoid "feathering" the lines. Pull a wood pencil or lead holder to avoid ripping the paper surface. The lead in fine-line mechanical pencils breaks easily, so you should push them. Some designers leave a small space (gap) between each line segment (the space is unnecessary with grid paper).

Vertical lines are drawn with the same general technique. For vertical lines, move the pencil from the top toward the

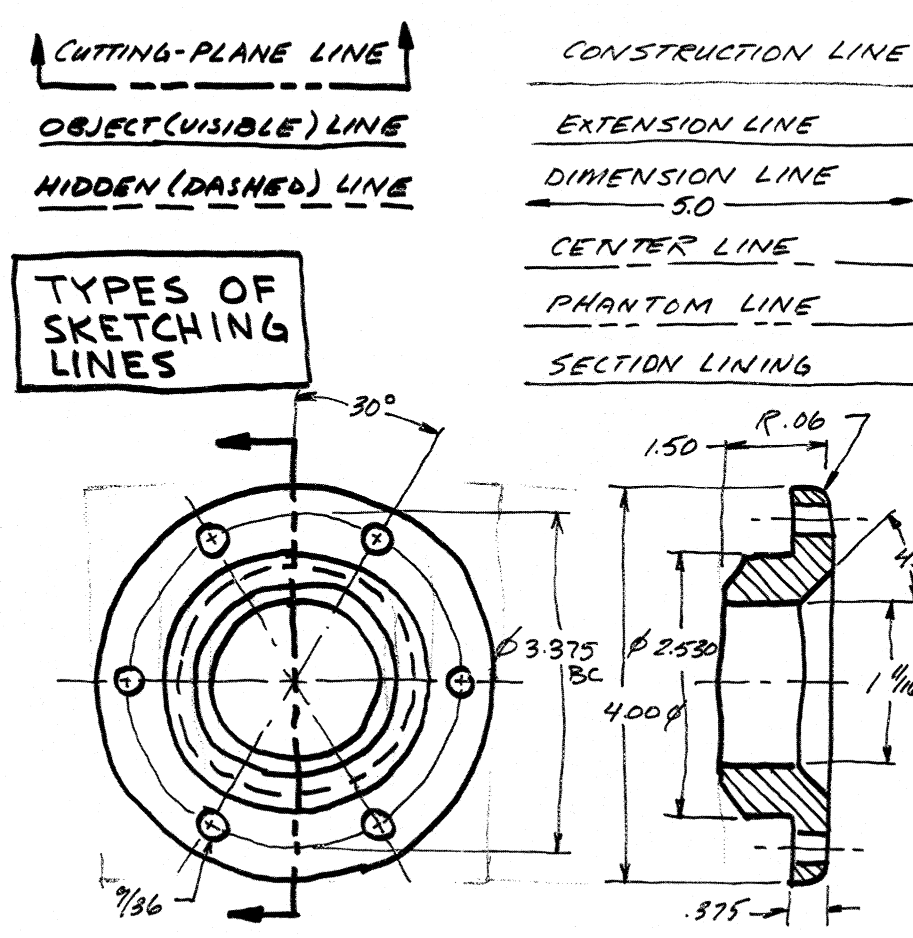
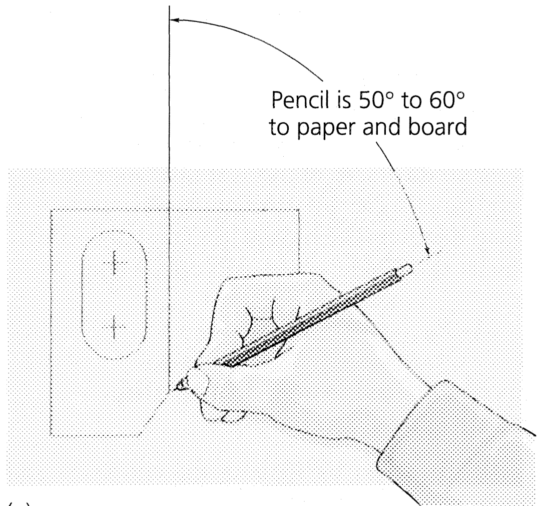
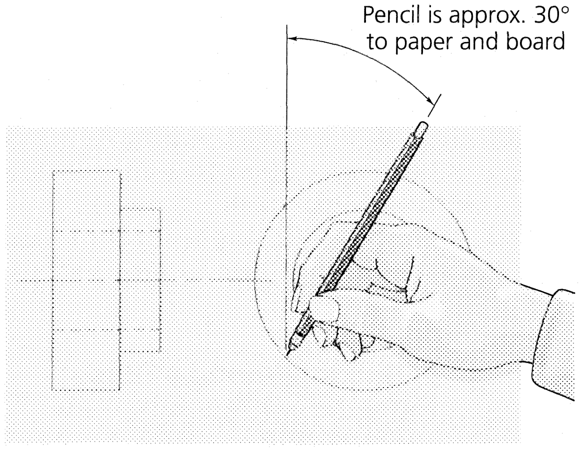


FIGURE 9.4 Line Types for Sketches



(a)



(b)

FIGURE 9.5 Angle of Pencil When Sketching

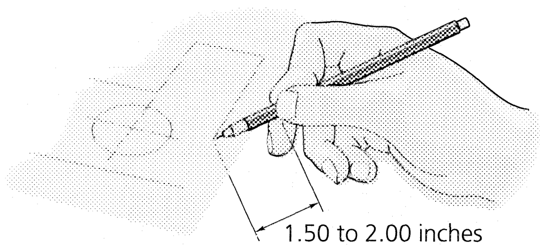


FIGURE 9.6 Holding a Pencil for Sketching

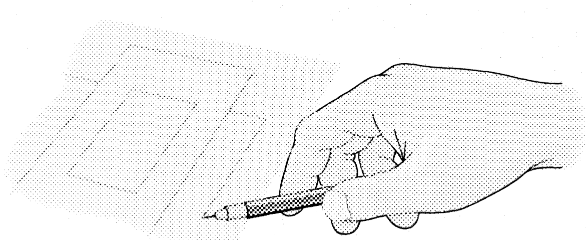


FIGURE 9.7 Flat Pencil Position for Sketching

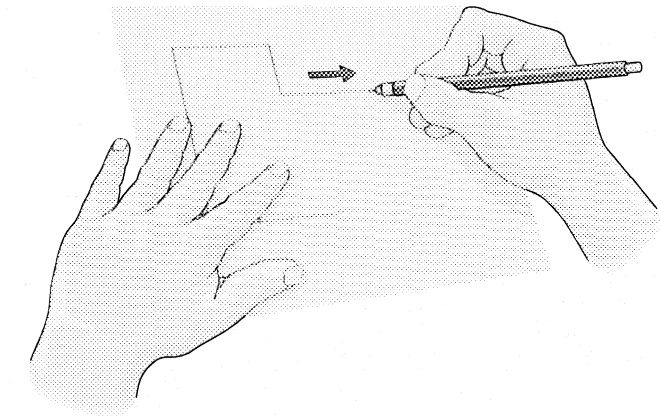


FIGURE 9.8 Sketching Horizontal Lines

bottom of the paper (Fig. 9.9). Again, grid paper helps ensure that the drawn lines will be vertical. Turn the paper to any convenient position to help speed the process. Some designers prefer to move the pencil away from the body from bottom to top or left to right. Try different methods to find the one that works best for you.

9.3.2 Sketching Inclined Lines

Angled lines are drawn by establishing the endpoints, lightly sketching the line, and finally darkening the line. Sketch inclined lines away from you if they are angled to the right (Fig. 9.10) or toward you (or turn the paper) if they are

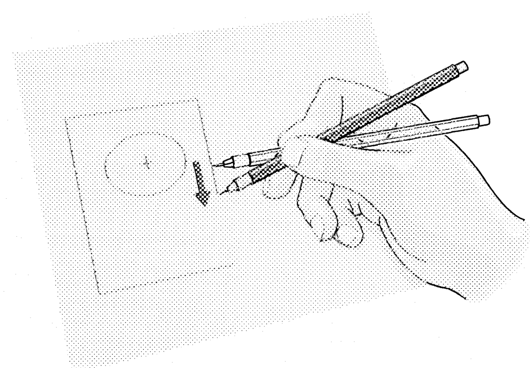


FIGURE 9.9 Sketching Vertical Lines

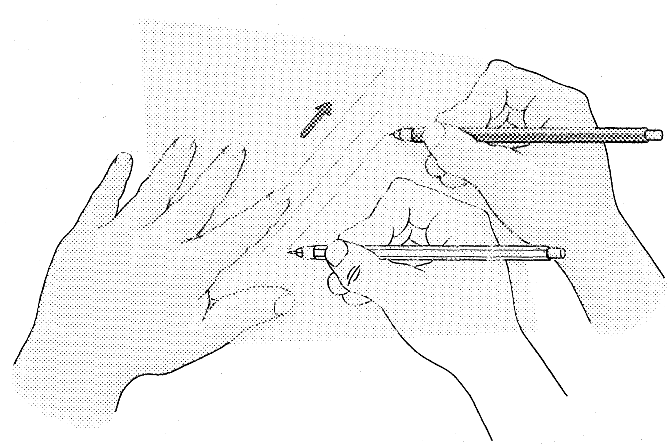


FIGURE 9.10 Sketching Inclined Lines

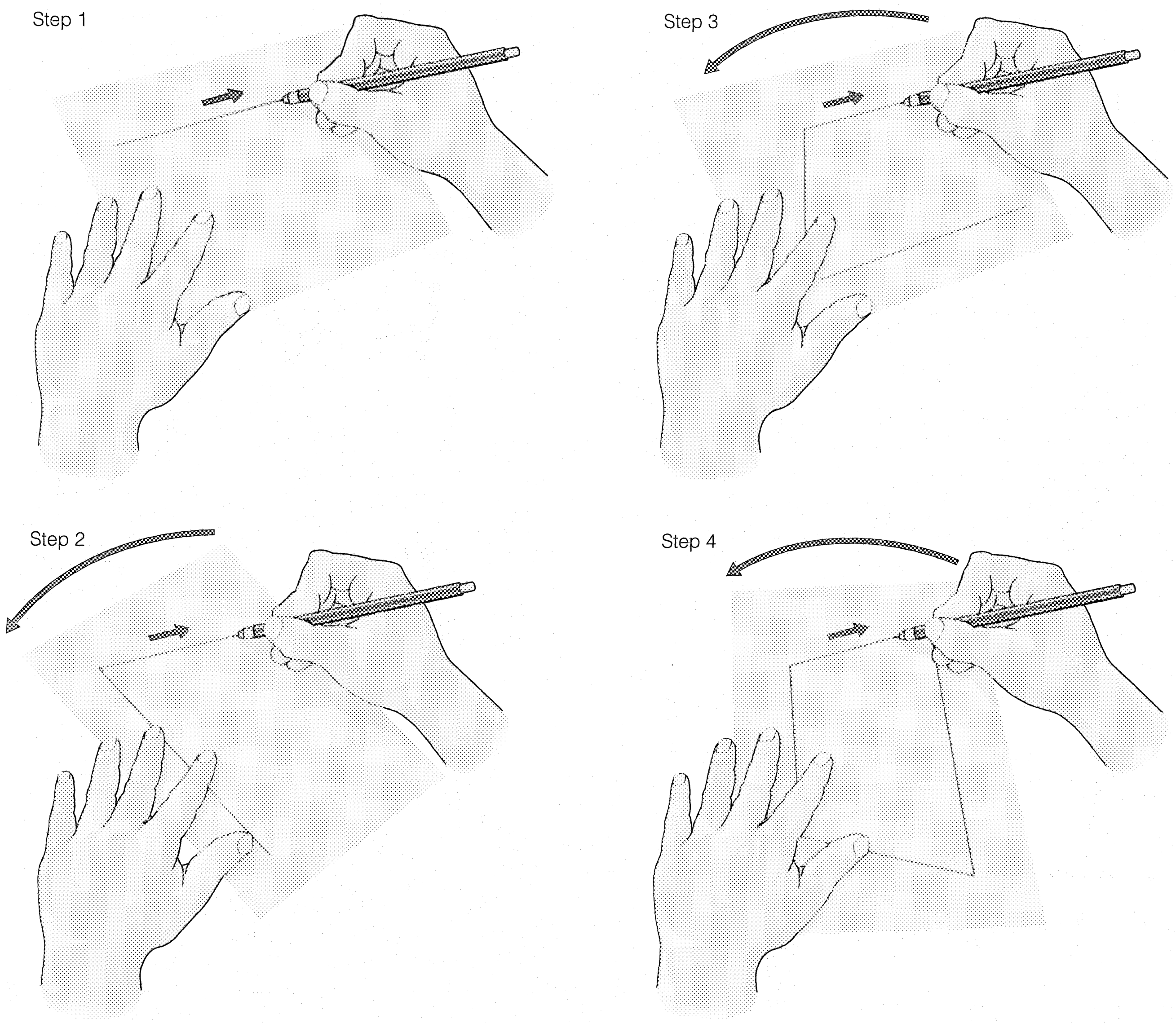


FIGURE 9.11 Sketching by Turning Paper Counterclockwise

angled to the left. Use the opposite technique if you are left-handed.

Since horizontal lines are the easiest to draw, turn the paper so the line you are sketching is close to horizontal (Fig. 9.11). However, large sketches are often taped to the table, so you should also learn to sketch without turning the paper. If your sketch is small or it is attached to a sketch pad or clipboard, you can turn the sketch at any convenient angle, as shown in Figure 9.12.

Angles can be estimated by drawing two lines perpendicular (90°) to one another. Bisecting this angle gives a 45° measurement. Similarly, dividing the 45° angle into three divisions provides a 15° angle and a 30° angle (Fig. 9.13). Always locate the endpoints of a line by dimensions.

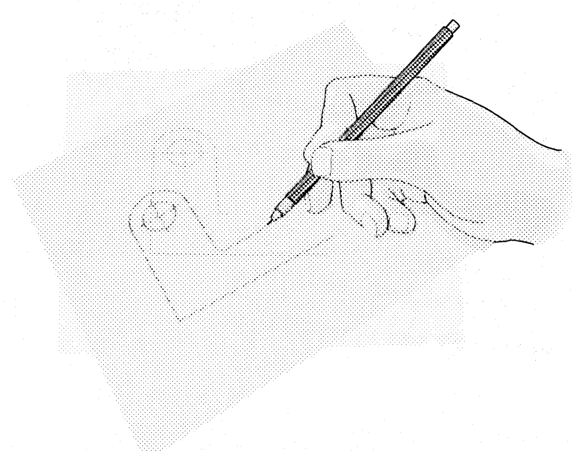


FIGURE 9.12 Turning Paper May Make Sketching Easier

Focus On . . .

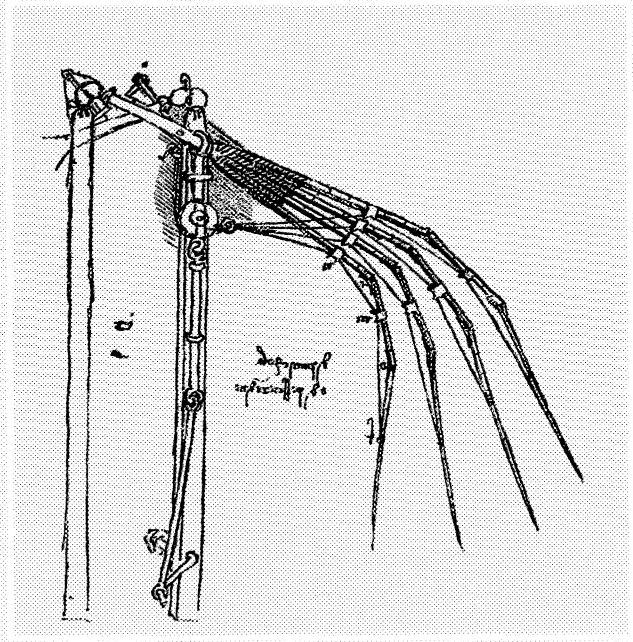
LEONARDO DA VINCI, "THE SKETCHER"

Sketches, illustrations, and technical drawings visually represent the designer's ideas so they may be understood by others. The thought required to sketch an idea and the discussion of ideas with others are good ways to refine proposed solutions to engineering problems.

Prehistoric people recorded their experiences by drawing on cave walls. These cave drawings showed hunting scenes and included people, animals, and tools such as spears and arrows. Who knows, they may have even believed these drawings had the power to make events come true.

A freehand sketch has always been a fast and easy way to put on paper ideas formulated in the mind. Leonardo da Vinci sketched hundreds of plans for his inventions. Today, manufactured parts often begin with a freehand sketch.

You probably think of Leonardo da Vinci as one of the greatest painters of the Italian Renaissance. It is true that he was trained to be a painter and he did produce some of the world's greatest paintings, including the *Mona Lisa*. He also designed machines that were far ahead of his time, such as a flying machine and a parachute. He became one of the most versatile geniuses in history because of his achievements, including scientific inventions.



A mechanical sketch of a flying mechanism by Leonardo da Vinci.

Leonardo's backward notes.

In approximately 1482, Leonardo went to Milan to be the court artist to the Duke of Milan. One of his duties was as a military engineer. He designed artillery and the diversion of rivers. He also designed sets for court pageants. When he was forced to leave that post because of the French invasion, he returned to Florence to serve as a military engineer to that court. During this time, he traveled throughout central Italy preparing sketches for maps that would become important to the history of cartography. Although he never did construct a building, he was held in the highest esteem as an architect. He drew plans ranging from the dome of the Milan cathedral to an enormous bridge over the Bosporus.

During his later years, Leonardo did little painting; instead he produced many sketches of experimental machines and other inventions. These rank among his greatest masterpieces because of their sense of motion and his use of shade and shadows.

Leonardo recorded his ideas in several notebooks, many of which include sketches and drawings that reveal his skill as a drafter and designer. About 4200 pages of his notebooks still exist. However, should you decide to read them, be sure to bring a mirror. Leonardo wrote his notes backwards!

If all engineers, designers, and drafters recorded their ideas in a similar diligent and elegant fashion, we too might be well known for our graphic communications skills. Leonardo showed us all the value of a sketch or two. His, of course, were also masterpieces.

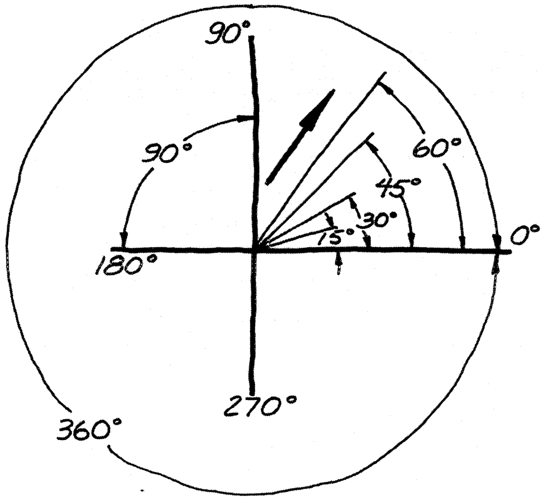


FIGURE 9.13 Typical Angles Used in Sketching

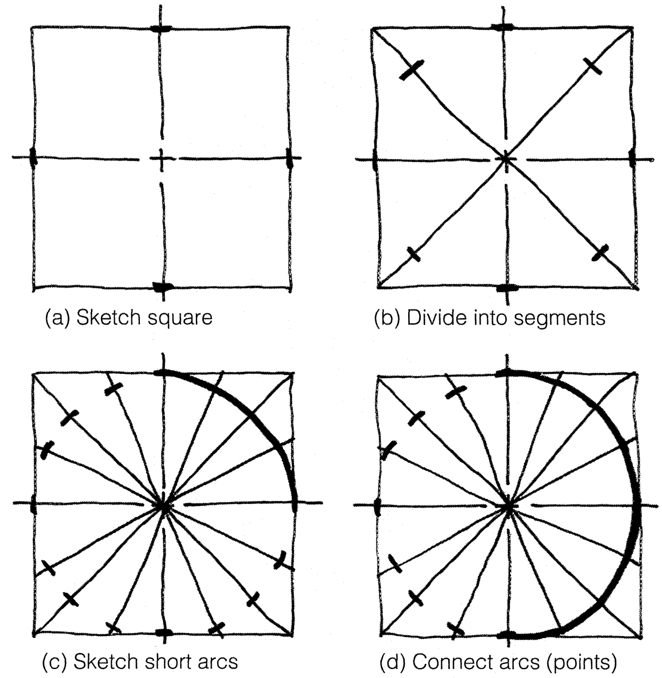


FIGURE 9.14 Sketching a Circle

9.3.3 Sketching Arcs and Circles

Learning to sketch arcs and circles can be frustrating. Always start by locating the center point of the circle or arc, then draw the centerlines of the circle. Measure or estimate the size of the circle, and lay out the diameter along the centerlines, as shown in Figure 9.14. Block out the circle by drawing a square that encompasses it [Fig. 9.14(a)]. Next, draw diagonals and lay out the diameter on the diagonals [Fig. 9.14(b)]. If the circle is large, divide the circle into smaller segments and measure the diameter [Fig. 9.14(c)]. Connect the points by sketching short arcs to complete the circle [Fig. 9.14(d)]. If the sketch is small, rotating the paper helps keep the circle round (Fig. 9.15).

Use the same general technique to sketch arcs. In Figure 9.16, several arcs and circles were required. Centerlines were used for every arc and circle, and all circles and arcs were blocked before they were drawn.

Freehand sketching of irregular curves involves establishing an adequate number of points along the curve and then connecting the points with a smooth curve. A lightly sketched construction curve is drawn first; then the irregular curve is darkened. Grid paper makes it easier to establish the control line points (Fig. 9.17).

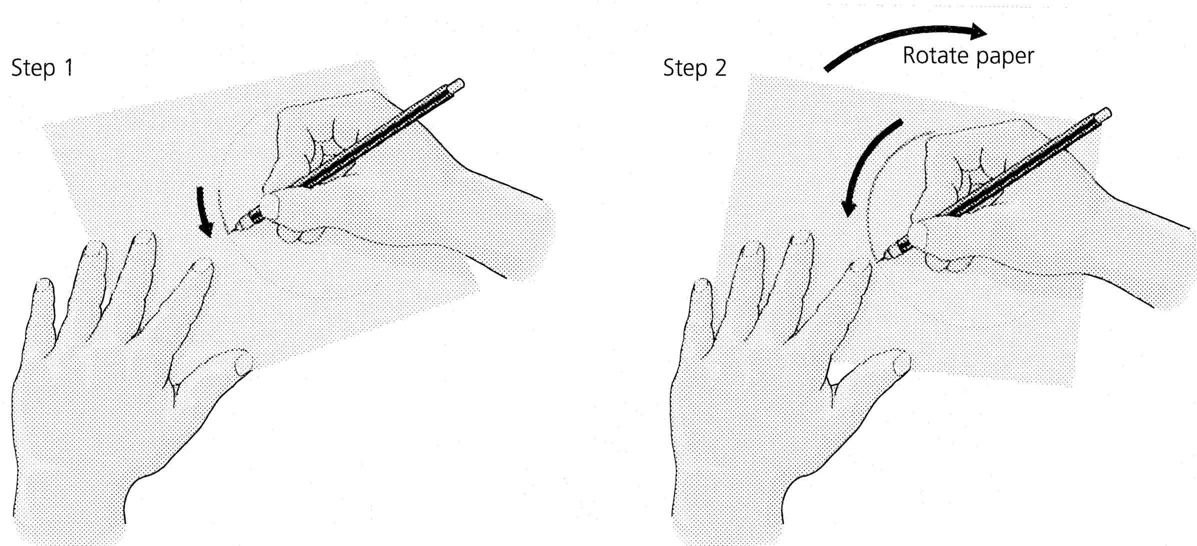


FIGURE 9.15 Sketching Circles by Rotating the Paper

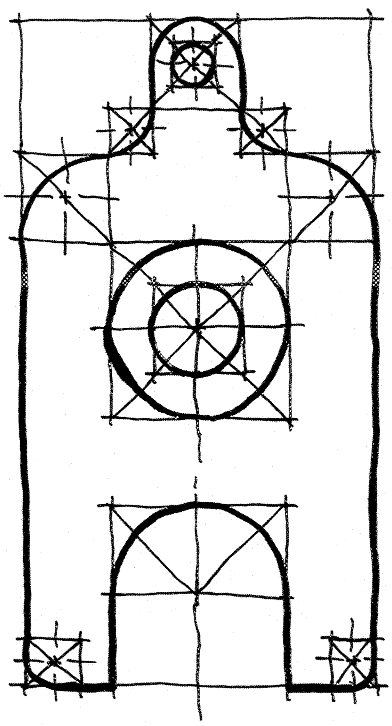


FIGURE 9.16 Blocking Out Circular Shapes When Sketching

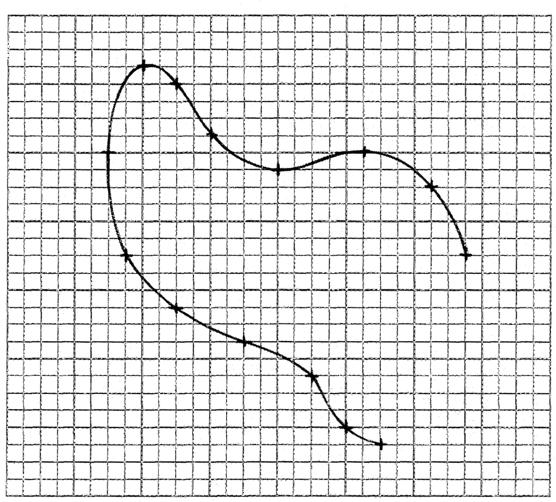


FIGURE 9.17 Sketching Irregular Curves

9.4 INTRODUCTION TO PROJECTION TECHNIQUES

Technical operations usually require 2D (paper) representations to communicate ideas and give physical descriptions of 3D shapes. These projections are divided into two categories, *pictorial* and *multiview.* **Pictorials** simulate 3D views of the part, while **multiviews** are 2D projections of the part. This simple division separates single-view drawings (pictorials—oblique, isometric, and perspective) from multiview drawings.

Chapter 10 covers multiview drawing in great depth. Chapter 13 provides in-depth coverage of all types of pictorial projection methods. This chapter introduces the types of projection associated with freehand sketching and how a sketch is used in industry.

Often, engineering working drawings are multiviews, while pictorials are used for technical illustrations. In sketching, however, both types may help refine design concepts. Figure 9.18 shows each of the four projection types for an angle block. Pictorial projections are single-view drawings that may serve as rough sketches of preliminary ideas, but they do not always lend themselves to communicating exact technical details. *Perspective* projections are constructed with projecting lines that converge at a point. Although this method provides the most lifelike appearance of the part, it does not show true dimensions. *Oblique* pictorials distort the depth of the part. The *isometric* method, which uses full-scale dimensions for all lines that are vertical or parallel to the axes, is the most common and useful method for engineering sketching.

Multiview drawings are not lifelike, because they show the parts in more than one view and are projections. Multiview projection presents the object's top, front, and sides in related adjacent views. The theory behind orthographic projection is that the object is rotated by turning it to the appropriate view. For example, rotating it 90° sideways provides a side view. In Figure 9.19, the part was rotated to the right, so the resulting view is a right side view. The three-view drawing (bottom) shows the part aligned between views. The three principal views (top, front, and side) can be used to project any number of needed views to provide engineering data. An *auxiliary* view is any projection

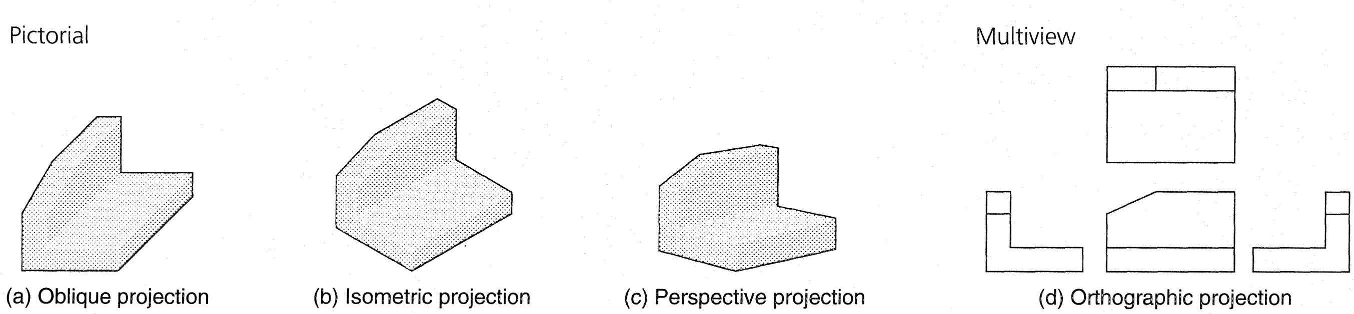


FIGURE 9.18 Projection Methods

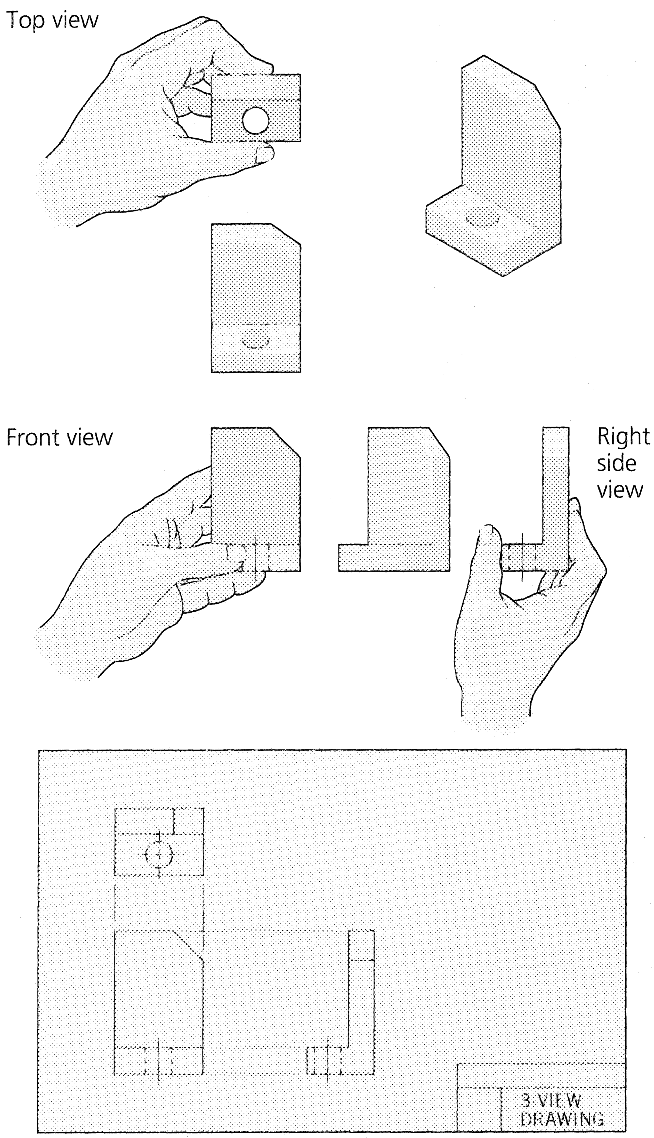


FIGURE 9.19 Three-View Orthographic Projection

other than one of the six principal views (top, front, right side, left side, back, and bottom).

In some cases, the combination of pictorial and multiview sketches define the part or assembly better than would just one method. In Figure 9.20 the sketch of the manufacturing processes required to produce a tank are described with a pictorial isometric sketch and a cutaway section view of the tank's interior.

9.5 MULTIVIEW PROJECTION

Multiview projection describes a part's features and dimensions in one or more views that are projected at 90° angles to each other (Fig. 9.21). This is the primary projection method in engineering work. Figure 9.22 shows a multiview sketch that communicates ideas, dimensions, and shapes for the manufacture of a rocker arm. The front and right side views are shown in the two-view sketch.

Multiview drawing uses orthographic projection to establish the spatial relationship of points, lines, planes, or solid shapes. Two methods are involved: the *normal method* and the *glass box method*. In the normal (natural) method, the object is viewed perpendicular to each of its three primary surfaces.

9.5.1 The Glass Box and Hinge Lines

In the glass box method, you imagine that the part is enclosed in a transparent box. A view of the part is established on its corresponding **glass box** surface (plane) by perpendicular projectors originating at each point on the object and extending to the box surface [Fig. 9.23(a)]. The glass box is hinged so that it can be unfolded onto one flat plane (the paper) [Fig. 9.23(b)]. Each projection shares a dimension with its adjacent view. For example, the top and front views share the width dimension. In this method, all six sides are revolved outward so that they are in the plane of the paper. All except the back plane are hinged to the front plane. The back plane, when used, is normally revolved from the left side view. Each plane is parallel to the plane opposite from it before it is revolved around its hinge line.

A **hinge line,** often referred to as a **folding line,** is the line of intersection between any two adjacent image planes (Fig. 9.24). The left side, front, right side, and back are all elevation views and show the height dimension. The top and bottom surfaces are in the horizontal plane. The depth dimension, width dimension, front, and back are established there. Each image plane (surface of the glass box) is connected at right angles to an adjacent view. For example, the top view is hinged to the front view, as is the right side view. Hinge lines are not shown on technical drawings or sketches.

In the United States and Canada, the six principal views of a part are drawn through third-angle projection, in which the line of sight goes through the image plane to the object (Fig. 9.25). Assume that the object is projected back along the line of sight to the image plane. The line of sight is at a right angle to the projection plane and is assumed to originate at infinity. To visualize this, place the plane between you and the object. Your position changes with every view so that your line of sight is always at a right angle to each image plane. A point is projected on the image plane where its projector (line of sight) pierces that image plane. Point 1 in Figures 9.24 and 9.25 is located on the part and is projected onto the three primary image planes.

9.5.2 Selection of Views

Selecting the proper views and their orientations requires consideration of the actual part and its natural or assembled position. The front view customarily shows the primary features of the part in elevation. Selection of the top view is usually obvious. It is best to use the minimum number of

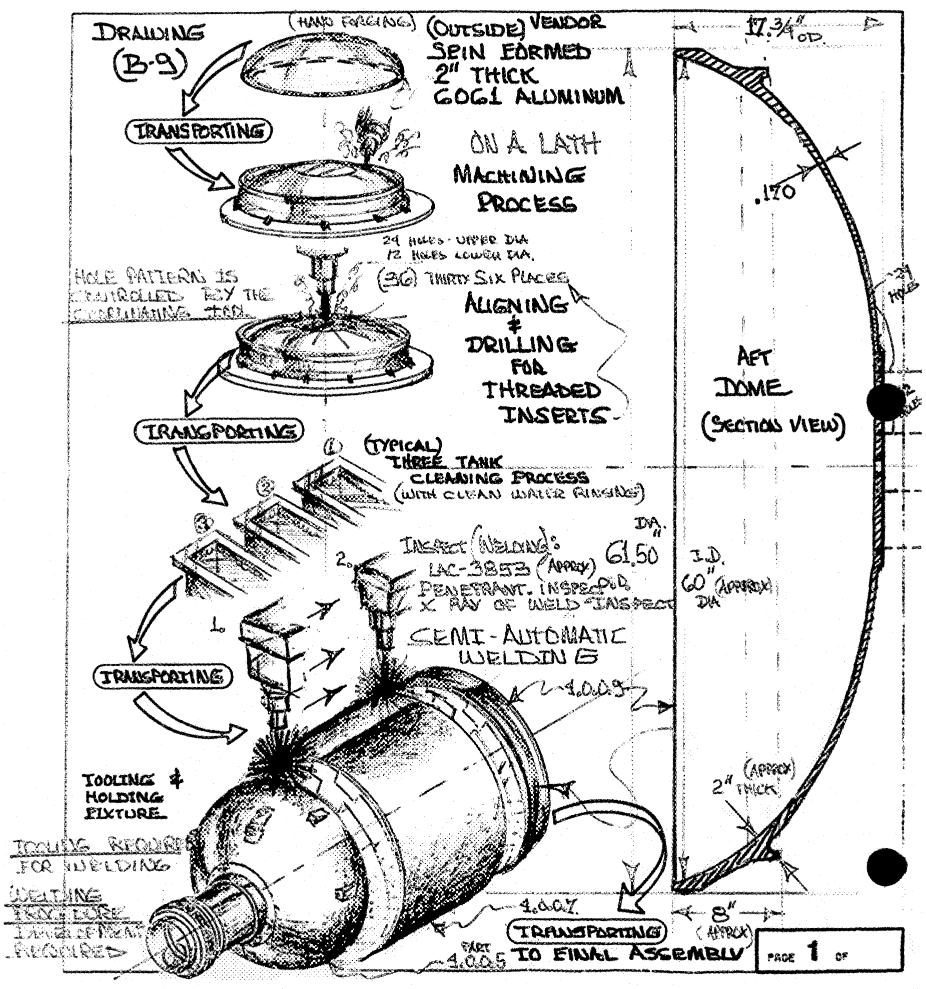


FIGURE 9.20 Manufacturing Sequence Sketch

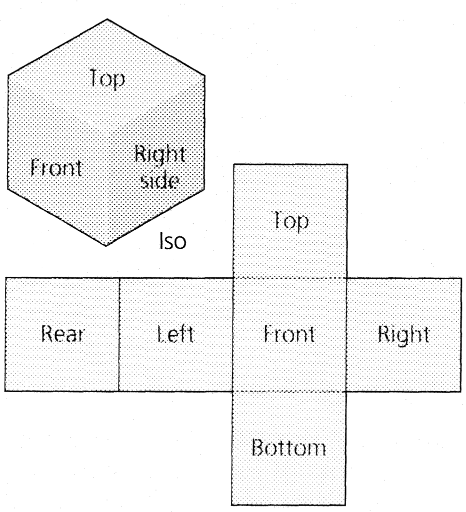


FIGURE 9.21 The Six Standard Views of an Object Plus an Isometric View

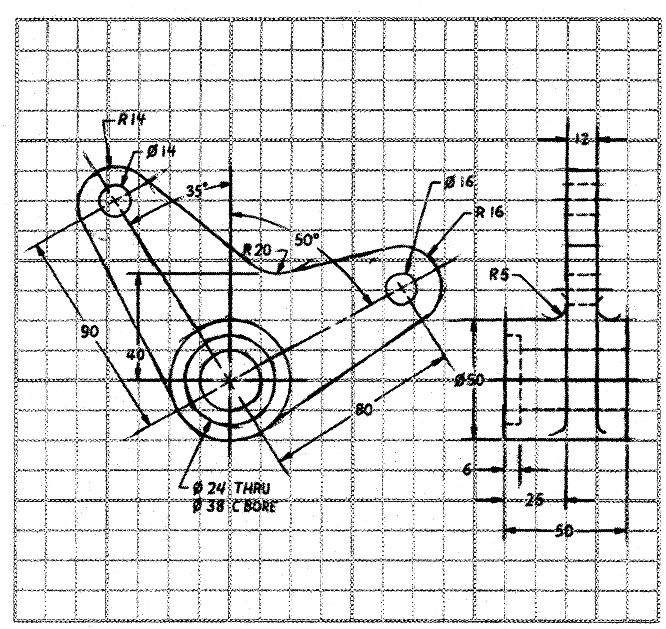


FIGURE 9.22 Two-View Drawing

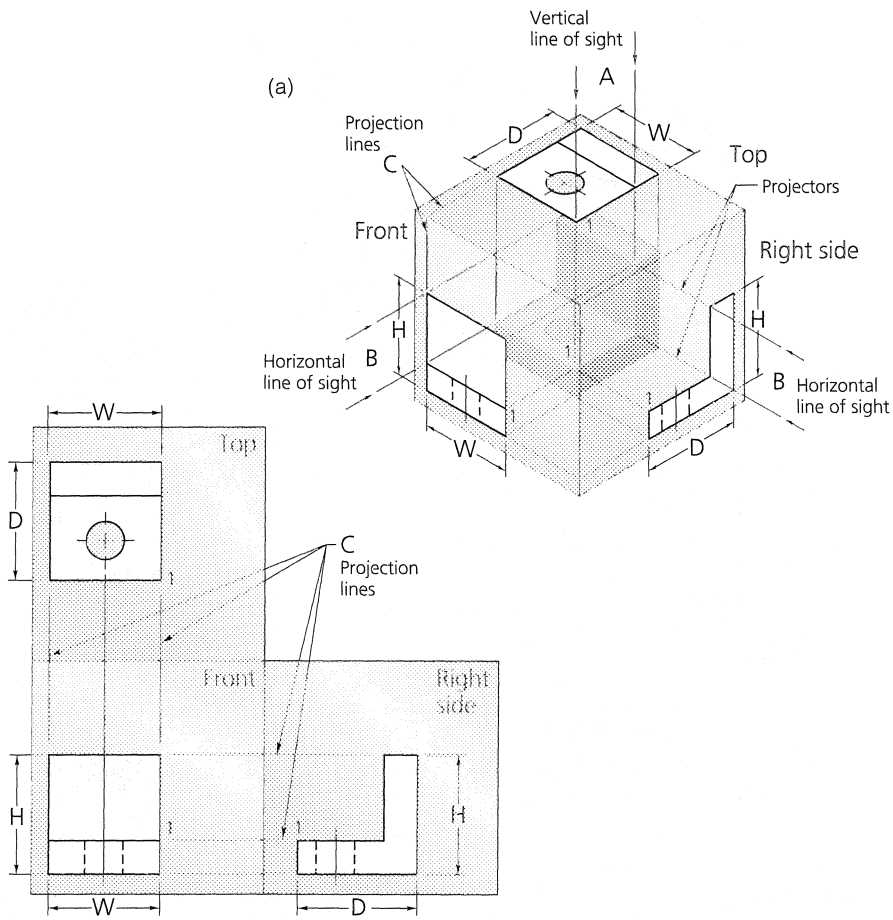


FIGURE 9.23 The Glass Box Method

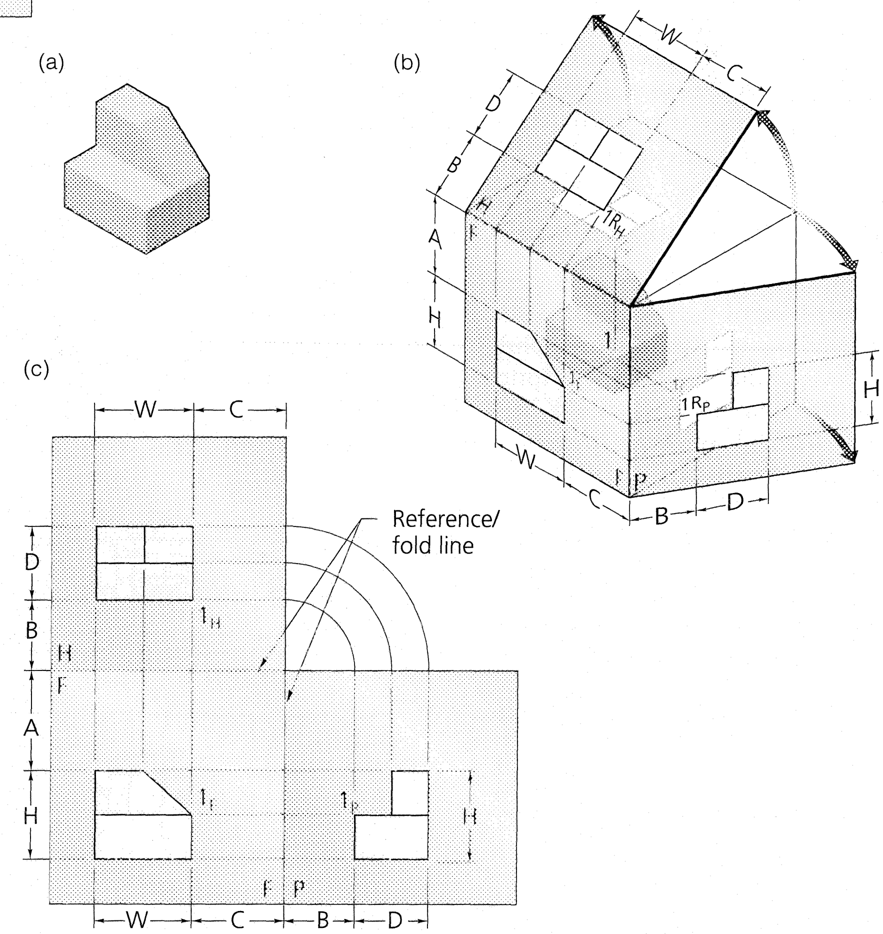


FIGURE 9.24 Unfolding the Glass Box

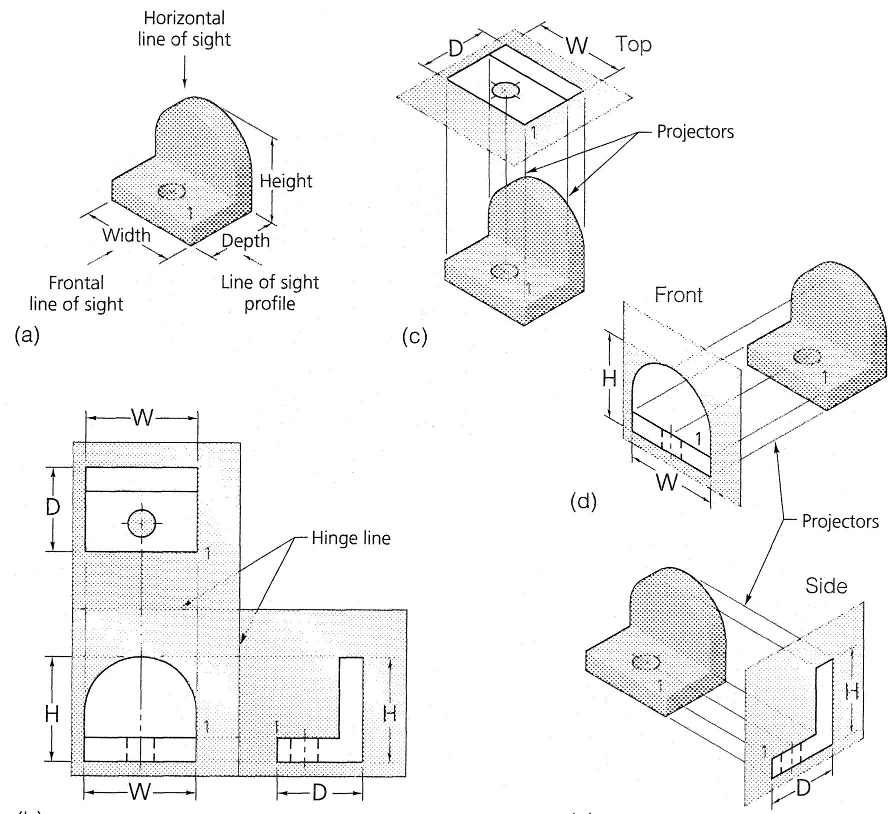


FIGURE 9.25 **Line of Sight for Views**

views necessary to describe the object completely. For example, only one or two views are needed for cylindrical parts because the diameter dimension will describe width and depth, and features along the length are described in the longitudinal view (Fig. 9.26). Engineering sketches generally require at least two views.

9.5.3 Multiview Sketching

Figure 9.26 shows the three stages of sketching. The overall dimensions of the part were blocked out first in each view. Centerlines were added to establish circular or symmetrical aspects of the part. Next, the spring coils were drawn with construction lines. Finally, the lines were darkened. Figure 9.27 shows these same steps applied to a two-view mechanical part.

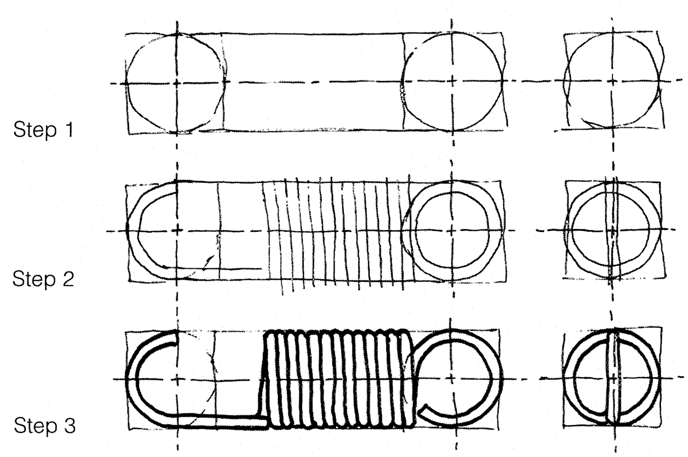
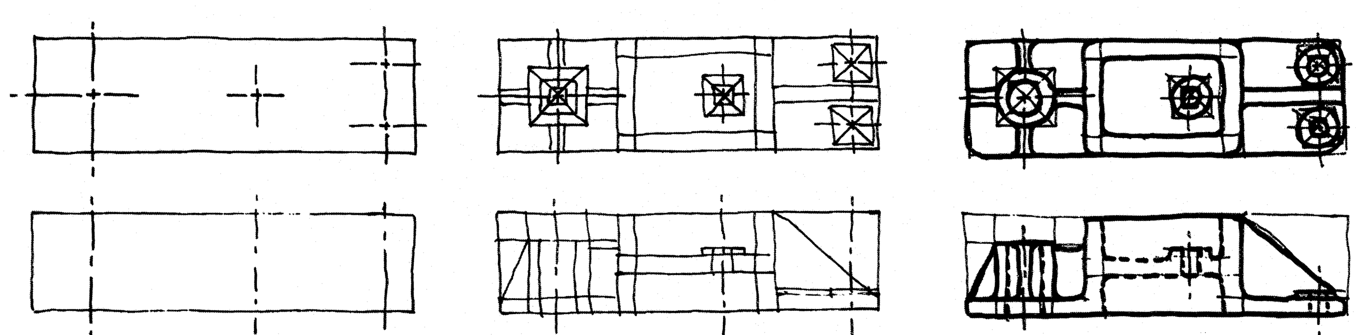


FIGURE 9.26 **Blocking Out a Part**



Step 1: Blocking out overall dimensions and centerlines

Step 2: Completing secondary features

Step 3: Darkening part features and centerlines

FIGURE 9.27 **Blocking Out a Two-View Sketch**

Figure 9.28 is a multiview sketch of a part that required all three views. Each view is "in line" with its adjacent view, as are all the features of the part. Adjacent views of edges, holes, and other shapes are established by projecting lines between the views. Construction lines are extended view to view. Since alignment of the views is critical in multiview sketching, grid paper makes the sketching process easier and faster.

Sketches are not complete without dimensions. In Figure 9.29 a three-view sketch is shown along with the completed mechanical detail. Both the sketch and the finished detail incorporate the dimensions that are required to manufacture the part accurately. In reality, either the detail or the sketch could have been used to manufacture the part with the same result.

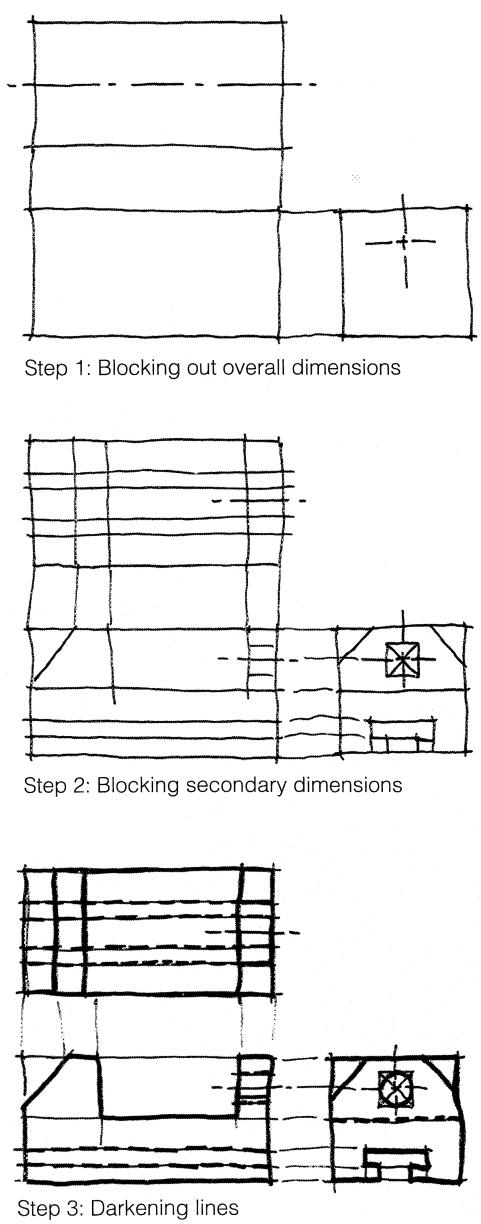


FIGURE 9.28 Blocking Out a Three-View Sketch

Because of the widespread use of computers in technical work, computer-aided design is now involved in many projects. However, sketching is and will continue to be the most effective and most popular way to communicate graphic ideas. Many companies now use a correctly dimensioned engineering sketch to speed the drafting stage of the design through the manufacturing cycle. This is called *simplified drafting* and has gained widespread acceptance in our highly competitive world.

You May Complete Exercises 9.1 Through 9.4 at This Time

9.6 PICTORIAL PROJECTION

Pictorial drawings are widely used for display illustrations and product literature. Isometric drawing is the most common pictorial technique. Pictorial projection includes isometric, oblique, and perspective methods.

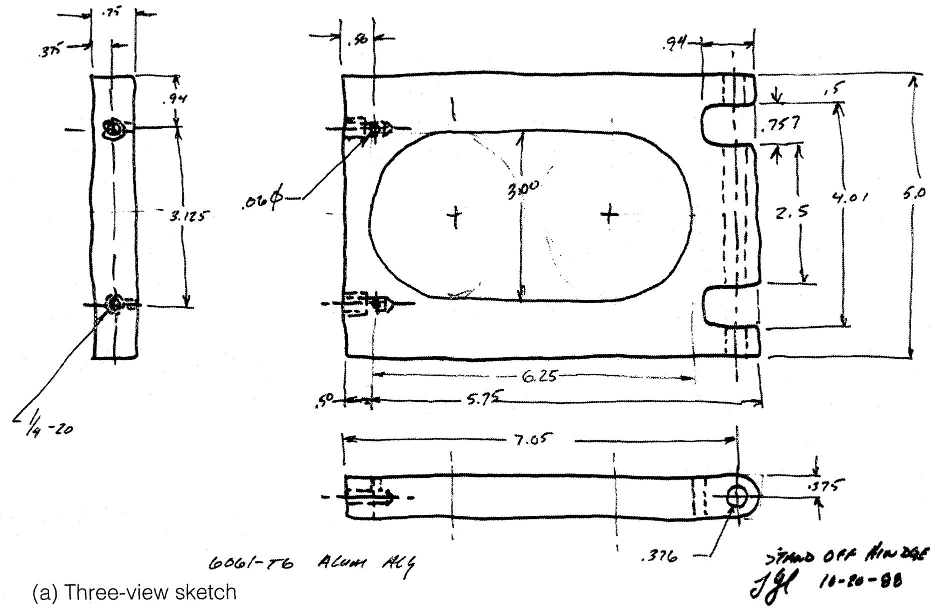
Isometric projection is based on the theory that a cube representing the projection axes is rotated until its front face is 45° to the frontal plane and then is tipped forward or downward at an angle of 35°16′. All three primary faces are displayed equally. In Figure 9.30 the part has been enclosed in a glass box and projected onto each of its corresponding surfaces. The viewing plane 1-2-3 is parallel to the projection plane. This is an isometric view. In true isometric projection, the three axes make equal angles with the projection plane and all three axes are equally foreshortened and make equal angles of 120°. A true isometric projection is about 81% the size of an isometric drawing. In actual industry practice, isometric drawing, not isometric projection, is employed.

Isometric drawing is commonly used in sketching (Fig. 9.31). Isometric drawings are constructed along three axes, one vertical and the other two at 30° to the horizontal, going both right and left (isometric axes, Fig. 9.32). All lines in isometric drawings that are on or parallel to the three axes are drawn true length and are isometric lines. Lines not on or parallel to the axes are constructed with offset dimensions and are called nonisometric lines. Nonisometric lines are not true length.

9.6.1 Isometric Construction

Isometric construction using the box method is illustrated in Figure 9.33. The procedure for drawing an isometric box is shown in Figure 9.33(a) using 30° triangles. Starting at point A, the three axes are drawn. The edges of the box are constructed from the height, width, and depth. In an isometric drawing the dimensions are not foreshortened.

After the part is boxed in, the remainder of the drawing is completed. Dividers (or a scale) are used to transfer dimensions shown in Figure 9.33(c) to the isometric view of Figure 9.33(b). All measurements are taken along isometric



(a) Three-view sketch

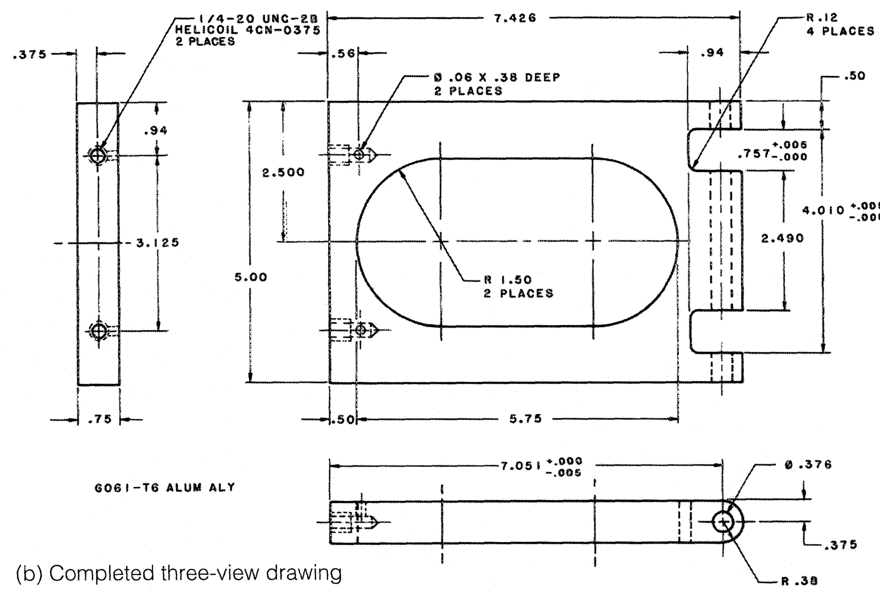


FIGURE 9.29 View Alignment

(b) Completed three-view drawing

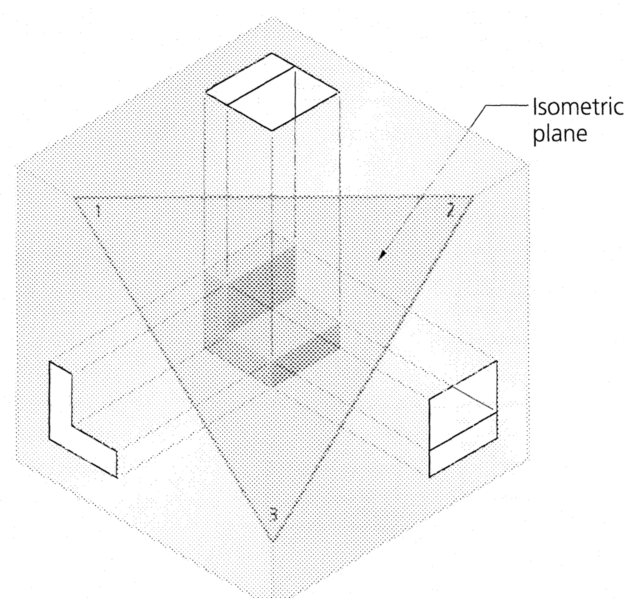


FIGURE 9.30 Isometric Projection

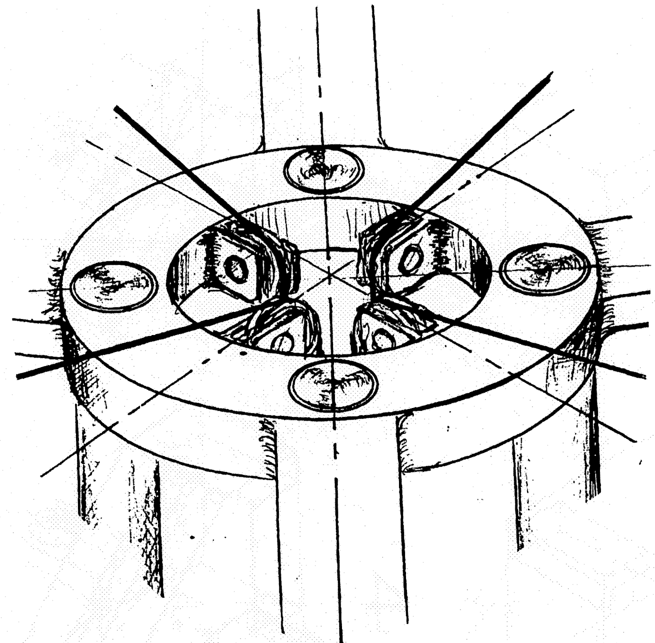


FIGURE 9.31 Isometric Sketch

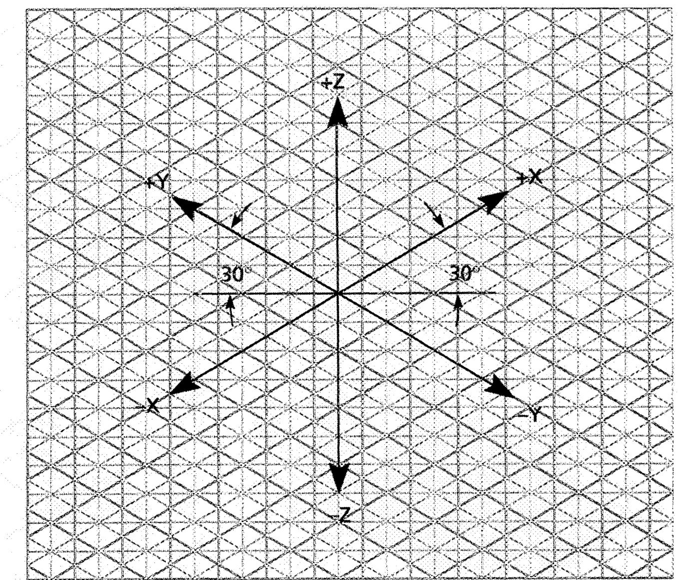


FIGURE 9.32 Isometric Axes

lines. Dimension D1 is measured along the vertical axis; dimensions D2, D3, and D4 are in the horizontal plane and are measured along or parallel to one of the receding axes. After the centerlines are located, the circles and arcs are drawn.

9.6.2 Isometric Angles

Because of the distortion created by the isometric view, few angles appear as true angles. Angles appear larger or smaller than true size and must be established by offset dimensions. For example, the plane in Figure 9.34 has angles of 45° and 30°, both of which are constructed from offset dimensions,

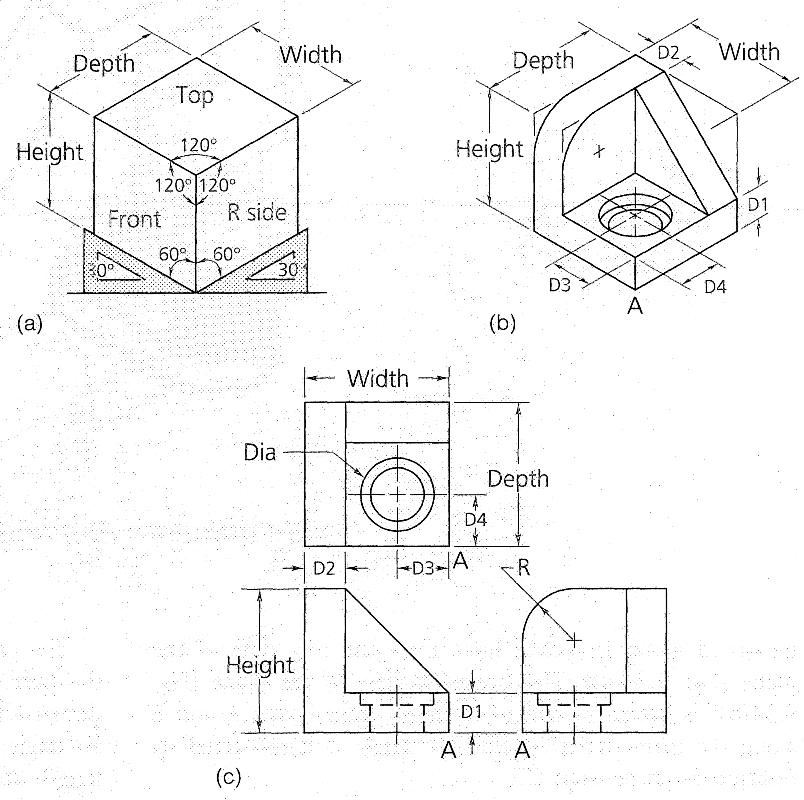
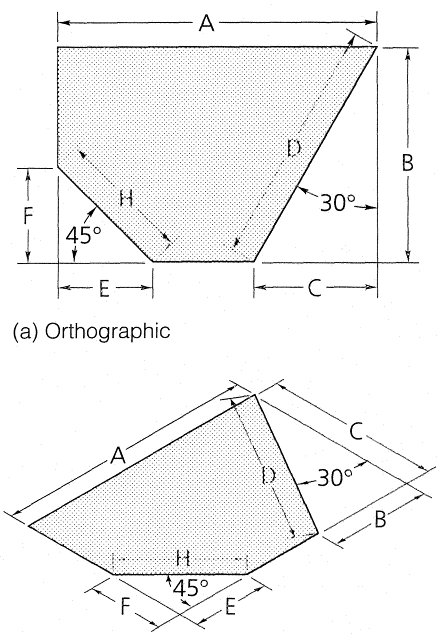


FIGURE 9.33 Isometric Projection



(a) Orthographic

(b) Isometric

FIGURE 9.34 Isometric Angles

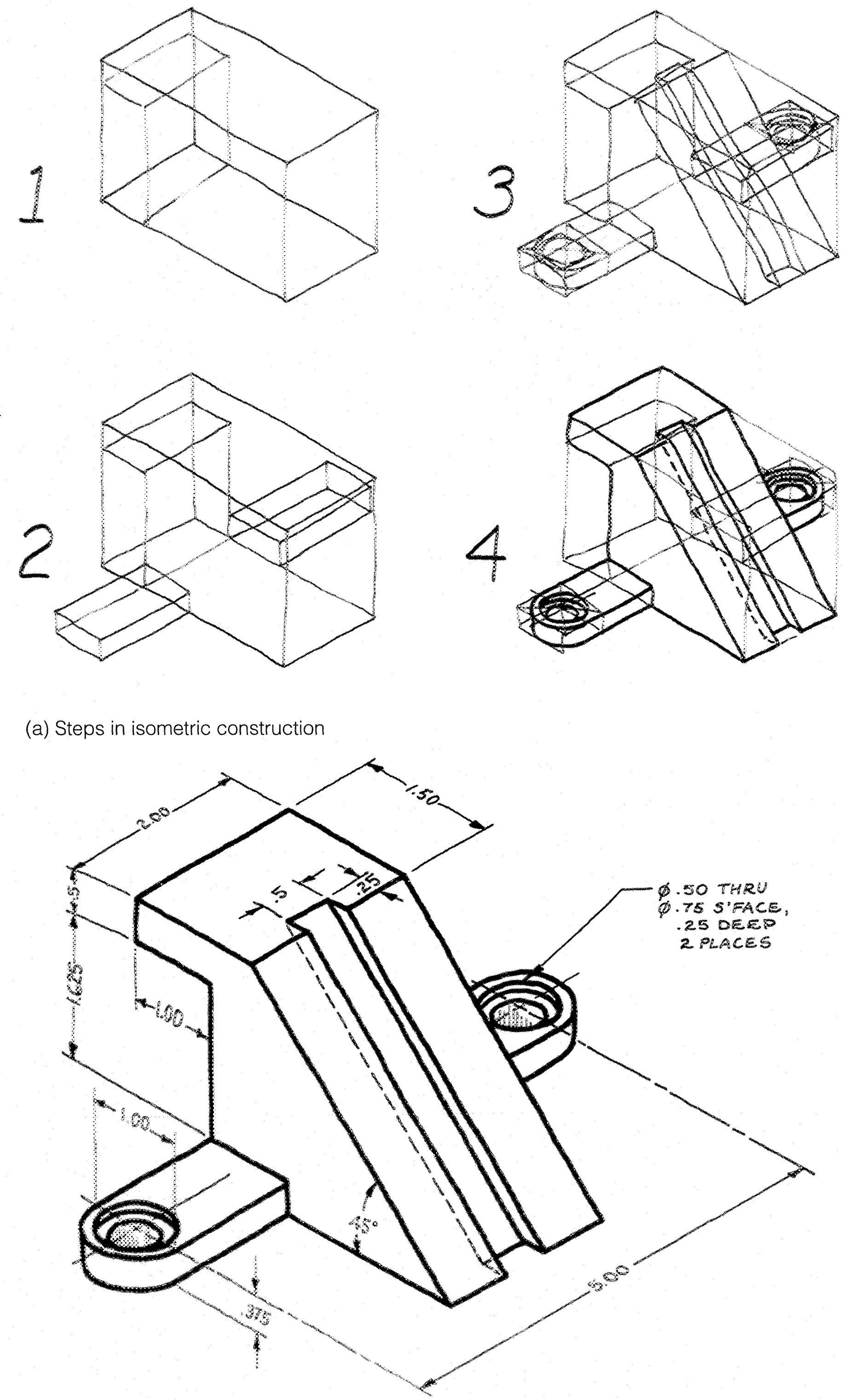


FIGURE 9.35 Isometric Construction

(a) Steps in isometric construction

(b) Completed sketch with dimensions

measured along isometric lines from the top view of the plane [Fig. 9.34(a)]. The isometric view of the plane [Fig. 9.34(b)] is boxed in with true-length dimensions A and B along the isometric axes. The 30° angle is constructed by transferring dimension C.

The part in Figure 9.35 has an angled surface. To draw the part in an isometric view, it is necessary to use offset dimensions to establish the endpoints of the edges that are at an angle. Endpoint dimensions can be taken along true-length lines.

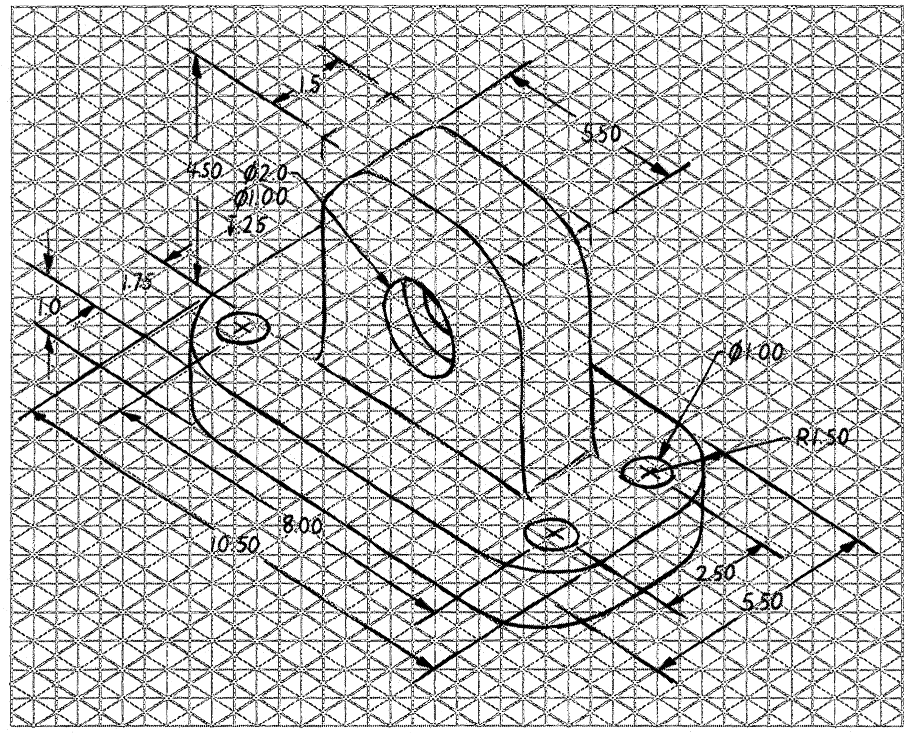


FIGURE 9.36 Isometric Sketch Using Grid Paper

9.6.3 Isometric Circles and Arcs

Circles and circular arcs on isometric drawings appear elliptical (Fig. 9.36) unless they fall exactly on or parallel to the isometric viewing plane. Many methods are employed to construct isometric ellipses: template, trammel, four-center, and point plotting. For sketches, freehand techniques are sufficient (you can also use a template).

The four-center method (Fig. 9.37) does not create a perfect ellipse, but is accurate enough for most purposes. This method is used to draw circles or arcs on any isometric face (Fig. 9.38):

1. Draw lines DA and DC along the two receding axes (at 30°). Draw line AB parallel to DC, and line CB parallel to

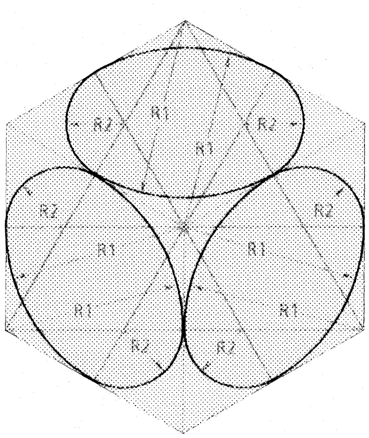


FIGURE 9.37 Isometric Ellipses Drawn by the Four-Center Method

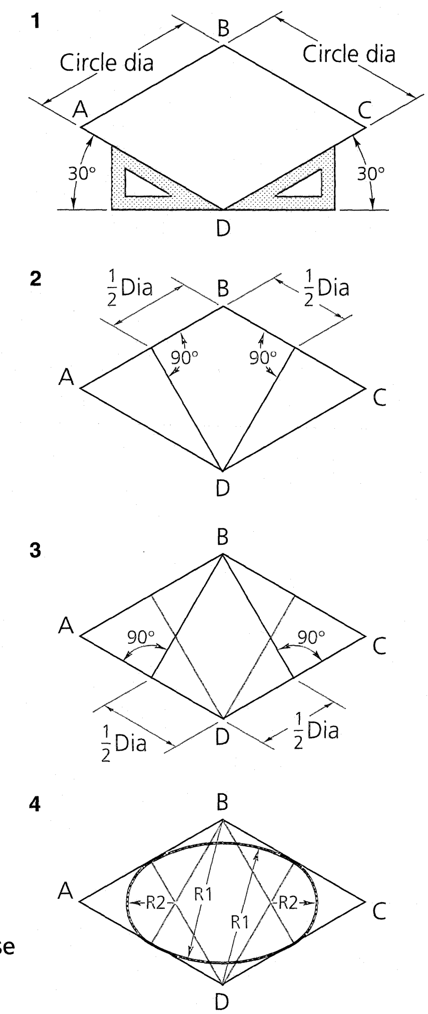


FIGURE 9.38 Ellipse Construction for Isometric Drawings

Applying Parametric Design . . .

SKETCHING FEATURES

Sketching is done in the **Sketcher**. Almost all traditional lines, circles, arcs, and their variations can be accomplished on the screen without the need to create exact and perfectly constructed geometry. The system will assume a variety of conditions, such as tangency, similar sizes for same-type geometry, parallelism, perpendicularity, verticality, horizontality, touching endpoints, tangent points, and symmetry. The simple part in Figure A was created by sketching lines and arcs.

You can create two types of **lines**: geometry lines and centerlines. Geometry lines are used to create feature geometry. Centerlines are used to define the axis of revolution of a revolved feature, to define a line of symmetry within a section, or to create construction lines. To sketch lines:

1. Choose Line from the GEOMETRY menu. The LINE TYPE menu appears.
2. Choose Geometry or Centerline from the top portion of the menu to indicate the type of line you want.
3. Choose a command from the bottom portion of the menu to indicate how you wish to create the line:

2 Points Create a line by picking the start point and the endpoint. Geometry lines created with this command will automatically be chained together.

Parallel Pick an existing line to determine the new line's direction, then pick the start point and the endpoint. For a centerline, only a single pick is needed to determine the parallel placement of the line, and the ends of the centerline will be chosen to fit model or section outlines.

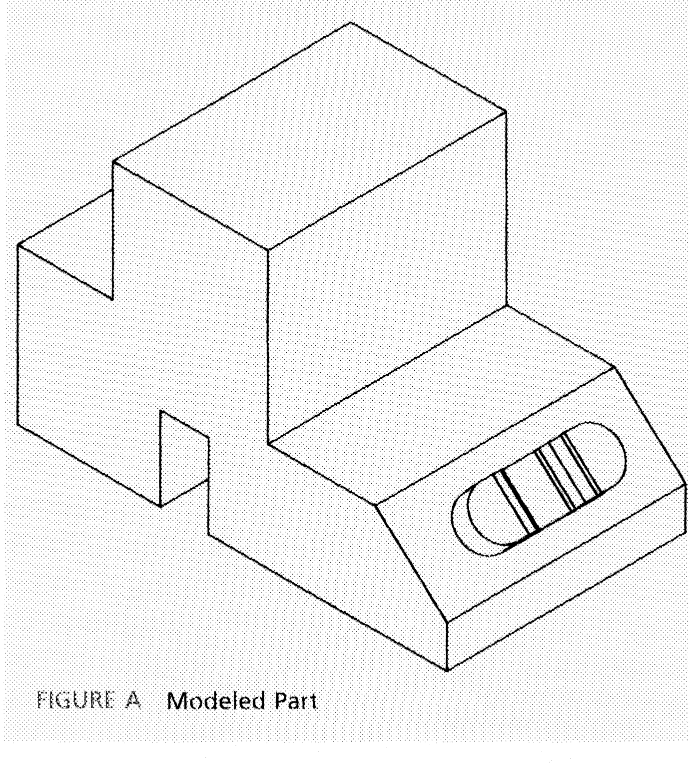


FIGURE A Modeled Part

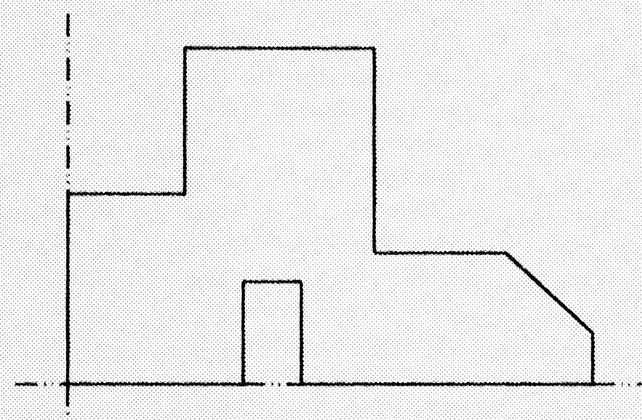


FIGURE B Sketched Outline of Part Using Lines

Perpendicular Pick an existing line to determine the new line's direction, then pick the start point and the endpoint. For a centerline, only a single pick is needed to determine the perpendicular placement of the line, and the ends of the centerline will be chosen to fit model or section outlines.

Tangent Pick an endpoint of an arc or spline to start the new line and determine its direction, then pick the endpoint of the line. For a centerline, only a single pick is needed to determine the tangent placement of the line, and the ends of the centerline will be chosen to fit model or section outlines.

2 Tangent Pick two arcs, splines, or circles to determine the direction of the new line. The line is automatically created between the selected entities. A 2 Tangent line created to construction entities will not split the entity. This button is the model and stays selected until you explicitly select another command. A 2 Tangent centerline, created as a 2 Tangent line defined with two circles, will not split the circles.

Pnt/Tangent Pick a point anywhere in the current section, then pick an arc, spline, or circle to which the line must be tangent. The line will be created automatically.

Horizontal Creates a line that is horizontal relative to the orientation of the section. For a geometry line, the endpoint is automatically the starting point of a chained vertical line. For a centerline, only a single pick is needed to determine the vertical locations of the line.

Vertical Creates a line that is vertical relative to the orientation of the section. For a geometry line, the endpoint is automatically the starting point of a chained horizontal line. For a centerline, only a single pick is needed to determine the vertical locations of the line.

The lines for the outline of the part shown in Figure B are sketched without regard to actual sizes. After the lines are sketched, dimensions are added (see Fig. C). The part is aligned to the default datum planes (see Fig. D). After

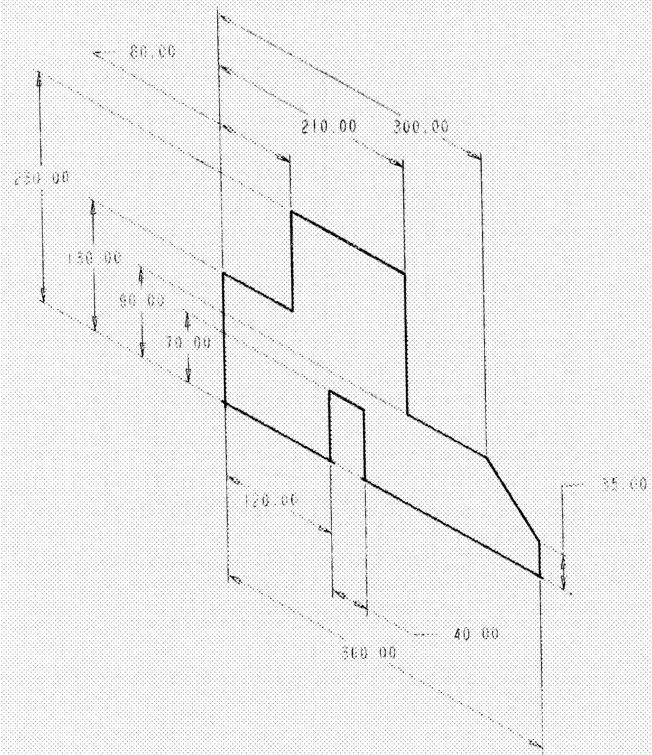


FIGURE C **Dimensions Added to the Sketch**

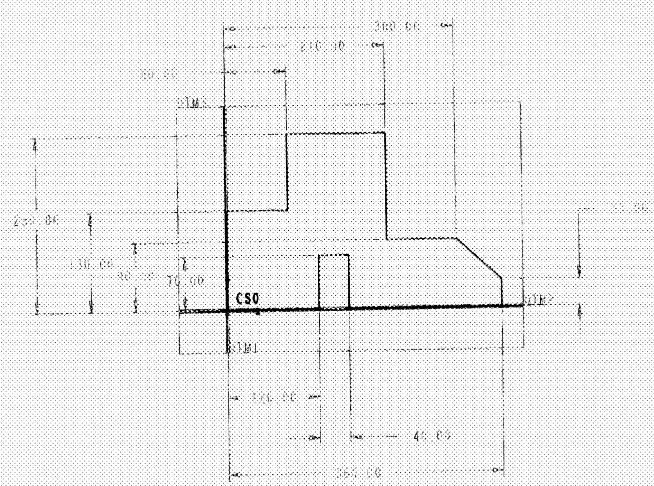


FIGURE D **Part Aligned and Constrained by the Default Planes**

regeneration, the depth of the part is input and the extruded protrusion base feature is complete (see Fig. E).

The next feature to be created lies on the angled surface (Fig. F). The slanted plane is used as the sketching plane and an arc is placed via one of the available options (Fig. G). The opposite arc is then sketched (Fig. H). The lines between the arcs and a **centerline** complete the sketch (Fig. I). Dimensions are added to the sketch and modified to the design values (Fig. J). The depth of the protrusion is given and the feature is complete (Fig. K).

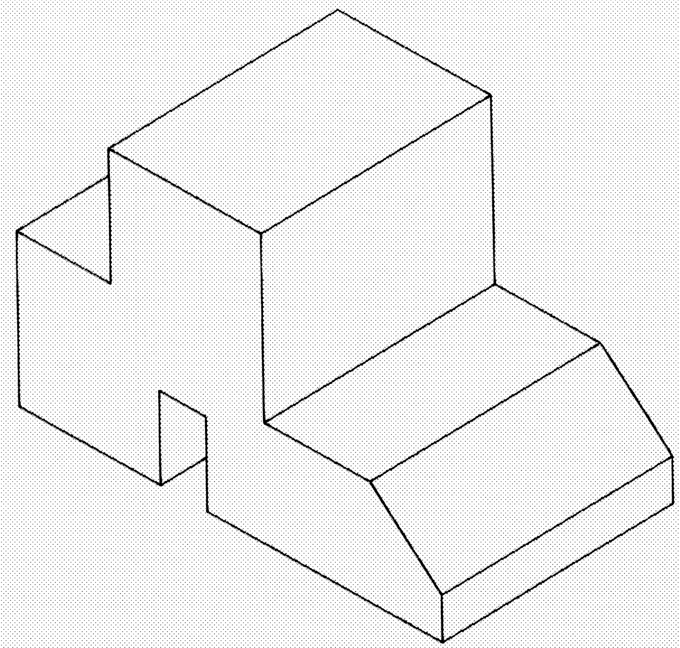


FIGURE E **Extruded Protrusion Base Feature**

Arcs are sketched via the menu or the mouse. To sketch arcs:

1. Choose Arc from the GEOMETRY menu. The ARC TYPE menu appears.
2. Choose one of the following options from the ARC TYPE menu:

Tangent End This is the same as creating an arc with Mouse Sketch, except you must use the left mouse button. Pick an

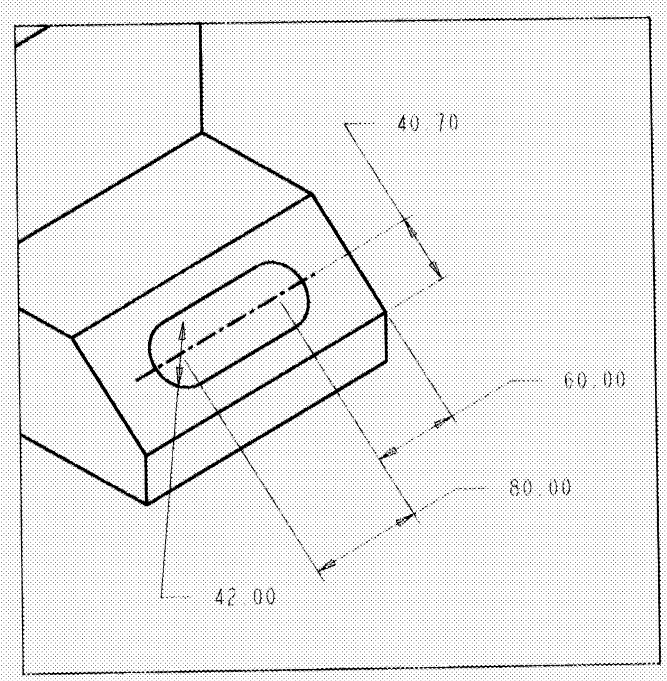


FIGURE F **Sketched Boss Feature Showing Lines and Arcs**

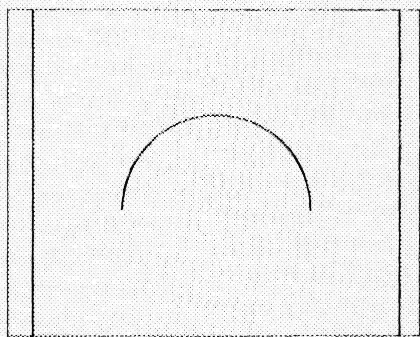


FIGURE G Sketched Arc on Angled Surface

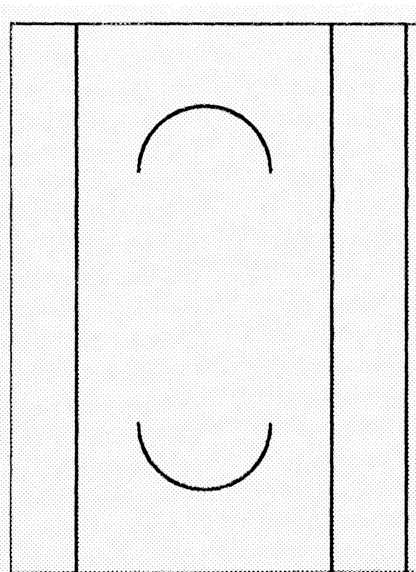


FIGURE H Second Arc Added to the Sketch

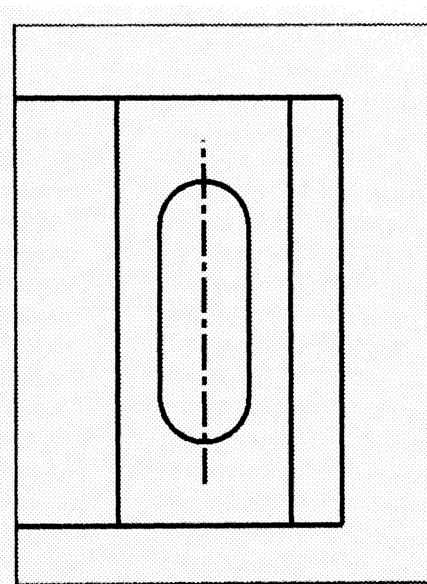


FIGURE I Lines and Centerline Added to the Sketch

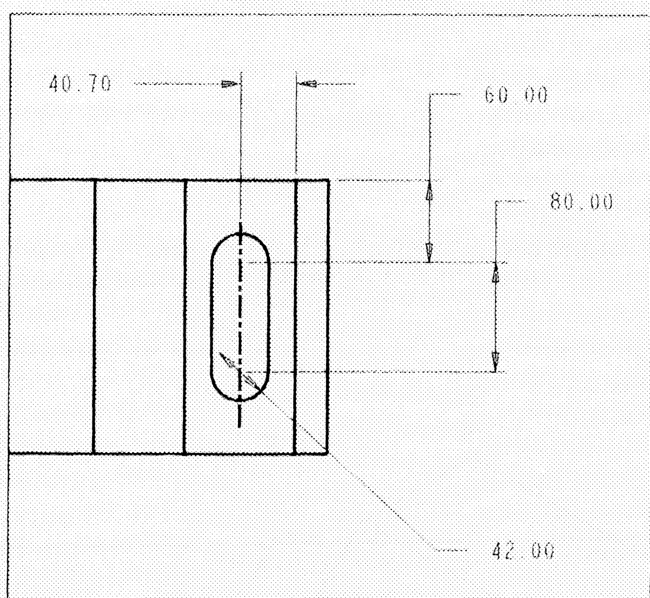


FIGURE J Dimensions Added to the Sketch

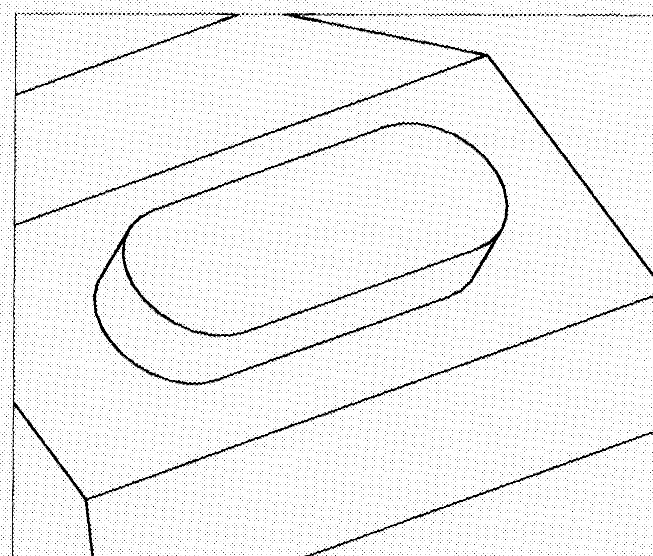


FIGURE K Completed Boss Feature

end of an entity to determine tangency, then pick the endpoints of the arc.

Concentric Pick an existing circle or arc as a reference, then pick the endpoints of the new arc. As you create the arc, a radial line will appear through its center to assist in aligning the endpoint.

3 Tangent Select three entities for the new arc to be tangent to, then create the arc in the same direction as the reference picks.

Fillet Pick two entities between which to create a tangent arc.

Ctr/Ends Pick the center point of the arc, then pick the arc's endpoints.

3 Points Pick the endpoints of the arc, then pick a point on the arc.

The last feature of the part is crated by sketching the slot cut on the edge of the boss protrusion (see Fig. L). The dimensions for the cut are added and then modified to their correct design values (Fig. M). The slot feature will go through the boss feature (Fig. N). The completed slotted boss is shown pictorially (Fig. O). The part is now complete and is displayed as a shaded model (Fig. P).

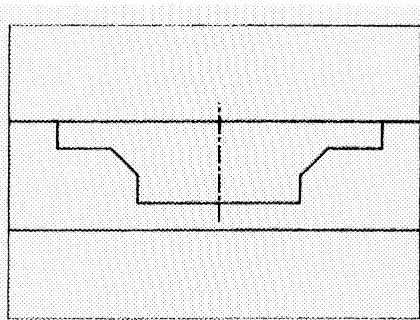


FIGURE L **Sketched Cut Using Lines**

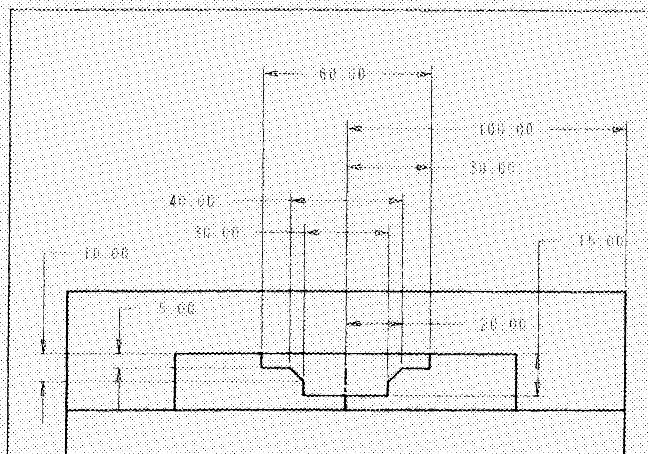


FIGURE M **Slot Dimensions Added and Modified**

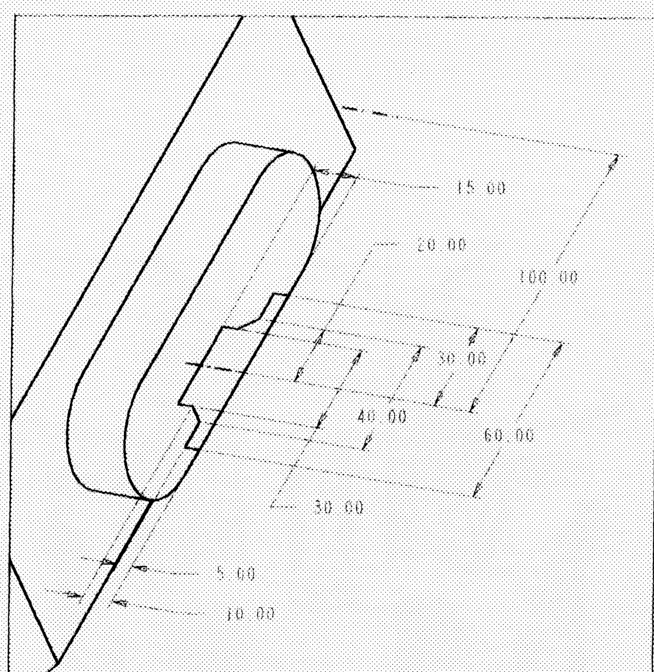


FIGURE N **Pictorial of Slot Sketch and Dimensions**

This part (Fig. Q) is an example of the type of modeling that can be accomplished with just two simple types of geometry, lines and arcs.

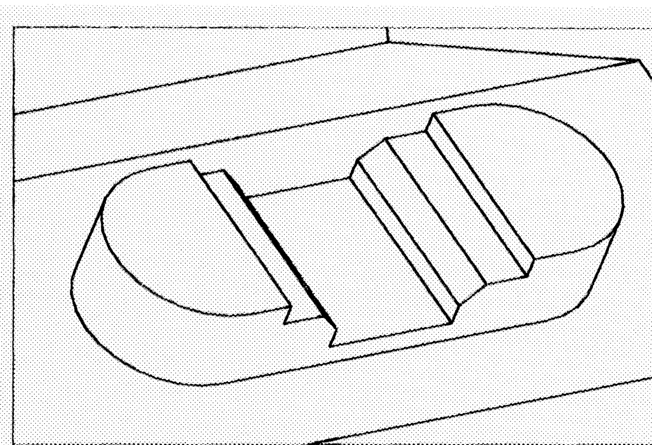


FIGURE O **Pictorial View of Completed Boss**

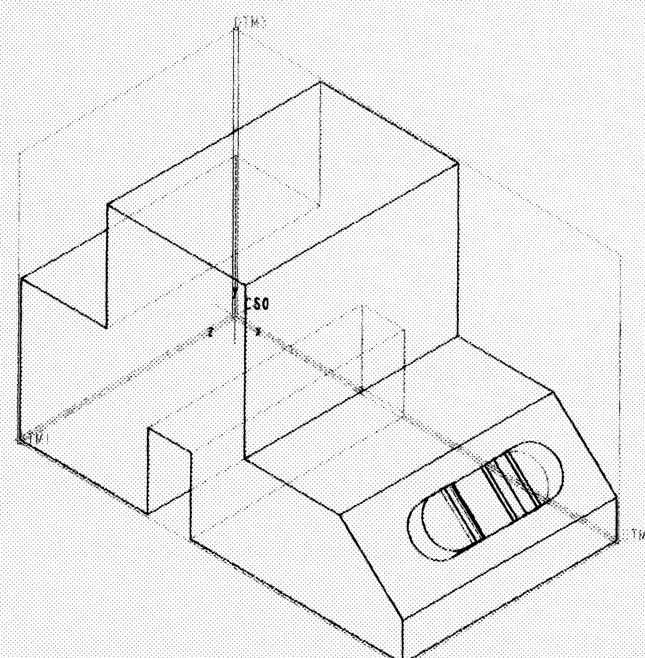


FIGURE P **Completed Part**

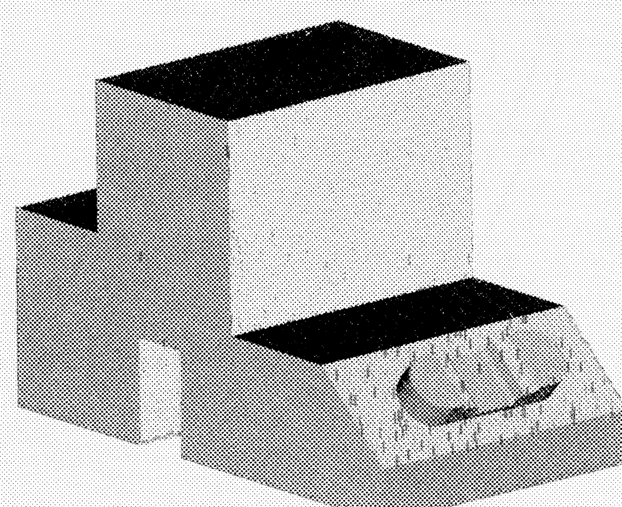


FIGURE Q **Shaded Display of Part**

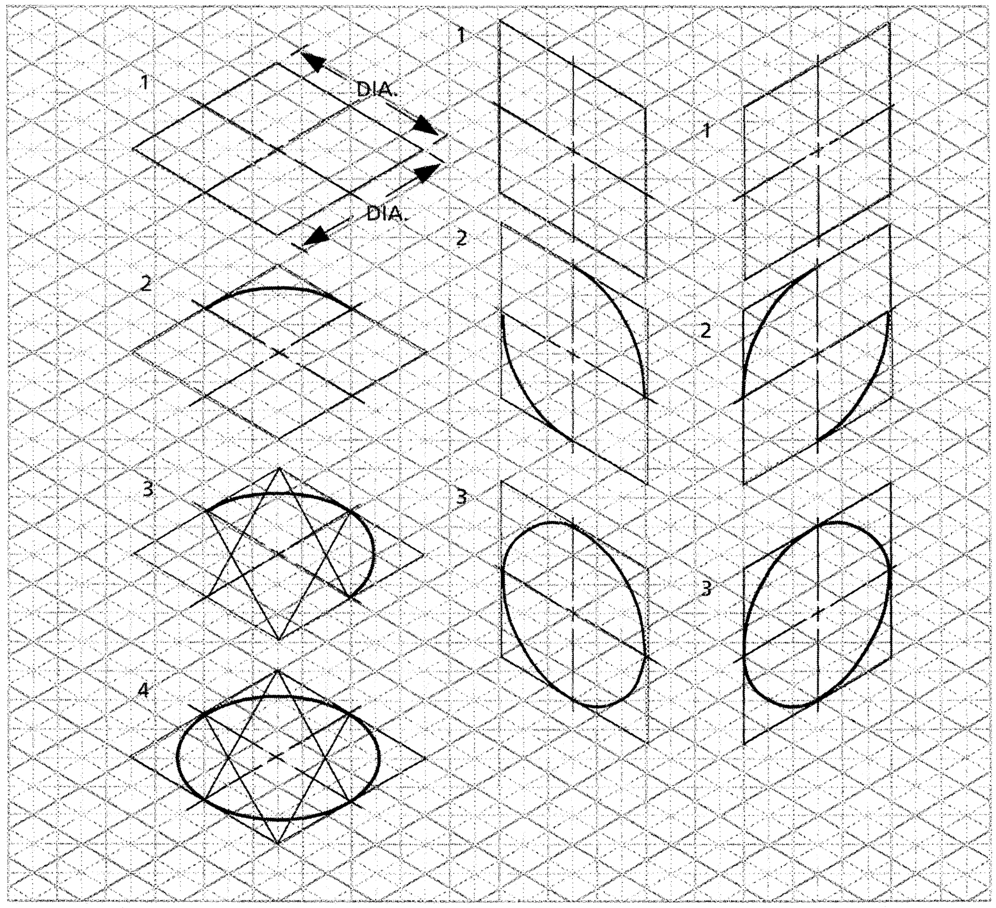


FIGURE 9.39 Sketched Isometric Arcs

AD. *The isometric square has sides equal to the diameter of the circle.*

2. Draw construction lines from point D perpendicular to line AB at its midpoint and perpendicular to line CB at its midpoint. These lines are perpendicular bisectors of each side.

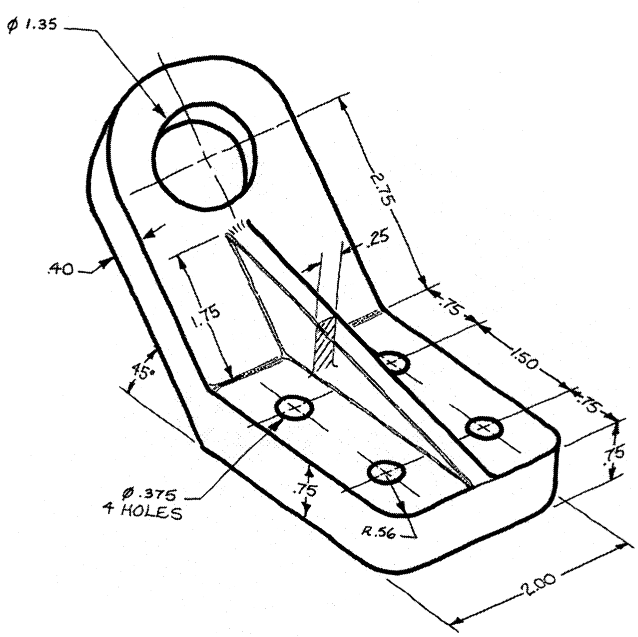


FIGURE 9.40 Isometric Sketch of Part

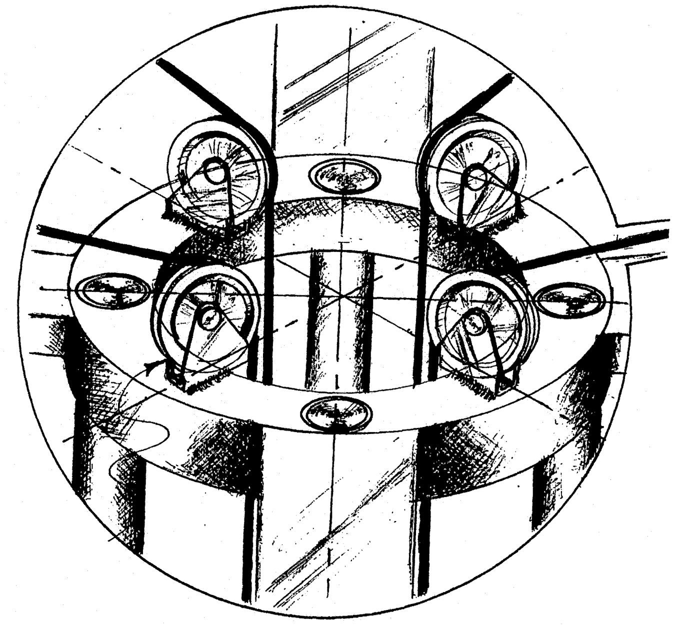


FIGURE 9.41 Isometric Sketch of Pulley System Showing a Variety of Circular Parts Not in an Isometric Plane

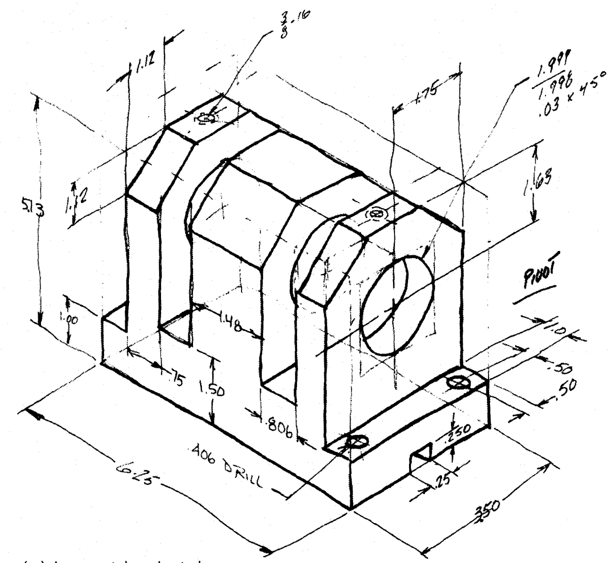
3. Repeat step 2 using point B and lines AD and CD
4. Use points D and B to construct arcs R1. R2 originates at the intersection of the perpendicular bisectors.

Isometric ellipses are easily sketched using this method (Fig. 9.39). Circles, arcs, or curves that do not lie in isometric planes, as in Figures 9.40 and 9.41, must be plotted with offset dimensions. A series of points is established along the curved outline (see Chapter 13). Offset dimensions for these points are transferred to the isometric drawing and are laid off along isometric lines.

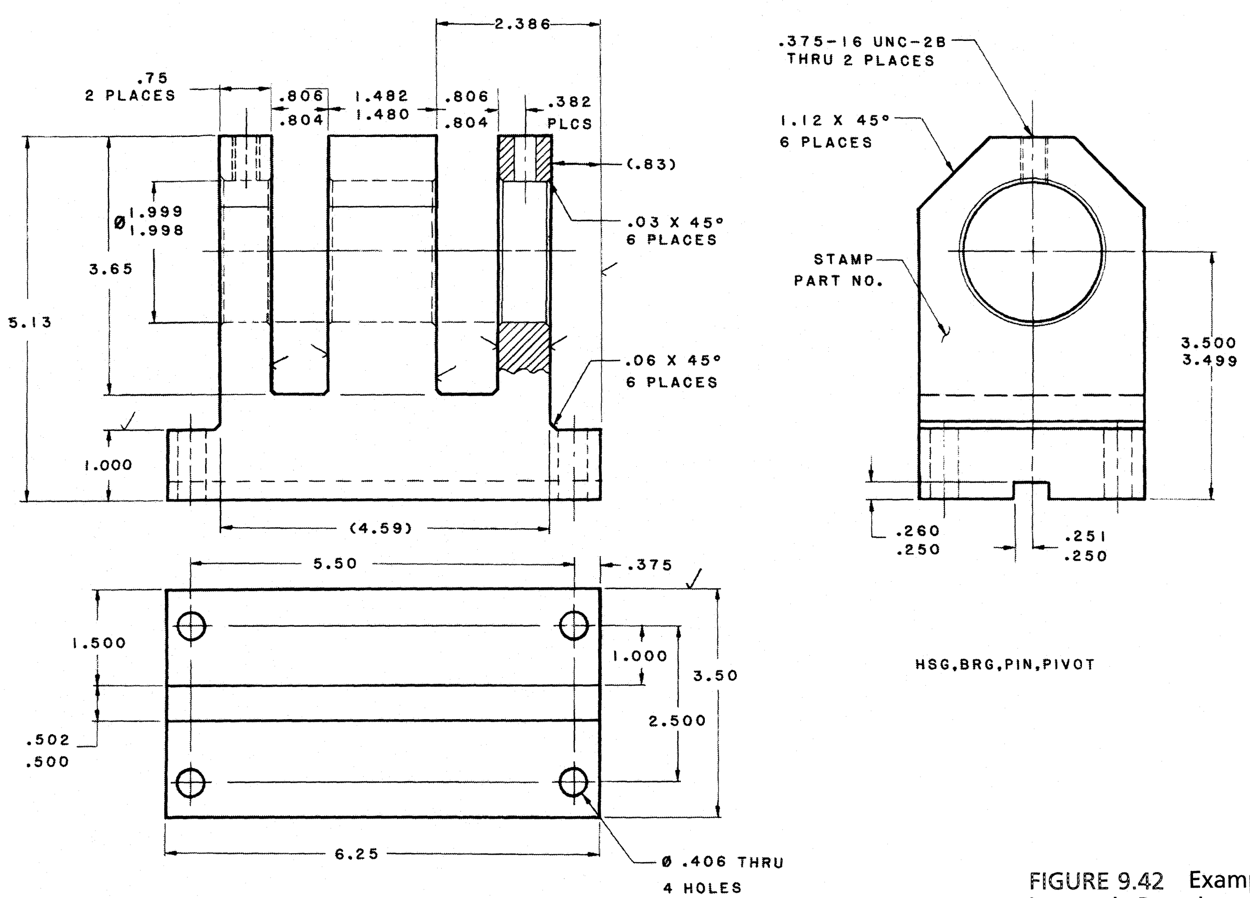
The last step in any sketch is to provide the required dimensions. Figure 9.42 shows a sketch of a mechanical part, along with the finished detail.

9.7 OBLIQUE PROJECTION

Oblique drawings are produced from parallel projectors that are angular to the projection plane. The primary difference between isometric and oblique is that a receding axis is required for oblique drawings. The other surface is drawn true shape and size. The three axes are vertical (+Z), horizontal (+Y), and receding (+X) (Fig. 9.43). In oblique projection the front face of the part is placed parallel to the image plane. The other faces of the part are on receding axes (1° to 89°), as shown in Figure 9.44. Here, the front face of the block and the diameter of the hole are drawn true shape and size.



(a) Isometric sketch



(b) Instrument drawing

FIGURE 9.42 Examples of Isometric Drawing

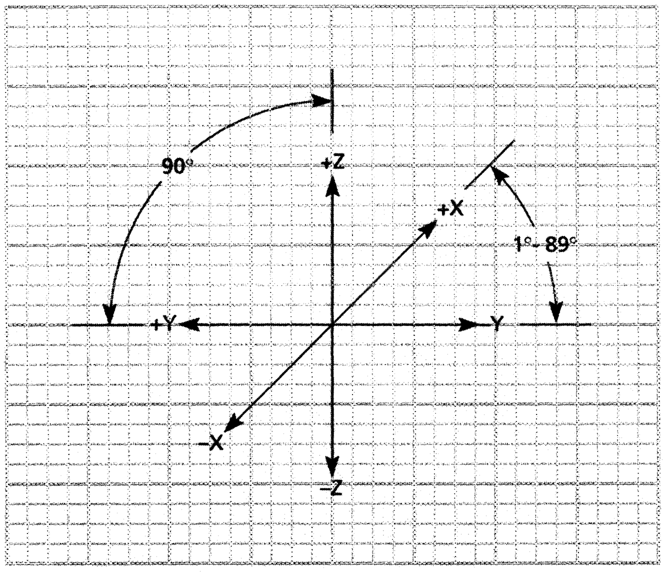


FIGURE 9.43 **Oblique Axes**

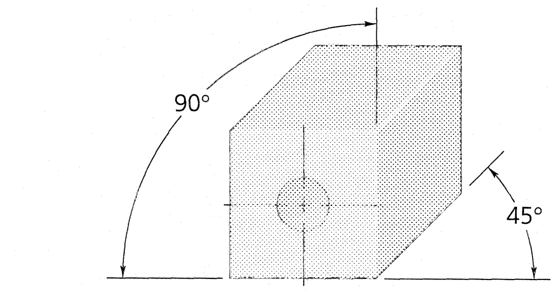
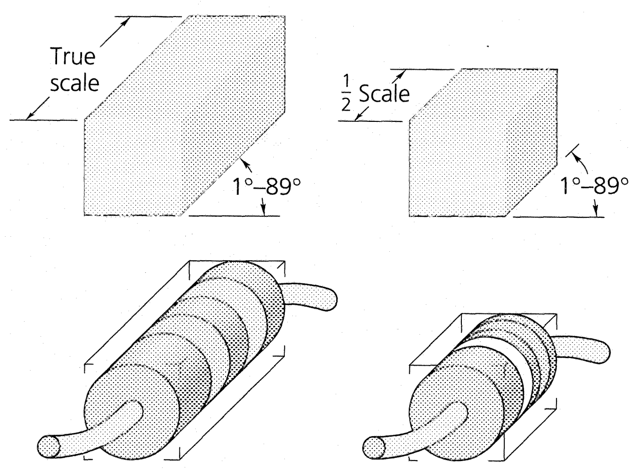


FIGURE 9.44 **Oblique Projection**

The two basic categories of oblique projection are *cavalier* and *cabinet* (Fig. 9.45). In a cavalier projection (a), receding lines are not foreshortened (full scale). In a cabinet projection (b), the receding lines have been foreshortened one-half their original length ($\frac{1}{2}$ scale). The most common angles are 15°, 30°, 45°, 60°, and 75°. The most common angle for the receding axis is 45°.

Parts drawn with oblique projection are oriented so that the surface with the most curved features lies in the front plane.



(a) Cavalier (b) Cabinet

FIGURE 9.45 **Oblique Projection**

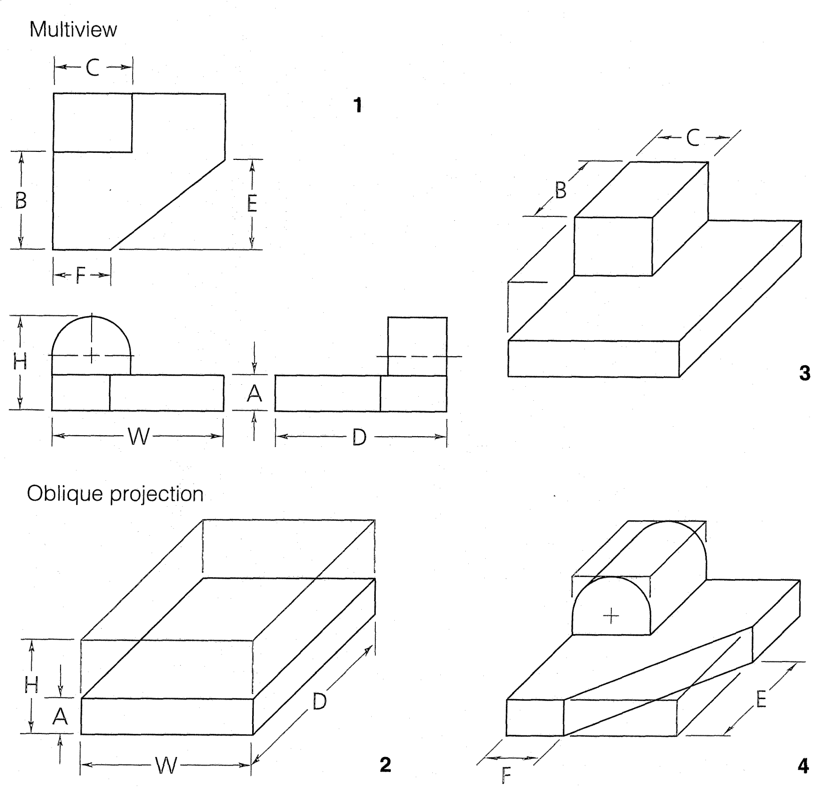


FIGURE 9.46 **Construction Steps in Oblique Projection**

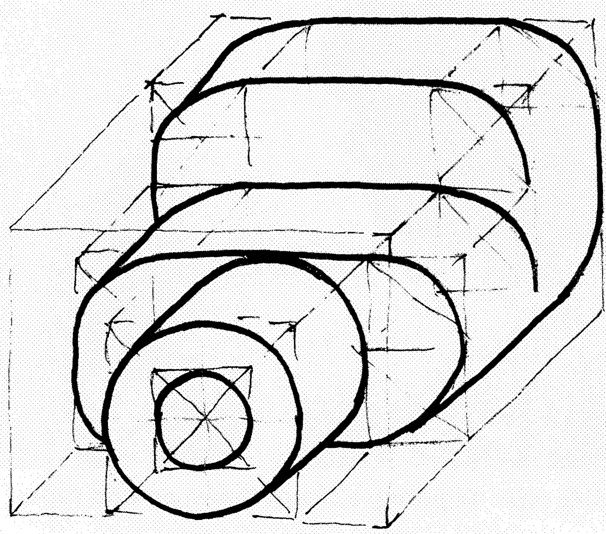


FIGURE 9.47 Oblique Sketch

Circles and arcs are true projections in this position. Oblique projection is extremely useful for parts with parallel curved or irregular features. The construction process for inclined lines and planes is similar to that for isometric drawings: Locate each feature's endpoints along lines that are parallel to one of the axes. For slanted surfaces, locate both ends of the surface and connect the points.

In Figure 9.46 the following steps were used in the construction of the part:

1. Transfer each true-length dimension from the multiview drawing to the oblique view.
2. Establish the overall dimensions of the part using the height, width, and length dimensions, and block out the part as shown (45 ° was used as the receding angle). Use dimension A to establish the top surface of the part.
3. Use dimensions B and C to locate and establish the circular lug of the part. Dimension C is the diameter of the circular feature.
4. Set off dimensions E and F along their respective edges, and draw the corner cut as shown. Then draw the circular curve using the center point, and darken the part.

9.7.1 Oblique Sketching

Start the oblique sketch as you would a multiview or isometric drawing: Block out the overall dimensions, each feature, and box in the curves (Fig. 9.47). Figure 9.48 shows the four steps in oblique sketching: (1) Block out the part, starting with the front or the rear face, and establish the width and the height. (2) Establish the depth of each face, and then *carve out* the primary features. (3) Locate each circular form with centerlines, and box in the curves. (4) Complete the sketch by finalizing the features and darkening

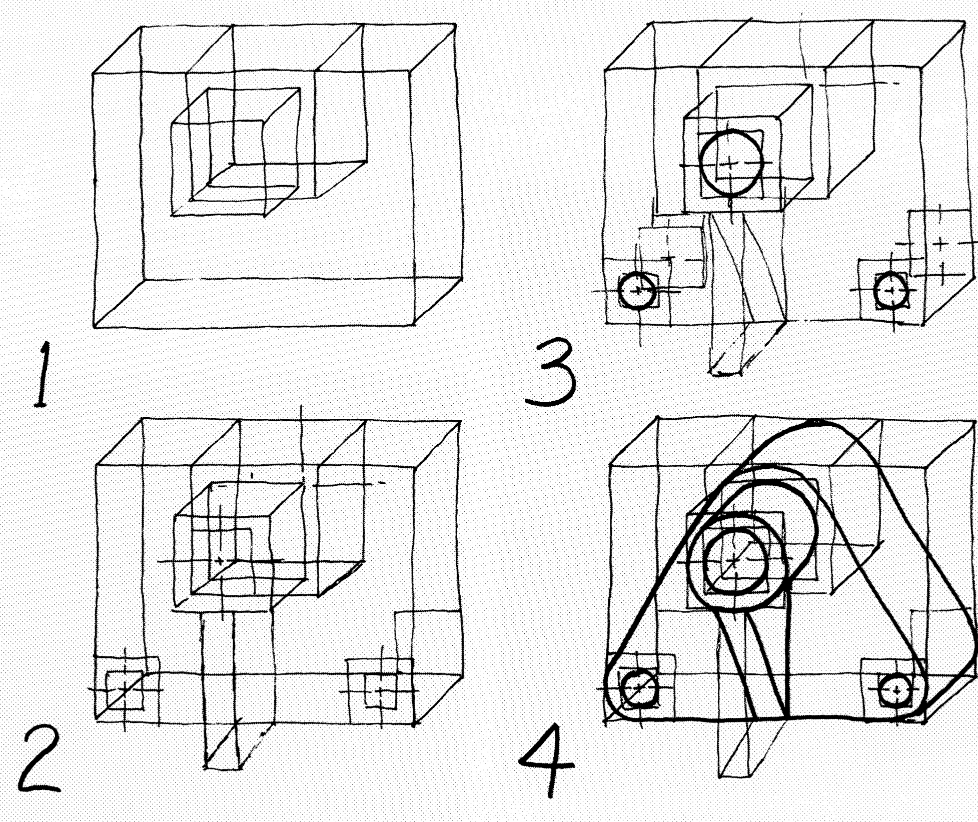


FIGURE 9.48 Step-by-Step Oblique Construction

in all object lines. Normally, hidden edges and surfaces are not shown. In most instances, construction lines are left on the sketch or only slightly dimmed before darkening the object lines.

You May Complete Exercises 9.5 Through 9.8 at This Time

9.8 CAD AND SKETCHING

Sketches are utilized in three ways when a CAD system is used for engineering and design:

- Directly on the CAD system using a sketch or sketcher command
- To lay out diagrams that will later be digitized, usually on grid paper
- As a designer's or engineer's tool for graphically exploring a design on paper before using the system

The typical CAD workstation has an area next to it dedicated as a reference surface (Fig. 9.49). Here, the designer or engineer uses the time-proven method of freehand sketching during the initial design stages of a project. Sketches enable you to develop and explore ideas and to design alternatives. The sketch can then serve as a reference for 3D CAD modeling or 2D CAD drawing and detailing.

Freehand sketching directly on the CAD system is also possible with some computer graphics systems, though this capability is still somewhat limited. By turning on construction lines on one layer and using a different color, you can sketch directly on the system. The layer with the construction lines is turned off before plotting the sketch.

FIGURE 9.49 Designers Sketching at a Reference Table Next to a CAD/CAM Workstation

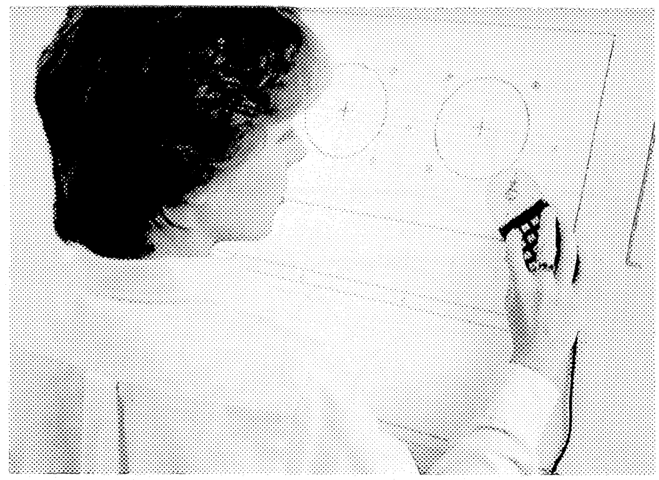


FIGURE 9.50 Digitizing a Sketch Drawn on Grid Paper

Digitizing existing drawings and diagrams (Fig 9.50) can be done directly from a freehand sketch with the proper equipment. A digitizing table is used to input existing instrument drawings or freehand sketches. Freehand sketches are normally completed on grid paper and then taped to the digitizing surface. The drafter can establish any scale for the project. The drawing or sketch is then digitized (Fig. 9.51) using a puck, pen, or other input device to create the 2D drawing.

Besides the digitizing of sketches, some CAD systems have the ability to create sketches via specific commands. The **SKETCH** command on AutoCAD permits freehand drawings to be created as part of a drawing. Freehand drawings are distinguished from normal AutoCAD drawings in that they are automatically entered as the puck, mouse, or pen is moved, rather than being built from points, lines, arcs, etc. Freehand drawing via a CAD system is best suited for such items as signatures (Fig. 9.52), map contour lines, and other types of irregular material.

The freehand drawing facility captures your sketching as a series of lines. You can perform limited editing on these

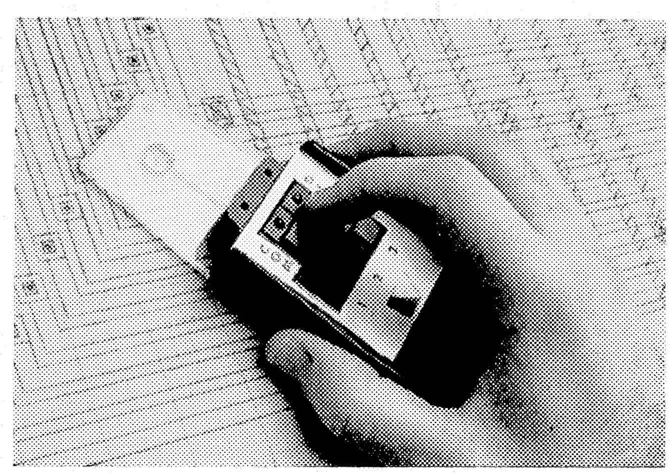


FIGURE 9.51 Digitizing a 2D Sketch

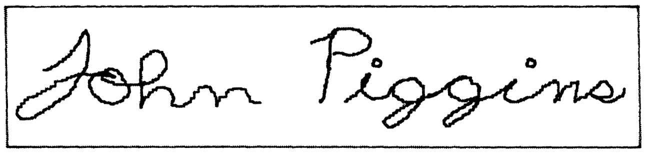


FIGURE 9.52 Signature Drawn with AutoCAD's SKETCH Command

lines before recording them in the AutoCAD drawing database. Once recorded, all the normal facilities of AutoCAD can be used on the freehand material—you can move it, delete all or part of it, make it part of a **Block,** and so forth.

Freehand drawings can be made only with a pointing device (digitizing tablet, mouse, pen, etc.). Freehand drawing, especially with very fine accuracy, generates a large number of lines. Although every effort is made to reduce the number of lines generated (by combining lines in the same direction), it is possible in 20 seconds of freehand sketching to create a drawing with as many lines as a normal drawing that took 20 hours to enter.

To complete a simple freehand sketch, turn **Ortho, Snap,** and **Tablet** modes off and enter the **SKETCH** command:

```
Command: SKETCH
Record increment <current>:
```

You determine the increment distance based on the drawing size and the required resolution. Enter the distance, in drawing units, over which movement of the pen or puck justifies generating a line segment. The smaller the distance, the more accurate (smoother) the sketch, but also the larger the database storage involved. In general, 0.1 will generate a reasonably high-resolution sketch on AutoCAD. This method of sketching is not as fast or as accurate as freehand sketching on grid paper. The final sketch with a CAD system tends to have jagged lines, as in Figure 9.52, where the signature is not very smooth, even though an increment of 0.05 was specified.

For the foreseeable future, sketching will remain a freehand manual process, as it has been since humans "scribbled" on cave walls and flat stones, used sticks to scratch construction ideas in sand, and drew engineering marvels on parchment and papyrus.

QUIZ

True or False

1. The most common angles for sketching oblique drawings are 15°, 20°, 25°, and 40°.
2. When drawing a circle, it is common practice to rotate the paper.
3. When blocking in a circle or an isometric ellipse, the sides are equal to the diameter.
4. 6H lead is best for sketching.
5. Sketch vertical lines starting from the bottom and move up.
6. Never show centerlines for round or curved portions of a part if it is drawn as an oblique or isometric projection.
7. Pictorial sketches are essential to the design process because they allow the designer to explore different possibilities, shapes, and orientations of the part.
8. Grid paper should be used whenever possible when sketching.

Fill in the Blanks

9. ________ or ________ lead is the best grade for sketching.
10. The pencil is held about ________ from the ________ when sketching.
11. Right-handed drafters draw horizontal lines by moving the pencil from ________ to ________ .
12. Circles are sketched by first drawing a ________ .
13. ________ and ________ lines are used to lay out the outline of a part before darkening the lines.
14. ________ and ________ lines are sketched by moving the pencil from ________ to ________ .
15. ________ , ________ , and ________ are used to represent a part pictorially during the ________ stage of a project.
16. ________ lines are drawn vertical or receding at ________ degrees to the horizontal for isometric drawings.

Answer the Following

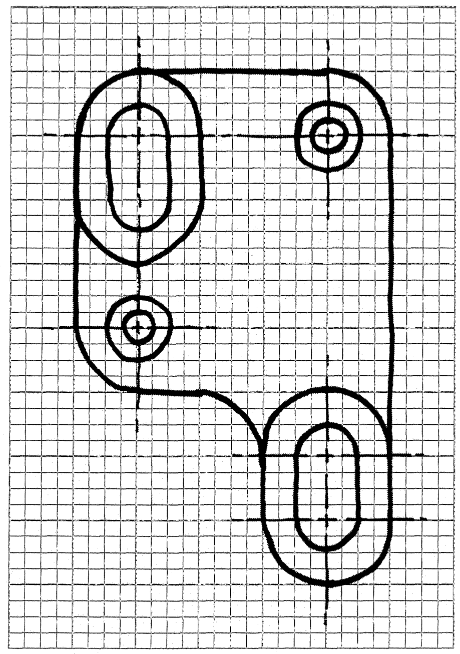
17. List the steps in sketching a circle.
18. Describe the difference between isometric and oblique projection.
19. How is sketching used in conjunction with CAD in the design process?
20. How would you sketch an ellipse that lies in the horizontal plane?
21. What are the six standard views? Which views are most commonly represented on an orthographic drawing?
22. Why are parts always blocked out before darkening the lines?
23. How does the shape of a part help determine the use of isometric or oblique projection techniques?
24. Describe the process of sketching irregular curves.

EXERCISES

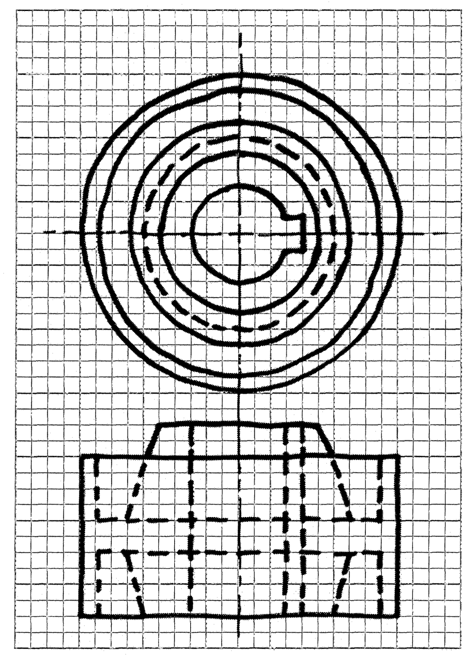
Transfer the given information to an "A"-size sheet of .25 in. grid paper. Complete all views, and solve for proper visibility, including centerlines, object lines, and hidden lines. Exercises that are not assigned by the instructor can be sketched in the text to provide practice and to enhance understanding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

After Reading the Chapter Through Section 9.5.3, You May Complete the Following Exercises (These exercises can also be used for isometric and oblique problems after you have completed the chapter.)

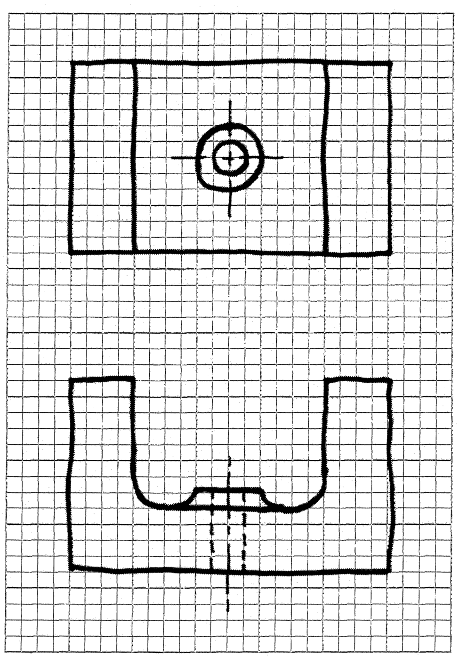
Exercise 9.1 Sketch the one-view drawing.
Exercise 9.2 Sketch the two-view drawing.
Exercise 9.3 Sketch the circular part.
Exercise 9.4 Sketch the two-view section drawing.



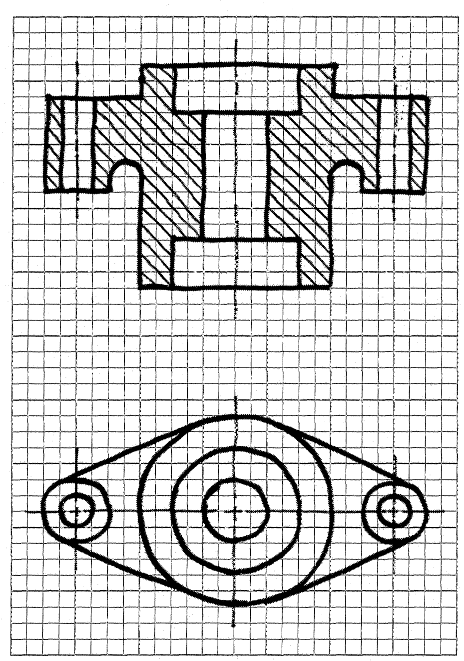
EXERCISE 9.1



EXERCISE 9.3



EXERCISE 9.2



EXERCISE 9.4

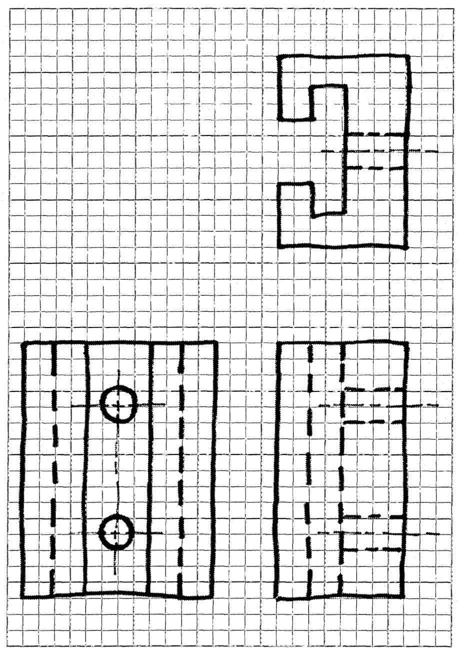
After Reading the Chapter Through Section 9.7.1, You May Complete the Following Exercises

Exercise 9.5 Sketch the three-view part, and complete an isometric sketch of the part on isometric grid paper.

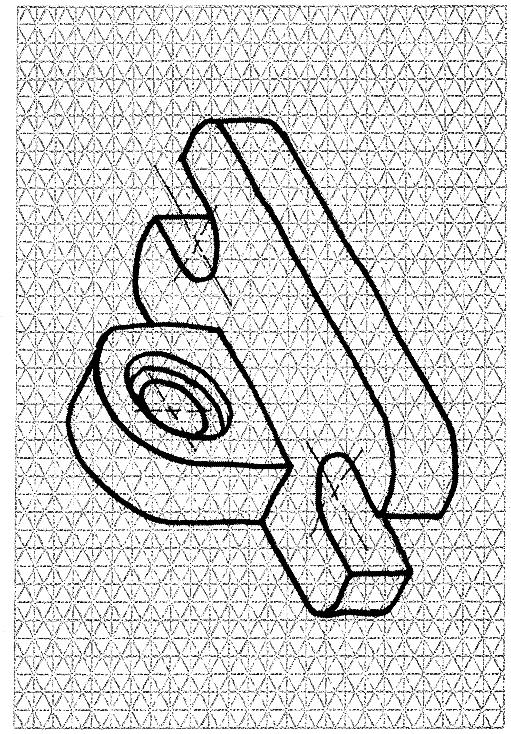
Exercise 9.6 Sketch an isometric view of the part on isometric grid paper, and complete a two-view drawing.

Exercise 9.7 Sketch an isometric view of the part on isometric grid paper. Also complete a three-view drawing.

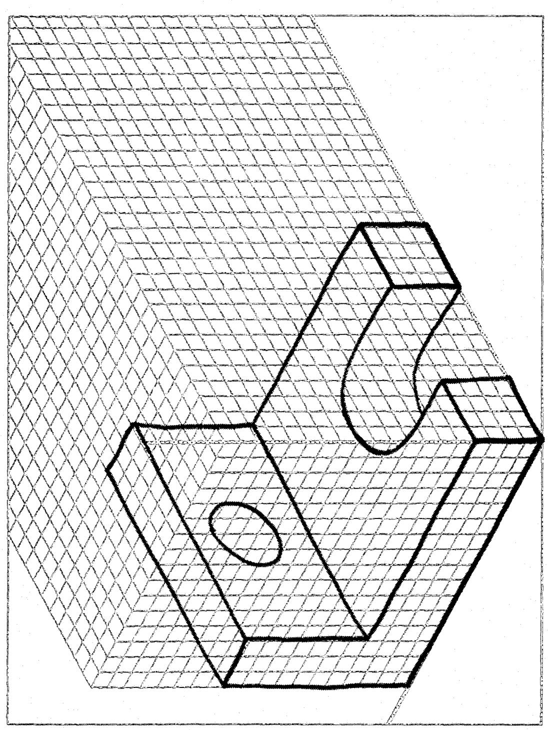
Exercise 9.8 Sketch an oblique cabinet view of the part (use 45°), and complete a two-view drawing project at two times the book scale.



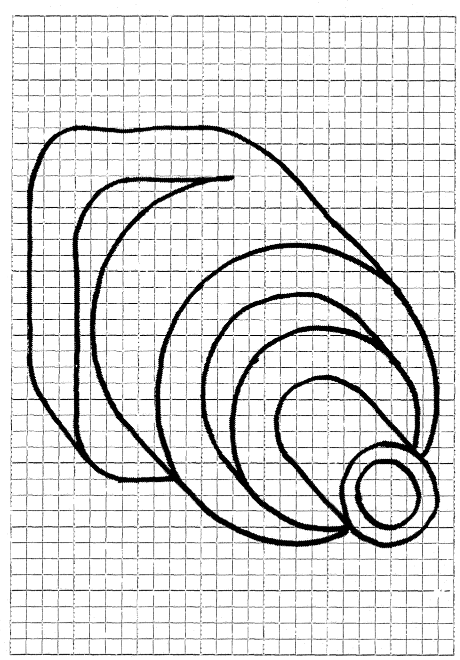
EXERCISE 9.5



EXERCISE 9.7



EXERCISE 9.6

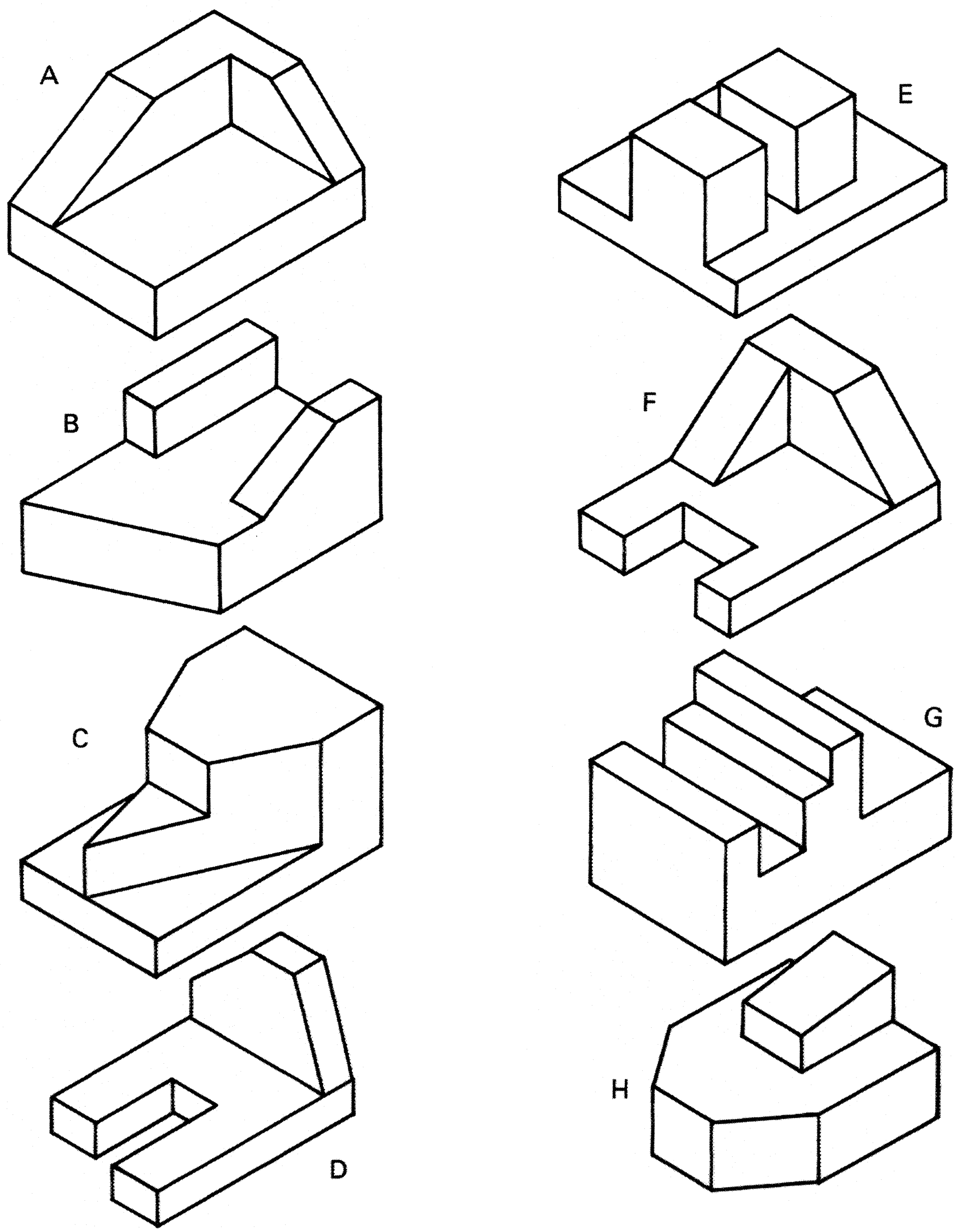


EXERCISE 9.8

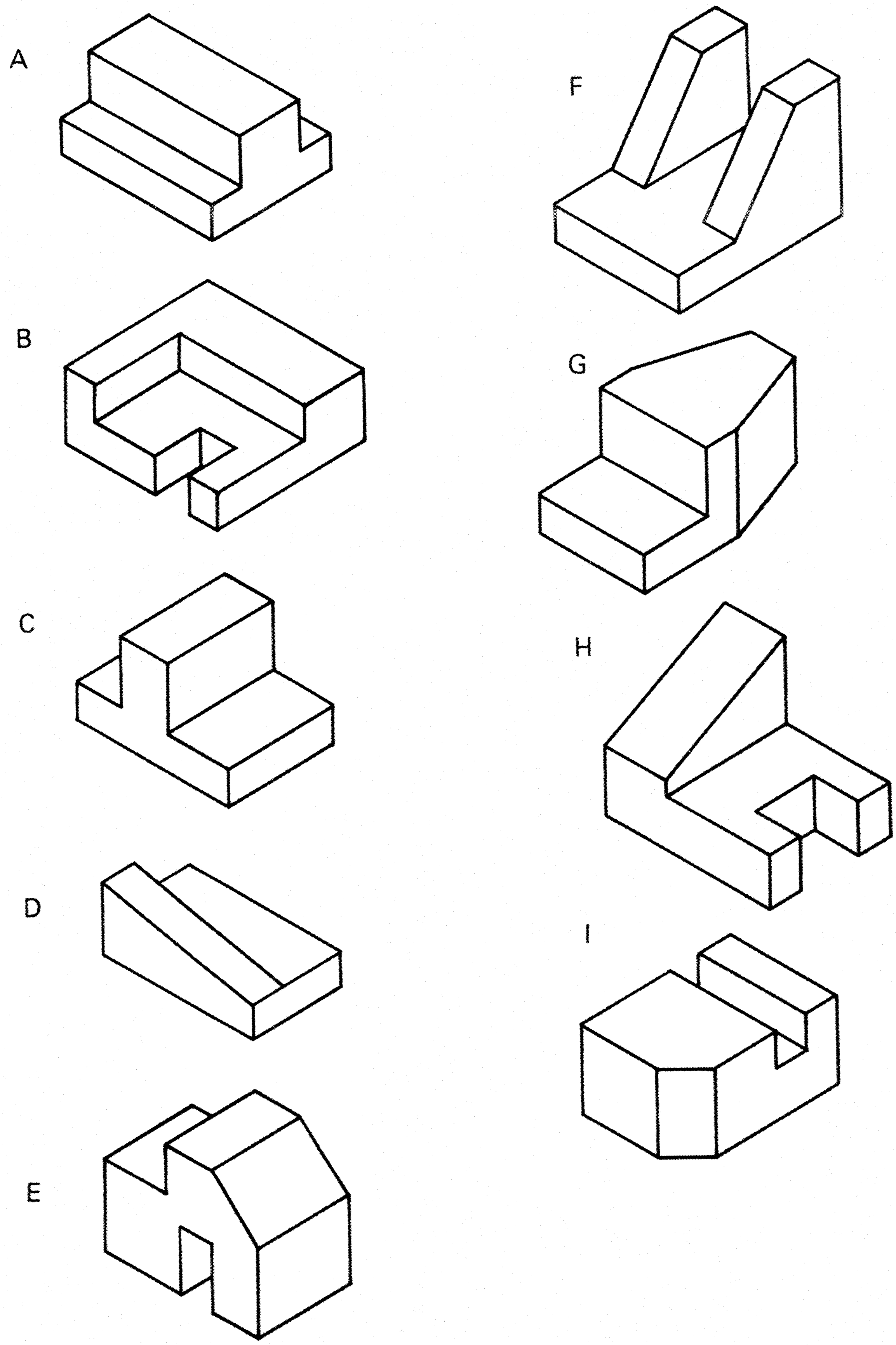
PROBLEMS

Problems 9.1(A) through (H) Freehand sketch the assigned problems using multiview projection. Draw the projects at two times the book scale. Problems can be drawn using any units type. Two or three views may be required for the problem.

Problems 9.2(A) through (I) Same as Problem 9.1.

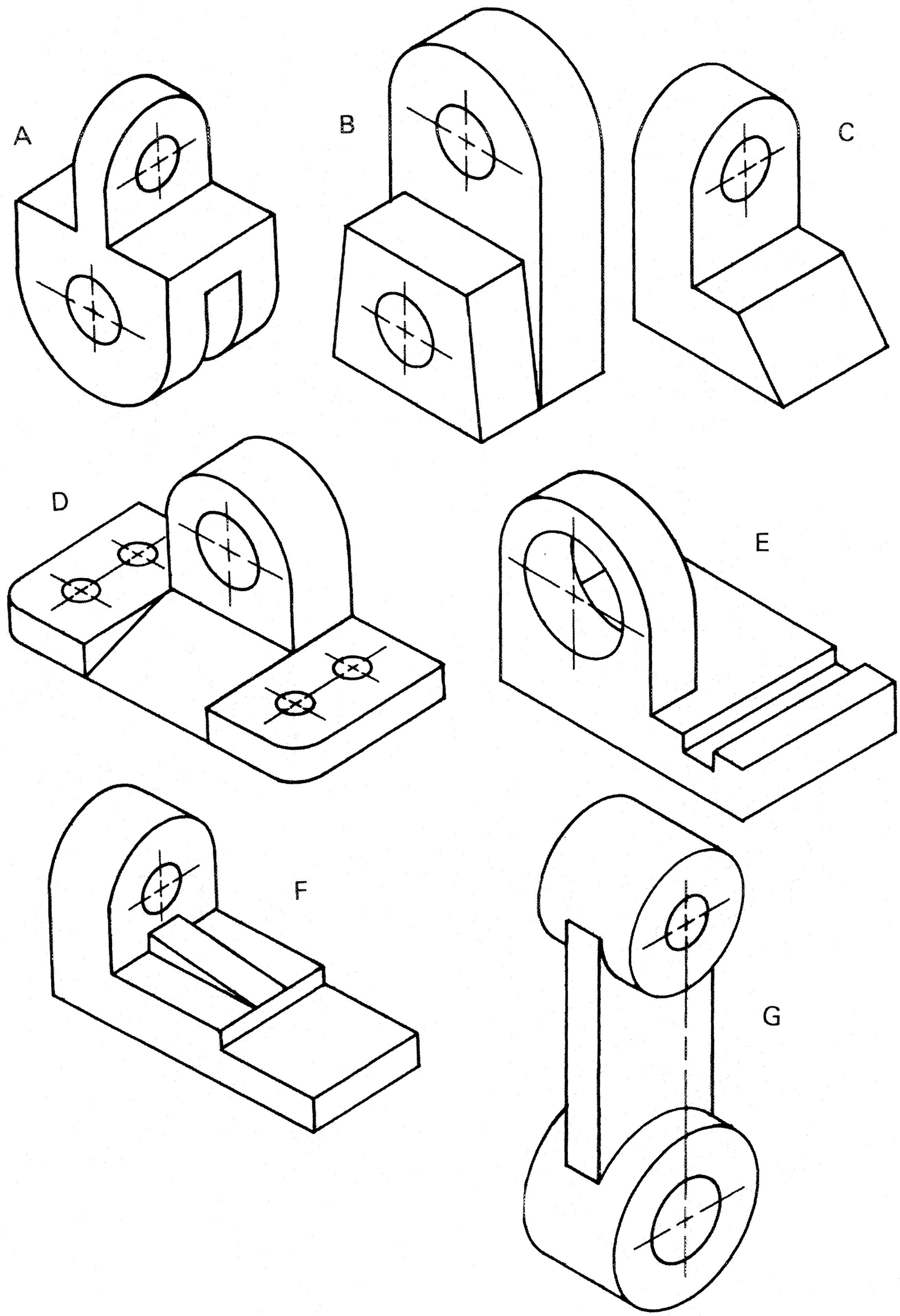


PROBLEMS 9.1(A) THROUGH (H)



PROBLEMS 9.2(A) THROUGH (H)

Problem 9.3(A) through (G) Using freehand sketching, draw each of the assigned problems. Use oblique projection. Be careful to choose the proper surface for the front face of the part. Draw at two times book scale.

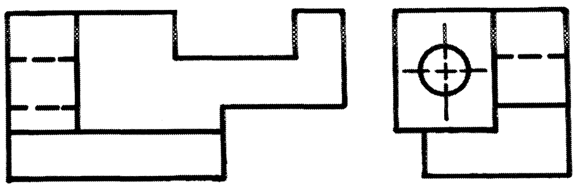


PROBLEMS 9.3(A) THROUGH (G)

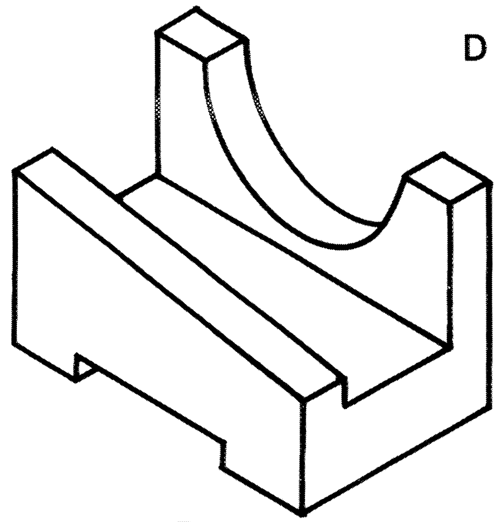
Problems 9.4(A) through (C) Complete the three views of each problem. On the same sheet sketch an isometric view. Draw at two times book scale.

Problems 9.4(D) through (F) Sketch three views of each problem. Draw at two times the book scale.

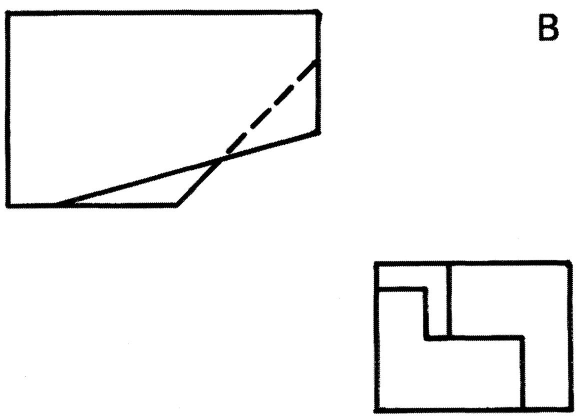
A



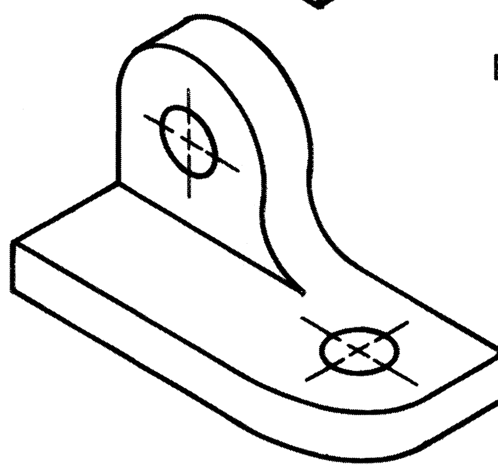
D



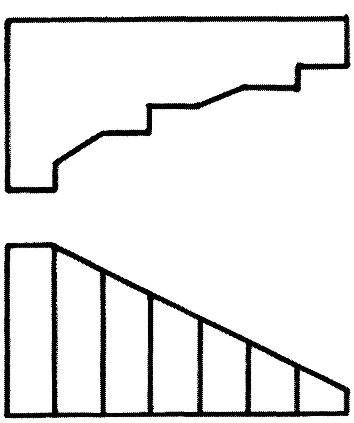
B



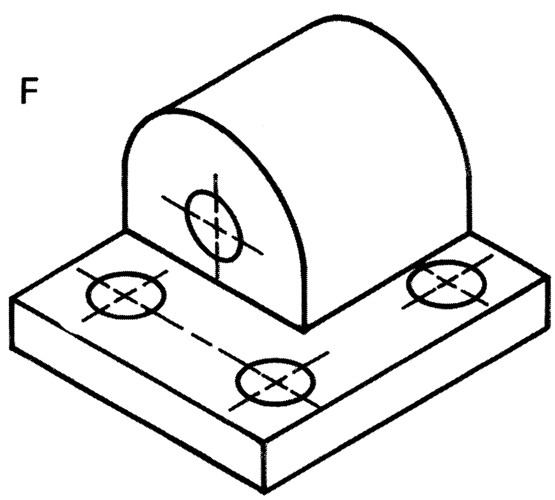
E



C

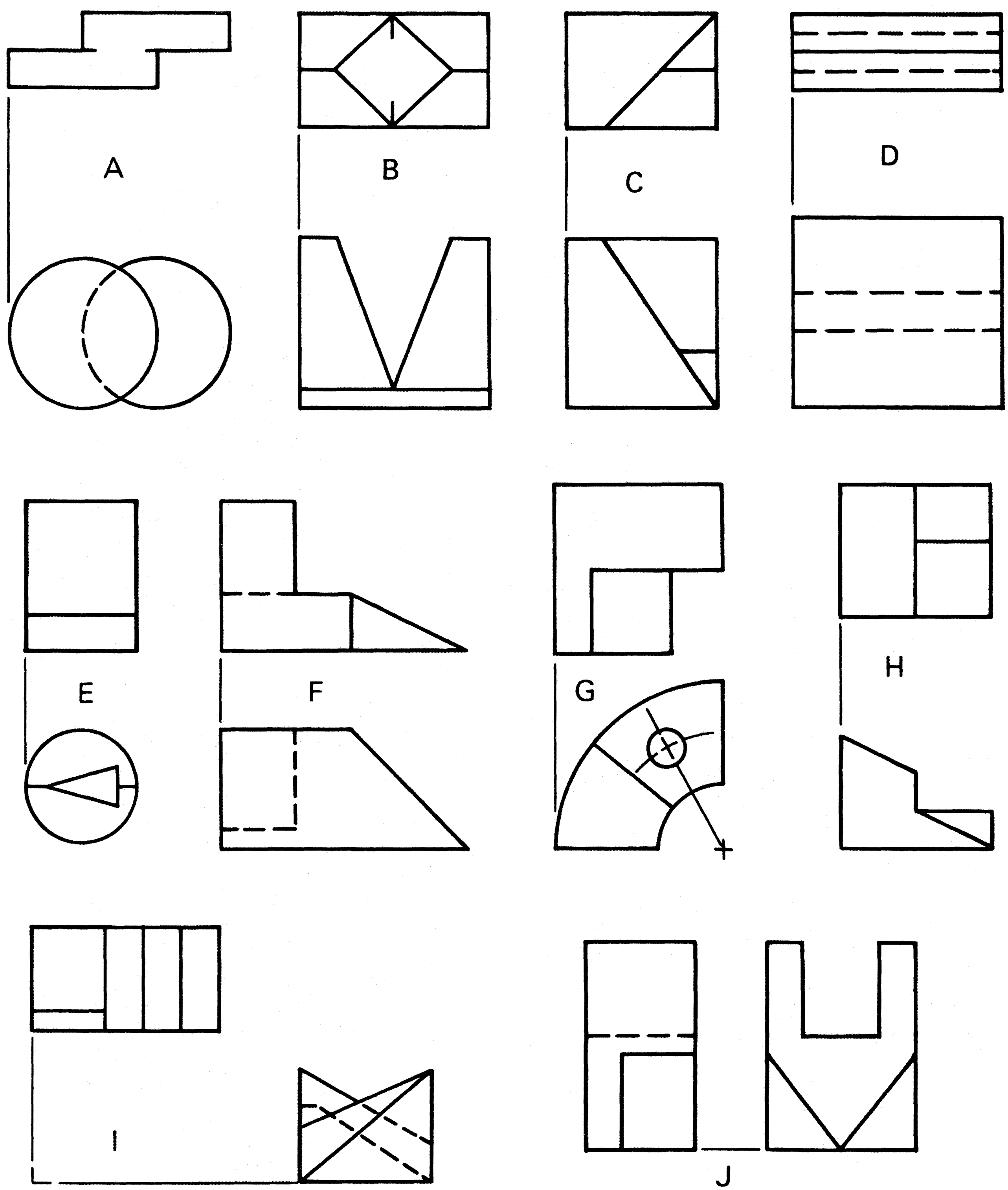


F



PROBLEMS 9.4(A) THROUGH (F)

Problem 9.5(A) through (J) Complete the given views and project a third view of each problem. Do an isometric sketch of each problem on a separate sheet of paper. Draw at three times the book scale. The isometric sketch will help you solve for the three views of the part.



PROBLEMS 9.5(A) THROUGH (J)