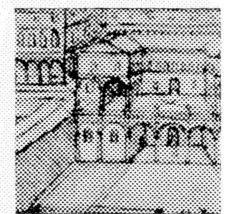
PICTORIALS: 3D REPRESENTATIONS AND 3D MODELING

Chapter 13



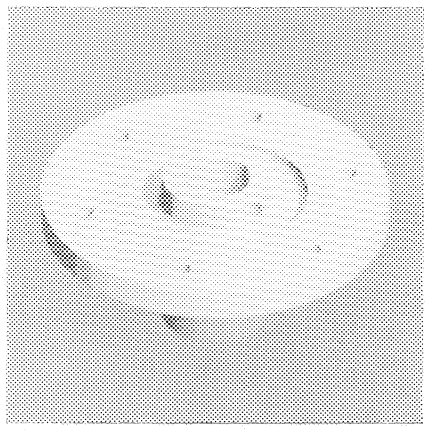
LEARNING OBJECTIVES

Upon completion of this chapter you will be able to:

1. Understand the difference between 3D representations and 3D modeling.
2. Recognize pictorial drawings as single-plane projections representing 3D shapes.
3. Understand the ways in which pictorial drawings may be most useful.
4. Understand and apply the functions of hidden lines, centerlines, and techniques for dimensioning on pictorials.
5. Become familiar with drafting conventions for illustrating certain part features pictorially.
6. Understand the basic capabilities and methodology of 3D modeling.
7. Recognize the three-dimensional modeling capabilities of CAD as applied to pictorials.
8. Define and be able to produce axonometric, oblique, and perspective drawings.

13.1 INTRODUCTION

Pictorial drawing is the oldest written method of communication known, but the character of pictorial drawing has continually changed with the advance of civilization. In this chapter, the types of pictorials commonly employed by engineers, designers, and illustrators are described. Pictorial drawings (whether manual or CAD) are single-plane projections. In other words, they present three primary surfaces to the viewer at the same time. Pictorials created on 3D CAD systems are actual to-scale **models** of a part, a structure, or some other object. The part in Figure 13.1(a) is a 3D shaded model of the flange bracket detail shown in Figure 13.1(b). The shaded pictorial was generated from the part database and displayed as a pictorial via a rendering program. The part was modeled and detailed with AutoCAD. A manual technical illustration is shown in Figure 13.2. Here, the exploded assembly pictorial was drawn by hand and inked with technical pens.



(a) Shaded image of a 3D model

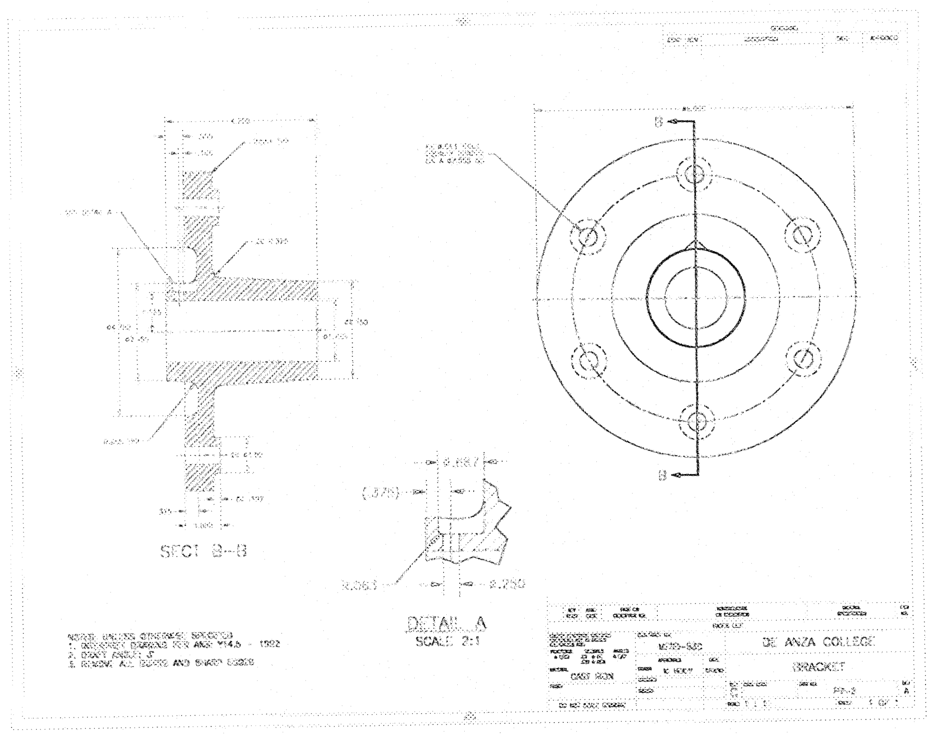


FIGURE 13.1 Bracket (b) Detail drawing generated from the 3D model

Pictorials are useful in design, construction or production, erection or assembly, service or repairs, and sales. Pictorial sketching (Figure 13.3) was discussed in Chapter 9, where the use of pictorials was limited to the initial stages of the design process. Engineers and designers use pictorial sketches to refine and communicate 3D designs before they are formally drawn or modeled.

The choice of pictorial drawing is dependent on its intended application. Pictorials are used in a variety of ways throughout industry and business.

Uses of Pictorial Drawings

- To *explain* complicated engineering drawings to people who are not trained or who lack the time to read the conventional multiview drawings
- As a *supplement* to the engineering detail [Fig. 13.1(a)]
- To *train* new shop employees with the aid of illustrated training manuals
- To *speed up* and clarify the *assembly* of a machine
- For *ordering* new parts, as in parts catalogs and service manuals (Fig.13.2)

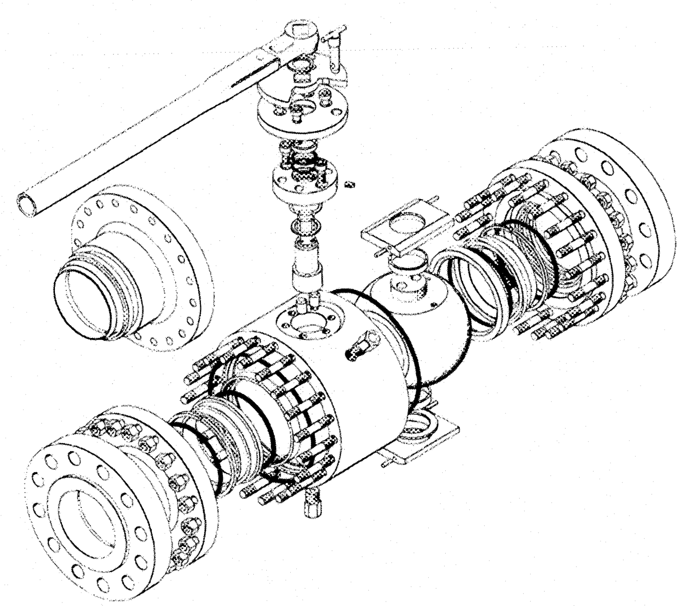


FIGURE 13.2 Technical Illustration of a Ball Valve from a Sales Catalog

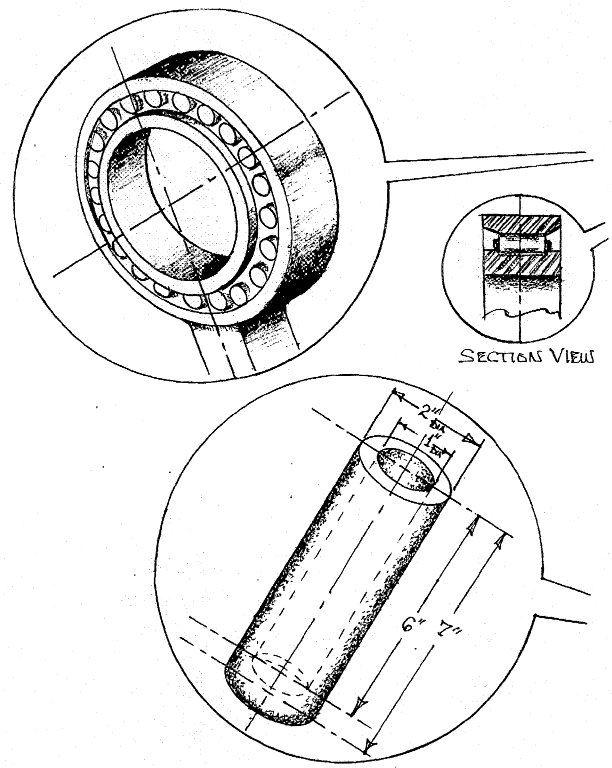


FIGURE 13.3 Pictorial Sketch

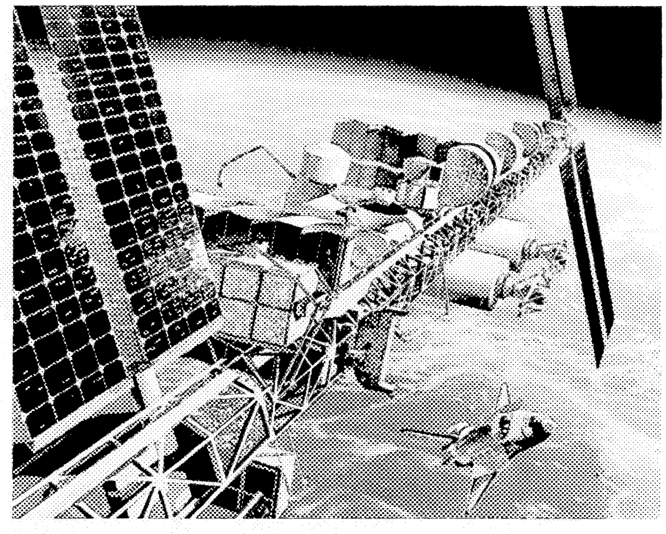


FIGURE 13.4 Space Station Solar Panels and Support System

- As design *sketches* to clarify a concept or process (Fig. 13.3)
- To *transmit ideas* from person to person, from shop to shop, or from sales to purchasing
- To *display* futuristic *designs* or concepts (Fig. 13.4)
- As an *educational aid* in developing visualization.
- To *present new* product *designs* (Fig.13.5)
- To *render* a product, project, or display [Fig. 13.6(a)–(c)]
- To *help* the designer or engineer *work out problems in 3D space,* such as clearances, intersections, interferences, and routing [Fig. 13.7(a)–(e)]
- To *design and modify complex subassemblies and assemblies* [Fig. 13.8(a) and (b)]

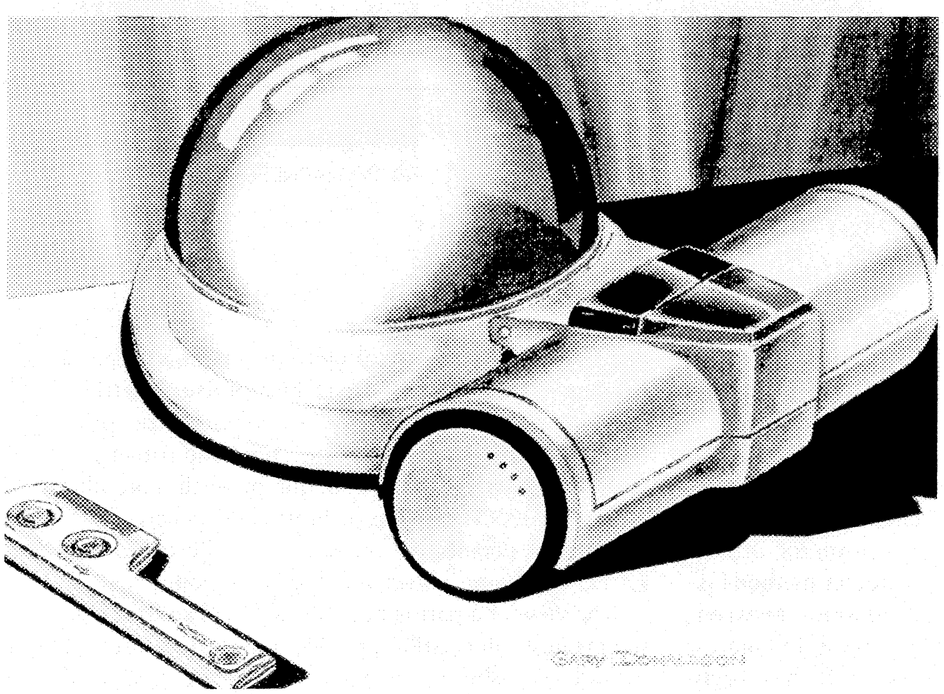
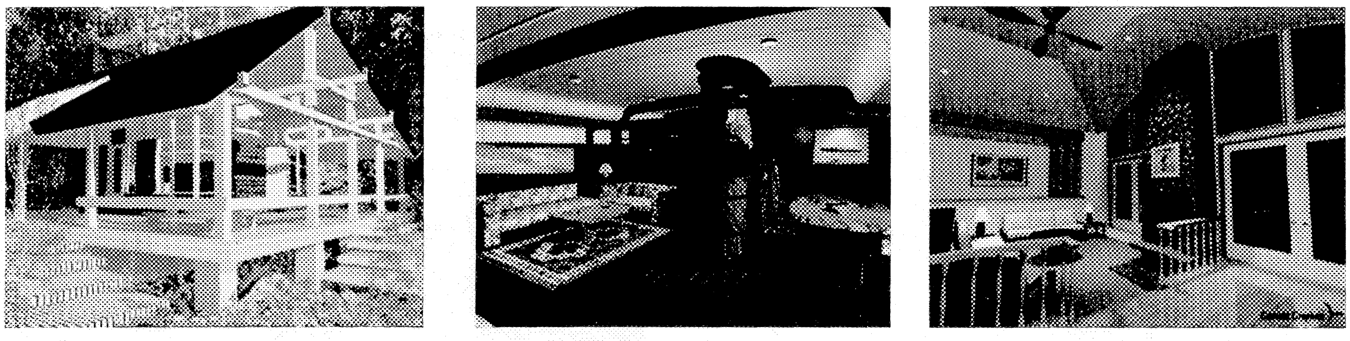
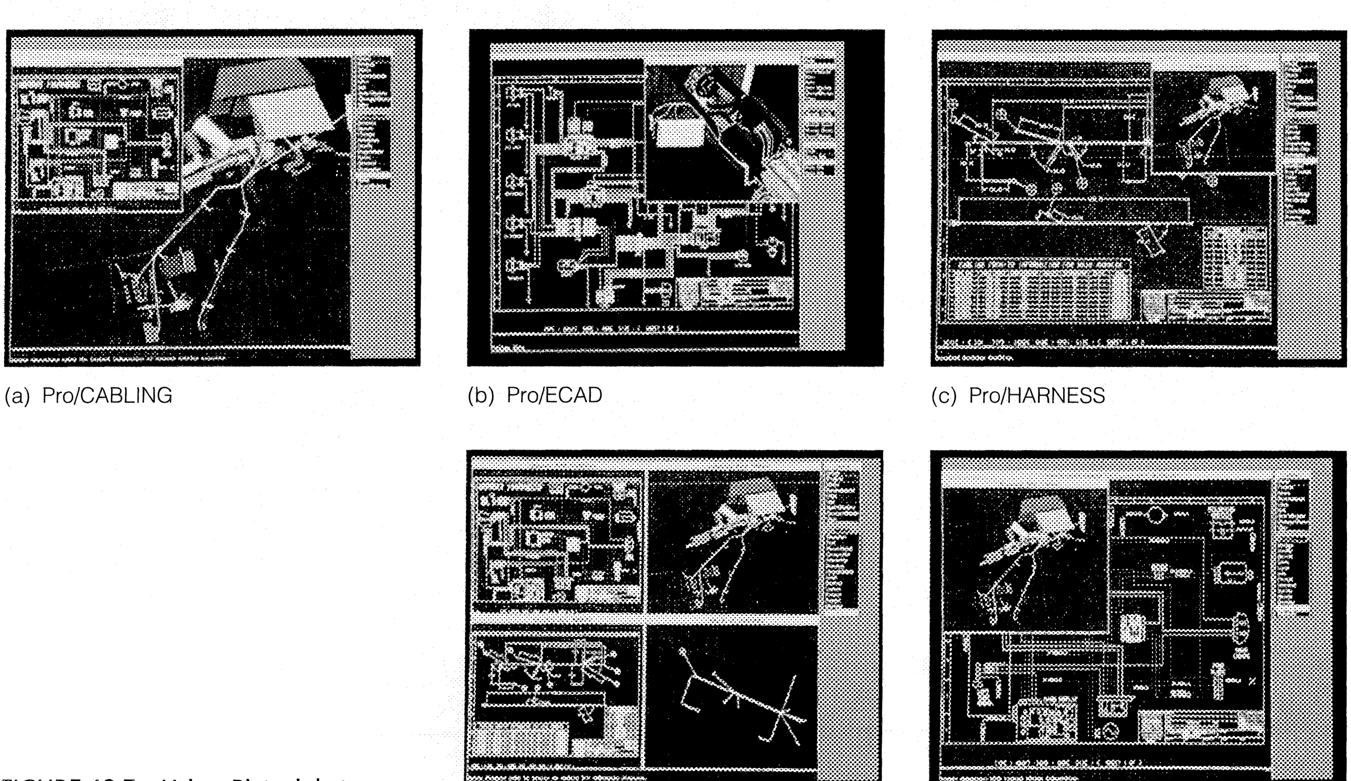


FIGURE 13.5 Research and Development Illustration of a New Product



(a) Sauna (b) Yacht interior (c) Sunroom

FIGURE 13.6 Room Interiors



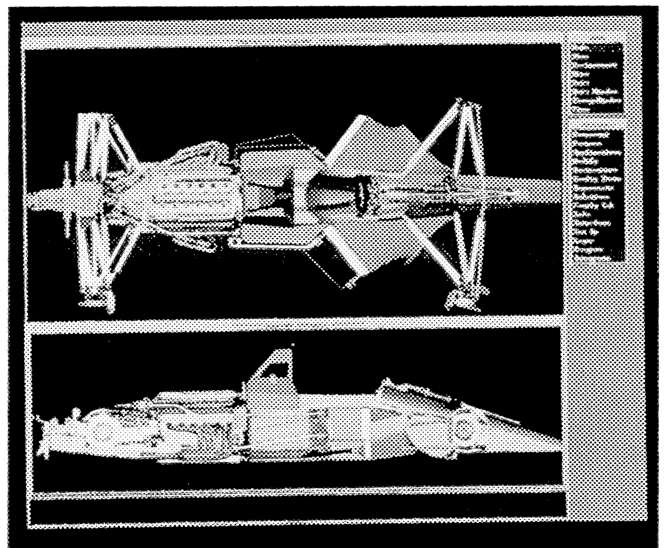
(a) Pro/CABLING (b) Pro/ECAD (c) Pro/HARNESS (d) Pro/HARNESS (e) Pro/HARNESS

FIGURE 13.7 Using Pictorials to Enhance 3D Design

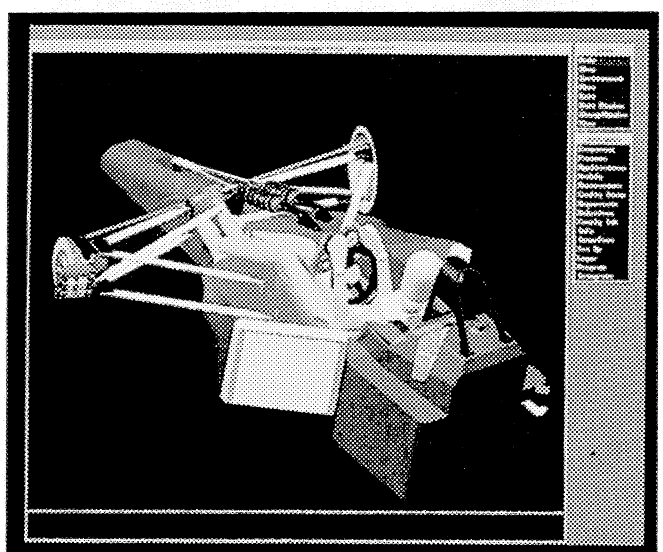
13.2 TYPES OF PICTORIAL DRAWING

Pictorials can be divided into three general projection methods: axonometric, oblique, and perspective. These differ from each other in the fundamental scheme of projection (Fig. 13.9). Axonometric projection is a form of orthographic projection. Each of the three projection methods is subdivided by varying some of the relationships between point of sight, plane of projection, and the object. In Figures 13.10, 13.11, and 13.12, the same assembly has been displayed by each of the projection types and their accompanying versions. The four versions of axonometric projection are illustrated in Figure 13.10: isometric projection, isometric drawing, dimetric projection, and trimetric projection. The three versions of oblique projection are illustrated in Figure 13.11: cavalier, cabinet, and general. The three versions of perspective projection are illustrated in Figure 13.12: one-point, two-point, and three-point.

The view of a part is normally selected so that it will give the greatest information possible, unless its natural position or its relationship to other parts must take precedence.



(a) Two views of Indy car



(b) Internal view of Indy car

FIGURE 13.8 Indy Car Design

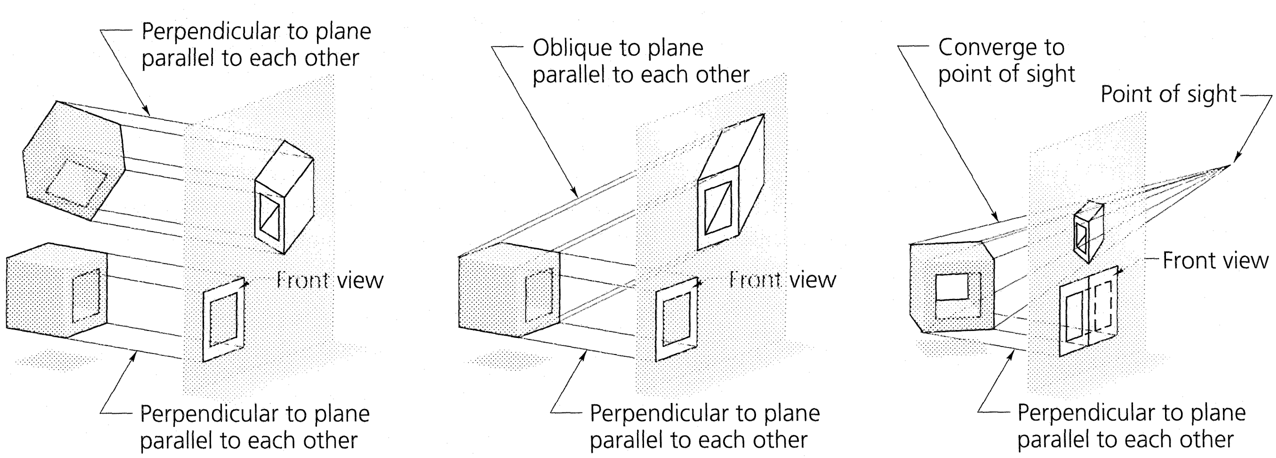
13.3 AXONOMETRIC PROJECTION

A projected view in which the lines of sight are perpendicular to the plane of projection but in which the three faces of a rectangular object are all inclined to the plane of projection is called an **axonometric projection.** The projections of the three principal axes may make any angle with each other except 90°. Three types of axonometric projection are used: isometric, dimetric, and trimetric. Isometric is the most common.

An **isometric projection** is a pictorial drawing in which the three principal faces and the three principal axes of the object are inclined equally to the plane of projection. The plane of projection is called the *isometric plane.* The three axes on the drawing also make equal angles with each other [Fig. 13.13(a)], but may be placed in a variety of positions. A true orthographic projection of an object on the isometric plane is an isometric projection. The scales on all three axes are equal and foreshortened in the ratio of approximately 0.8 to 1.0. The term *axes* refers to the projections of the principal axes, unless otherwise stated.

A **dimetric projection** [Fig. 13.13(b)] is drawn with two axes making equal angles and the third axis at any selected angle. A **trimetric projection** uses three different scales (one for each axis) and has three different angles for the axes [Fig. 13.13(c)]. Trimetric projection is the most lifelike method, but it is also the most time-consuming and difficult to draw.

Since most pictorials are drawn with isometric projection methods, the following discussion will concentrate on this type. Dimetric projection and trimetric projection are completed with the same general layout procedures as isometric projection. Therefore, the following techniques will apply for all three types.



(a) Axonometric (b) Oblique (c) Perspective

FIGURE 13.9 Kinds of Projection

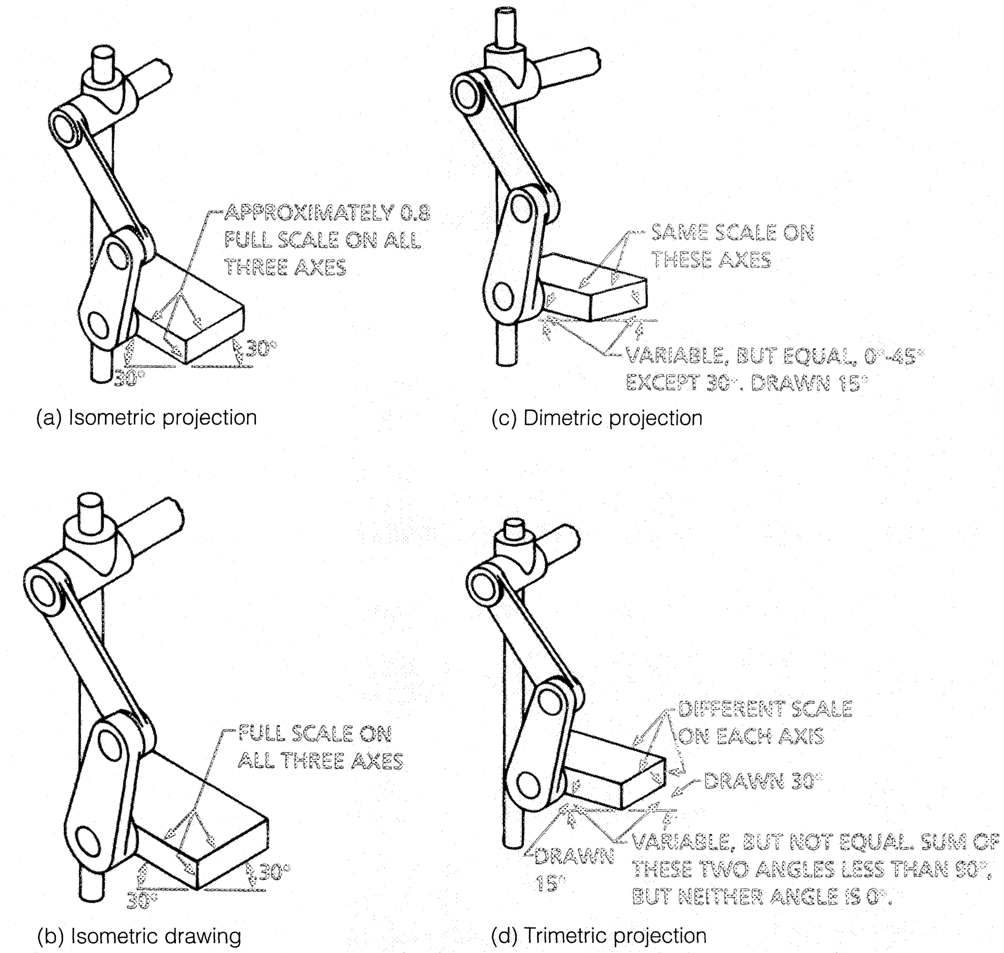


FIGURE 13.10 Types of Axonometric Projection

13.3.1 Isometric Drawings

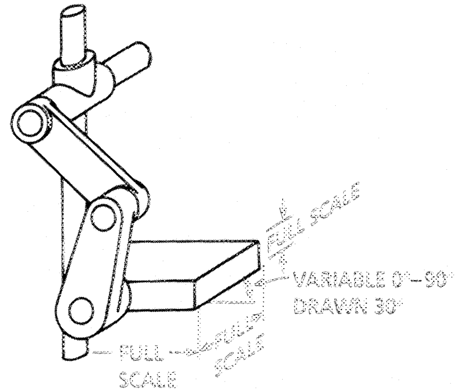
For **isometric drawing,** the distances on each axis are measured true length, with any standard scale, thus making a drawing larger than isometric projection (which is normally 81% of the original in size). Isometric drawing is the form in which the isometric technique is most commonly used. Isometric projection and isometric drawing are both based on the assumption that a cube representing the projection axes will be rotated until its front face is 45° to the horizontal plane and then tipped forward or downward at an angle of 35°16′. The axes make equal angles of 120° to one another [Fig. 13.14(a) and (b)]. The resulting rotation displays all three primary surfaces equally. Figure 13.14(d), (e), and (f) show the isometric cube in three different orientations. All three axes make equal angles with the projection plane and can be drawn easily using 30°/60° triangles [Fig. 13.14(c)]. The three faces of the cube are identical in size and shape. The projected lengths of each edge are not foreshortened.

Because isometric drawings are constructed along the three axes (one vertical and the other two at 30° to the horizontal to the right and left), each dimension is measured true length (not foreshortened) along an axis. *All lines in isometric drawings that are on or parallel to the three axes are drawn true length.* Lines not on or parallel to the axes are constructed with *offset dimensions.*

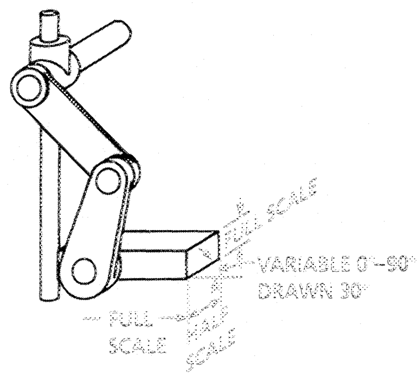
The orientation of the axis determines which faces of the part are visible. Figure 13.14 shows three of many alternatives for placing the axes. The most typical orientation is Figure 13.14(d), where the cube is viewed so that the top, front, and side of the object are visible. In Figure 13.14(e), the axes have been turned on their side, and in (f), the bottom, front, and side are visible because the axes are shown from beneath the isometric cube. A variety of arrangements are possible for the isometric axes as long as they remain at 120° to one another.

13.3.2 Isometric Construction

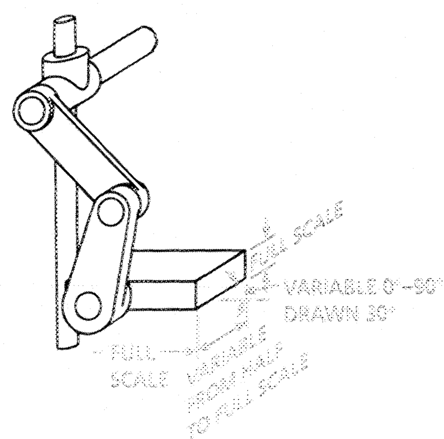
Isometric construction using the box method is illustrated in Figure 13.15. The three axes are drawn first: one vertical, one at 30° receding to the right, and one at 30° receding to the left [Fig. 13.15(b)]. The edges of the box are constructed from the height, width, and depth dimensions transferred from the multiview drawing of the part [Fig. 13.15(a)].



(a) Oblique projection (cavalier)

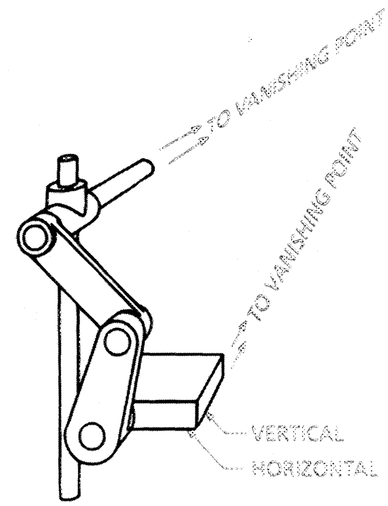


(b) Oblique projection (cabinet)

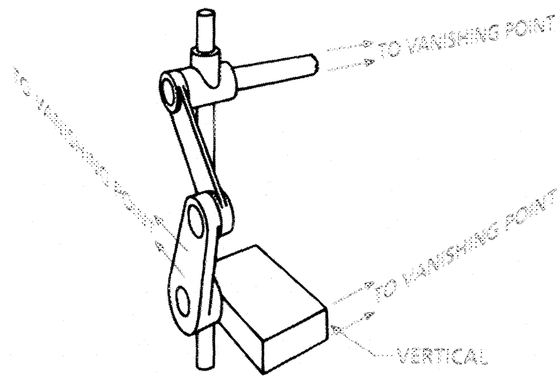


(c) Oblique projection (general)

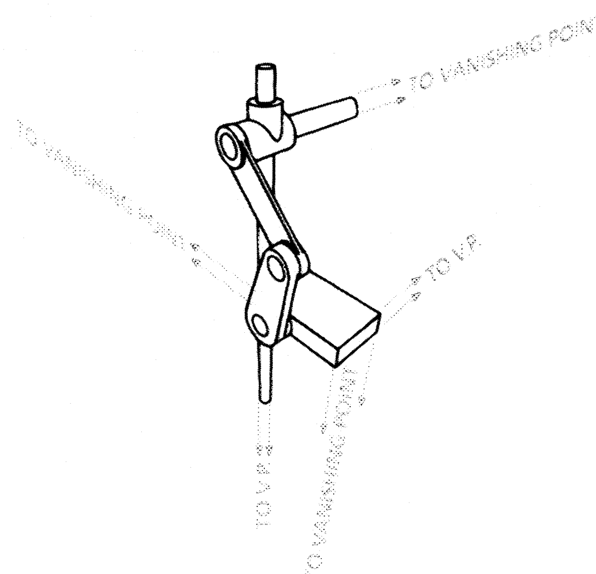
FIGURE 13.11 Types of Oblique Projection



(a) Perspective (one-point)



(b) Perspective (two-point)



(c) Perspective (three-point)

FIGURE 13.12 Types of Perspective Projection

Remember, in an isometric drawing the dimensions are not foreshortened. Therefore, if the distance is on or parallel to one of the axes, each measurement is full scale from the multiview projection (or directly from the part). Dimensions can be marked off with a scale or transferred with dividers.

All lines on isometric drawings that are parallel to or on one of the three axes are true length. This type of line is called an **isometric line.** Lines that are not parallel to or on an axis, called **nonisometric lines,** will not show true length

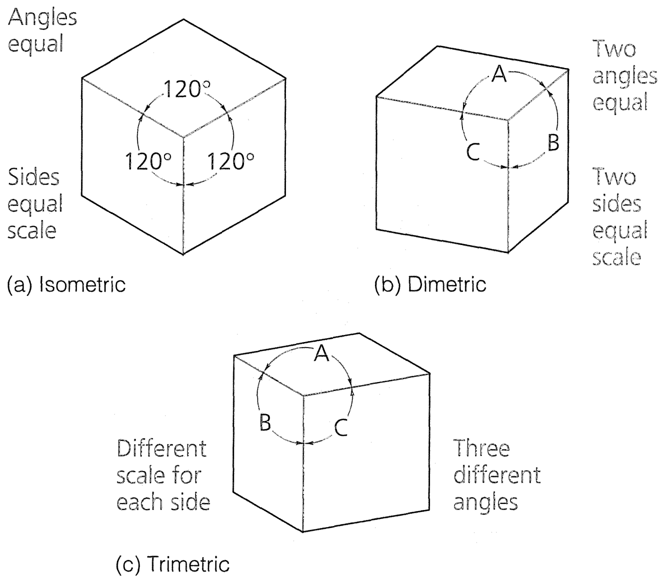


FIGURE 13.13 Axonometric Axes

on the isometric drawing. *Nonisometric lines must be established from their endpoints, located along isometric lines.*

After the part is boxed in, the remainder of the drawing is completed. Dimensions A, B, C, and D are taken from the multiview drawing of the part in Figure 13.15(a) and transferred to the isometric box [Figure 13.15(c)]. This establishes the steplike features of the part. The remaining features are created in the same manner [Fig. 13.15(d)]. After the part is complete, all axes and construction lines are erased and the object lines darkened [Fig. 13.15(e)].

The procedure of "blocking in" a part is similar to carving a model out of some soft material with a knife. One aid in this process is isometric grids (Fig. 13.16). In the figure, the part has been drawn in three views on grid paper and then transferred to the isometric grid. Since the part's features all fall on grid lines, no measurements are necessary to complete the project. Transferring the part from the three-view drawing to the isometric drawing involves simply counting grid lines.

Figure 13.17(a) shows a three-view drawing of an integrated circuit, complete with dimensions. Figure 13.17(b) provides the steps for blocking in and drawing an isometric pictorial of the integrated circuit.

13.3.3 Nonisometric Lines

The two lines that make the V-shaped feature in Figure 13.18 are not parallel to one of the three axes. Therefore, these lines are **nonisometric lines.** Nonisometric lines cannot be scaled, but their endpoints are easily located through the box method and offset dimensions. Depending on their orientation, nonisometric lines may become longer or shorter on the isometric drawing. In this figure, the nonisometric lines are at the same angle, but slanting from different directions. In the isometric view [Figure 13.18(d)], the two lines now make different angles. Also, one is longer than the original line and the other is shorter. This distortion is typical of nonisometric lines.

In Figure 13.18, using the part's overall dimensions, the isometric box is drawn first in (b). Using dimensions transferred from Figure 13.18(a), the primary features are then "carved" [Figure 13.138(c).] Nonisometric lines are established by locating their endpoints using offset dimensions [Fig. 13.18(d)]. The angled (nonisometric lines) features are established using offset dimensions I and J. In Figure 13.18(e), construction lines have been removed and the part darkened.

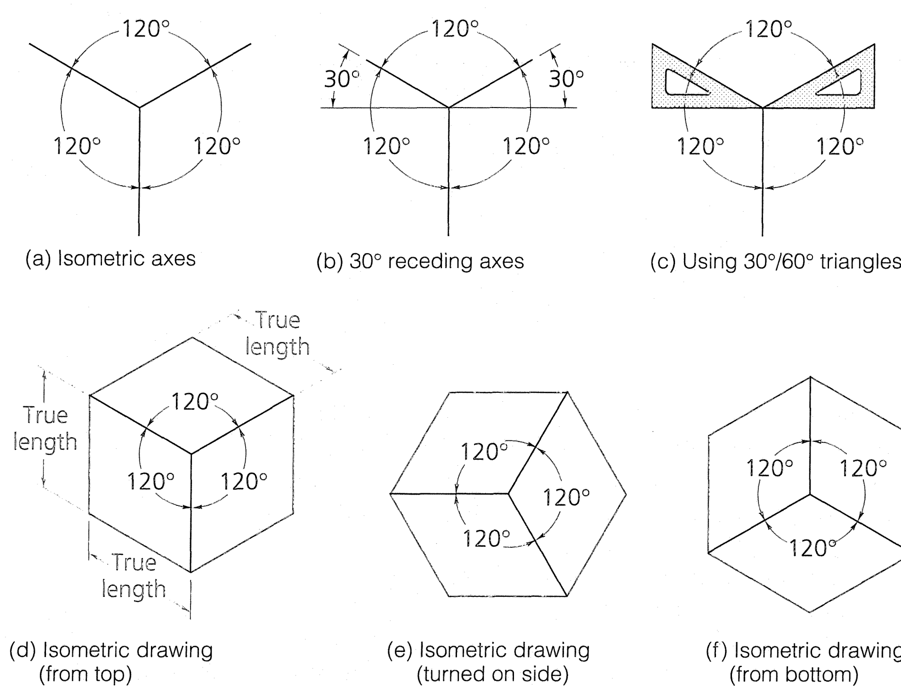


FIGURE 13.14 Isometric Axes

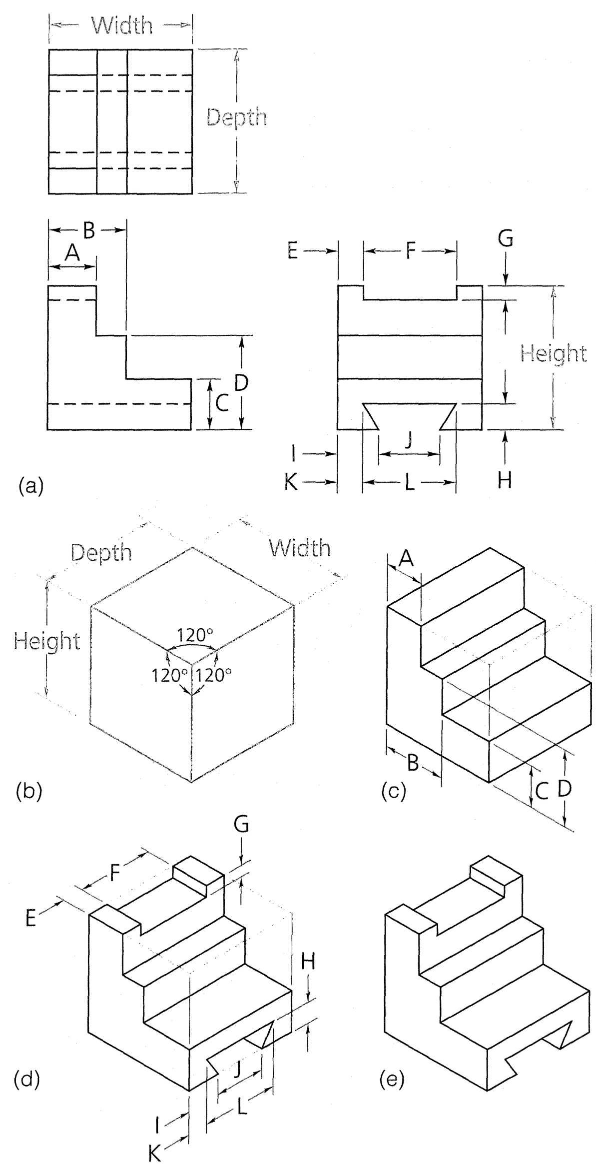


FIGURE 13.15 **Isometric Construction Using the Box Method**

13.3.4 Isometric Angles

The three major axes along an isometric box (or cube) are at 120° angles to one another. In reality, all lines of a cube are at 90° or are parallel to each other. Because of the distortion created by the isometric view of the box, few angles appear as true angles. Angles, as with nonisometric lines, must be established by means of offset dimensions. Angles appear larger or smaller than true size on isometric drawings in relation to their position in the view. The lines that make an angle are nonisometric lines. Angles cannot be measured from the multiview drawing and transferred directly to the isometric view. Instead, they must be drawn by locating their endpoints along isometric lines using offset dimensions, as when drawing nonisometric lines.

The block in Figure 13.19 has an angled surface. To draw the part in isometric, it is necessary to use dimensions A, B, and C. These dimensions can be taken along true-length lines. Points 1, 2, 3, and 4 are established in the isometric view using these dimensions. The angle is not true size (it is smaller than the original) in the isometric view. The steps in this figure were the same ones used to draw a part containing nonisometric lines with the box method and offset dimensions. Remember, **offset dimensions** are always taken parallel to one of the three axes or along isometric lines.

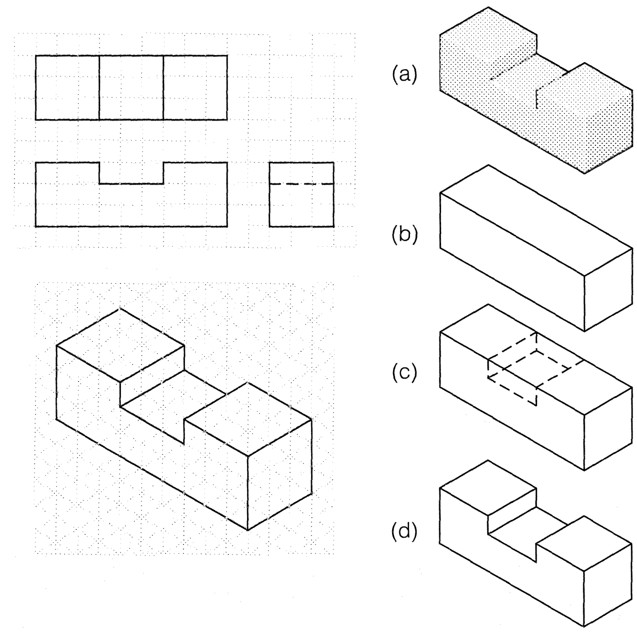


FIGURE 13.16 **Box Construction and Grids for Isometric Drawings**

13.3.5 Irregular Objects on Isometric Drawings

Any shape can be drawn isometrically with the box method and offset dimensions. In Figure 13.20, a pyramid has been drawn this way. Given the three-view drawing of the part in Figure 13.20(a), the isometric drawing is started as in the previous examples. The isometric box is drawn with the part's three primary dimensions [Fig. 13.20(b)]. Using offset dimensions A and B, the base is established first (points 1, 2, and 3). Point O is located with offset dimensions C and D [Figure 13.20(c)].

13.3.6 Circles and Arcs on Isometric Drawings

All circles and circular arcs on isometric drawings appear elliptical. A variety of methods are available for isometric ellipse construction: template, trammel, four-center, and point plotting, to name a few. A template should be used for

(a) Three-view drawing

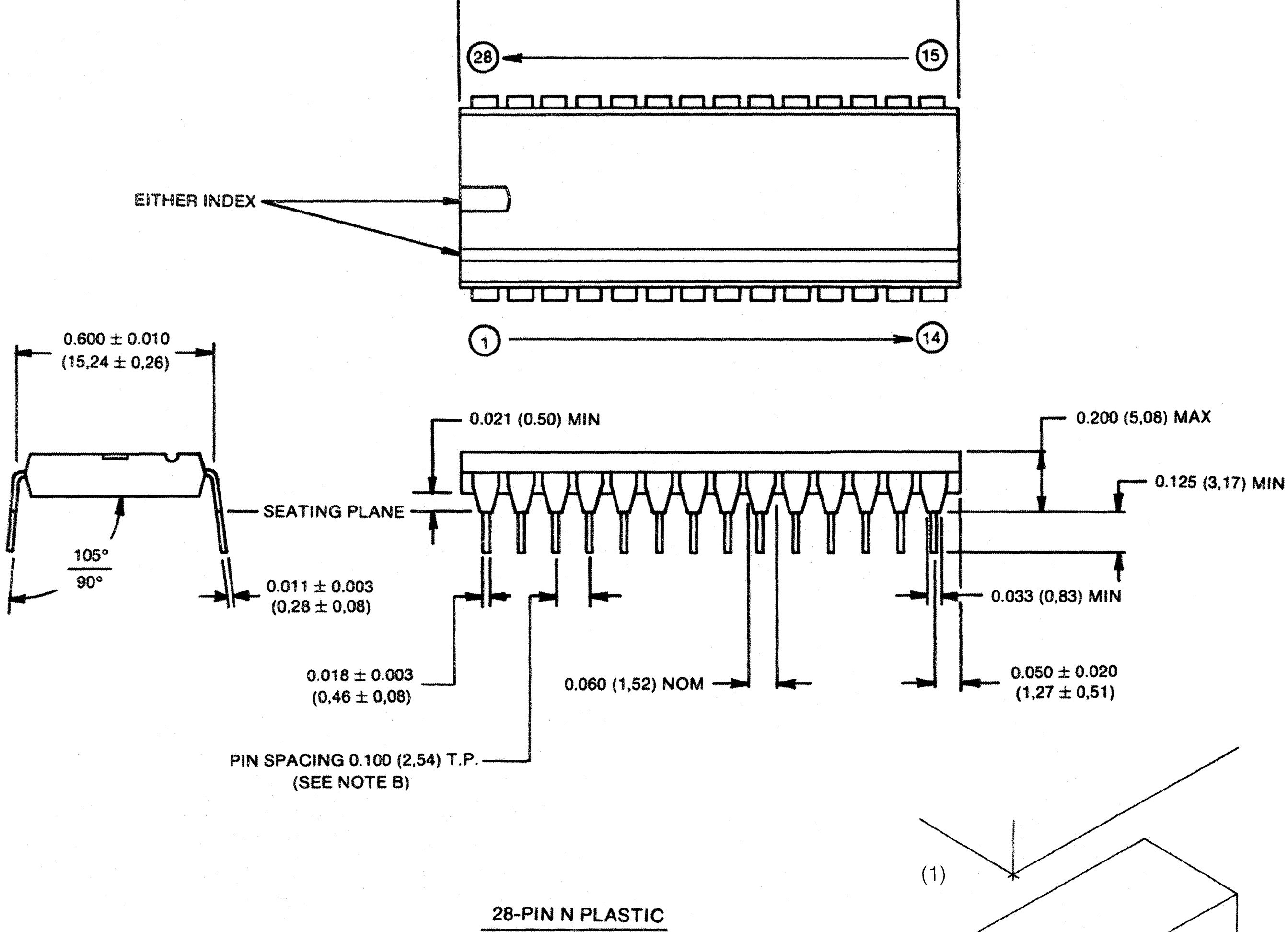
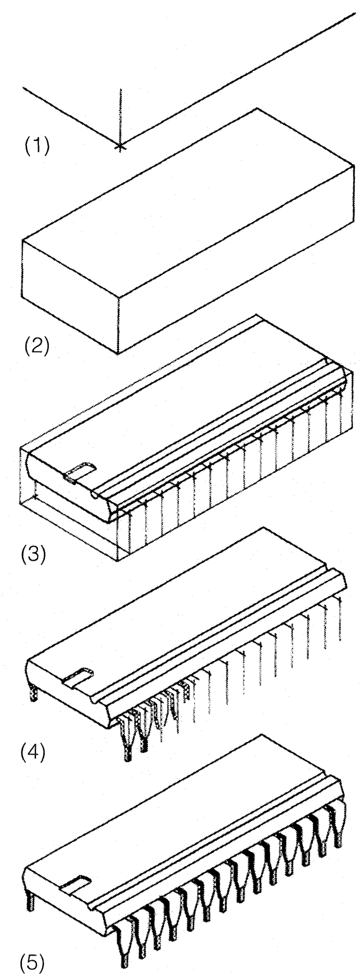


FIGURE 13.17 Integrated Circuit



(b) Isometric drawing

instrument drawings whenever the size of the ellipse can be matched with available equipment. For sketches, freehand techniques are normally sufficient.

If templates are not available, the trammel and point plotting methods are the most accurate, but they are time-consuming procedures for constructing circles and arcs in isometric drawings.

The **four-center method** shown in Figure 13.21 does not create a perfect ellipse, but is accurate enough for most purposes and for constructions that cannot be made with an ellipse template. This method can be employed to draw circles or portions of circles (arcs) on any isometric face/plane (Fig. 13.22). The following steps describe the construction of an isometric ellipse (Fig. 13.21).

1. Lines DA and DC are drawn along the two receding axes (at 30°). Line AB is parallel to DC, and line CB is parallel to AD. Each of the lines will be the same length as the diameter of the circle [Fig. 13.21(a)].

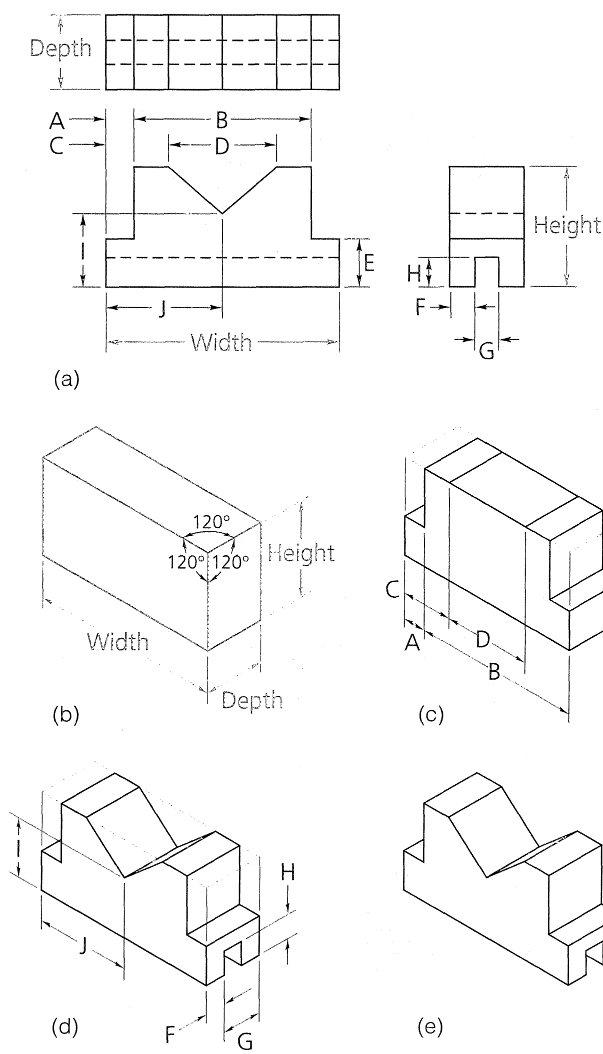


FIGURE 13.18 Nonisometric Lines in an Isometric Drawing

2. Construction lines are drawn from point D perpendicular to line AB at its midpoint and perpendicular to line CB at its V midpoint [Fig. 13.21(b)].
3. Step 2 is repeated for point B and lines DA and DC [Fig. 13.21(c)].
4. The intersection of the construction lines is used to draw R1. The radius is equal to the distance from the intersection of the construction lines to one of the numbered points (1, 2, 3, or 4). The radius will thus be tangent to two edge lines (AB and AD or CB and CD) [Fig. 13.21(d)].
5. Points D and B are used to draw arc R2. Arc R2 originates at the intersection of the construction lines for both sides of the ellipse (B or D). Radius R2 will be tangent to two sides each (BA and BC or DA and DC) [Fig. 13.21(e)].
6. Lastly, all construction lines are erased and the ellipse is darkened [Fig. 13.21(f)].

Portions of circles (arcs) are sometimes required for parts that have fillets and rounds. The same procedure for construction is used for these cases. In Figure 13.23, the round has been constructed as a portion of an isometric ellipse. In Figure 13.23(a) the round is shown as a true shape. The ellipse is boxed in using two times the radius as each side for construction lines in Figure 13.23(b), and the radius of the bend is located to establish the tangent points in (c). Since just one-quarter of the circle (one-quarter ellipse) is being drawn, only one radius is necessary [Fig. 13.23(d)]. Unless the ellipse is an odd size or very large, a template is normally used for this construction, as shown in Figure 13.23(e). When using a template to construct the ellipse, care must be taken. The major axis of the ellipse will be at 30° unless the curve falls in the top or bottom face of the part.

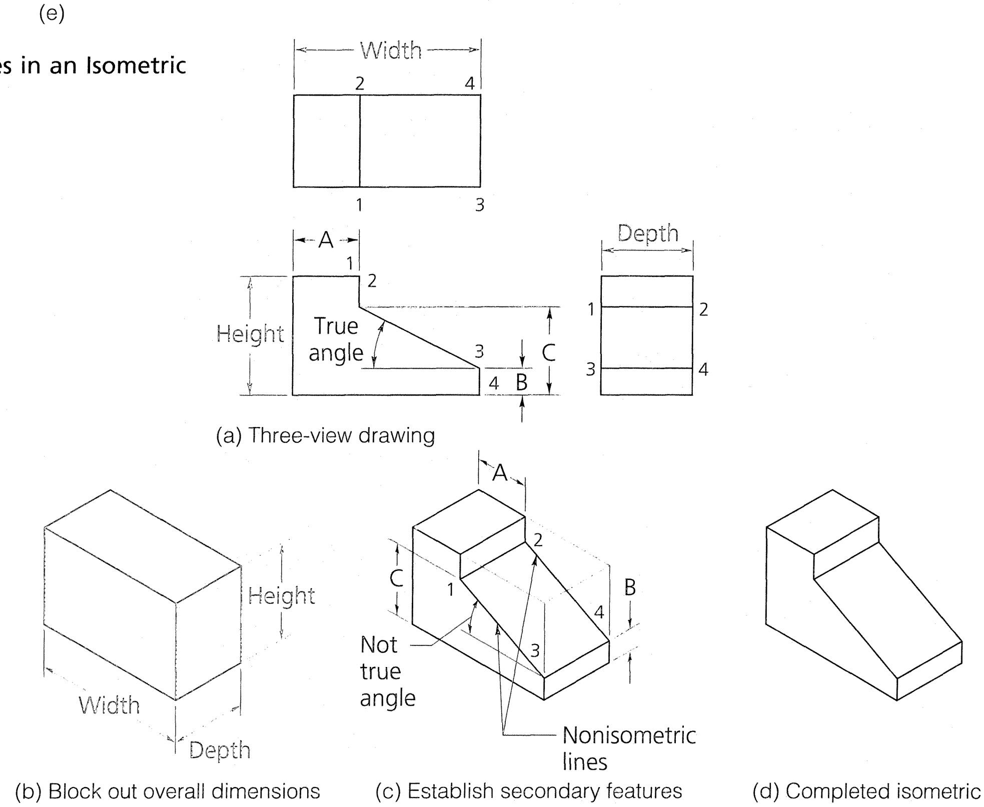
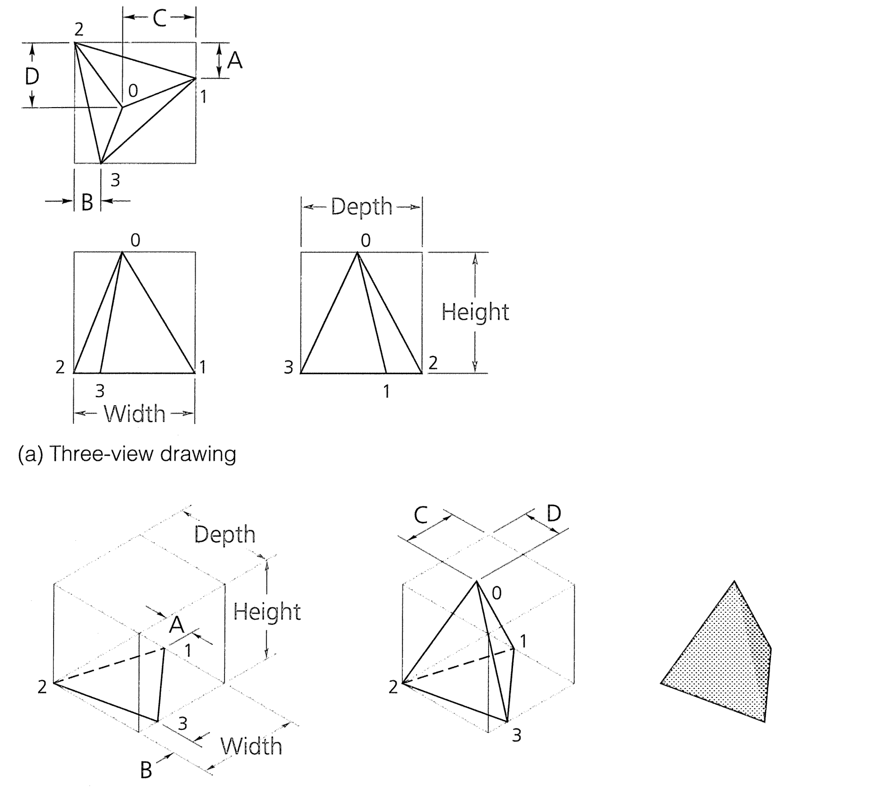


FIGURE 13.19 Offset Dimensions for Isometric Construction

.3.3.7 Using Offset Dimensions for Ellipse Construction

'he **offset dimension method** locates a series of points long the curve of an ellipse. This method is more accurate ıan the four-center method, but is time-consuming and the quality of the finished curve is dependent on the designer's skill with an irregular curve. Two versions of this method are given here. The first method (Fig. 13.24) divides the circle



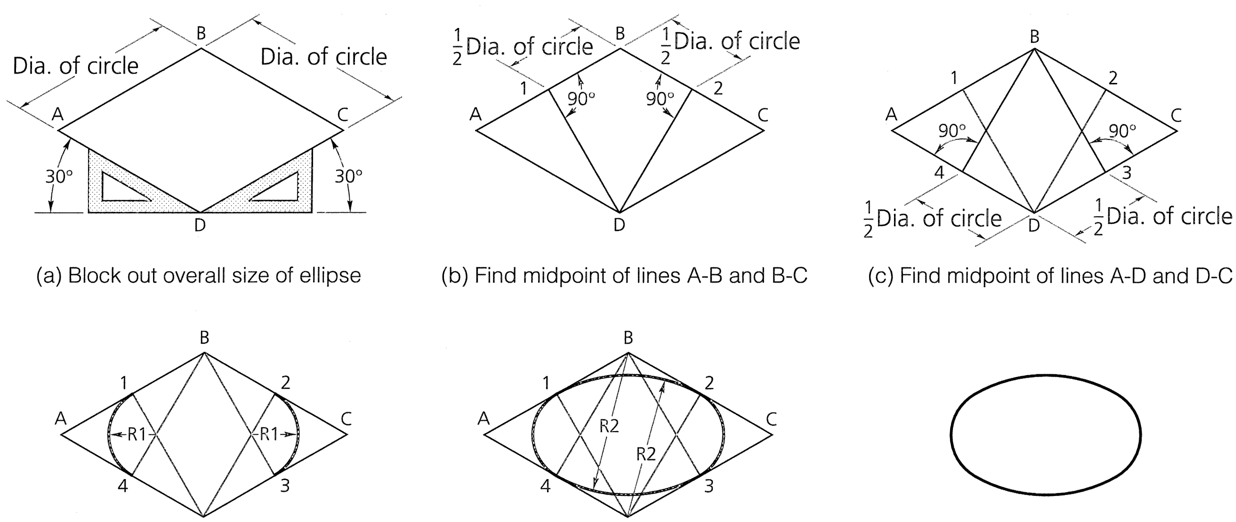
(a) Three-view drawing

(b) Block out height, width, depth, and base features

(c) Locate apex

(d) Darken in isometric

GURE 13.20 Construction of Irregular Objects in an Isometric Drawing



(a) Block out overall size of ellipse

(b) Find midpoint of lines A-B and B-C

(c) Find midpoint of lines A-D and D-C

(d) Swing R1 arcs

(e) Swing R2 arcs

(f) Erase construction lines and darken ellipse

GURE 13.21 Drawing a Four-Center Ellipse evenly by drawing equally spaced construction lines emanating from the center of the circle to where they intersect the circle's circumference. The circle is drawn first, along with its centerline, in Figure 13.24(a); then the circle is boxed in [Fig. 13.24(b)]. Equally spaced construction lines are drawn from the center of the circle to the circle's circumference [Fig. 13.24(c)]. The number of equally spaced lines drawn is dependent on the accuracy desired: the more points established on the circumference, the more accurate the ellipse. Here, a 30° spacing was used to establish twelve evenly spaced points along the circumference (points 1–12). The box shape is drawn isometrically in Figure 13.24(d), and each of the points is transferred from (c) using offset dimensions. Dimensions *D* and *C* establish point 1. Dimensions *A* and *B* establish point 2. Points 3 and 12 are located at the tangent points of the circle and the box, and are established on the isometric view at the intersection of the box and the centerline. To complete the ellipse, each of the four quadrants can be drawn by the same method [Fig. 13.24(e)]. Actually, since quadrants 1 and 3 are the same as are quadrants 2 and 4, only two quadrants need to be established; the opposite side can be mirrored. The darkened, finished ellipse is shown in Figure 13.24(f).

The second method (Fig. 13.25) is similar, but the points are arbitrarily fixed along the circumference of the circle and offset dimensions are taken as shown in Figure 13.25(c). The steps of Figure 13.25(a), (b), (d), (e), and (f) are the same as the first method. The points should be located on the circumference and of sufficient number so that a smooth curve can be established.

Circles, arcs, or curves that do not lie in isometric planes must be plotted with offset dimensions. This procedure requires that a series of points be established along the curved outline. Offset dimensions for each point are transferred to the isometric drawing and marked off along isometric lines.

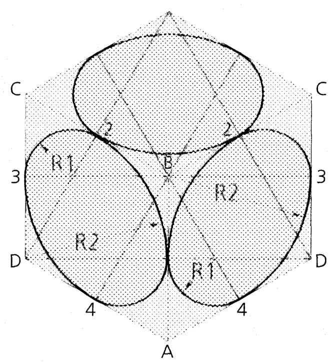


FIGURE 13.22 Four-Center Ellipses on the Surfaces of an Isometric Cube

13.3.8 Curves on Isometric Drawings

A space curve can be constructed isometrically using offset dimensions and box construction. The methods are not much different from those employed to draw space curves on multiview drawings with orthographic projection. The difference is in the use of receding axes drawn at 30°. Otherwise, all measurements are marked off the same for both types of drawings. The offset method can be used for any shape. Points are located along the curve and their positions transferred to the pictorial with dividers or a scale. A sufficient number of points are established to lay out the curve accurately. After the points are located on the pictorial, a light curve is drawn freehand through the points, and an

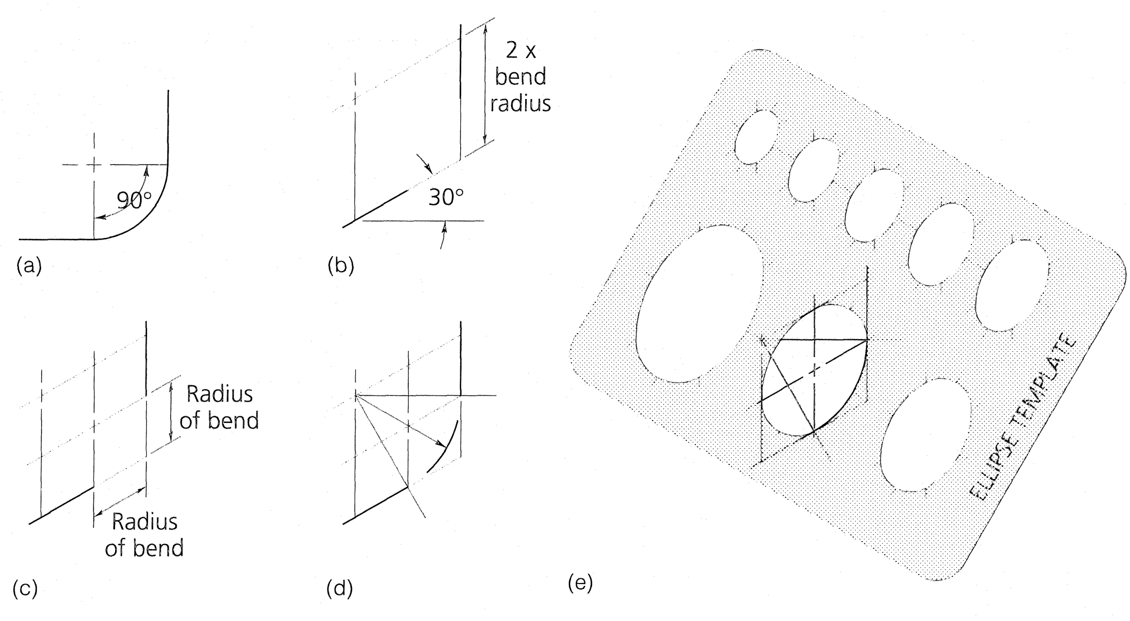
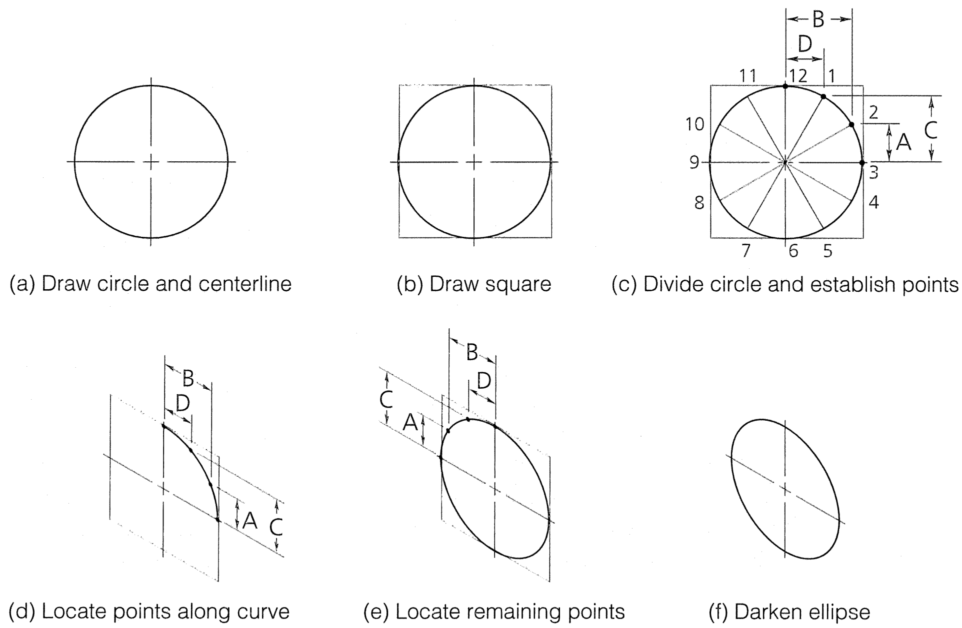


FIGURE 13.23 Construction of Arcs in Isometric Drawings



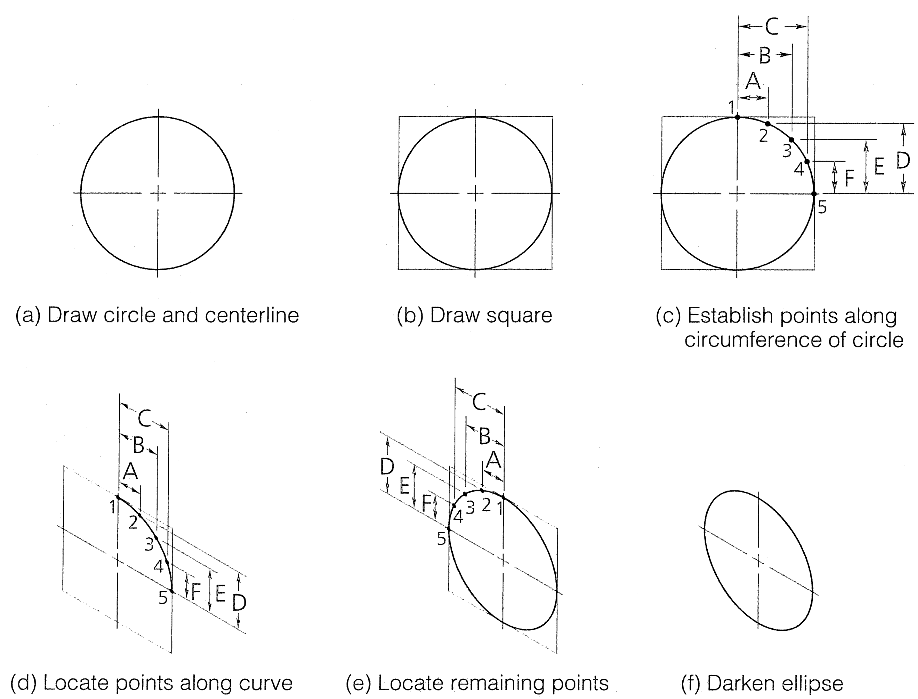
IGURE 13.24 Construction of rue Ellipse Using Offset imensions

rregular curve is used to draw the curve with the appropri- te line thickness.

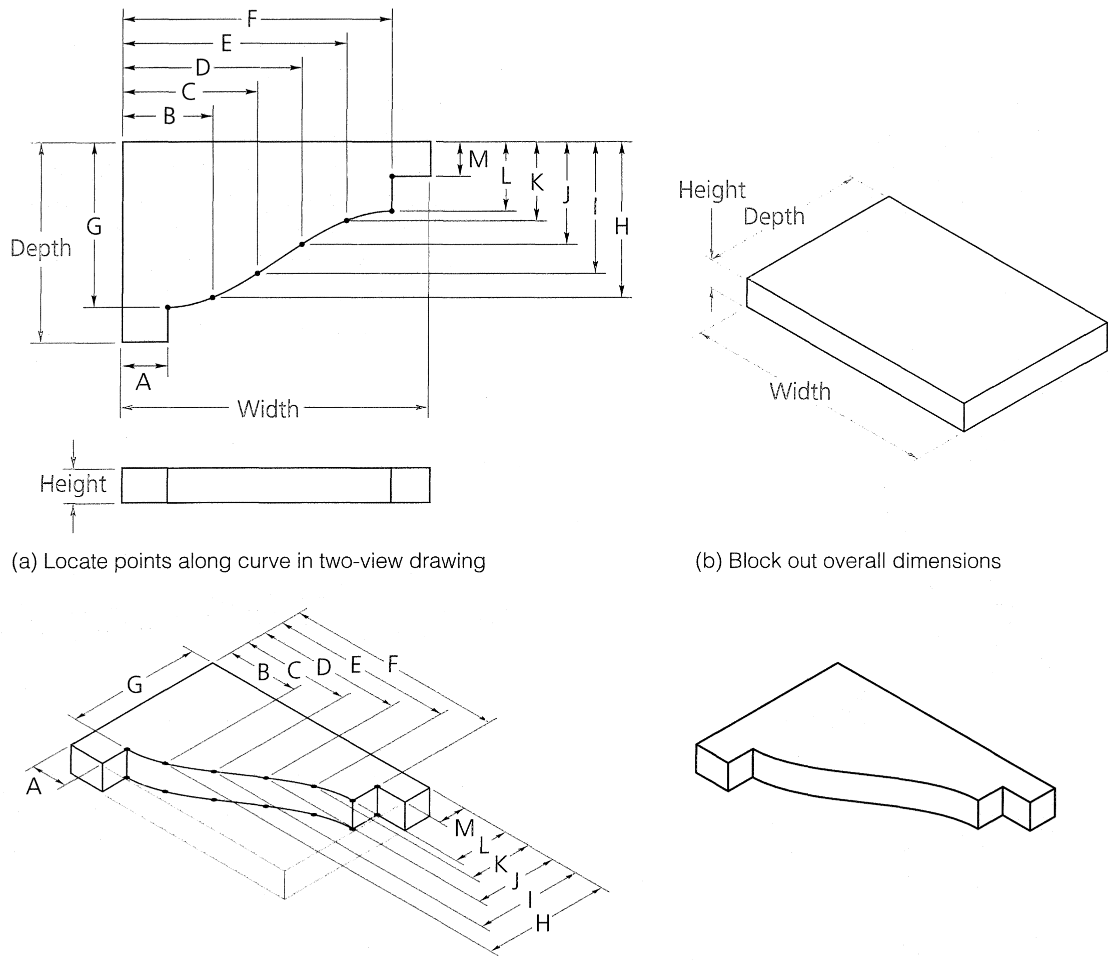
In Figure 13.26(a), the top and the front views of the part re drawn first. Points are then established along the curve n the top view. Offset dimensions A through M are located rom the edges of the part. The height, width, and depth are sed to box in the part in the isometric drawing [Fig. 3.26(b)]. Using offset dimensions A through M, each of the oints 1 through 7 is located from the edges of the part [Fig. 3.26(c)]. All point dimensions are taken parallel to their corresponding axis (edge). Point 1 is located by dimensions A and G; point 2 by dimensions B and H; point 3 by dimensions C and I. Vertical lines are drawn from points 1 through 7 to establish the part's thickness [Fig. 13.26(c)]. To complete the pictorial construction, lines are erased and the outline is darkened [Fig. 13.26(d)].

13.3.9 Hidden Lines on Isometric Drawings

Hidden lines are omitted on most pictorial illustrations unless required for clarifying interior features. When an



GURE 13.25 True Ellipse onstruction by Plotting oints



(a) Locate points along curve in two-view drawing

(b) Block out overall dimensions

(c) Locate points

(d) Erase construction lines and darken lines

FIGURE 13.26 Space Curves in Isometric Drawing

illustration requires hidden lines, a sectioned pictorial should be considered. In Figure 13.27, three variations of a pictorial drawing are shown. In Figure 13.27(a), the part is shown with all hidden lines; in (b), only visible lines are shown. This would normally be the case for most pictorials. In the last variation (c), the part is shown as it would appear on a CAD display modeled in 3D as a wireframe model. Here, the part has all edges shown.

When using a CAD system with wireframe modeling, you must correctly visualize the part and which sides are shown at any given moment. In Figure 13.27 we are looking at the top, front, and right side or the bottom, front, and left side. Since the examples in (a) and (b) show the first possibility, it is hard to see that the second viewing direction is also valid when viewing (c). Most CAD systems have a hidden line removal capability (**HIDE** command) that eliminates the problem of determining the correct viewing direction of a wireframe model.

13.3.10 Centerlines on Isometric Drawings

Centerlines that are needed to locate and identify circular or symmetrical aspects of a part are found on isometric drawings. Many pictorials do not show centerlines so that the part looks more realistic. If dimensions are required on the drawing (Fig.13.28), centerlines are included.

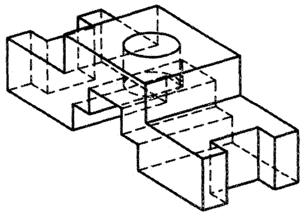
During the construction of a pictorial, centerlines identifying the origin of a part's symmetrical or curved features are as necessary as any other construction line. In most cases, they may be erased after the pictorial is constructed and before it is darkened in with pencil or inked.

13.3.11 Dimensioning Isometric Pictorials

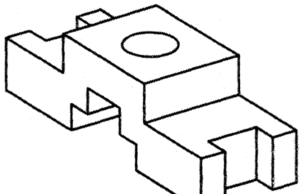
Dimensions on isometric drawings can be either aligned or unidirectional. Aligned dimensions look pictorially correct but are harder to draw. Unidirectional dimensions and notes are easier to add to the illustration and are often either typeset or labeled mechanically.

In Figure 13.28, the part's dimensions are shown by both methods. Figure 13.23(a) was drawn with aligned dimensions. Guidelines for the lettering are drawn parallel to the feature being dimensioned and in the same isometric plane. The arrowheads for aligned dimensions are drawn with their backs parallel to the extension line, as shown in the enlargement in Figure 13.28(c). With a 3D CAD system, you can automatically insert dimensions pictorially.

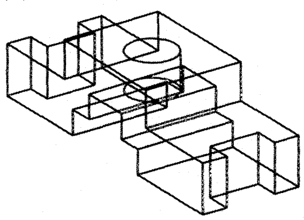
Unidirectional dimensions are positioned horizontally



(a) Part with hidden lines



(b) Part with hidden lines removed



(c) Wireframe model of part on 3D CAD system

FIGURE 13.27 Hidden Lines in an Isometric Drawing

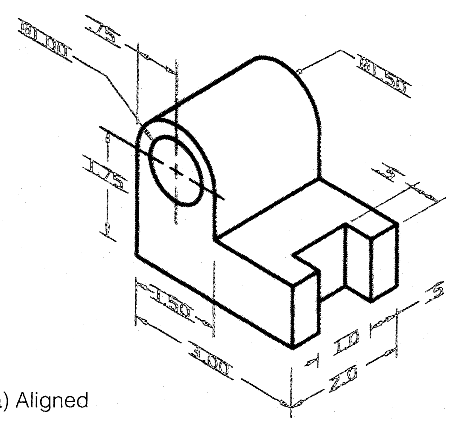
and are therefore easiest to construct [Fig. 13.28(b)]. Guidelines are drawn horizontally and dimensions are added with vertical lettering.

For all pictorials, the dimension line, extension lines, and the dimension text (unless unidirectional lettering is used) should lie in the same plane as the line or feature being dimensioned [Fig. 13.28(a)]. Arrowheads should be long and narrow, with a ratio of 3:1, and should lie in the plane of the dimension and extension lines [Fig. 13.28(a)]. For unidirectional dimensioning, the lettering should be made with vertical letters and should read from the bottom of the sheet [Fig. 13.28(b)].

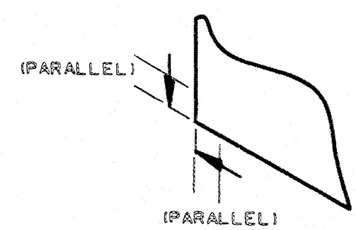
You May Complete Exercises 13.1 Through 13.4 at This Time

13.4 OBLIQUE PROJECTION

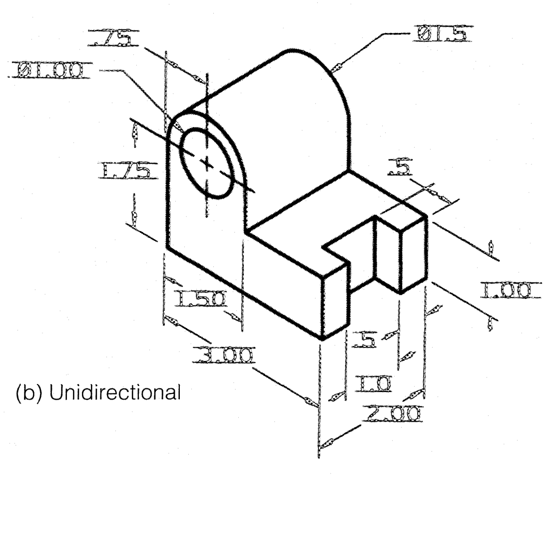
A projected view in which the lines of sight are parallel to each other but inclined to the plane of projection is called an **oblique projection** (Fig. 13.11). For practical purposes, the principal face is placed parallel to the plane of projection, thus making it and parallel faces show in true shape. In all forms of oblique projection, the receding axis may be drawn



(a) Aligned



(c) Arrowhead construction



(b) Unidirectional

FIGURE 13.28 Dimensioning Isometric Drawings

Focus On . . .

PICTORIALS

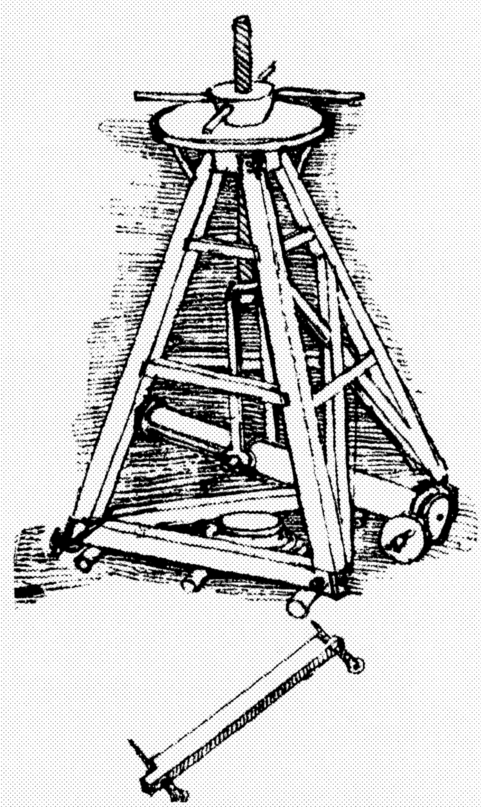
Pictorial drawings help the viewer to visualize a part better and gain a better understanding of its components and features. These drawings help bridge the gap between photograph and part.

Engineering drawings contain a wealth of information about a product: its size, shape, location of features, construction materials, and assembly specifications. However, engineering drawings (multiview projections) are not always easy to read for a nontechnical person untrained in those projection techniques. A more realistic-looking 3D drawing (technical illustration) is produced for situations in which engineering drawings are not the most appropriate presentation, such as in marketing meetings and in maintenance documentation.

Early tries at technical illustration by the Egyptians and Greeks didn't really show all three dimensions in one view. Around 1500 A.D., pictorial drawings that showed all three dimensions in one view evolved. Leonardo da Vinci was the most famous of this group of inventors/illustrators. His artistic ability and scientific foresight provided the means for true technical illustration. Techniques were further refined during the Industrial Revolution. After 1940, technical illustration became popular design and development tools.

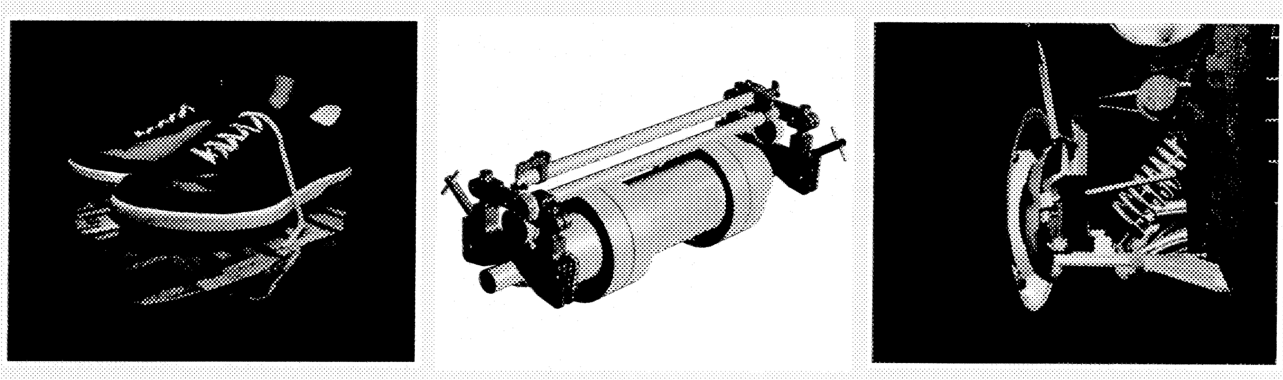
Today, technical illustrators produce pictorial drawings that aid in product development, assembly, marketing, illustration, repair, and maintenance. It is a fact that technical illustrations accelerate production, improve communications during development, and reduce product cost. Even people with limited technical knowledge can understand complex assemblies and interrelationships of parts and features. Indeed, pictorials prove to us "a picture is worth a thousand words."

The solid models on advanced CAD stations today are an extension of this same principle. Not only do these parts look three dimensional, they are 3D mathematically. The advanced renderings of parts on these sophisticated systems makes the parts look very real on the computer screen—in fact, they look almost as good as photographs. The 3D database can be used to make renderings and illustrations and to generate a tool path to machine the part. Improved visualization is one of the key factors driving the increasing popularity of these systems.



One of Leonardo da Vinci's mechanical sketches.

However, the lesson learned by humankind long ago is that we really need to be able to "see" a part to understand it. Leonardo da Vinci was a master of this long before the computer age.



3D solid models.

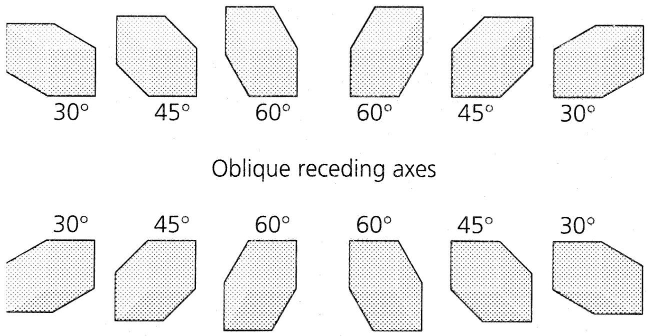


FIGURE 13.29 Axis Choice for Oblique Projection

n any direction (Fig. 13.29). By changing the axis angle and the choice of front face, any orientation required to exhibit the part properly and clearly can be attained.

Oblique drawings are similar to isometric drawings. However, they are produced from parallel projectors that are not perpendicular to the projection plane. The primary difference lies in the use of only one receding axis and the

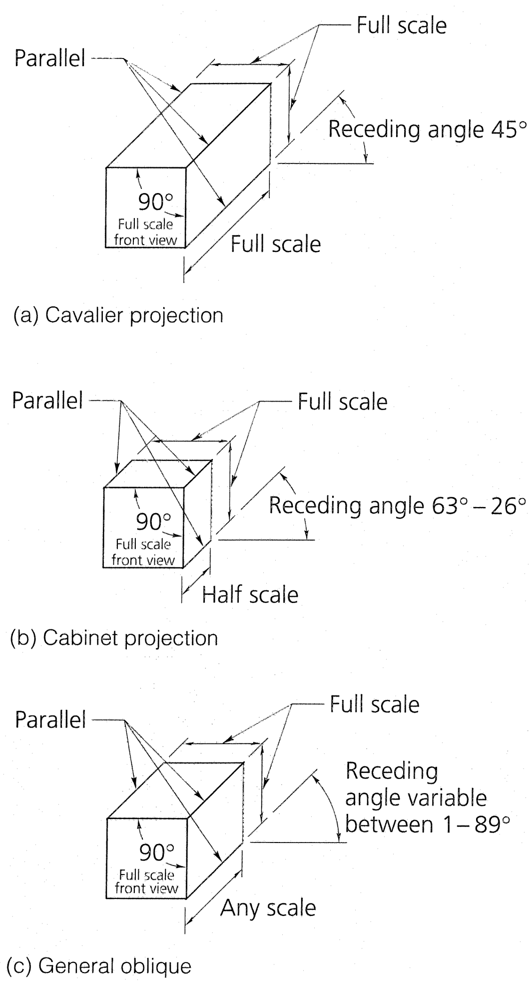


FIGURE 13.30 Types of Oblique Projection

ability to draw one surface as true shape and size in the front plane.

There are three versions of oblique projection, differing only in the comparative scales used along the receding axis and the angle of the receding axis (Fig. 13.30). An oblique projection on which the lines of sight make an angle of 45° with the plane of projection is called a **cavalier projection** [Fig. 13.30(a)]. The front is drawn full scale and true shape, as with all forms of oblique projection. The same scale is used on all axes; therefore, the receding faces are drawn full scale (but not true shape). An oblique projection in which the lines of sight make an angle of between 63° and 26° with the plane of projection is called a **cabinet projection** [Fig. 13.30(b)]. *The scale on the receding axis is one-half of the scale on the other axes.*

An oblique projection in which the lines of sight make any angle other than 45° or 63° to 26° is called a **general oblique.** The scale on the receding axis should be something between full scale and one-half scale of the horizontal and vertical axes, as shown in Figure 13.30(c). The choice of the receding angle (1° to 89°) is determined by the shape of the object and the most descriptive view orientation.

The distortion often noticeable in oblique projection may be decreased by reducing the scale on the receding axis. Cylinders and cones should have their axes on the receding axis to reduce distortion and to make it possible to draw circles with a compass or template.

Oblique projection is most commonly used for objects that have a series of circles, curves, or irregular outlines in the same or parallel planes. By placing curved outlines in the front face, they can be drawn true shape and full scale without distortion. A standard circle template or a compass can be used in the construction. In other words, the front face of an oblique projection is exactly the same as the front view of a part drawn in a multiview projection.

13.4.1 Oblique Construction

Objects that are drawn with oblique projection should be oriented so the surface with curved lines lies in the front plane bounded by the axes that are at 90°. Since all surfaces that lie on the front plane or that are parallel to it are drawn true shape and size, this orientation lessens drawing time. Circles and arcs are therefore true projections.

Oblique construction is started the same as a multiview or an isometric drawing—by blocking in the overall dimensions. The drawing is started by establishing the width and the height of the front and the rear face. Next, the depth of each face is established, and then the part's edges are constructed. Circular features are then located and their dimensions blocked in. Finally, the part is darkened. Figure 13.31 shows four steps in this process.

The front face of the part in Figure 13.31 (with all curved features) is drawn true shape and size, and all measurements on this front face are true length. The measurements are

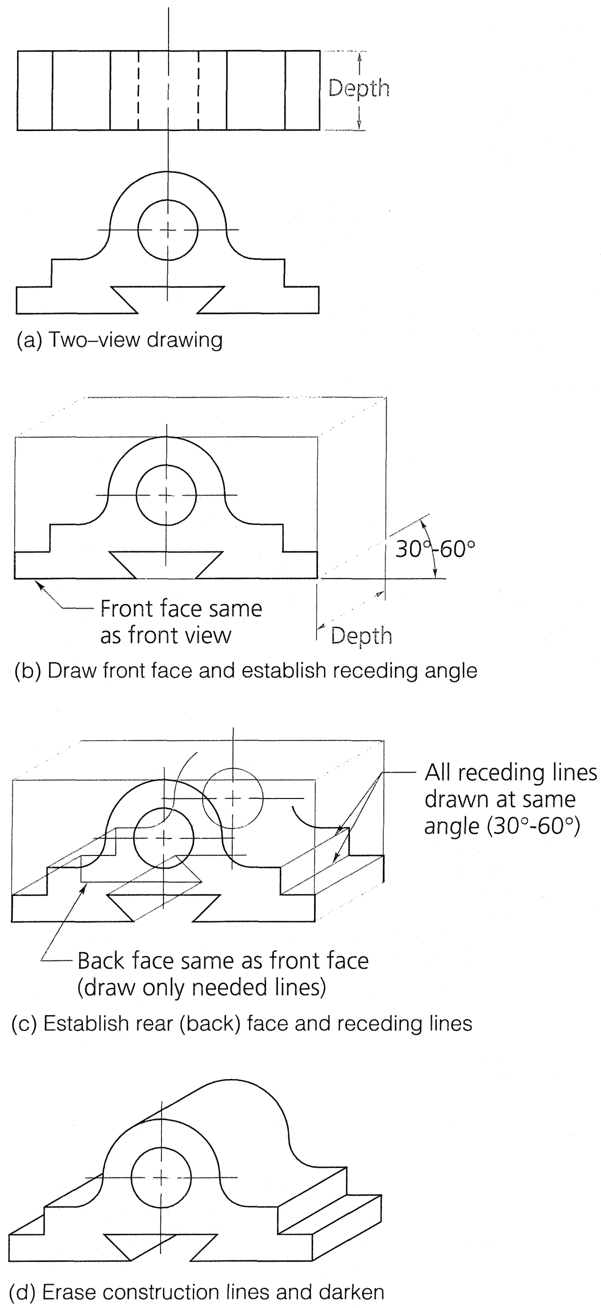


FIGURE 13.31 Step-by-Step Construction of Oblique Drawing

taken from the front view in (a) and transferred to (b). The angle for the receding axis is then determined. (The most common angle for the receding axis is 45°.) After the angle is determined, the rear face of the part is blocked in. The features of the rear face can then be drawn, as in Figure 13.31(c). Because the front and the rear faces are parallel and therefore identical, only the portions of the rear face that show along the receding face need to be drawn. The receding edge lines are drawn parallel to the receding axis between the part's corners and tangent to the curved feature on the top edge [Fig. 13.31(c)]; the part is completed by darkening in the visible edges [Fig. 13.31(d)].

13.4.2 Using Offset Measurements for Oblique Drawings

When the object to be drawn is placed so that the curved features do not fall in the front face or two or more faces have curved features, offset measurements must be used to establish the curves. After the feature's points are plotted, an irregular curve is employed to draw the curve. The curved features are divided vertically and horizontally with construction lines [Fig. 13.32(a)]. The intersection of vertical construction lines with related horizontal construction lines establishes points on the curved feature.

The oblique projection is started by establishing a front face. The choice of front face in this example was made solely on the need to demonstrate this procedure. The construction lines are transferred to the oblique view and the points plotted in Figure 13.32(b). A smooth curve is drawn through the points using an irregular curve. Since the opposite portion can be mirrored, only half of each hole needs to be plotted.

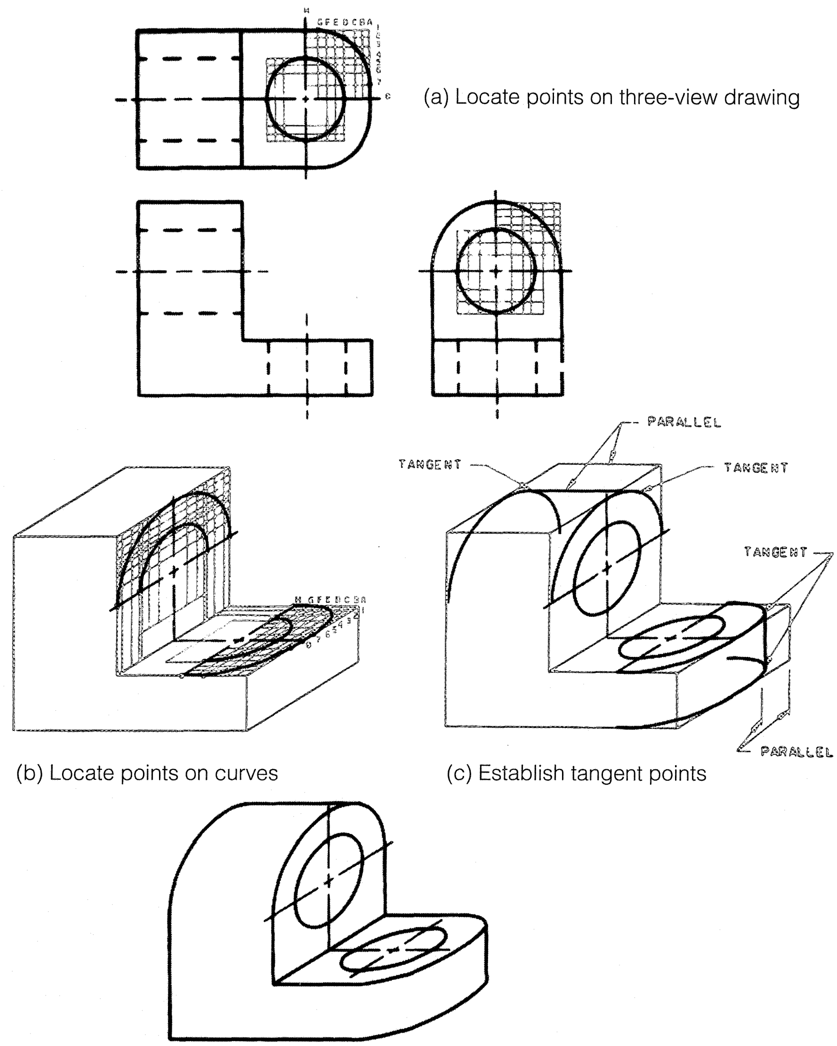
The projection is completed by drawing the end lines parallel to the axes and tangent to the curves in Figure 13.32(c), and darkening in the lines in (d). If a cabinet drawing is involved, the depth dimensions are halved and the same procedure used in its construction.

The construction process for oblique projection with slanted, inclined lines and inclined planes is similar to that for isometric drawings. Their endpoints are located along lines that are parallel to one of the axes.

13.5 PERSPECTIVE PROJECTION

A pictorial drawing made by the intersection of the picture plane with lines of sight converging from points on the object to the point of sight that is located at a finite distance from the picture plane is called a **perspective.** Perspective drawings are pictorials that appear similar to photographs. The use of perspective projection gives the illustration a photolike realism. The observer is stationed at a fixed position relative to the object being drawn, as with a photograph.

Perspective projection provides illustrations that approximate how a particular object looks to the human eye or as a camera would record the object on film. Since a perspective drawing approximates how an object really looks, it is not dimensionally correct and cannot be scaled. The only lines that can be scaled are those lines on the object that actually lie in the picture plane. Technical illustrations for advertisements, sales catalogs, technical manuals, and architectural renderings make extensive use of this form of pictorial



(a) Locate points on three-view drawing

(b) Locate points on curves

(c) Establish tangent points

(d) Erase construction lines and darken lines

FIGURE 13.32 Offset Dimensions for Curves Not in or Parallel to the Front Face of the Part

projection. All lines in perspective drawings converge at one, two, or three points on the horizon (*vanishing points*) and, therefore, are not parallel, as in oblique and axonometric projection.

There are three basic categories in perspective projection: parallel, angular, and oblique (Fig. 13.33). A perspective in which two of the principal axes of the object are parallel to the picture plane and the third is perpendicular to the plane is called **parallel,** or **one-point, perspective** [Fig. 13.33(a)]. A perspective in which one axis of the object (usually the vertical axis) is parallel to the picture plane and the other two axes are inclined to it is called an **angular,** or **two-point, perspective** [Fig. 13.33(b)]. A perspective in which all three principal axes of the object are oblique to the plane or projection is called an **oblique,** or **three-point, perspective** [Fig. 13.33(c)].

13.5.1 Perspective Pictorial Drawing Terminology

The following terms are used with perspective drawings (Fig. 13.34):

Station point (SP) The assumed position of the observer.

Picture plane (PP) A vertical plane that is 90° to the line of sight from the station point (SP). The picture plane is usually placed between the object and the station point.

Horizon line (HL) The line of intersection made by a horizontal plane located at eye level and the picture plane. The horizon line is raised or lowered as the observer is raised or lowered. The horizon line remains in a horizontal position.

Ground line (GL) The intersection of the ground surface plane and the object or structure contacting the ground surface.

Vanishing point (VP) A point located in space where the ground line appears to intersect with the horizon line. The number of vanishing points depends on the type of perspective. There will be either one, two, or three vanishing points (in this figure there are two, VPL and VPR).

Height line (HL) A vertical line in the plane of sight that falls on the line of sight on which measurements of height

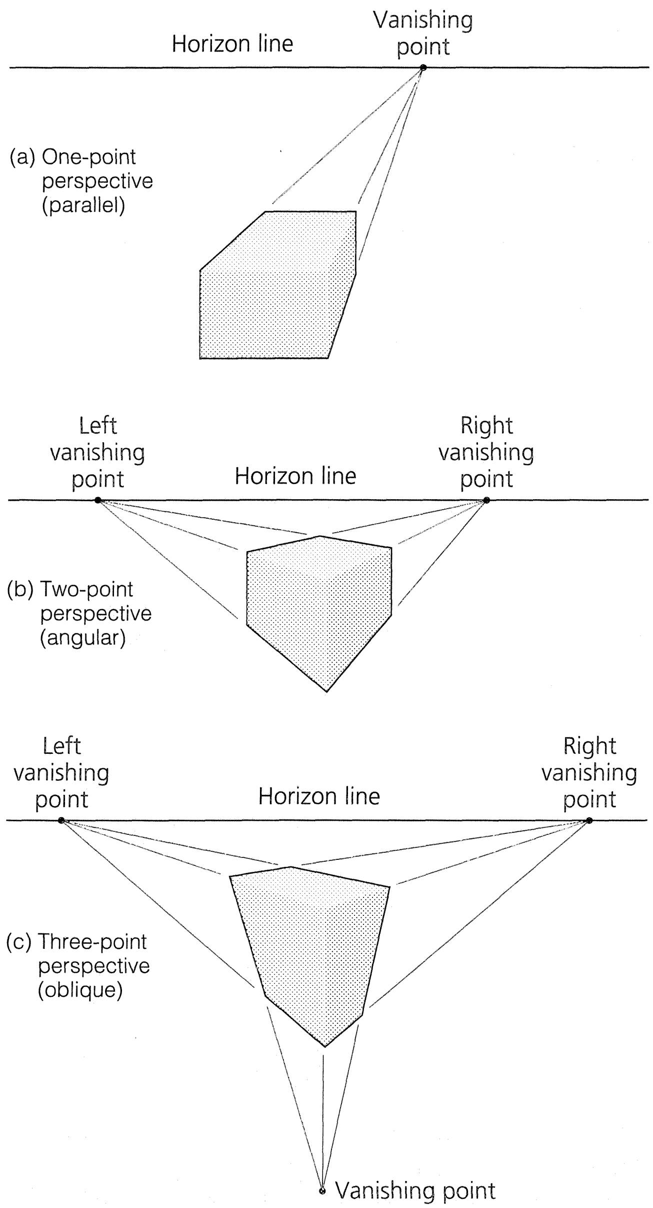


FIGURE 13.33 Types of Perspective

correspond to the height in the elevation (distance 1-2 in Fig. 13.34). It is drawn perpendicular to the edge view of the picture plane.

13.5.2 Locating the Horizon Line

The horizon line is usually located on eye level with the observer in the horizontal plane. The use of a higher horizon line will change the perspective appearance. Figure 13.35 shows three possible positions of the horizon line. The ground line remained the same for each projection, but the horizon line (VP 1 H, VP 2 H, VP 3 H) was adjusted to see the part from the bottom, the front, and the top, respectively.

13.5.3 Locating the Station Point in Perspective

To avoid undue distortion in perspective, the station point should be located so that the cone of rays has its apex at the point of sight and includes the entire object. This should have an angle at the apex not greater than 30°, shown as the angle of vision in Figure 13.34. When the object is close to the horizon, a greater angle may be used. Pleasing results are usually obtained if the point of sight is located centrally in front of the object and at a height that will show the desired amount of the horizontal surfaces.

13.5.4 One-Point Perspectives

The **one-point (parallel) perspectives** use a frontal plane that is parallel to the front face of the part (Fig. 13.36). The frontal plane is perpendicular to the line of sight. Parallel perspective pictorial drawings are employed most commonly to illustrate interior views of rooms or other spatial conditions. On solid objects, one-point perspectives more nearly resemble oblique pictorial drawings. The frontal surface of the pictorial box is perpendicular to the lines of sight (axis of vision) and shows height and width in true distance. The depth lines of the object converge toward a common vanishing point (CV). The top, front, and side faces are drawn first with full-scale dimensions. Height dimensions are taken from the side view for the front face.

After the top view and the front face of the part are drawn, the station point and the common vanishing point are established. Visual rays are projected from each point of the part to the station point. The intersection of a visual ray and the picture plane establishes points along the picture plane that are then projected to the front view. The intersection of these points and rays extending from the front view to the CV establishes the depth features of the part. This same procedure was used in drawing Figure 13.37. Here, the CV was placed to the right of the part, not centered as in the previous example.

Since the curves are drawn without distortion in the front plane, parts with curved features on one surface are easily drawn with this form of perspective (Fig. 13.38). Receding curves are located in the perspective view (front) by establishing the intersection of each curve's center point with the picture plane, points 1^1, 2^1, and 3^1.

13.5.5 Two-Point Perspectives

The **two-point perspective** is also called an **angular perspective.** In Figure 13.39, the top view shows the width and depth of the part. The height is either projected to the height line (common to the line of sight) or measured directly (to scale) on this line. There are two height lines on this example because two mating parts are drawn. Each has the same top view (except for visibility). The height line for the top piece goes from the TOP LINE to point 1. The bottom piece has a height line from the GROUND LINE to point 1.

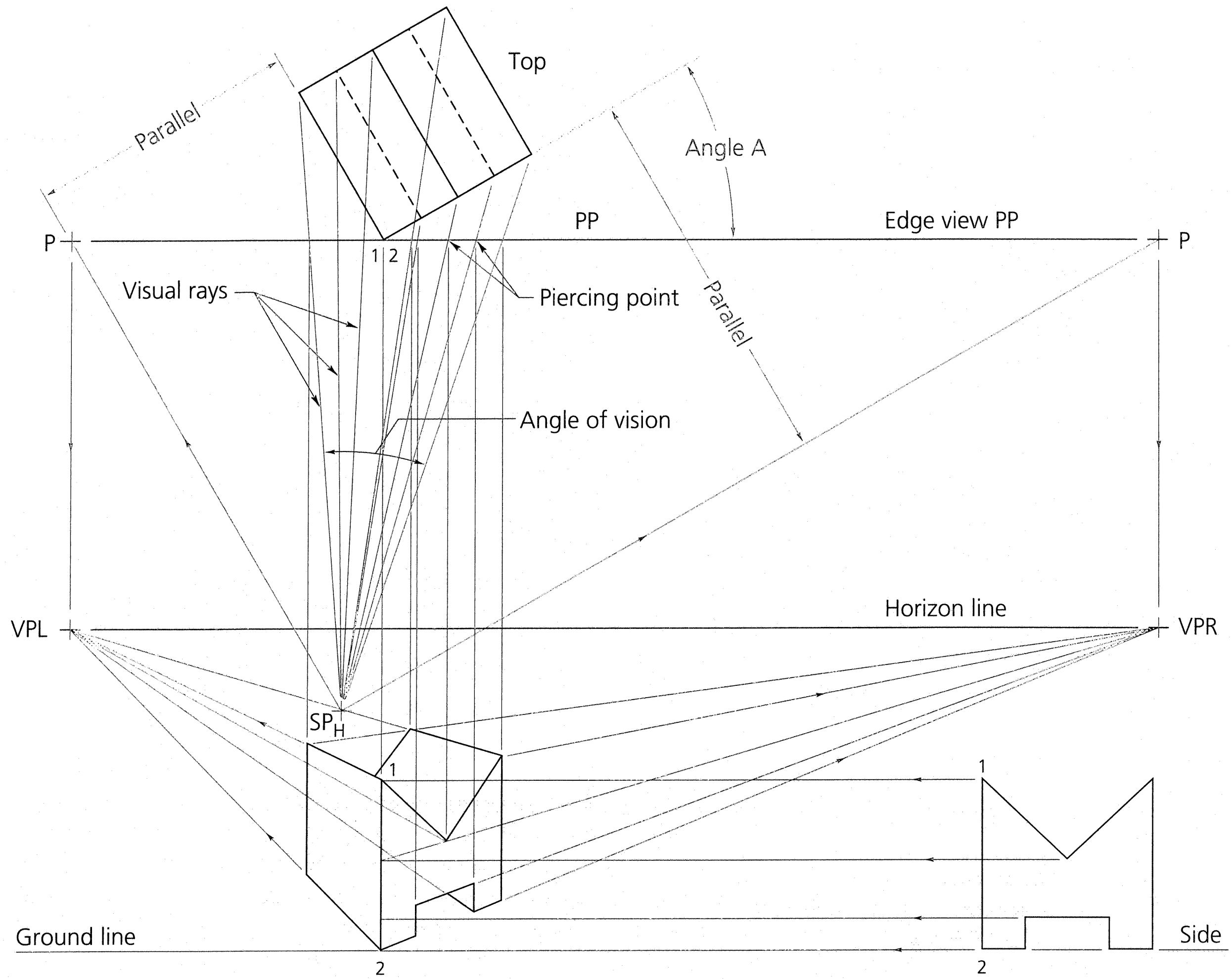


FIGURE 13.34 Angular Perspective Projection

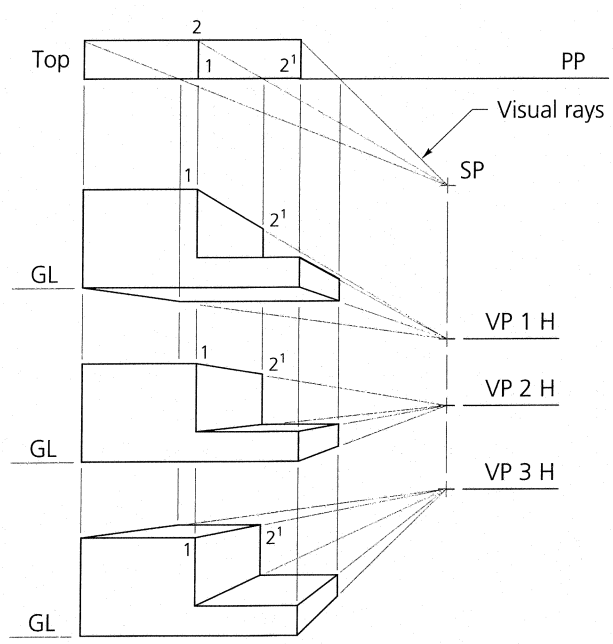


FIGURE 13.35 Variations in the Placement of the Horizon Line

The side views of each piece are reversed (and have different visibility).

The part is angled in the top view so that it makes 30° with the picture plane (PP). Lines are drawn from the SP parallel to the part's edge in this view. The angle formed by the SP and the two vanishing points in the top view will be 90° (angle A-SP-B). The intersection of each line and the PP determines the right and left vanishing points, which are then projected to the front view on the horizon line.

All lines on the drawing are determined by projection from location points on the height lines, with lines extended to the proper vanishing points. Lines to the left of the plan view go toward the left vanishing point (VPL). Lines to the right on the plan view are drawn toward the right vanishing point (VPR). Projection from these points must follow the direction of the planes in the plan view of the part. If these lines change planes in following the contour or details of the part, the direction must also change from that point toward the vanishing point for that plane.

A line from the station point toward the outer points, both left and right, should make an angle with the height line (line of sight) of 15° or less. In other words the angle

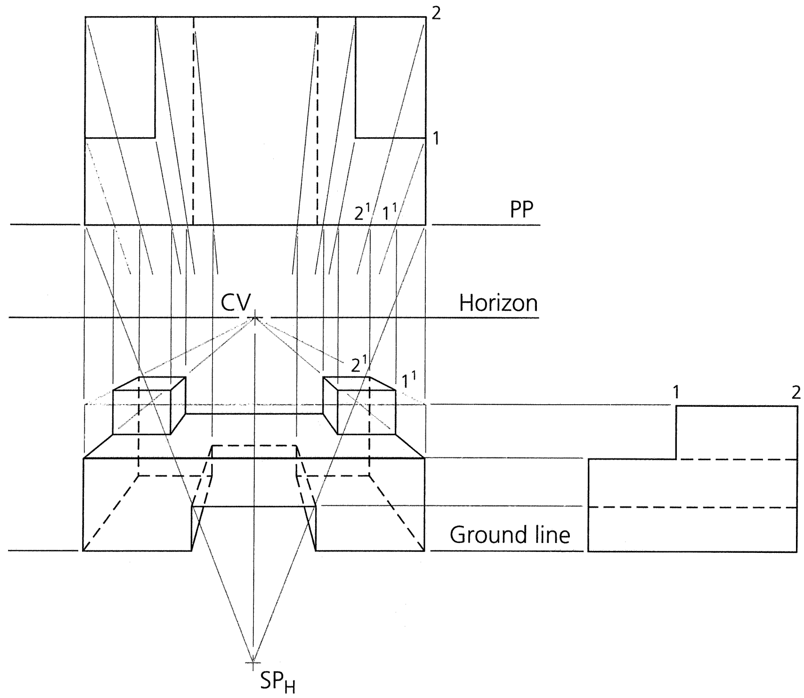


FIGURE 13.36 **One-Point Perspective Projection**

made by the visual rays should not exceed 30° in most cases. In Figure 13.39 a slightly larger angle was used. A larger angle is likely to create excessive distortion. The horizon line should be located at some distance from the picture plane to avoid overlapping of the plan view and the perspective pictorial drawing. This was *not* done in Figure 13.34 and, therefore, the construction there overlaps. Figure 13.39 is easier to read because the construction lines of the two views do not overlap.

The vertical dimensions of the part are measured to scale or projected from the side view directly. The scale of the plan view and of the elevation should be the same, to avoid distortion on the pictorial drawing.

Increasing the distance between the plan view of the part and the edge view of the picture plane decreases the size of the perspective and gives an appearance of distance from the part. However, increasing this distance does not compensate for a change in height and can cause distortion. The plan view should touch the edge view of the picture plane (Fig. 13.39).

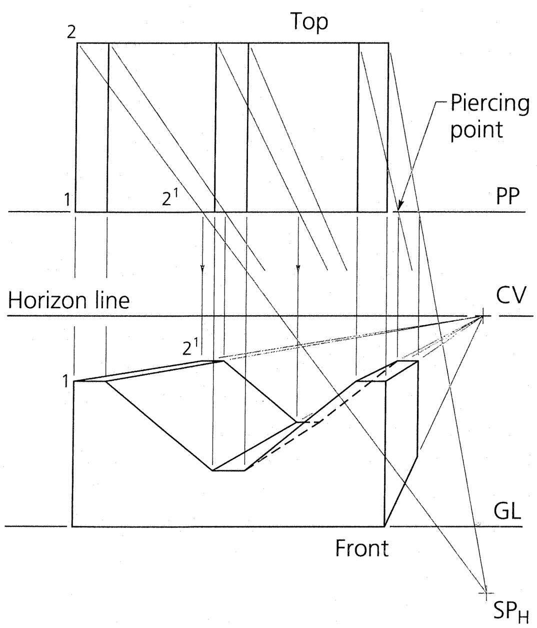


FIGURE 13.37 **Parallel Perspective Projection**

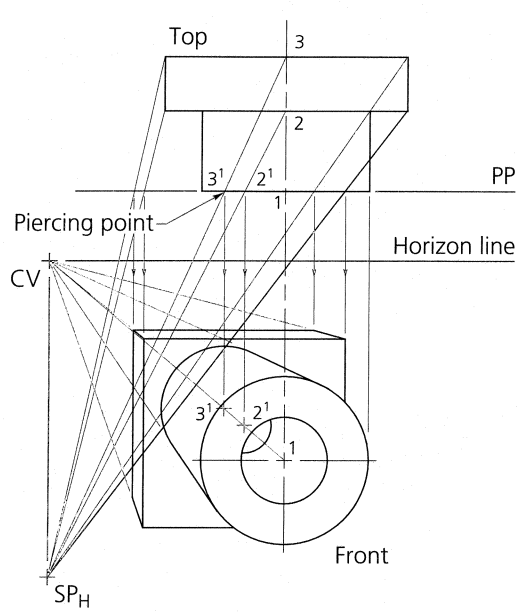


FIGURE 13.38 **Circles in Parallel Perspective Projection**

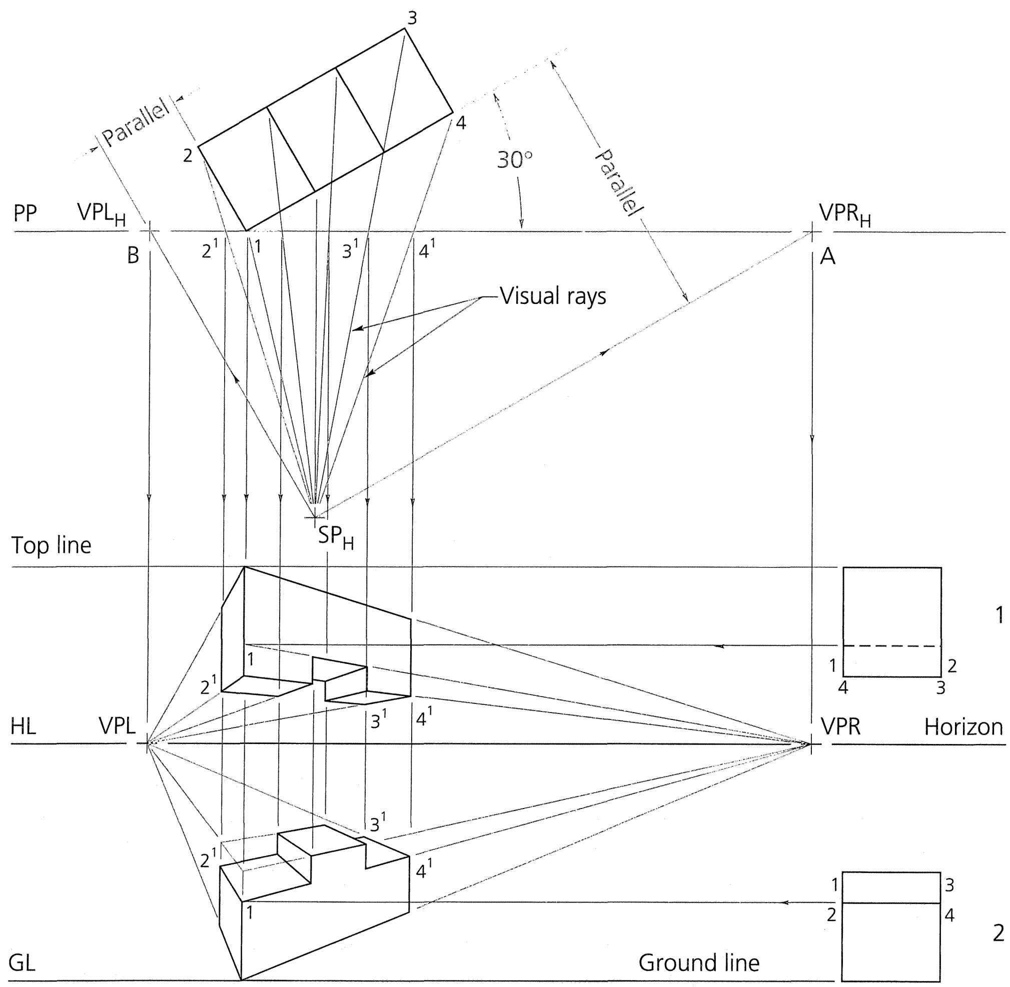


FIGURE 13.39 Two-Point Perspective Projection

13.5.6 Three-Point Perspectives

Three-point perspectives are called **oblique perspectives** [Figure 13.33(c)]. They are useful in situations where the object would not appear excessively distorted, as for very tall buildings. Oblique perspective pictorial drawings aid perception very little. A three-point perspective requires that all three axes of the object be oblique to the plane of projection. Height cannot be measured directly on the height line, as with the two-point perspective. When creating a three-point perspective, grid underlays are helpful.

13.6 CONSTRUCTION OF PERSPECTIVES

The three examples presented in this section use the same basic object as the part to be drawn in perspective. Figure 13.40 is a one-point perspective of the part, Figure 13.41 is a two-point perspective of the same part, and Figure 13.42 is a two-point perspective of the same basic shape but with curved features.

13.6.1 One-Point Construction

The construction of a one-point perspective starts with the determination of the part's scale and position in space. The top surface and the right side of the part should be viewed. The top view is drawn first [Fig. 13.40(a)]. The picture plane is passed through the front face of the part. In other words, the edge view of the picture plane and the edge view of the front face are along the same line. The horizon line is drawn according to the desired eye level and whether the part is to be viewed from eye level, below eye level, or above eye level. The ground line is drawn last. The distance between the ground line and the horizon line determines how far above or below eye level the part will be observed by the viewer.

The front view is drawn second [Fig. 13.40(b)]. The distance between the picture plane line in the top view and

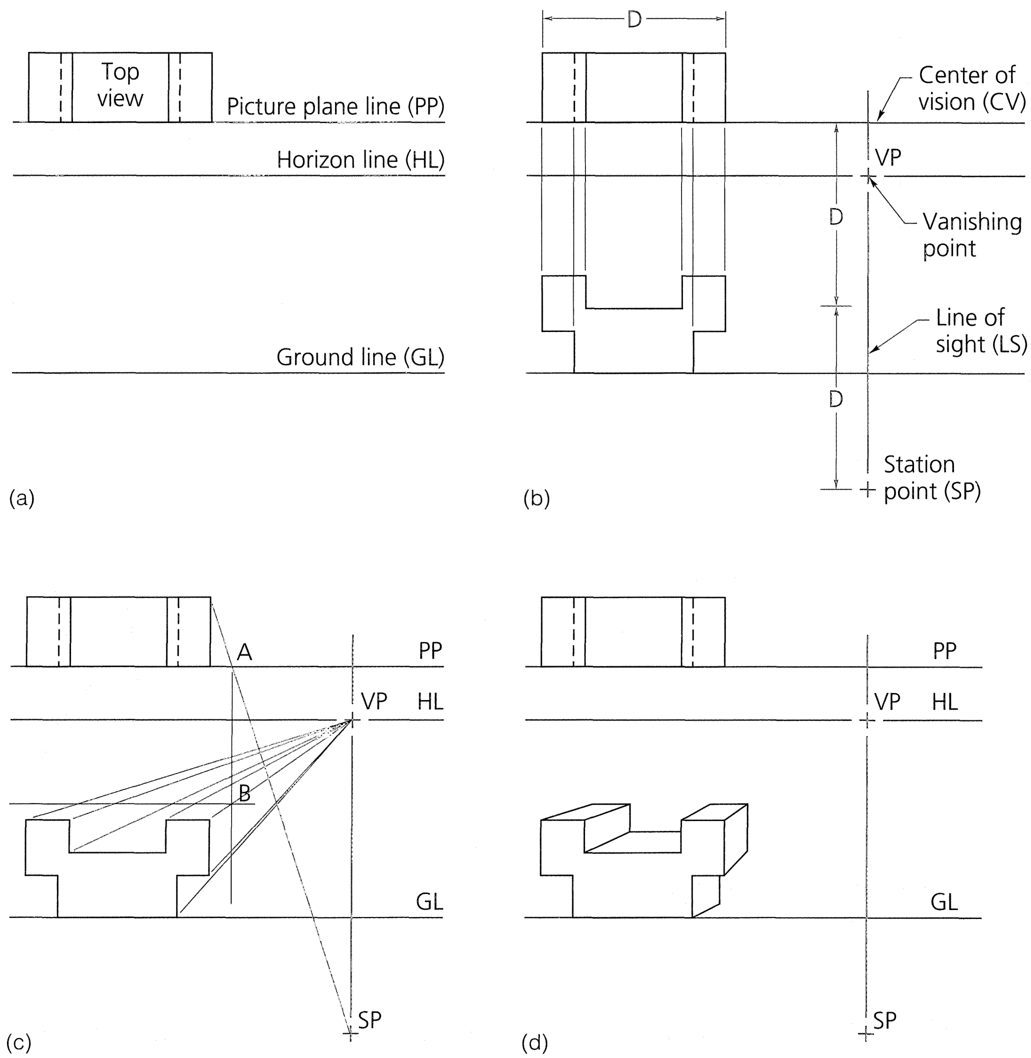


FIGURE 13.40 Constructing a One-Point Perspective

the station point is drawn two times (or more) the width of the part (dimension *D* in this figure). The location of the vertical line (line of sight) will be determined by which side the part is to recede to and how much of the part's side needs to be shown. Here, the vertical line is drawn on the right of the part, so the perspective recedes to the right. Dimension *D* establishes this distance. The line of sight is to the right of the part at a distance equal to dimension *D*. The farther to the side this line is drawn, the more the right side will show. The front view is then drawn using the width dimensions projected from the top view and completing the front face's features. The vanishing point is at the intersection of the line of sight (vertical line) and the horizon line.

Lines are then drawn from every point on the front view to the vanishing point (VP) in Figure 13.40(c). A line drawn from the station point to the back corner of the part in the top view intersects the picture plane (PP) to establish point A. A line is then drawn vertically from point A to the receding lines, which extend from the front view toward the VP. This will establish point B and the back vertical edge of the part. A horizontal line is then drawn from point B to the left to establish the back edge. All other features are drawn by the same procedure to establish points along the picture plane and to project to the front view [Fig. 13.40(d)].

13.6.2 Two-Point Construction

Figure 13.41 shows a two-point perspective of the same part as in Figure 13.40. Note the differences and similarities of the procedure and the finished illustration. Two-point perspectives require more work and the end result is more realistic.

Start by drawing the picture plane line (PP), as shown. The top view is then drawn with one of its corners touching the picture plane line. This will also establish the center of vision (CV). The front face is drawn at an angle of 30° to the PP, as shown. Other angles can be used. To show enough detail, the side with the most features should be at the

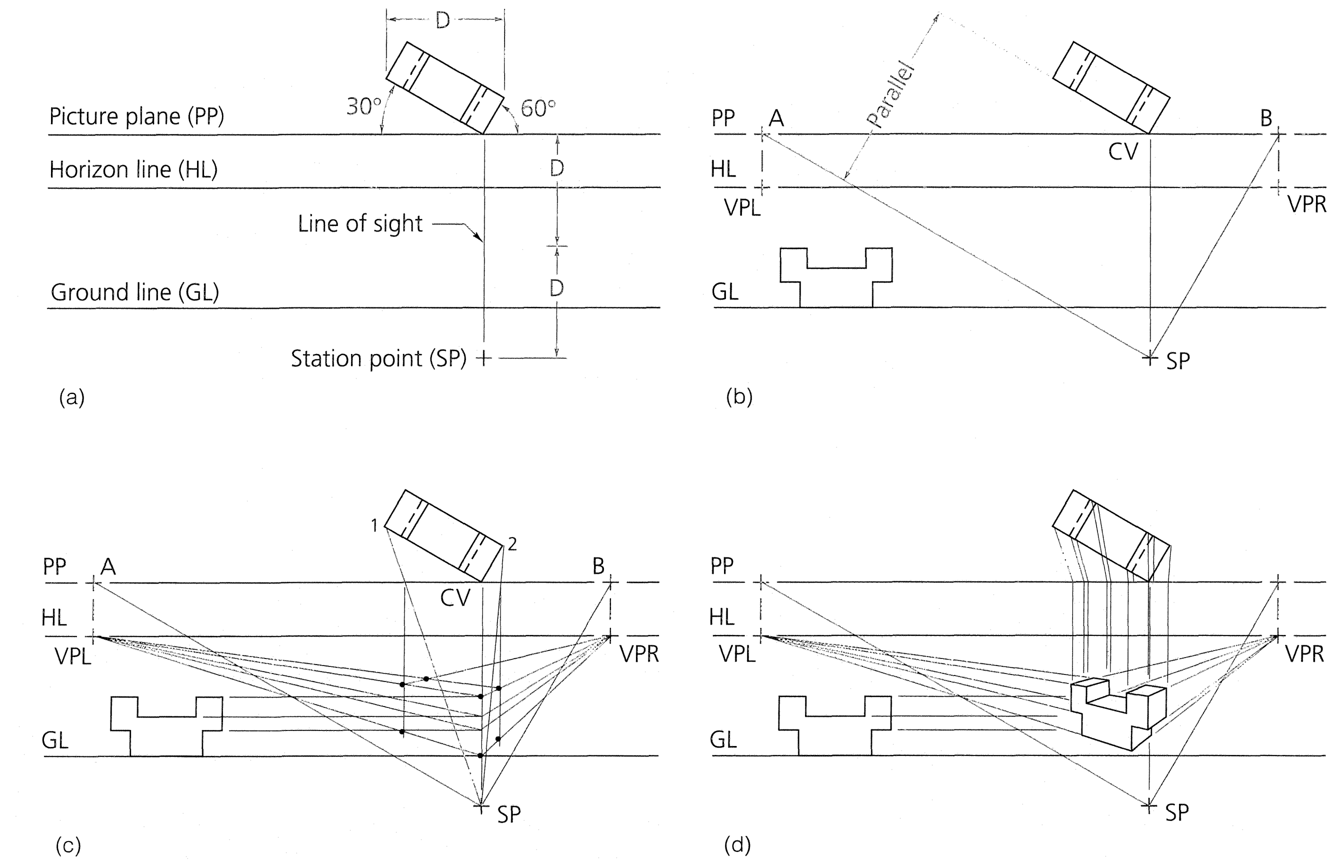


FIGURE 13.41 Constructing a Two-Point Perspective

smaller angle to the PP. The line of sight is drawn as a vertical line extending from the intersection of the part's front corner (the one touching the PP) in Figure 13.41(a). The horizon line, the ground line, and the station point are established as in Figure 13.40. Draw lines SP-A and SP-B from the station point parallel to the edges of the part to where they intersect the PP and establish points A and B [Fig. 13.41(b)]. Vanishing point left (VPL) and vanishing point right (VPR) are drawn by dropping vertical lines from points A and B to the horizon line.

The right or left side view is then drawn with its base on the ground line, as in Figure 13.41(c). Height lines are established in the perspective view by projecting each vertical dimension from the side view to the line of sight. The intersection of each line projected from the side view to the line of sight establishes points. Draw a line from each point to the VPL and the VPR. Draw lines from the SP to the corners of the top view at points 1 and 2. The intersection of these projectors and the PP establishes the outside limits of the perspective view when they are drawn vertically until they intersect their corresponding receding projectors. The part's outside dimensions have now been determined. By repeating the previous two steps, every feature of the part is established by projection. The perspective is then darkened, as in Figure 13.41(d).

13.6.3 Two-Point Construction of Curved Features

In Figure 13.42 the part has been constructed using the same procedure as in Figures 13.40 and 13.41. The parts are identical except that the cutout is now curved instead of rectangular. To draw a perspective of a part with curved features, points are established along the curve (Fig. 13.42). After the part's general shape is established, the points along each curve are projected and located by the same procedure

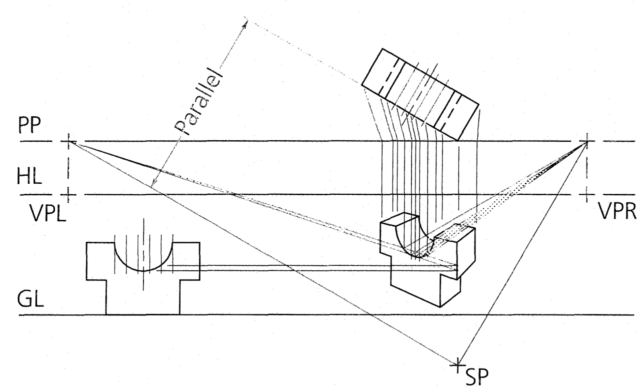


FIGURE 13.42 Constructing a Two-Point Perspective with Curved Features

as for locating an endpoint of a line or any straight feature. After a sufficient number of points is established along the curve in the perspective view, the curve can be drawn with the aid of an irregular curve.

13.7 TECHNICAL ILLUSTRATION

Various procedures and accepted drafting conventions for representing certain features are incorporated into pictorial illustrations. Pictorial sections and cutaways, breaks, fillet and round representation, and thread representation increase the lifelike qualities of an illustration. Shading and shadows are also added to pictorials to give a more lifelike representation of the part or system, as in Figure 13.43, where the degreaser system was pictorially sketched. Pictorial assemblies incorporate exploded views (Fig. 13.44) when it is necessary to show how a device fits together. CAD systems allow the user to create lifelike images in 2D (Fig. 13.45) and 3D images like the exploded view of the headset in Figure 13.46. These new and exciting possibilities include the ablity to display the part or system in any number of orientations and to add movement and animation to the engineering field.

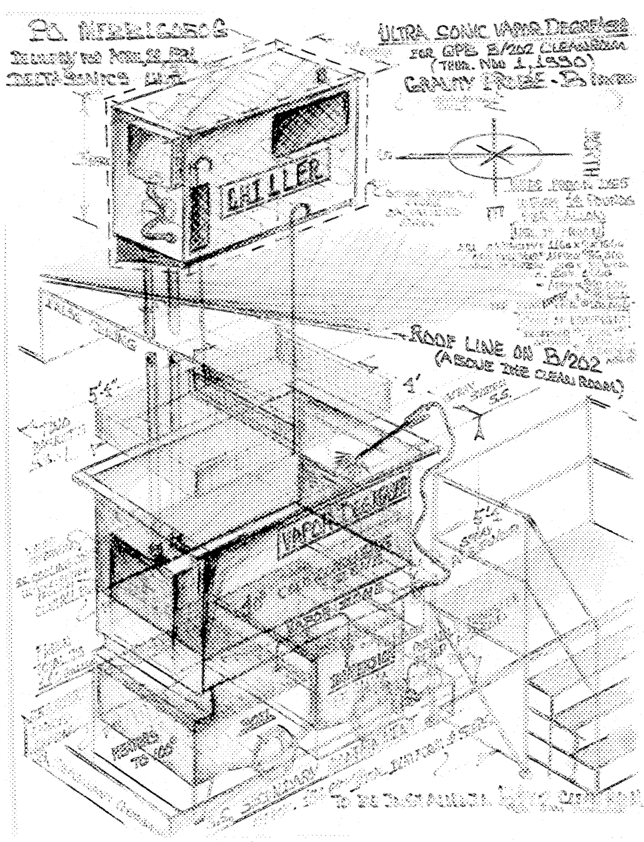


FIGURE 13.43 Pictorial Sketch of Vapor Degreaser

BELL CRANK HANDLE

PARTS LIST

NO.	REQ.	NAME
1	1	COUNT.WT.
2	1	BODY
3	1	CONNECTOR
4	1	HANDLE

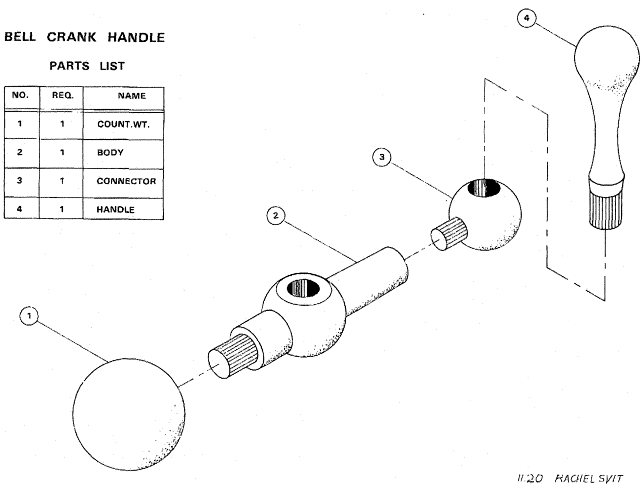


FIGURE 13.44 Bell Crank Handle

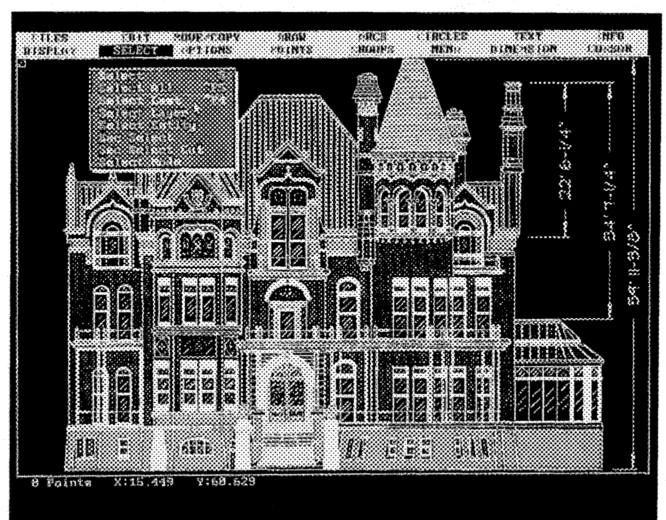


FIGURE 13.45 2D Illustration of a House

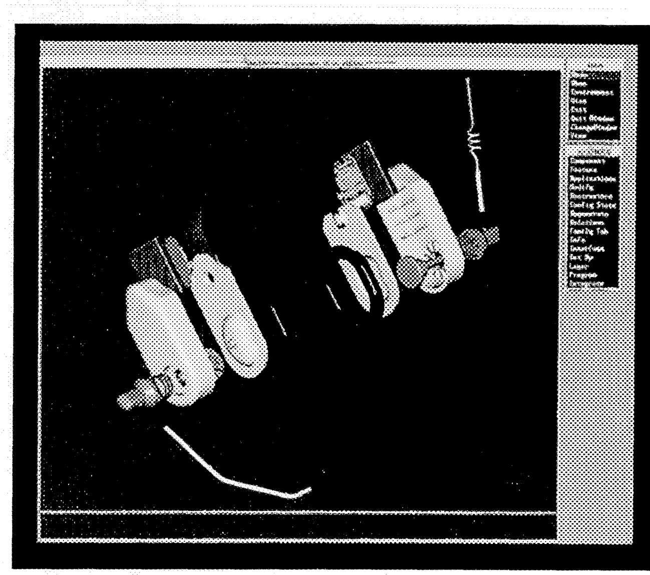
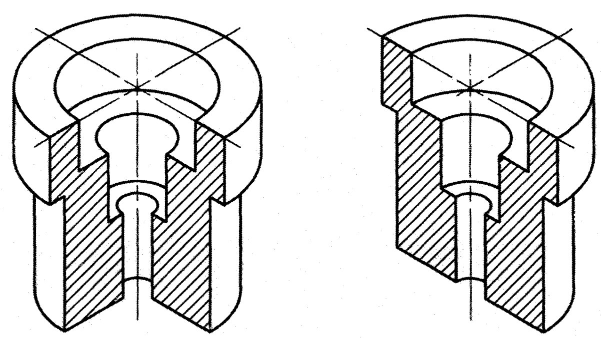


FIGURE 13.46 Exploded Shaded Assembly of a Headset

13.7.1 Sectional and Cutaway Views on Pictorials

Sectioned pictorials allow the viewing of the interior of a part or an assembly. When possible, section-cutting planes are passed through centerlines and parallel to one of the principal faces of the part. Figure 13.47 shows a half section (a) and a full section (b) of a part. Section lines in a half section should be drawn so that they would appear to coincide if the planes were folded together [Fig. 13.47(a)]. With a full section, all of the crosshatching should be drawn in the same direction—at the same angle and with the same spacing.

In assemblies, individual pieces are differentiated by appropriate symbols and by changing the direction of the section lining. When a section plane passes through a shaft, bolt, key, pin, or solid round item, it is desirable to run the section around that item and to show the entire bolt or shaft in the pictorial. Except for such cases, the section lines should indicate exactly what material has been cut. Figure 13.48(a) shows the first step in the creation of a section of a pictorial assembly. Each of the parts is blocked out. The assembly is then completed [Fig. 13.48(b)].



(a) Half section (b) Full section

FIGURE 13.47 Sectioned Pictorials

13.7.2 Pictorials and Break Lines

When the length of a part is beyond the size of the drawing format, and there are no features that require displaying and dimensioning, you may shorten the drawing with **break lines.** Position the break at a place on the part that does not interfere with the part features and the required dimensions. Freehand breaks are acceptable for most situations. Figure 13.49 gives the preferred and the acceptable methods for showing breaks.

13.7.3 Fillets and Rounds on Pictorials

Fillets and **rounds** usually can be highlighted or can be shown as straight or curved lines representing the filleted and rounded edges of a part, as in Figure 13.50. Highlighting and shading are drawn freehand. When a CAD system is used to generate a 3D model (Fig. 13.51), the model can display fillets and rounds in a variety of ways, including: as a wireframe with hidden lines and tangent curves shown [Fig. 13.51(a)]; as an edge representation with only the visible lines and tangent curves [Fig.13.51(b)]; as a solid

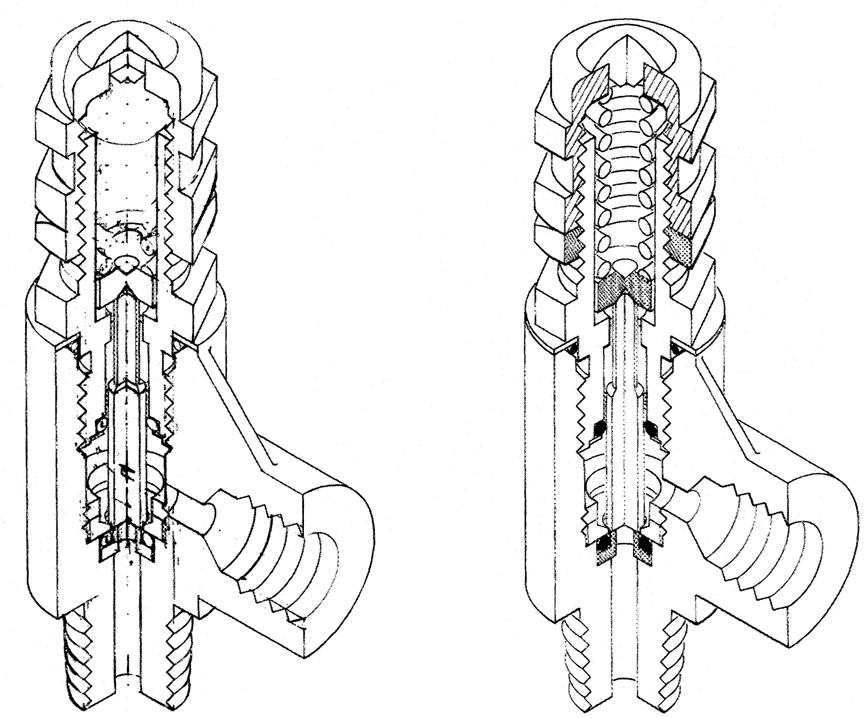


FIGURE 13.48 Pictorial Construction

(a) Construction of needle valve in an isometric drawing using a half section

(b) Completed pictorial of needle valve

model with fillets and rounds highlighted [Fig. 13.51(c)]. Fillets and rounds are covered in depth in Chapter 14.

13.7.4 Pictorial Thread Representation

Threads may be represented by a series of ellipses or circles uniformly spaced along the centerline of the thread. Shading increases the effectiveness of the thread appearance. In Figure 13.52, the exploded assembly has both internal and

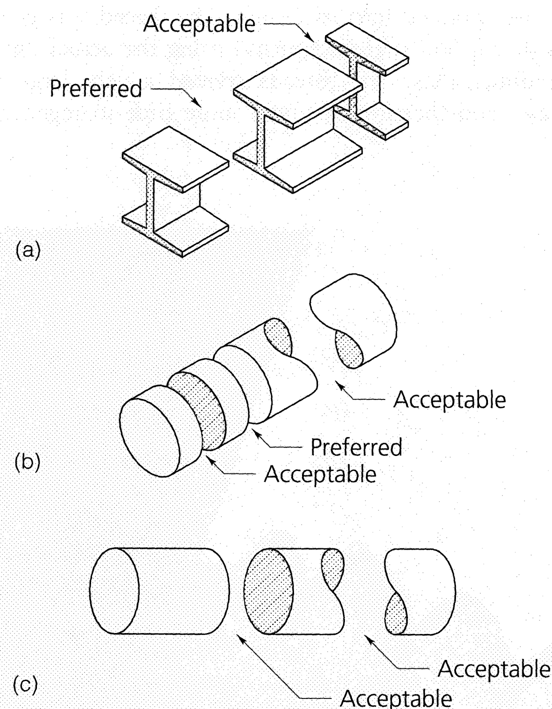
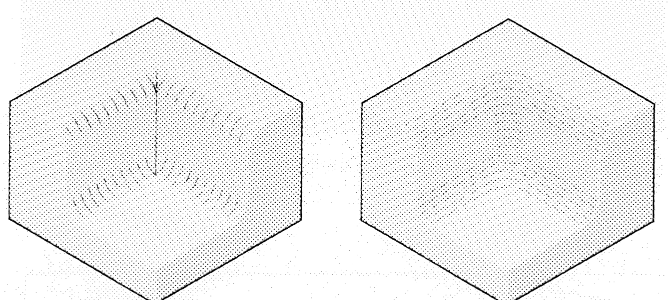


FIGURE 13.49 Break Lines for Pictorials



(a) Curved highlighting (b) Straight highlighting

FIGURE 13.50 Fillets and Rounds in Pictorials

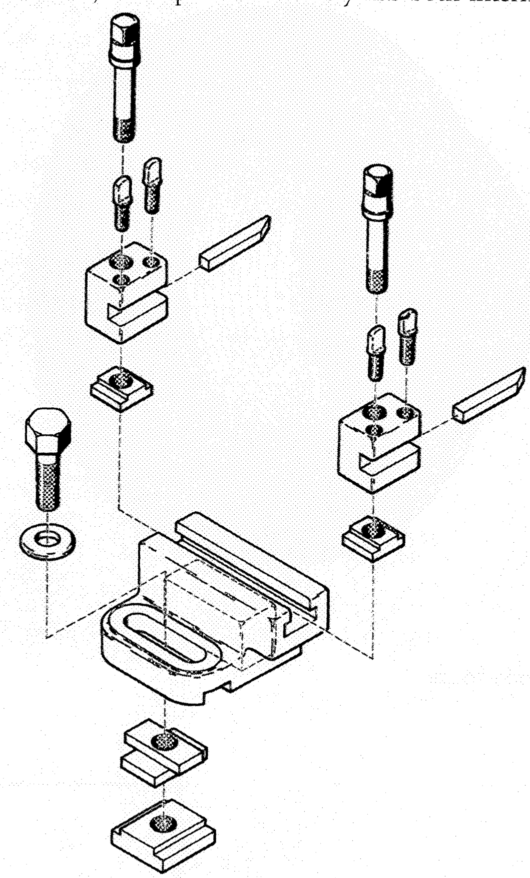
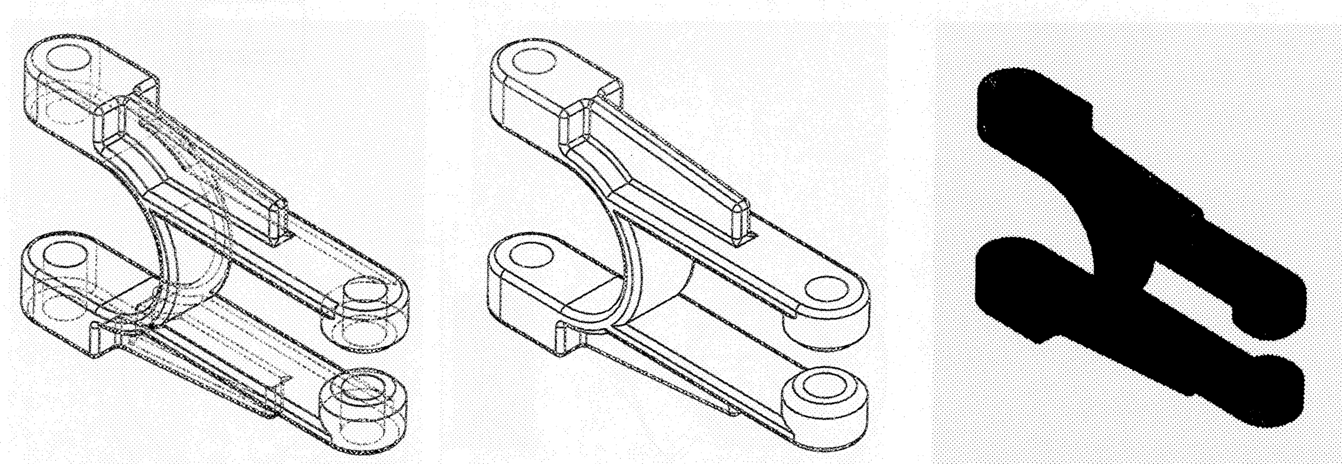


FIGURE 13.52 Exploded Assembly



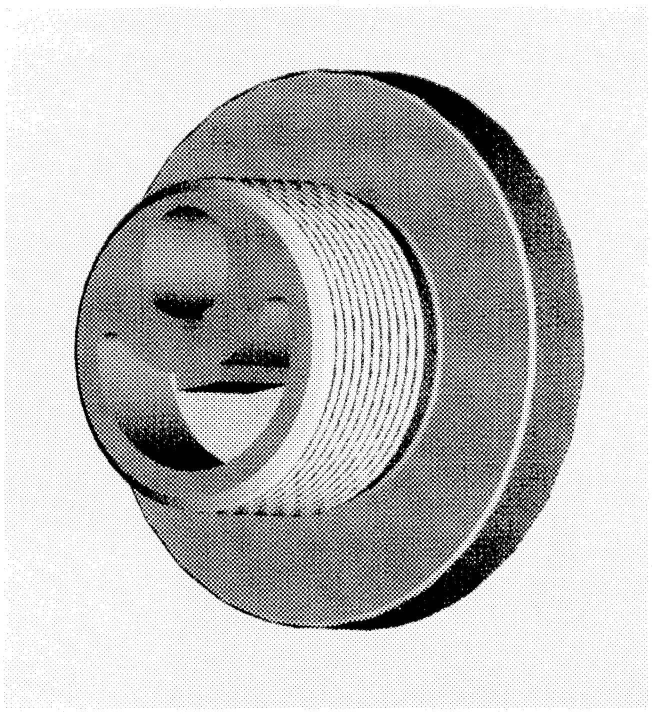
(a) With hidden lines and tangent curves (rounds and fillets)

(b) With tangent curves (rounds and fillets) and without hidden lines

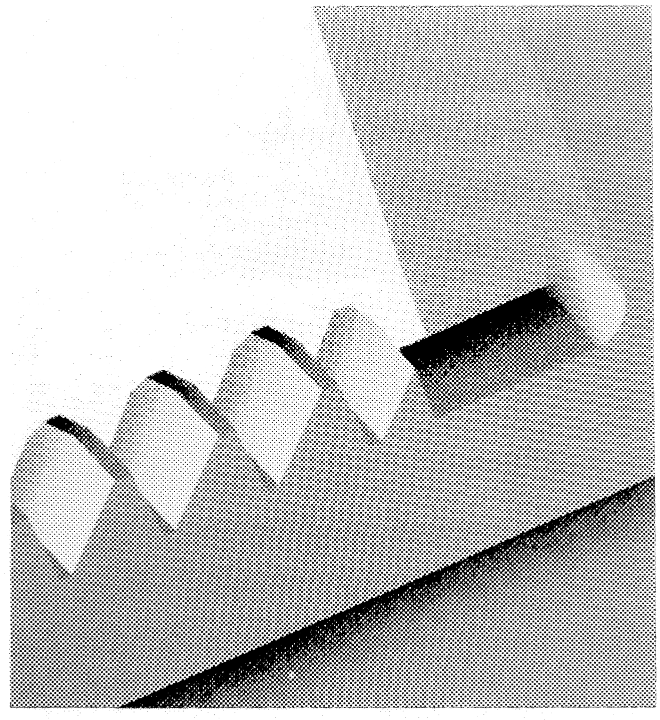
(c) Shaded image with rounds and fillets highlighted

FIGURE 13.51 3D Model

external threads. Threads should be evenly spaced, but it is not necessary to reproduce the actual *pitch* (distance between crests of the threads) or the exact number of threads. The solid model of the part in Figure 13.53 has a buttress thread incorporated into its design. The thread was generated with a *helical sweep* command using the actual thread profile dimensions. The threads created a very large file (database) and therefore required more time to regenerate



(a) Shaded image



(b) Closeup view of threads (shaded image)

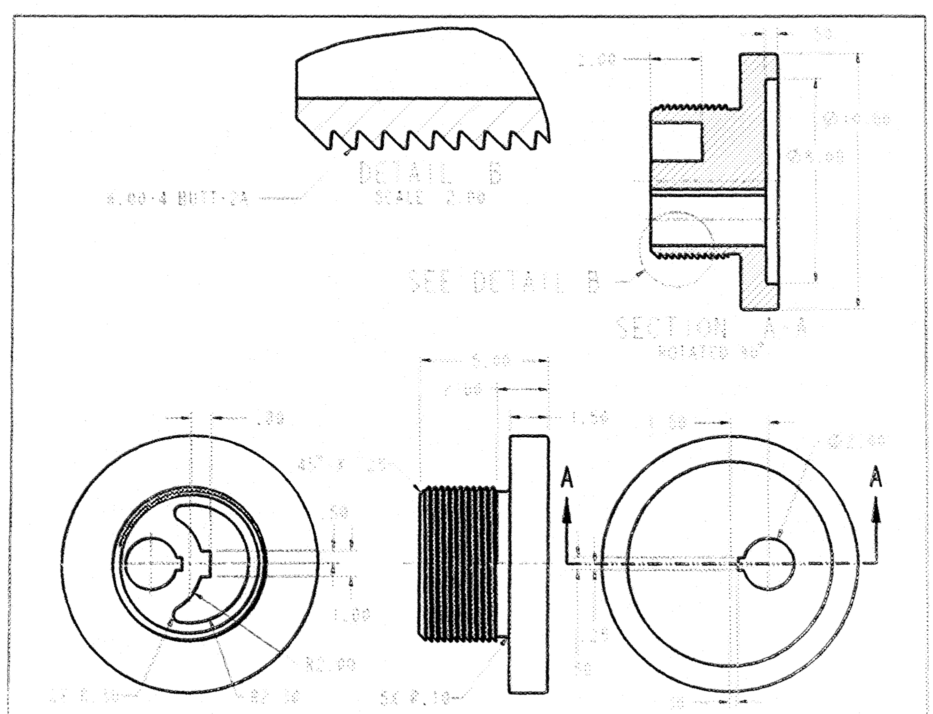
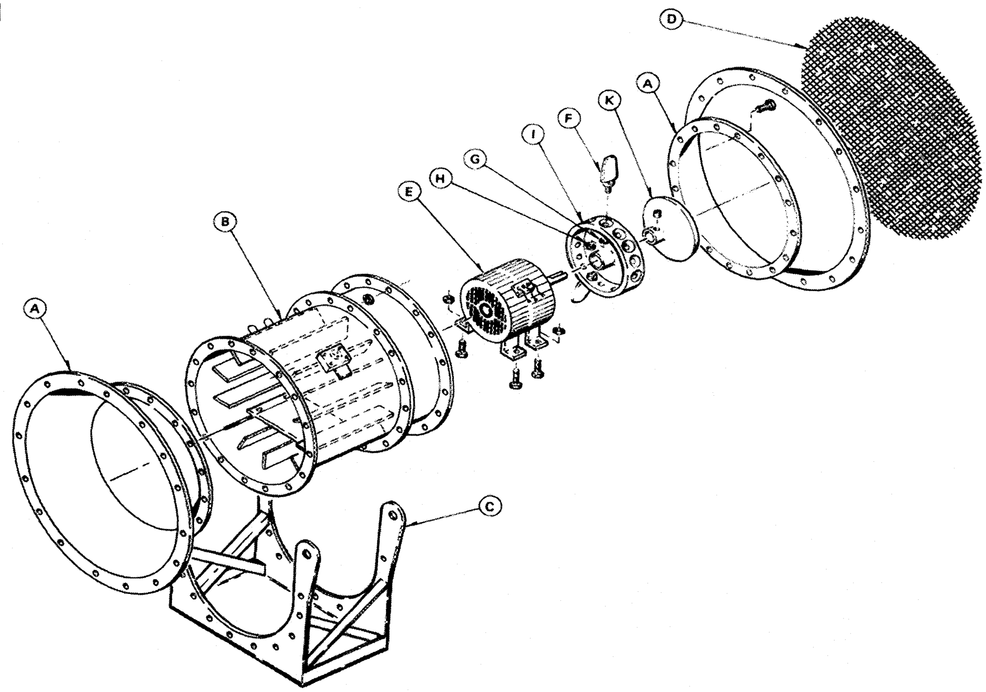


FIGURE 13.53 3D Part with Buttress Threads

(c) Detail drawing

FIGURE 13.54 **Exploded Pictorial**



while the part was in the design stage (25 minutes of laser plotting time). Normally the threads on a 3D solid model will be displayed as cosmetic threads (see Chapter 17).

13.7.5 Exploded Assembly Pictorials

A pictorial drawing showing the various parts of an assembly—separated but in proper position and alignment for reassembly—is called an **exploded assembly** (Fig. 13.54). Exploded pictorials are used extensively in service manuals and as an aid in assembling or erecting a machine or structure. Any type of pictorial drawing may serve this purpose, and the shading may be as simple or as complete as desired. In Figure 13.55 the exploded assembly is drawn isometrically. Highlighting and shading have been added to provide a realistic illustration of the pieces.

Each piece in an exploded assembly should be connected to its mating part by a centerline, as shown in Figures 13.54 and 13.55. If there is insufficient room to extend the exploded pieces out from each other in one line, the piece can be moved, as in Figure 13.52, where the bolt and washer have been brought forward and up. The *jogged centerline* still connects related pieces.

You May Complete Exercises 13.5 Through 13.8 at This Time

13.7.6 Pictorial Drawing Using CAD

The designer seated at a 2D CAD station gets no more feeling of three-dimensional definition from a single view on the screen than he or she would from a drawing on paper. However, the designer can visualize the three-dimensional part better when the features are seen in the view that indicates depth, even if in an artificial way. Creating a drawing with *perspective, isometric,* or *oblique* views helps to indicate depth to the designer.

Two-dimensional pictures can be made to look more realistic with **isometric views** of three-dimensional objects,

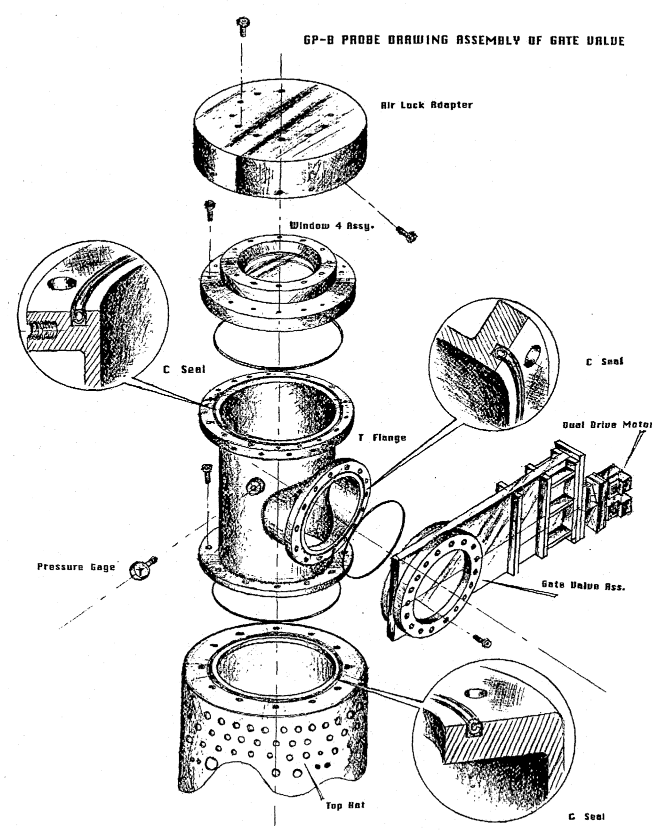


FIGURE 13.55 **Exploded Sketch of Gravity Probe Assembly**

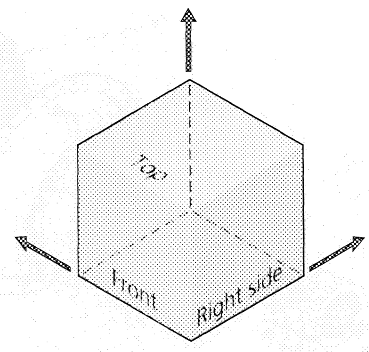


FIGURE 13.56 Isometric Axes

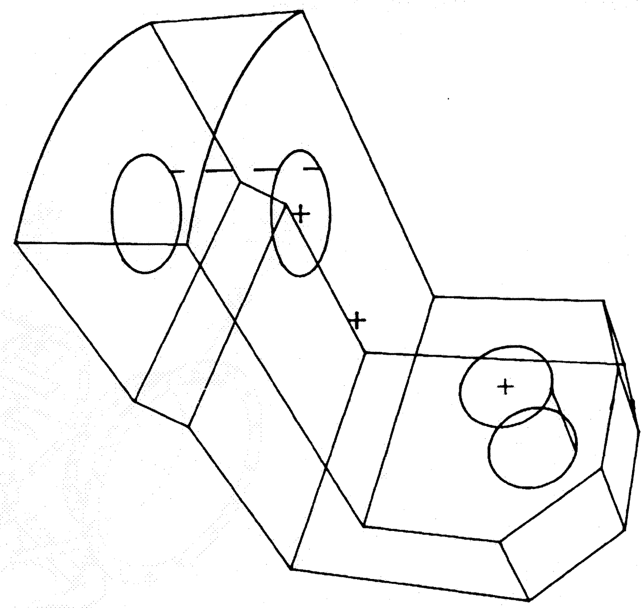


FIGURE 13.58 3D Model Displayed in Perspective

such as the cube shown in Figure 13.56. **Oblique views** also can create illustrations that simulate the third dimension (Figure 13.57).

The **perspective** capability is different from parallel orthographic projection in that it uses depth information. Perspective drawings can be constructed on CAD systems, but normally the perspective is generated from an existing view of a 3D modeled part. The size of the part is drawn in proportion to its distance from the viewer. The perspective projection of the part shown in Figure 13.58 demonstrates this scaling. The **PERSPECTIVE** command temporarily distorts the part so that it can be viewed in perspective. This option is limited to three-dimensional systems with modeling capabilities. Some systems automatically set the perspective parameters to give the part a certain amount of distortion. The location of the station point can be specified, or the default can be used. The default location assumes that the part is in the center of the screen. The distance between the station point and the picture plane can also be specified.

Some CAD systems still offer the choice of drawing in isometric. Isometric drawings are simulations, not true 3D drawings—once drawn, the object cannot be viewed from another angle or in perspective, hidden lines cannot be removed.

To construct a drawing in isometric via AutoCAD, an isometric plane, which utilizes the **SNAP** and **GRID** commands, can be activated. The isometric **SNAP** grid has three major axes. Assuming that rotation has not also been applied to the Snap grid, the axes are vertical, 30°, and 150° (Fig. 13.59).

ORTHO mode and keyboard pointing with the cursor movement keys can only deal with two of the three axes at a time. Therefore, AutoCAD assumes you are drawing on one of three isometric planes (left, top, or right), each of which has an associated pair of axes. The meaning of **ORTHO** mode and the action of the cursor movement keys are then modified to follow the current pair of axes.

You can set the **SNAP** to isometric by means of the **SNAP STYLE ISOMETRIC** command sequence. The current isometric plane also determines the orientation of the isometric

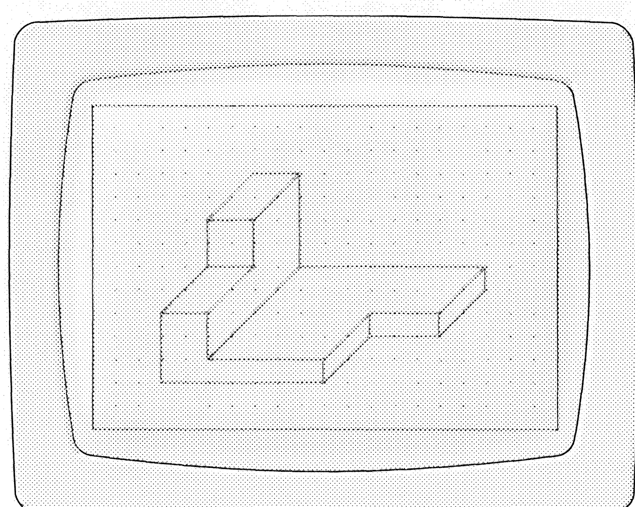


FIGURE 13.57 Oblique Drawing

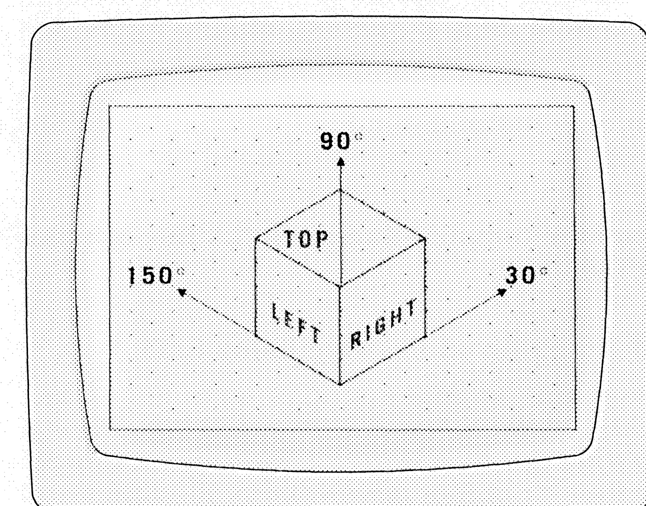


FIGURE 13.59 AutoCAD Isoplane Axes

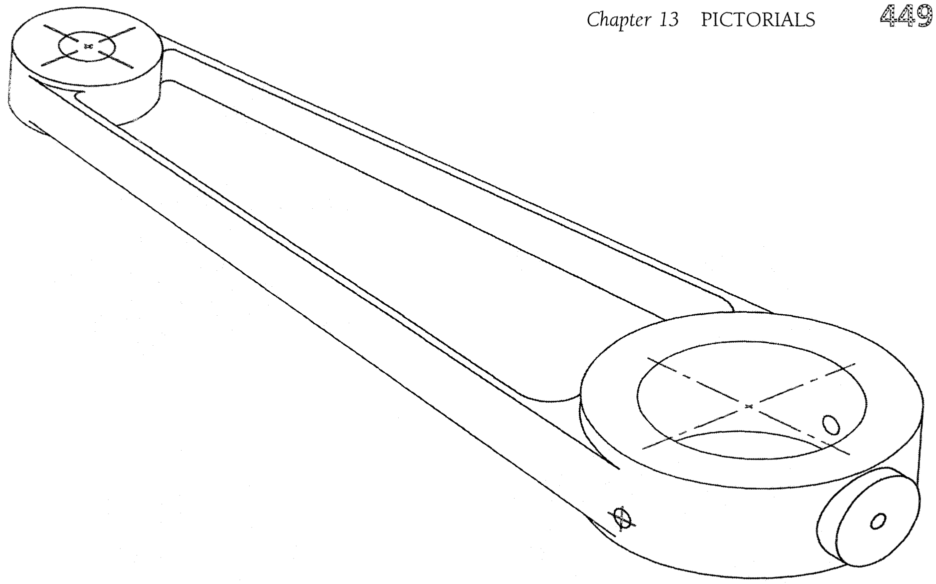


FIGURE 13.60 Isometric Drawing Created on AutoCAD

circles drawn via the **ELLIPSE** command. Figure 13.60 required the **ELLIPSE** command to represent the holes in the part. The **ISOPLANE** command selects the current isometric plane and thus the current pair of axes:

```
Command: ISOPLANE
Left/Top/Right/(toggle):
```

LEFT This option selects the left-hand plane, defined by the 90° and 150° axis pair. When **SNAP** mode is on, the up and the down cursor keys move you along the 90° axis, and the left and the right cursor keys move you along the 150° axis.

TOP The **TOP** isometric plane is the top face of the cube; it uses the 30° and 150° axis pair. When **SNAP** mode is on, the up and the down cursor keys move you along the 90° axis, and the left and the right cursor keys move you along the 150° axis.

RIGHT Responding to the prompt with **RIGHT** selects the right-hand plane, defined by the 90° and 30° axis pair. On this plane, the up and the down cursor keys move you along the 90° axis, and the left and the right cursor keys move you along the 30° axis.

The **ISOPLANE** command and the Isoplane toggle key are but two of the four methods available for setting the current isometric plane. The **SETVAR** command and **AutoLisp** are the other methods. The current isometric plane is maintained in the system variable **SNAPISOPAIR.**

While perspective, isometric, and oblique views are useful for depicting three-dimensional objects more realistically in a two-dimensional view, many CAD systems offer the capability to employ solid models. Here, the illustration of depth can be added through the use of color and shading. Figure 13.61 shows a solid model created on AutoCAD. The addition of color and shading enable the viewer to see the

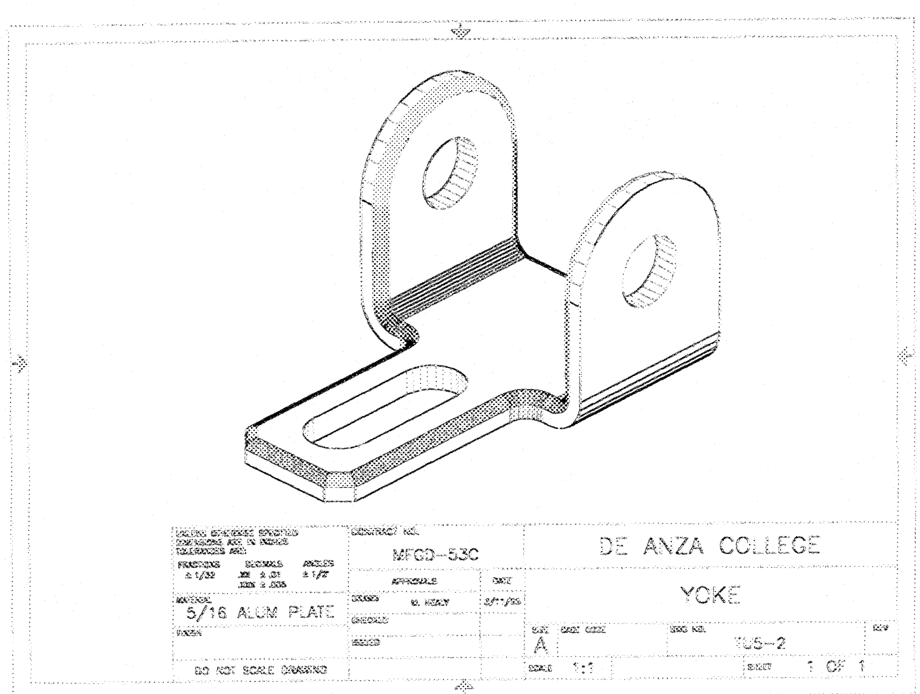
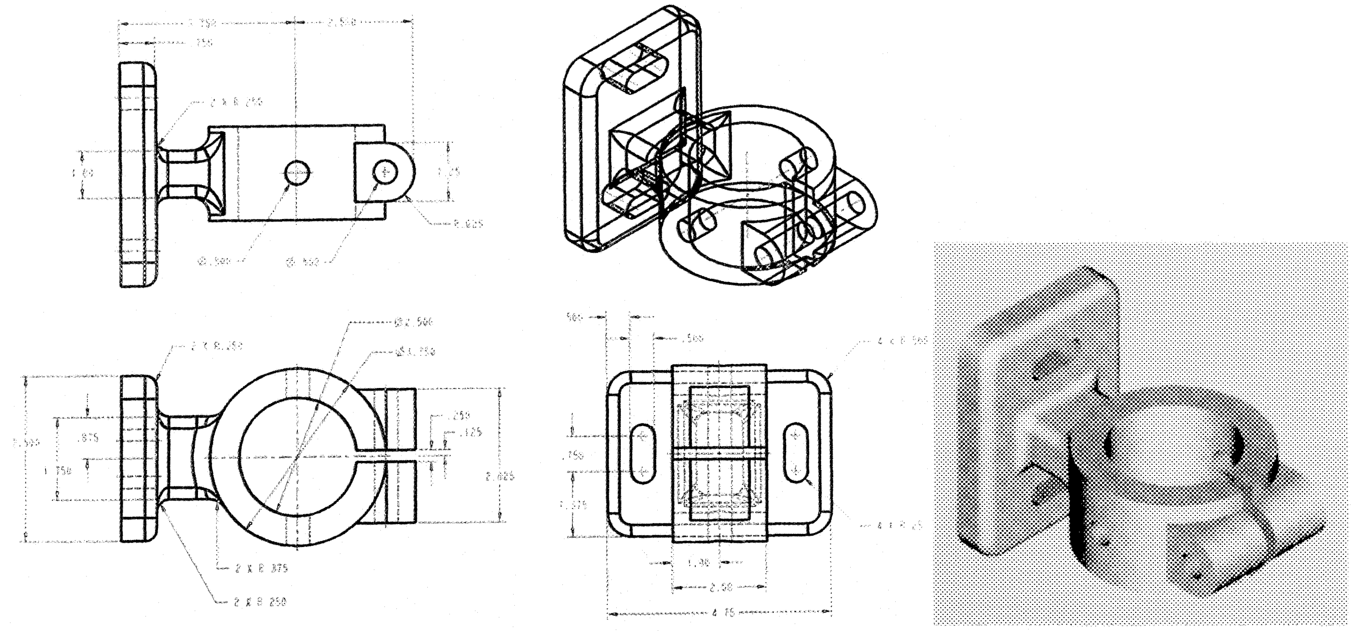


FIGURE 13.61 Part Created on AutoCAD Using Solids



(a) Detail of part

(b) Shaded image of model

FIGURE 13.62 Parametric Model

part's curved geometry. The settings used in this figure did not produce a smooth, high-quality plot. In Figure 13.62, on the other hand, the model is displayed with high-quality graphics. Here the part was detailed [Fig.13.62(a)] and displayed [Fig. 13.62(b)] via CAD with high-quality graphics and a laser color plotter.

13.8 3D MODELING AND 2D DRAWING

CAD systems with three-dimensional capabilities have been designed so their use is as natural as possible. This includes recognizing the different thought processes involved and normally associated with design and drafting. Typically, two operational modes exist: model mode and draw mode (Fig. 13.63). **Model mode** is for designing a part [Fig. 13.64(a)] and generating numerical control data [Fig.13.64(b) and (c)]. **Draw mode** can create detail drawings of the part by using the 3D model in views. The part in Figure 13.65(a) was designed as a 3D model [Fig. 13.65(b)] and detailed in draw mode [Fig. 13.65(c)]. Operations performed in model mode create model descriptions and model geometry of the parts designed. The database created in this mode for a particular part model is distinct and separate from that for all other models.

Model mode (**model space** for AutoCAD) can create a representation of a real-world three-dimensional object. A designer in model mode is creating an actual three-dimensional layout of model geometry. Draw mode (**paper space** for AutoCAD) creates drawing representations of the model by modifying or editing the model's geometry to change its visual appearance. Draw mode uses the model's database for information representing the model's geometry.

A designer in draw mode accesses the model database to create two-dimensional drawings from that database. Detail drawings with dimensions, text, and notes are created in draw mode. Drawing items can also be inserted in draw mode (draw entities) and are separate from the model items. One way to understand this more clearly is to imagine the model as being created within a transparent three-dimensional box in which the model's geometry exists physically in three-dimensional space. In contrast, a drawing created from the model in draw mode shows the model's projection on the sides of the box (Fig. 13.66).

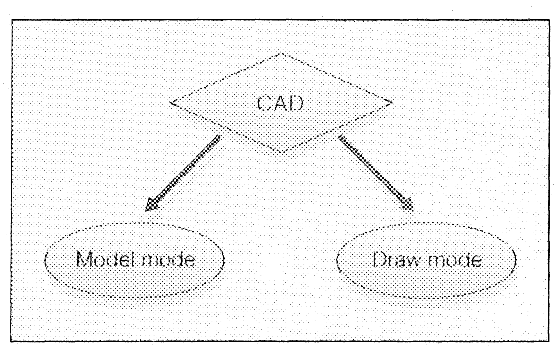
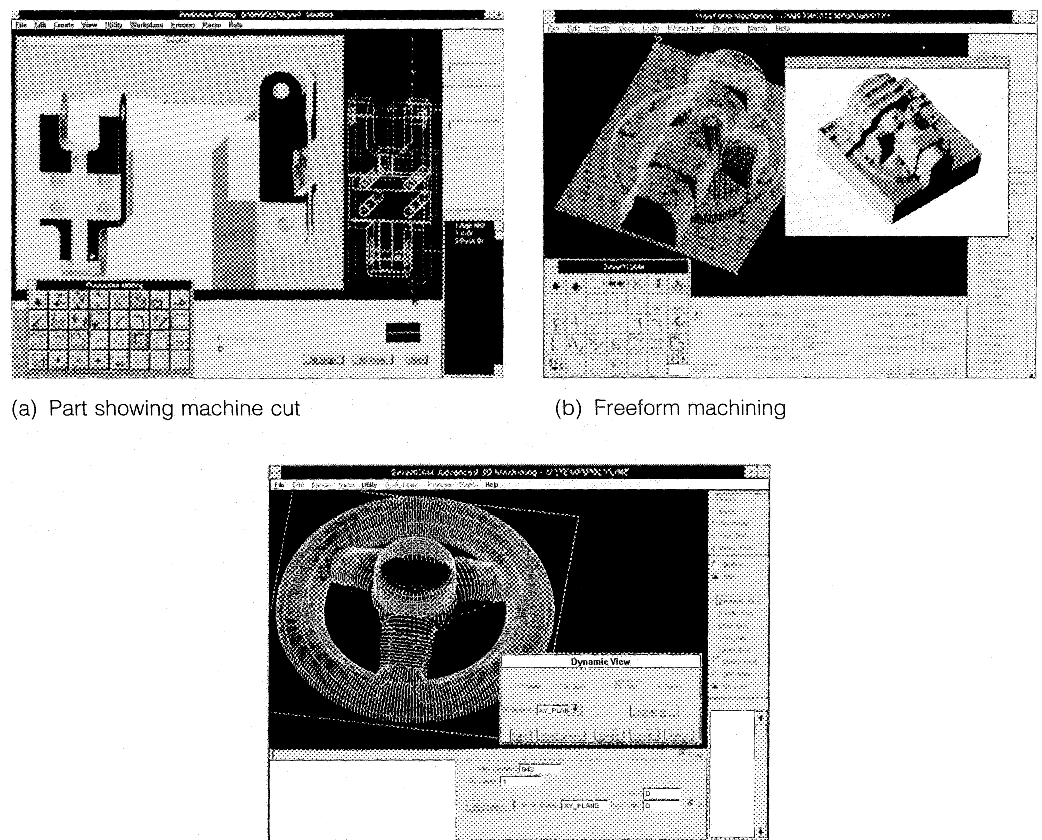


FIGURE 13.63 Model Mode and Draw Mode



(a) Part showing machine cut

(b) Freeform machining

(c) Advanced 3D machining

FIGURE 13.64 3D Machining Using CAM Program

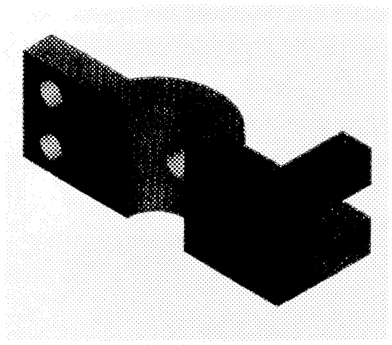
When draw mode is used to edit the model's visual appearance, the changes made are only pictorial, similar to air-brushing a photograph. In model mode, in contrast, the changes are made to the model and are reflected in all views. Changes made in draw mode allow the detailer to manipulate the model for drafting purposes. For example, solid lines could be changed into dashed lines to show that they are hidden or to indicate where a hole might be located. Lines can be erased temporarily for visual clarity or to prevent certain lines from being plotted (as hard copy) twice if they are normally seen one behind the other in a particular view. In this way the model's database retains its integrity while different views of the model are altered as needed for detailing purposes.

A large number of drawings can be created in draw mode and associated with a specific mode (Fig. 13.67). These drawings can be created at any size. It is important to realize that a drawing is a collection of views that depict the part. A view shows the part at a particular orientation (e.g., top, front, right). Three-dimensional CAD provides the capability to define any number of views for a drawing (Fig. 13.68). These views can be scaled at any display size. Additionally, a view will be defined by a border or frame whose size can also be redefined.

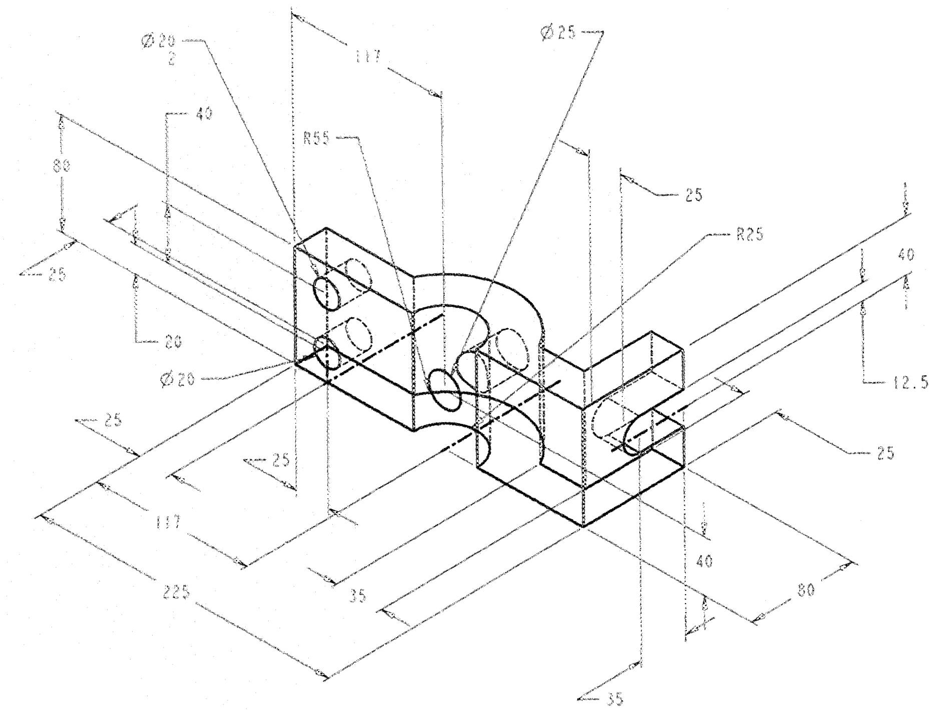
Annotation can be added to the drawing to enhance it. Annotation includes dimensions, text, and graphic items. Drawing items—lines, points, arcs, circles—look the same as model items, except that they are always associated with the drawing and never affect the model itself (and are never added to the model's database). The model's geometry is transferred pictorially to draw mode and the designer can alter it, but only the appearance is altered. In addition, the transferred geometry can be manipulated in any number of ways. For example, it can be scaled larger or smaller in size, sectioned, or manipulated into a particular view or set of views simultaneously. Two-dimensional geometry can be added (or deleted) for purposes of presentation. The geometry of the model can be hidden selectively to stimulate normal viewing. Pictorial views can also be generated from the 3D database. Pictorial illustrations for presentations, manuals, and other traditional uses are all generated automatically from the 3D database, not drawn individually with dimensions taken from a detail of the part.

13.8.1 Coordinate Systems

Model mode and draw mode have distinct spatial coordinate systems associated with them. Model space is the **three-dimensional coordinate system** in which model geometry is constructed. Model space is considered to be a real-world space existing in the three directions (axes). Those axes are X (horizontal, or left–right, equating to width), Y (vertical, or up–down, equating to height), and Z (back and forth, which



(a) 3D part shown as a shaded image



(b) Rotated view of part with dimensions displayed

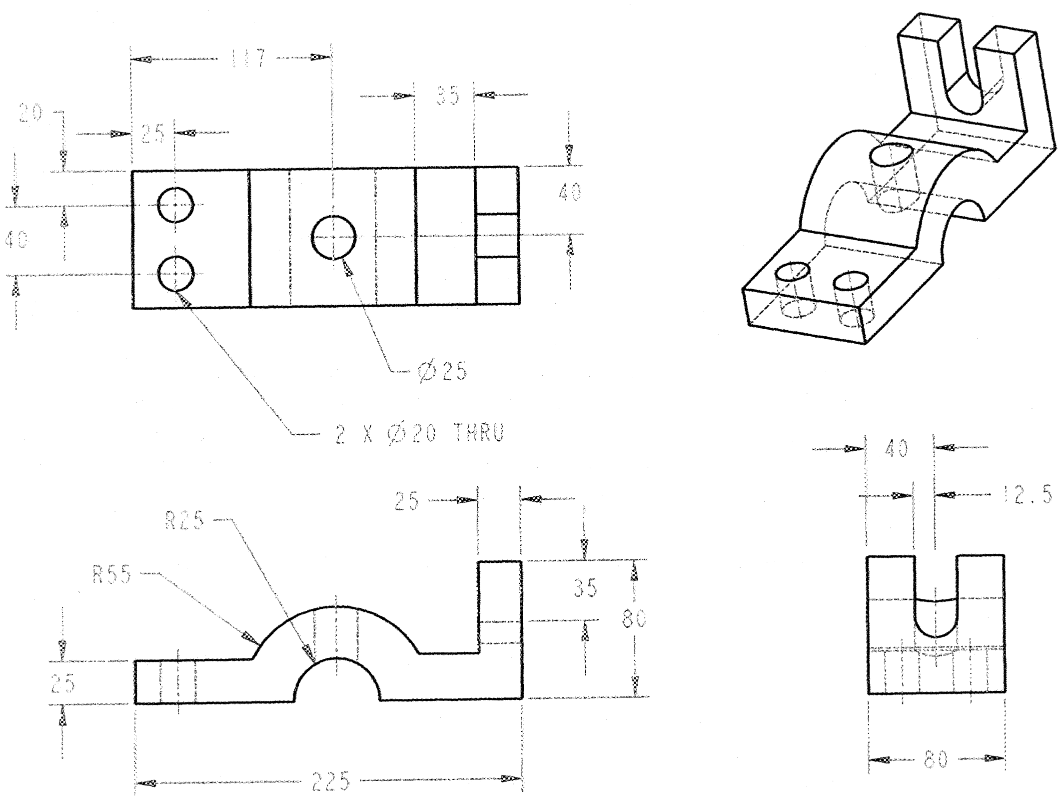


FIGURE 13.65 3D Design and Documentation

(c) Dimensioned detail

means in and out from the screen, equating to depth, as shown in Fig. 13.69). The location of the origin or all three axes is defined by the designer.

Drawing space is a **two-dimensional coordinate system** associated with draw mode. Drawing space is considered to be a flat, two-dimensional coordinate system existing in two

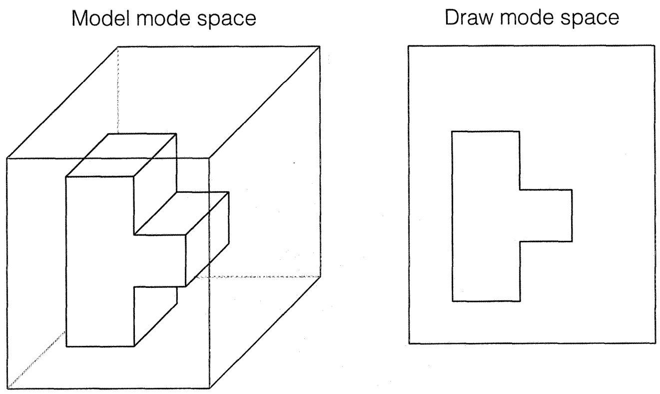


FIGURE 13.66 Model Mode Space vs. Draw Mode Space

directions, whose axes are X and Y (Fig. 13.70). The *illusion* of depth, or replicating the Z axis, can be created on a two-dimensional coordinate system. Figure 13.71 illustrates the difference between model space and drawing space and their associated coordinate systems. The origin, or the intersection of the X and Y axes, is located at the lower left corner of the drawing.

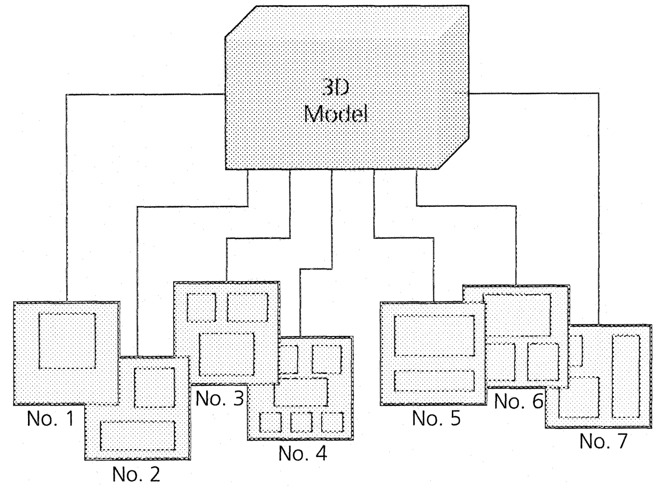


FIGURE 13.67 Using the 3D Database to Generate Drawings. Any number of drawings can be created automatically from the original 3D model. The drawings are associative to the part geometry: Any change on the model will also update the drawings. On parametric systems, the model, the assembly, and the drawings are linked and associative: Any change made on the assembly updates the drawings and the model of the part, and any change on the drawings will update the part model and the assembly.

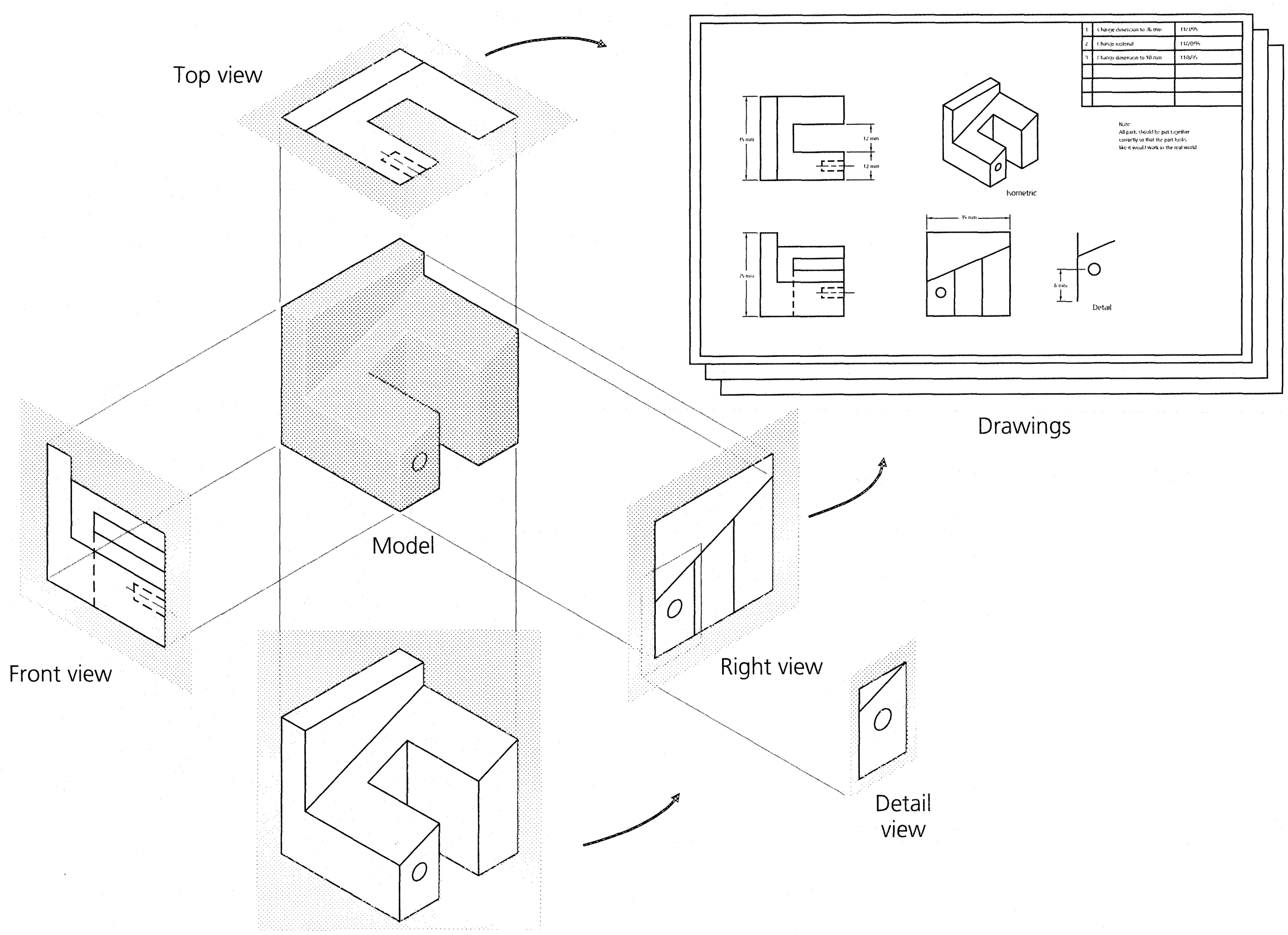


FIGURE 13.68 The Model Is Used to Generate the Views of the Drawing

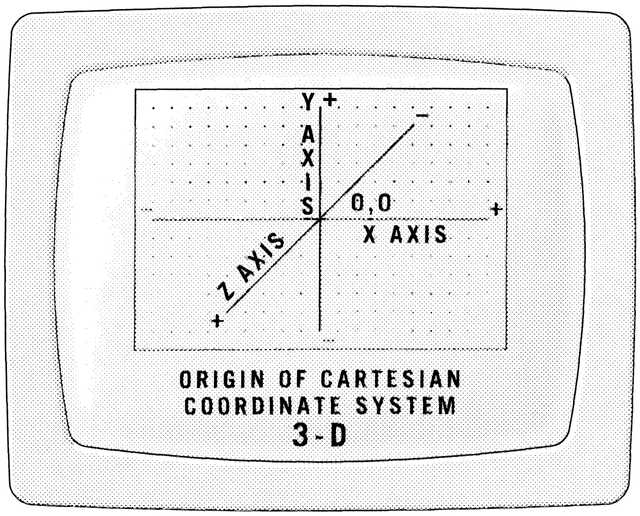


FIGURE 13.69 The Center of the Screen Is the Origin on Most CAD Systems. The Z axis comes out toward the viewer. This is the vertical axis, and when looking into it you are seeing the top view of the part (plan view).

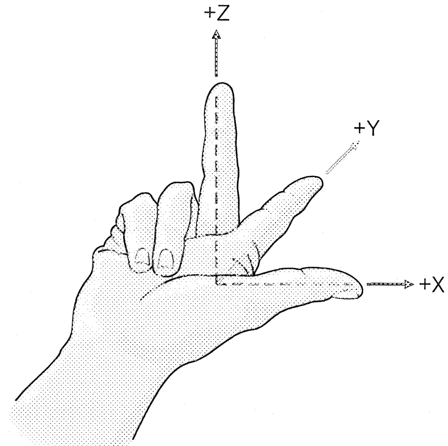


FIGURE 13.72 Right-Hand Rule. The middle finger represents the vertical (Z) axis, the thumb the X axis, and the pointing finger the Y axis. Use your right hand to help visualize the coordinate system in space (for the top—plan view) and when applying it to one of the standard orthographic views or an auxiliary surface.

13.8.2 Cartesian, Cylindrical, and Spherical Coordinate Systems

Coordinate systems are always displayed with X, Y, and Z axes. This can be visualized by using your right hand as shown in Figure 13.72 (the right-hand rule). The thumb is the X axis, the pointing finger is the Y axis, and the middle finger is the Z axis.

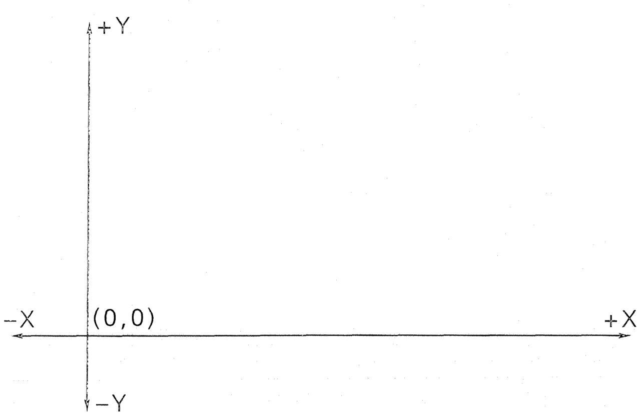


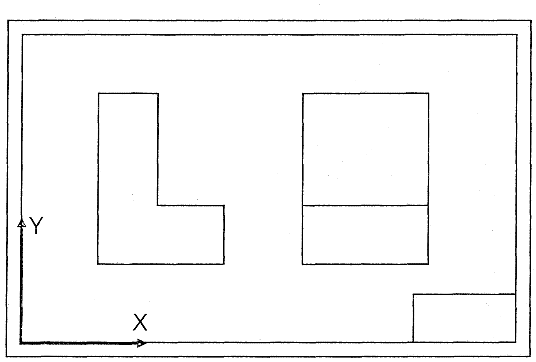
FIGURE 13.70 Cartesian Coordinate System

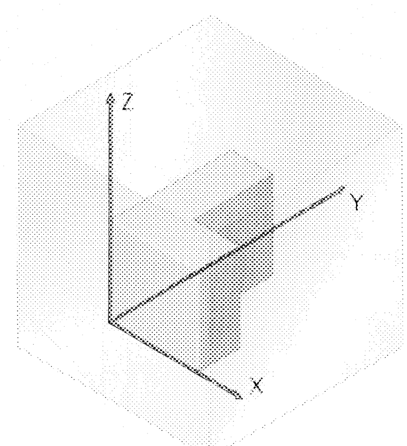
When referencing a coordinate system to make other features, the coordinate system can be interpreted as follows (Fig. 13.73):

Cartesian Coordinate values are interpreted as X, Y, and Z.

Cylindrical Coordinate values are interpreted as radius (r), theta (θ), and Z.

Spherical Coordinate values are interpreted as radius (r), theta (θ), and phi (ϕ).





Drawing coordinate system

Model coordinate system

FIGURE 13.71 Drawing Coordinate System vs. Model Coordinate System. The drawing coordinate system is for drawing entities such as the border, title block, revision block, notes, and cosmetic geometry. The model coordinate system is for establishing the part's features in space.

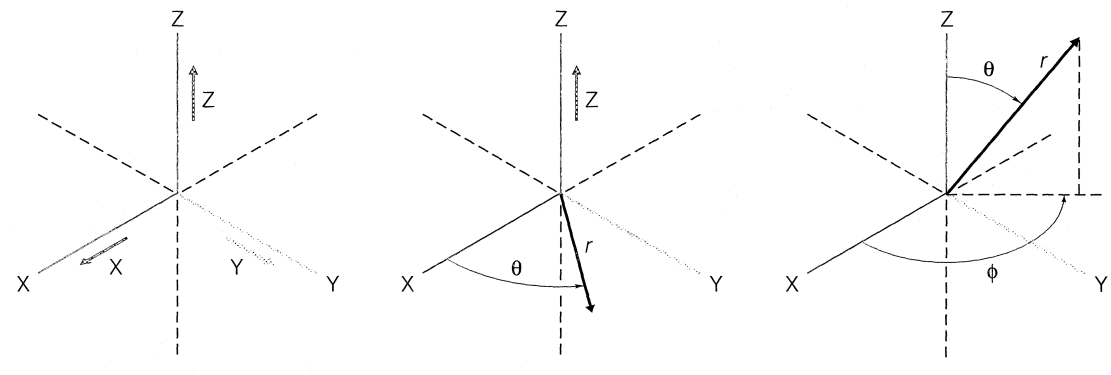


FIGURE 13.73 Cartesian, Cylindrical, and Spherical Coordinate Systems

13.8.3 Modeling with Coordinate Systems

On most CAD systems, coordinate systems are reference features that can be added to parts and assemblies for the following purposes:

- Calculating mass properties
- Assembling components
- Placing constraints for finite element meshing
- Providing manufacturing operation reference for tool paths
- Locating other features (coordinate systems, datum planes, planes and axes, imported geometry, etc.)

The following example creates a coordinate system [Fig. 13.74(a)] as base feature for Pro/Engineer:

1. Choose **Create** from the FEAT menu, then **Datum.**
2. Choose **Coord Sys** from the DATUM menu.
3. The coordinate system named CS0 will be created.

13.8.4 Datum Planes and Coordinate Systems

Three orthogonal **datum planes** can be created as the base feature, prior to adding any solid feature. This is helpful when the first solid feature is to be a sphere, a toroid, or sculptured surfaces; these tend not to have the planar surfaces needed to reorient the model or to specify sketching planes.

To create a default datum plane on Pro/ENGINEER use the following commands [Fig. 13.74(b)]:

1. Choose **Feature** from the PART menu.
2. Choose **Create** from the FEAT menu, then **Datum.** Then choose **Plane.**
3. The MENUDTM OPT pick **Default**.

Default creates three orthogonal datum planes intersecting at the default origin (X0,Y0,Z0). Three datum planes, with the names DTM1, DTM2, DTM3, appear in the center of the screen at right angles to one another [Fig. 13.74(b)]. The coordinate system and datum planes are then used to model the part geometry [Fig. 13.74(c) and (d)]. The drawing is created from the model, as shown in Figure 13.74(e), and then displayed pictorially as a shaded image [Figure 13.74(f)].

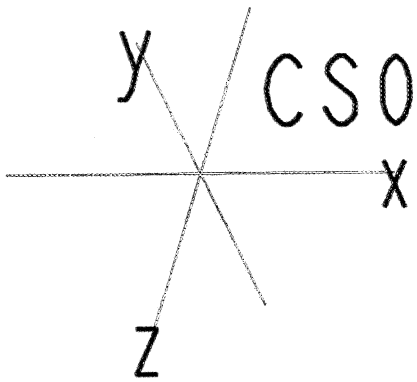
13.8.5 Construction Planes

The concepts and application of **construction planes** is another CAD feature that provides a real breakthrough for the designer. The use of construction planes makes the construction of model geometry easier and faster. Instead of employing datum planes to create geometry, many systems use construction planes. On AutoCAD, the default coordinate system is called the **world coordinate system (WCS)** and user-defined construction planes are referred to as the **user coordinate system (UCS).**

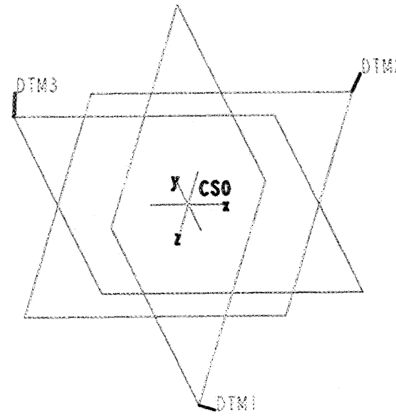
A construction plane is a planar surface that can be selected from the model's geometry. A construction plane can be predefined because it is associated with a specific view; e.g., for a FRONT view the construction plane is FRONT [(Fig. 13.75(a)]. A designer can also define a construction plane from preexisting geometry in any view that shows the model in three dimensions. This is called an **auxiliary construction plane** (or user coordinate system) [Fig. 13.75(b)]. You can think of an auxiliary construction plane as a secondary coordinate system. Once a construction plane is defined and selected, the following types of activities can take place.

- All digitizes (picks) are projected directly onto the construction plane.
- A coordinate system is set up when a construction plane is defined by the designer. The origin of the coordinate system is the same as the origin of the construction plane. Geometry can be created and/or located via this coordinate system; the coordinate system will be "local."
- Construction planes can be used to insert geometry or text, directly on the plane or parallel to it.

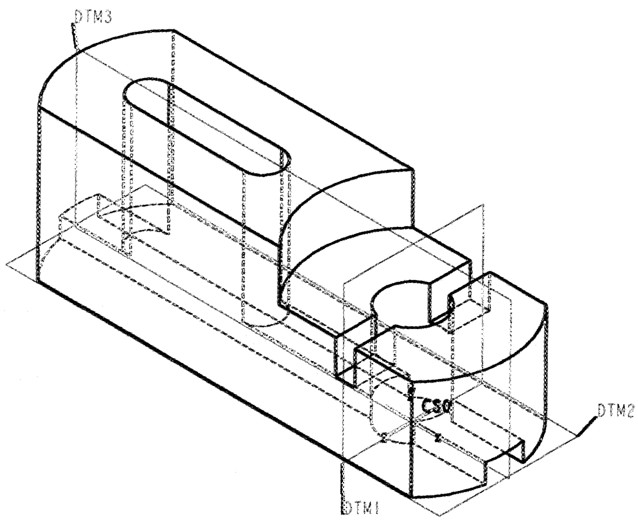
Construction planes serve as design aids. Along with the default top view (plan view) coordinate system (world coordinate system), an unlimited number of user construc-



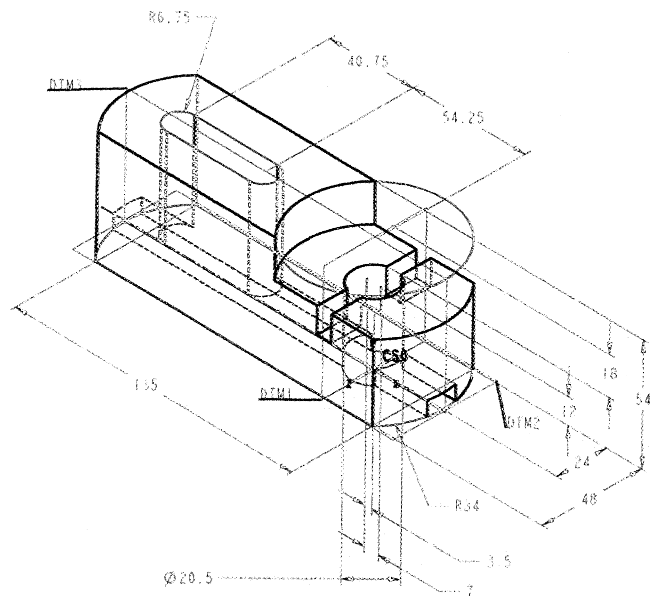
(a) CS0 (coordinate system zero)



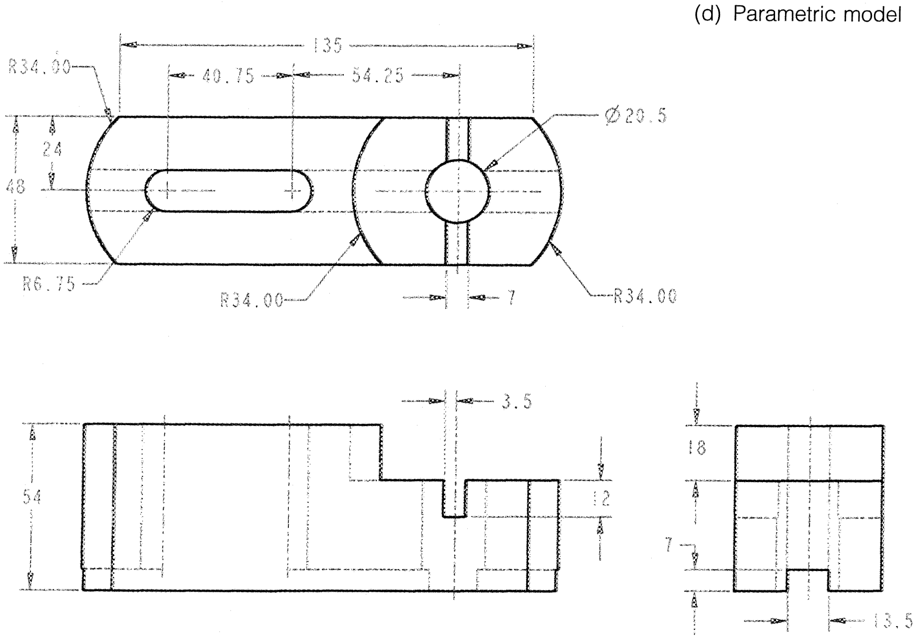
(b) DTM1, DTM2, and DTM3 default datum systems, with the coordinate system as the origin and the three datum planes intersecting at (0,0,0).



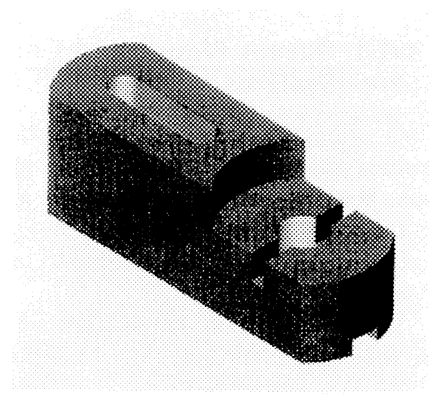
(c) Part geometry established using the datum planes and the coordinate system



(d) Parametric model

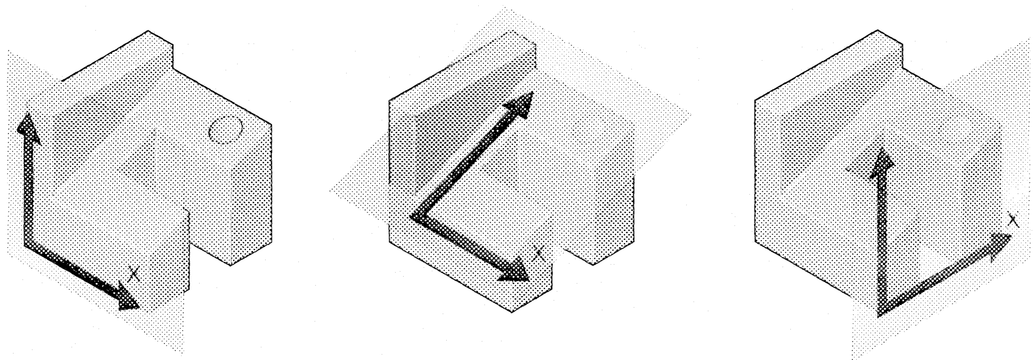


(e) Model used in **DRAW** mode



(f) Shaded model

FIGURE 13.74 Creating a Part



(a) Construction plane parallel to the front side (b) Construction plane parallel to the part's inclined surface (c) Construction plane parallel to the right side view

FIGURE 13.75 Using Construction Planes

tion planes can be activated or created. If multiple views are selected by the designer, each orthographic view will have an equivalent construction plane associated with it. For example, a front view has a construction plane on, and therefore parallel to, the front surface [Fig. 13.75(a)]. A right side view has a construction plane parallel and on its surface [Fig. 13.75(c)]. The inclined construction plane is shown on and parallel to the inclined surface in Figure 13.75(b). These types of construction planes allow geometry items to be projected directly onto them, and facilitate a two-dimensional approach to model construction. Figure 13.76 shows a three-dimensional model and a variety of construction plane coordinate systems that could be established to model the part.

The power of construction planes can be seen in a three-dimensional approach to part construction. For example, if the designer were constructing model geometry from one view along with the dynamics capabilities, designer-defined construction planes would be useful. Assume that the model was displayed in some rotated view. The designer wants to add some circles to one side or face of the model and the particular side is not directly parallel to the screen. Only the side or plane of the model to which the circles will be added would need to be defined. Once the side or construction plane is defined and selected, explicit coordinates can be specified or the location of the circles can be digitized, and the circles are added, displayed elliptically, as they should be.

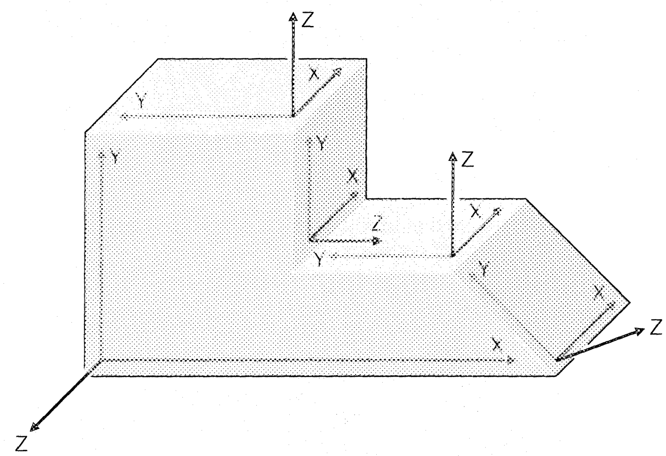


FIGURE 13.76 Auxiliary Coordinate Systems (User Coordinate System on AutoCAD). With commands, these can be placed on any surface of the part or oriented in space. The coordinate systems are construction planes and serve to place geometry and construct the 3D model.

13.9 CAD PICTORIAL ILLUSTRATIONS

After a part is modeled [Fig. 13.77(a)], it can be rotated, enlarged, or zoomed into a desired pictorial position and type (rotated or perspective). This capability eliminates the need to redraw the part as an isometric, oblique, or perspective drawing. In Figure 13.77(b), the model of the valve housing has had all hidden lines removed via a **HIDE** (hidden line removal) command. The model is now visually correct. The 3D model is used to create a drawing showing correct visibility, including all hidden lines, and centerlines, as shown in Figure 13.77(c). Note that the inclusion of a pictorial view (upper right corner) on the detail drawing enhances the understanding of the part's geometry, especially for nontechnical personnel. In this figure the dimensions are not included. The last illustration of this part shows a shaded image [Fig. 13.77(d)], which can be used for any of the traditional pictorial applications—display, manuals, catalogs, etc.

After the CAD illustrator displays the desired pictorial views, the 3D model can be embellished as required. Shading, color, or other artistic qualities help to prepare the final output as an illustration for technical manuals, advertisements, training manuals, sales literature, or design pre-

Applying Parametric Design . . .

PICTORIALS FOR PARAMETRIC MODELS

Pictorial illustrations are generated directly from the 3D model database. There is no need to recreate a part model via isometric, oblique, or perspective projection. The model can be displayed and oriented in any position automatically. You may select and orient the part to provide the required view orientation to display the part from underneath or from any side or position. Perspective projections are completely automated with a simple selection from a menu. The model can be spun around, reoriented, and even clipped to show the interior features. When assemblies are illustrated, you have the choice of displaying all components, and subassemblies, or any combinations of the design (see Fig. A).

Images of your parametric model are displayed with a wide variety of choices. The model can be displayed with any combination of hidden lines, coordinate systems, centerlines, or datum turned on or off, as required (see Figs. B and C). Each combination of selections can then be printed or plotted at the desired size and with the user-defined color settings.

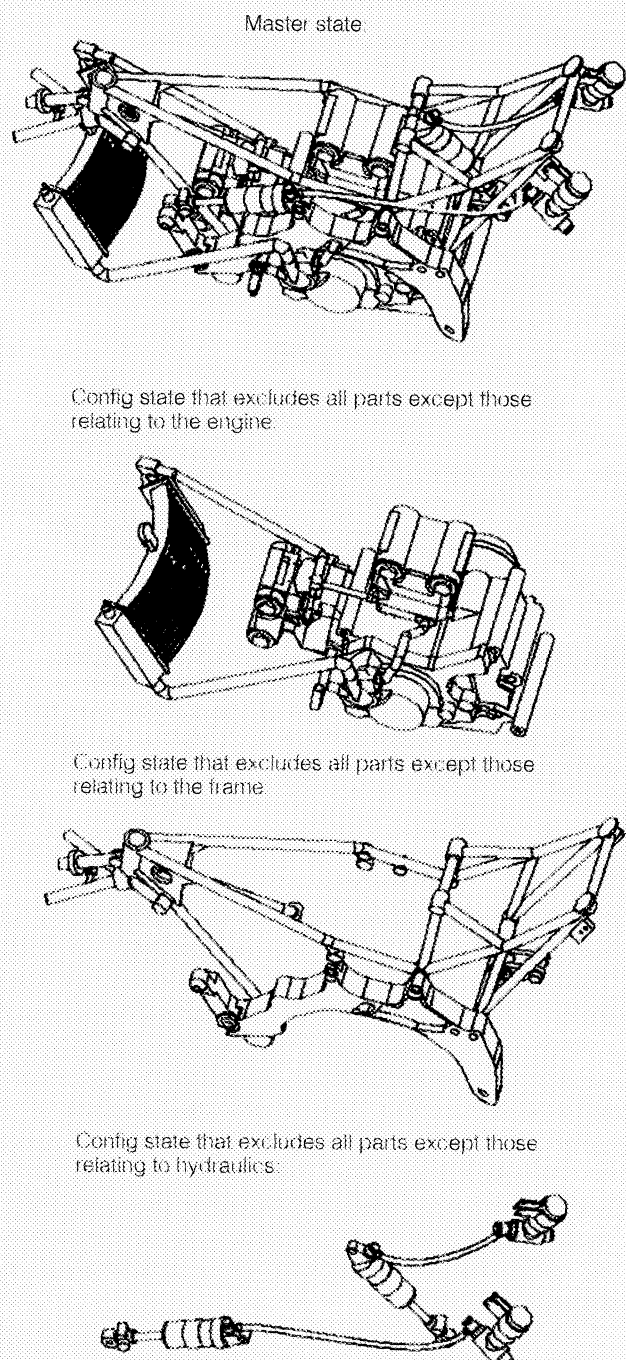


FIGURE A Excluding Parts, and Subassemblies, from an Assembly Illustration

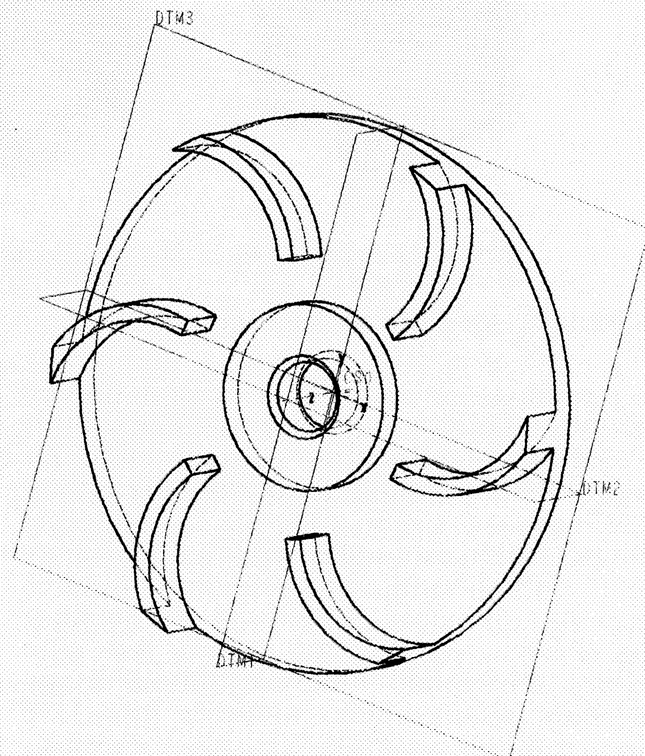


FIGURE B Flywheel with Hidden Lines, Datums, and Coordinate System Displayed

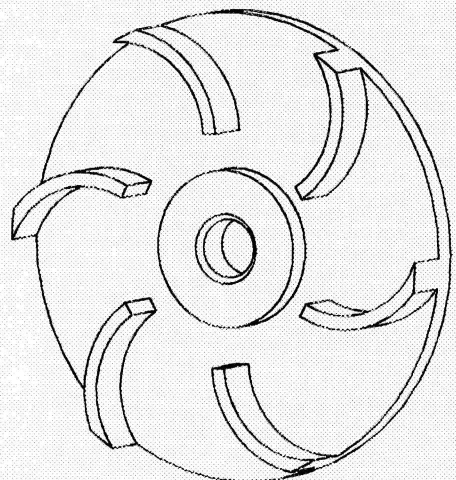


FIGURE C Flywheel with Only Visible Edges Displayed

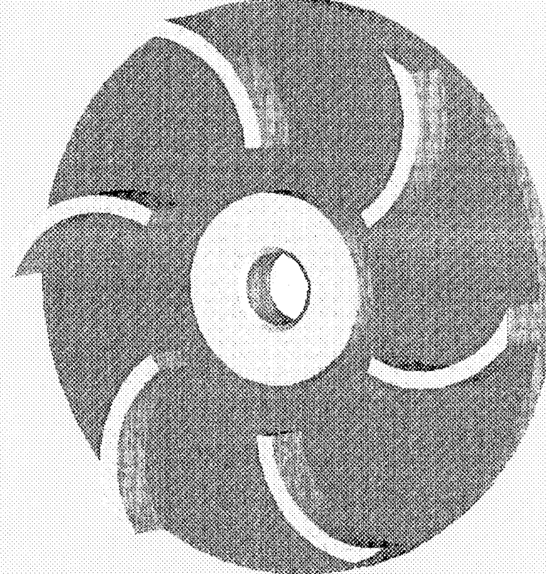


FIGURE D Shaded Flywheel

Shaded images can be generated automatically from the parametric model (see Fig. D). The quality and choice of shading can be selected from a menu and applied to your model (see Fig. E). If a color plotter or printer is available, you can shade your model in any desired color and highlight.

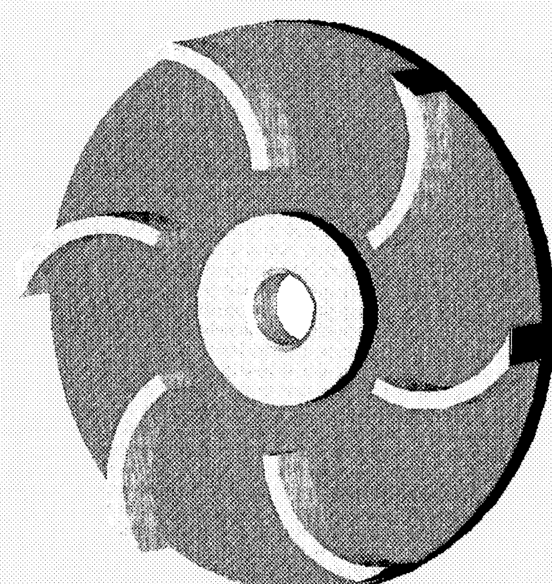
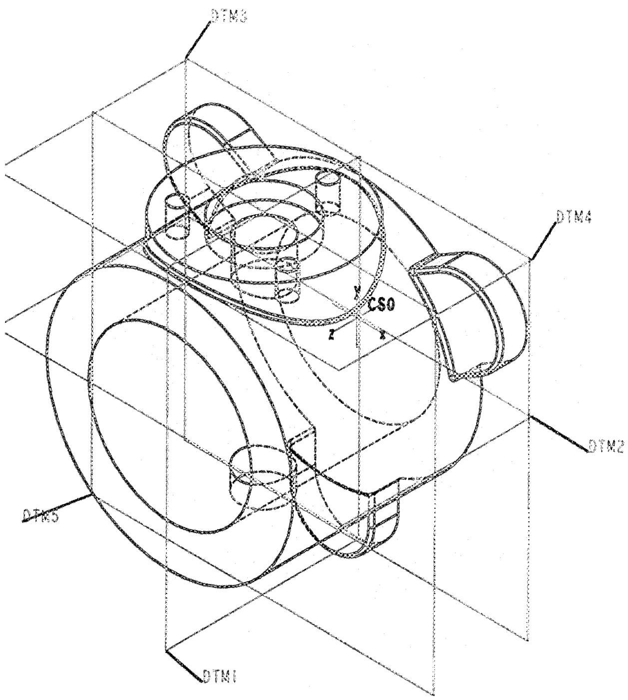


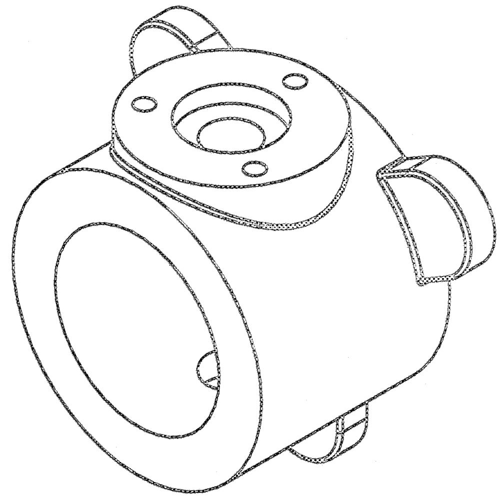
FIGURE E Shaded Flywheel with Different Display Settings

sentations. The airplane in Figure 13.78 was modeled as a solid and then displayed pictorially.

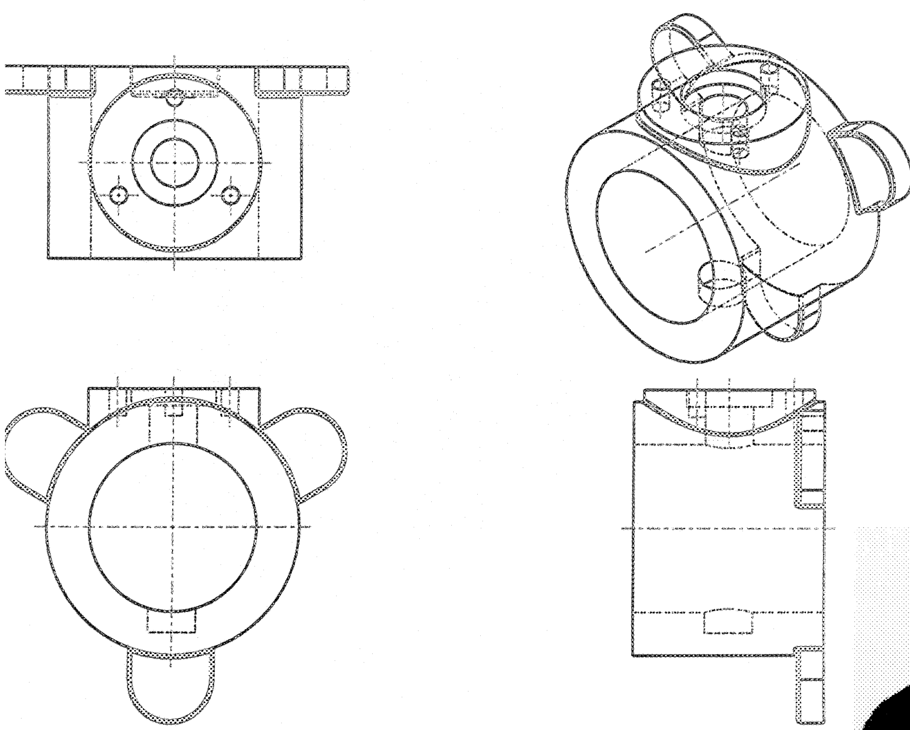
While perspective, isometric, and rotated views are useful in depicting 3D objects more realistically, most CAD systems offer the capability to use solid models (Figure 13.79). Here, a cutaway view of a helicopter is displayed. Depth is shown through color, shading, and perspective projection. Wire-frame, surface, and solid models can all be displayed pictorially via simple commands. Most CAD systems offer the option of an automated perspective command, with a variety of options for displaying the model (Figure 13.79). There are many rendering programs available for capturing high-quality images of your part model. The shaded image of the part in Figure 13.80(a) shows a view orientation and



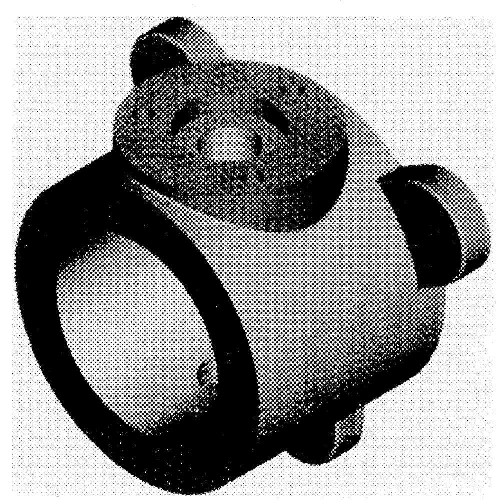
a) Part modeled with datum features and coordinate system



(b) Part shown with hidden lines removed



) Four views of the part, with hidden lines shown



(d) Shaded image of the part

GURE 13.77 Valve Housing

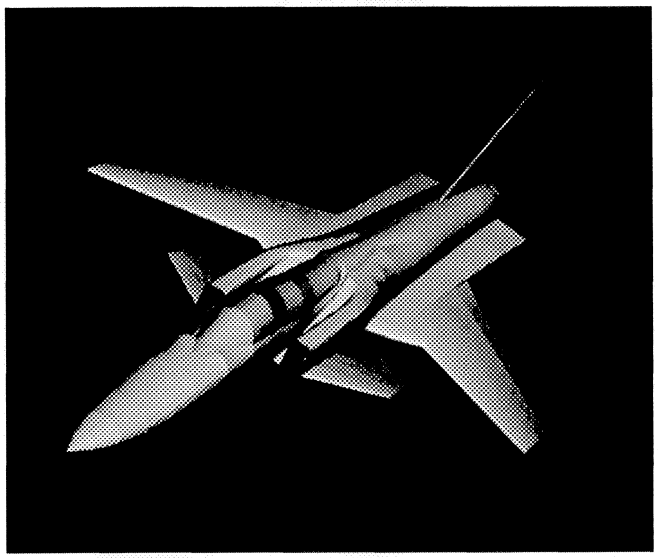


FIGURE 13.78 Shaded Image of Plane Design

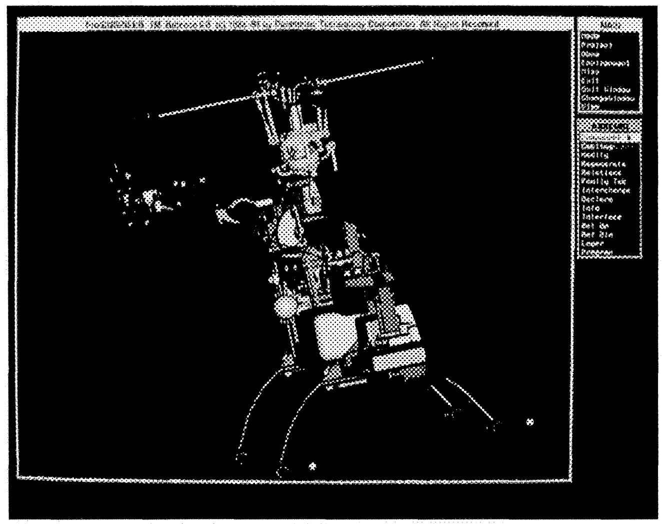
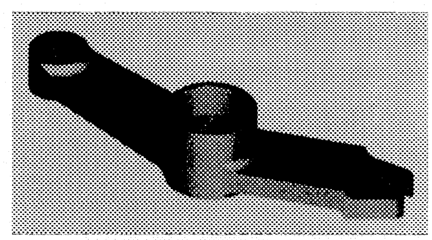


FIGURE 13.79 Perspective Image of Helicopter

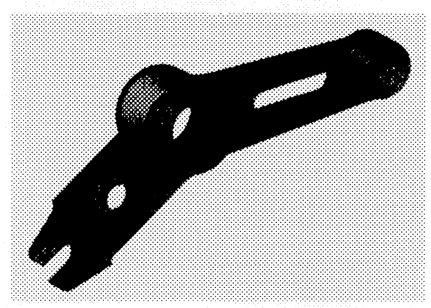
a color selection. Figure 13.80(b) shows another orientation and color mix. In Figure 13.81(a) we see a 3D view of a part with hidden lines displayed. The part can be shaded and the background color altered as required [Figure 13.81(b)]. Note that the dark background reduces the contrast around the slot. In Figure 13.81(c) the slot and holes have been recolored so the part stands out better from the dark background.

Assemblies also lend themselves to color and shading. The valve assembly in Figure 13.82 shows as a shaded model (a), as an exploded shaded image (b), with the arm as transparent (c), and with all parts except the valve desk as transparent (d). Figure 13.83 shows a pulley assembly (transparent) with the belt (a) and as an exploded assembly (b). Exploded pictorial assemblies and transparent plotting allow the designer to create high-quality visual aids with a CAD system and a rendering package.

Besides standard color and shading capabilities, a variety

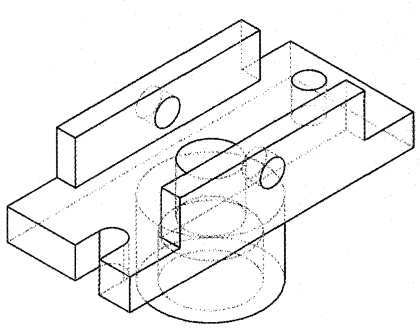


(a) Shaded pictorial of model using the default orientation

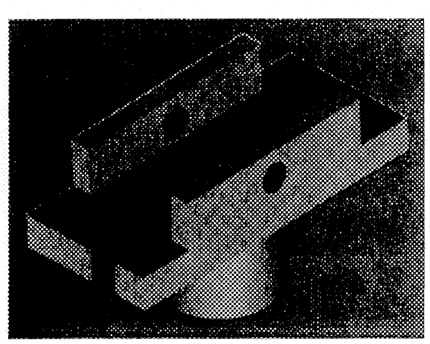


(b) New orientation and color mix for shaded model

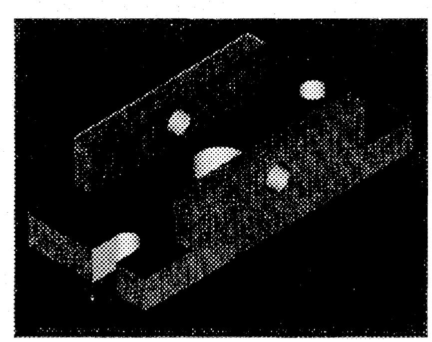
FIGURE 13.80 Shaded Images



(a) 3D image with hidden lines displayed

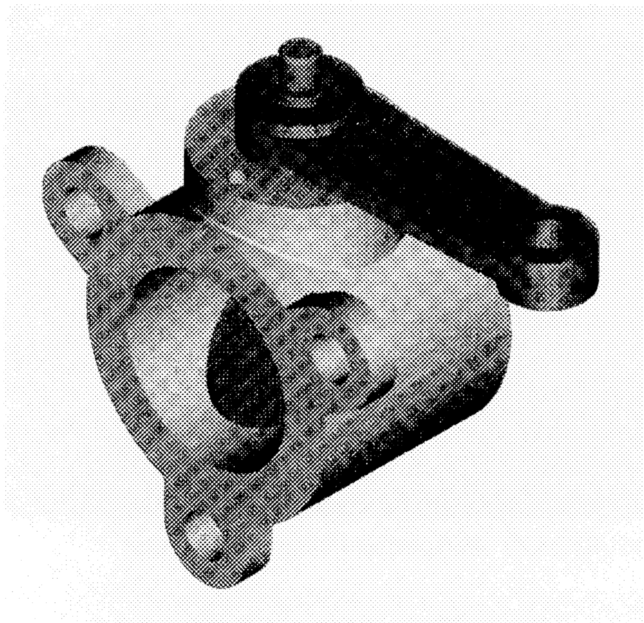


(b) Shaded color image with dark holes and slot

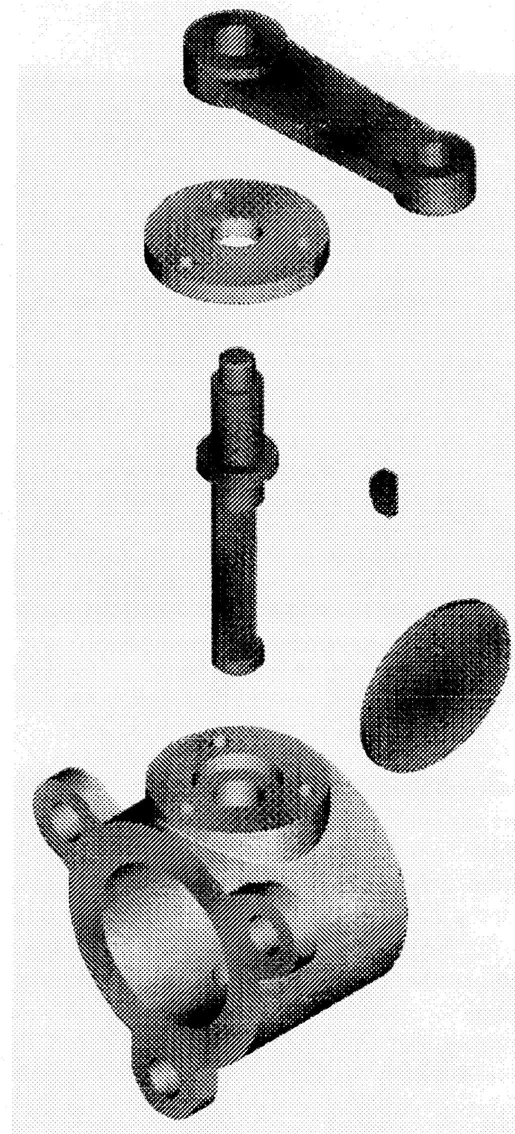


(c) Shaded image with light-colored holes and slot

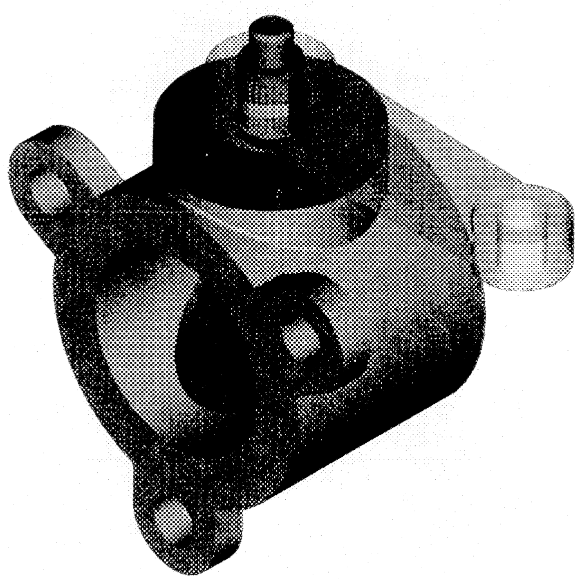
FIGURE 13.81 Model Displays



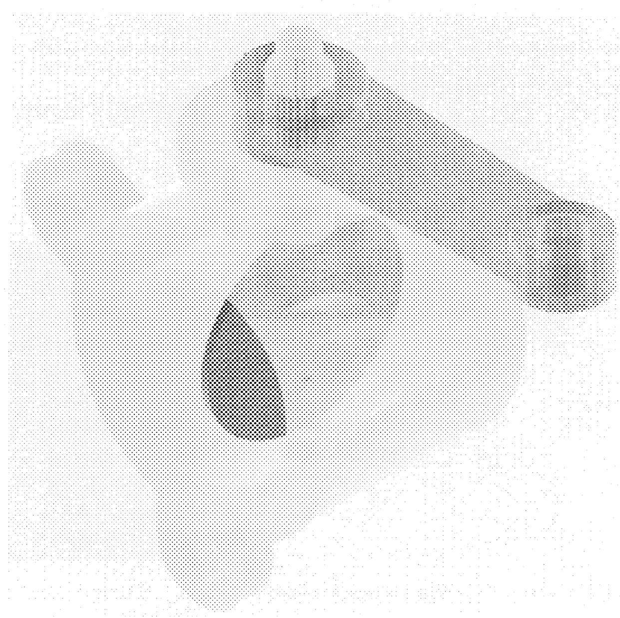
a) Shaded image of assembly



(b) Exploded shaded image of assembly

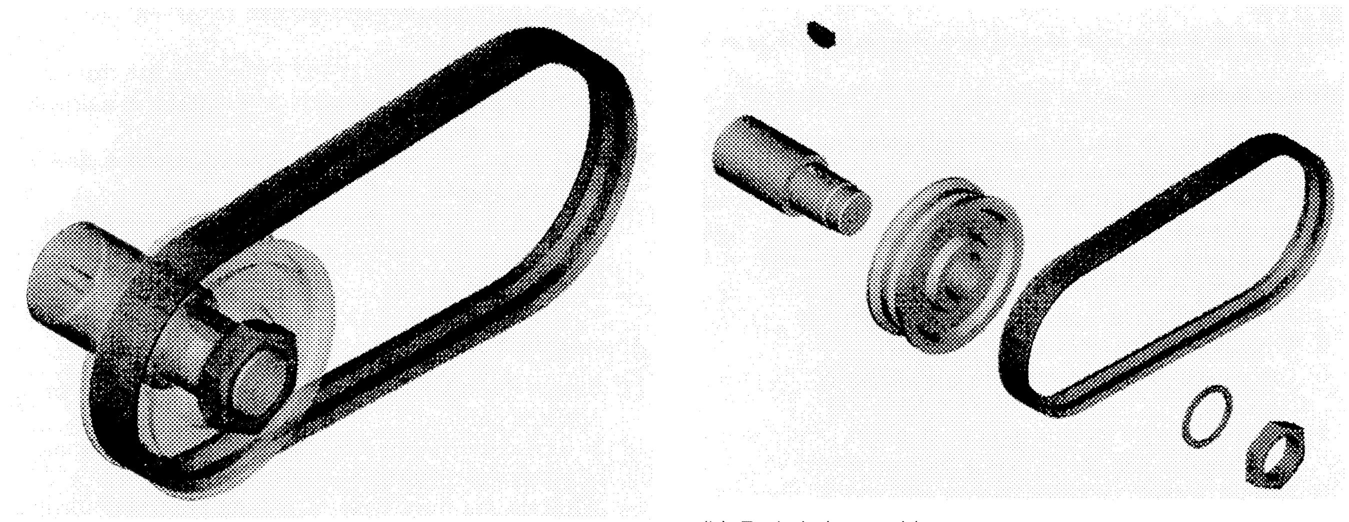


:) Shaded image with transparent handle



(d) Transparent model

GURE 13.82 Shaded Assembly



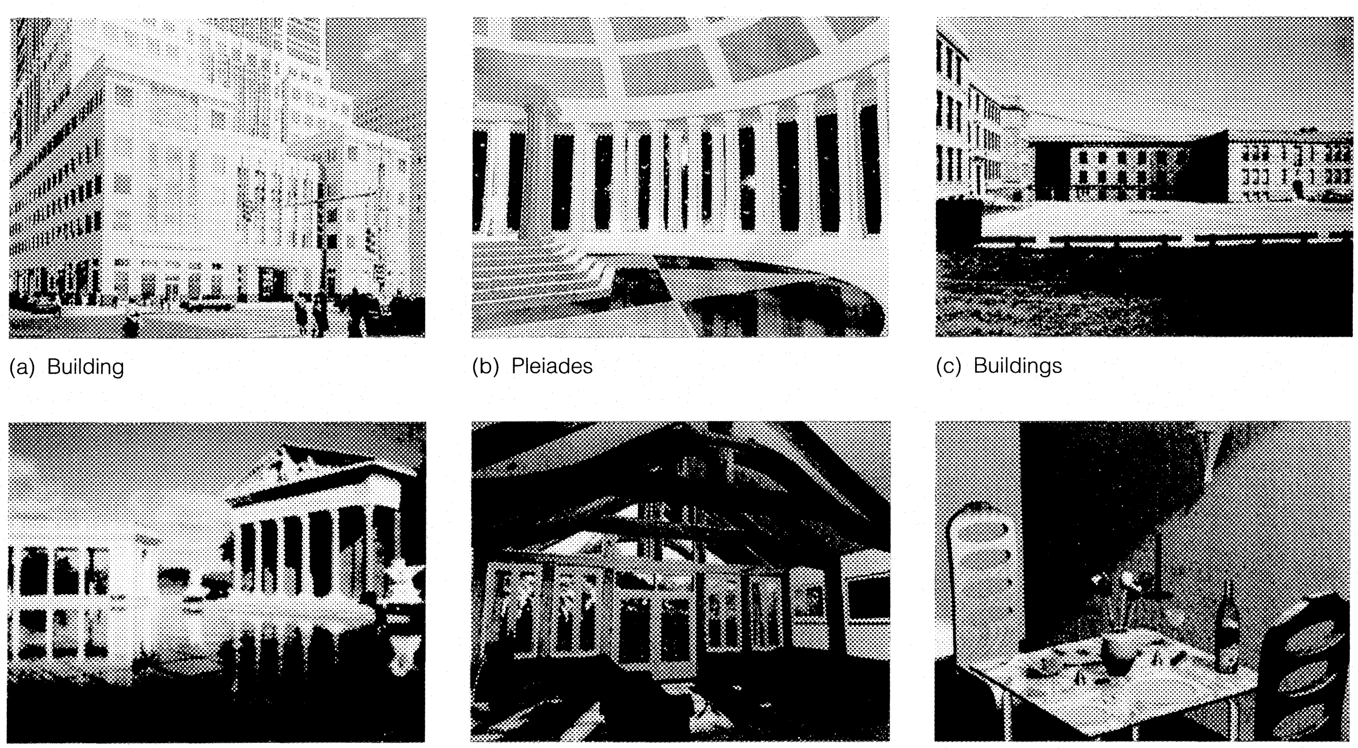
(a) Shaded assembly

(b) Exploded assembly

FIGURE 13.83 **Pulley and Belt Assembly**

of programs are available that create lifelike images of a design, such as the illustrations in Figure 13.84(a)–(f), which were created and rendered with AUTODESK products. The choices of color and view are limited only by your imagination, your hardware and software, and the plotter.

Regardless of the method used or the CAD system available, pictorials are found increasingly in engineering and design work. The need to describe in detail to nontechnical personnel the workings and assembly of products and machinery and a project's construction requirements will become even more important in the future. Therefore, the use of color, shading, rendering, and other pictorial capabilities will increasingly find there way into engineering and design.



(a) Building (b) Pleiades (c) Buildings

(d) Buildings and background (e) Interior view (f) Interior design

FIGURE 13.84 **Architectural Images Using Rendering Program**

QUIZ

True or False

1. Curves that do not lie in isometric planes must be constructed with offset measurements.
2. A general oblique drawing is constructed using 45° for the receding axis.
3. An isometric drawing is constructed using true-length measurements along all three axis lines.
4. A cavalier drawing is always foreshortened.
5. Centerlines are included on all pictorials.
6. Cabinet and cavalier drawings are types of perspective projections.
7. The point of sight for a perspective should be located so that the angle the cone of rays makes does not exceed 45°.
8. A trimetric projection uses different angles for all three axes.

Fill in the Blanks

9. A __________ view shows the interior of the part or assembly.
10. __________ dimensioning is found on most pictorial drawings.
11. __________ projection approximates how a part will look to the human eye.
12. In true isometric projection, all __________ __________ make equal __________ with the projection plane.

13 __________ lines are not parallel to or on one of the isometric axes.

14. A __________ __________ is a planar surface that can be placed on the model's geometry or is available as a default at the part's origin.
15. The __________ method of ellipse construction does not create an ellipse.
16. __________ features should be oriented so they lie in the __________ face of the object when __________ projection is used on a drawing.

Answer the Following

17. What is the difference between an isometric drawing and an isometric projection?
18. In what situation would an oblique drawing be used instead of an isometric drawing?
19. What are the three types of oblique projection? Describe each, and explain how they differ.
20. How does 3D CAD eliminate the need to construct traditional 2D pictorial drawings?
21. Give four uses of pictorial drawings.
22. Describe the three types of perspective projection, and explain their differences.
23. What are offset measurements, and when are they used in the construction of a pictorial drawing?
24. What is the difference between model mode and draw mode (model space and paper space for AutoCAD)?

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. 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.

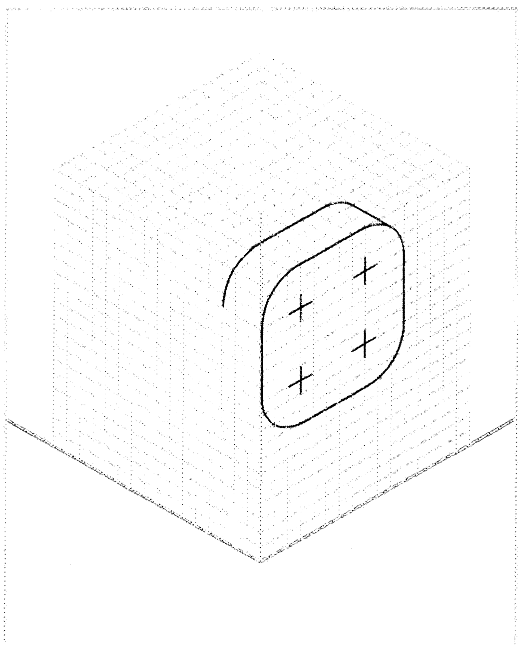
After Reading the Chapter Through Section 13.3.11 You May Complete the Following Exercises

Exercise 13.1 Using the part provided in Exercise 10.5B draw an isometric pictorial.

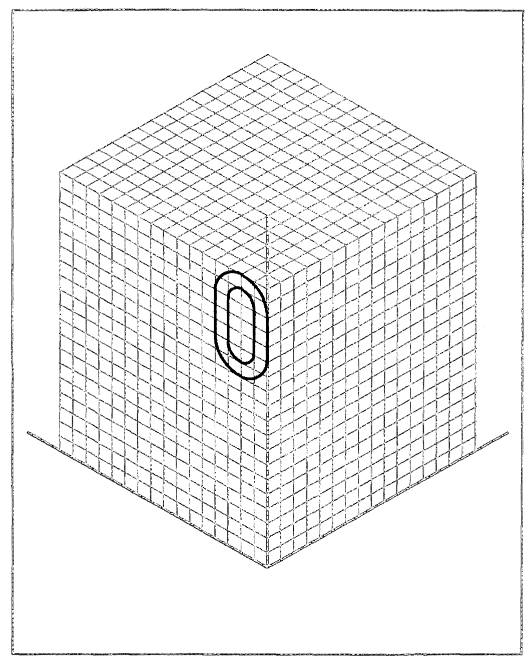
Exercise 13.2 Using the part provided in Exercise 10.9A draw an isometric pictorial.

Exercise 13.3 Using the part given in Exercise 10.10B draw an isometric pictorial.

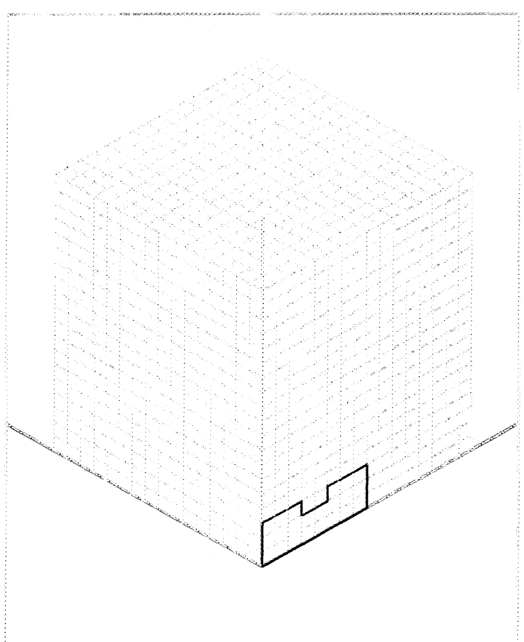
Exercise 13.4 Using the part provided in Exercise 10.3A draw an isometric pictorial.



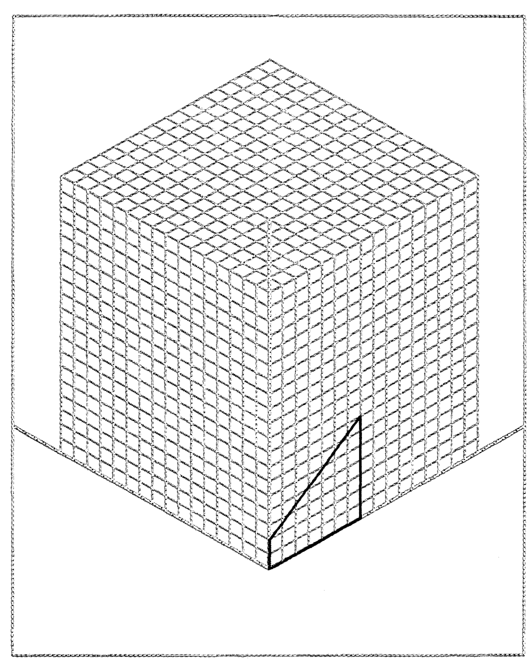
EXERCISE 13.1



EXERCISE 13.3



EXERCISE 13.2



EXERCISE 13.4

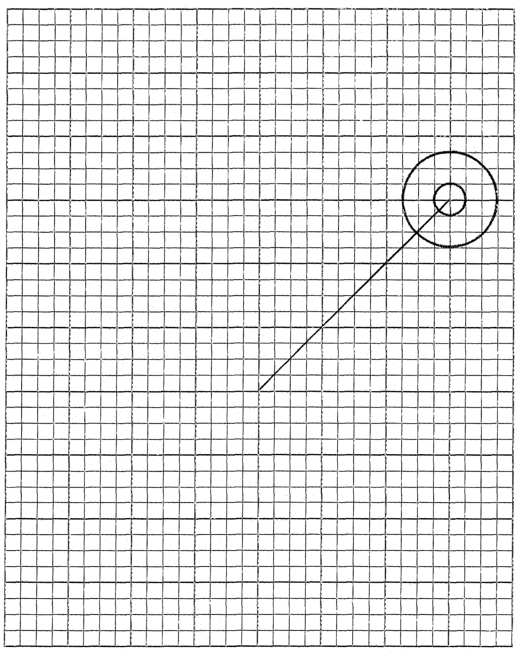
After Reading the Chapter Through Section 13.7.5 You May Complete the Following Exercises

Exercise 13.5 Using the part given in Exercise 10.13B complete an oblique pictorial.

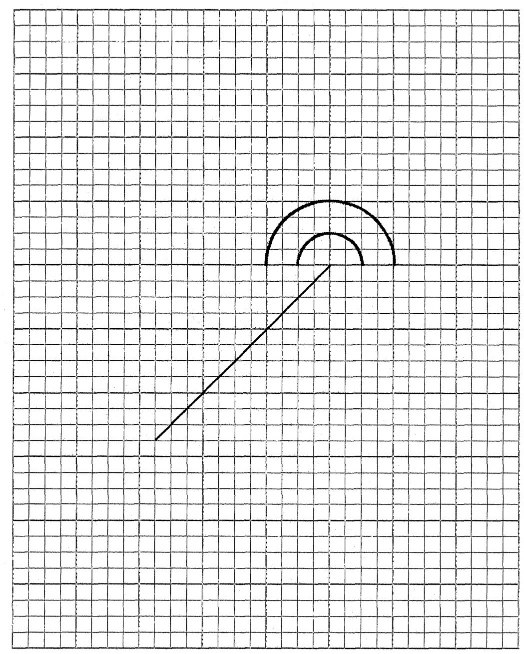
Exercise 13.6 Using the part given in Exercise 10.11B draw an oblique pictorial.

Exercise 13.7 Using the part provided in Exercise 10.4B draw an oblique pictorial.

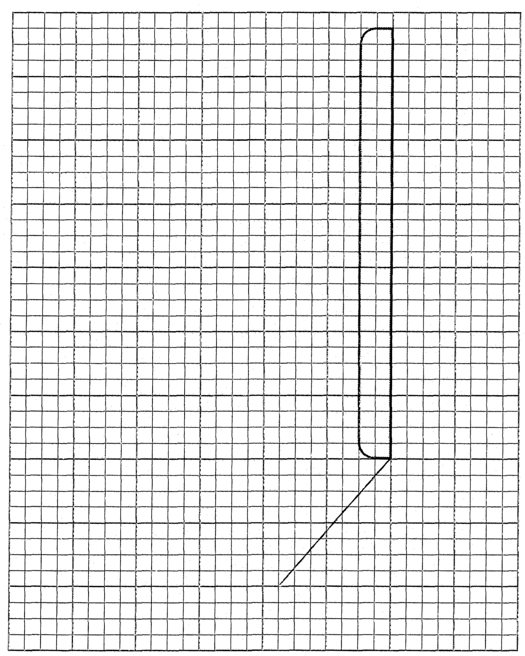
Exercise 13.8 Using the part provided in Exercise 10.1A draw a one-point perspective pictorial.



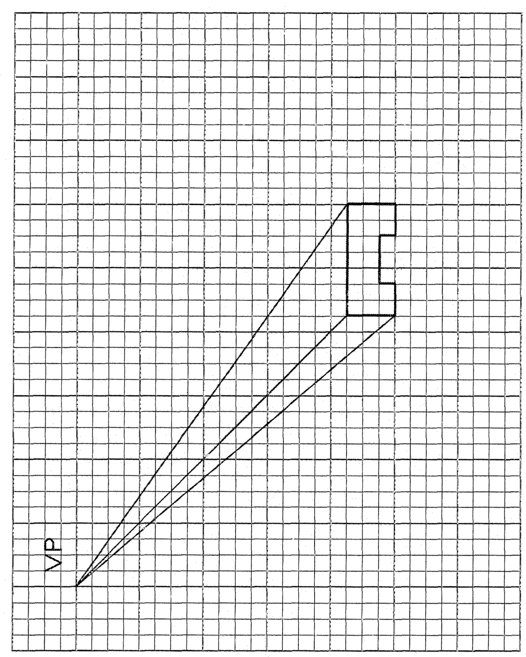
EXERCISE 13.5



EXERCISE 13.7



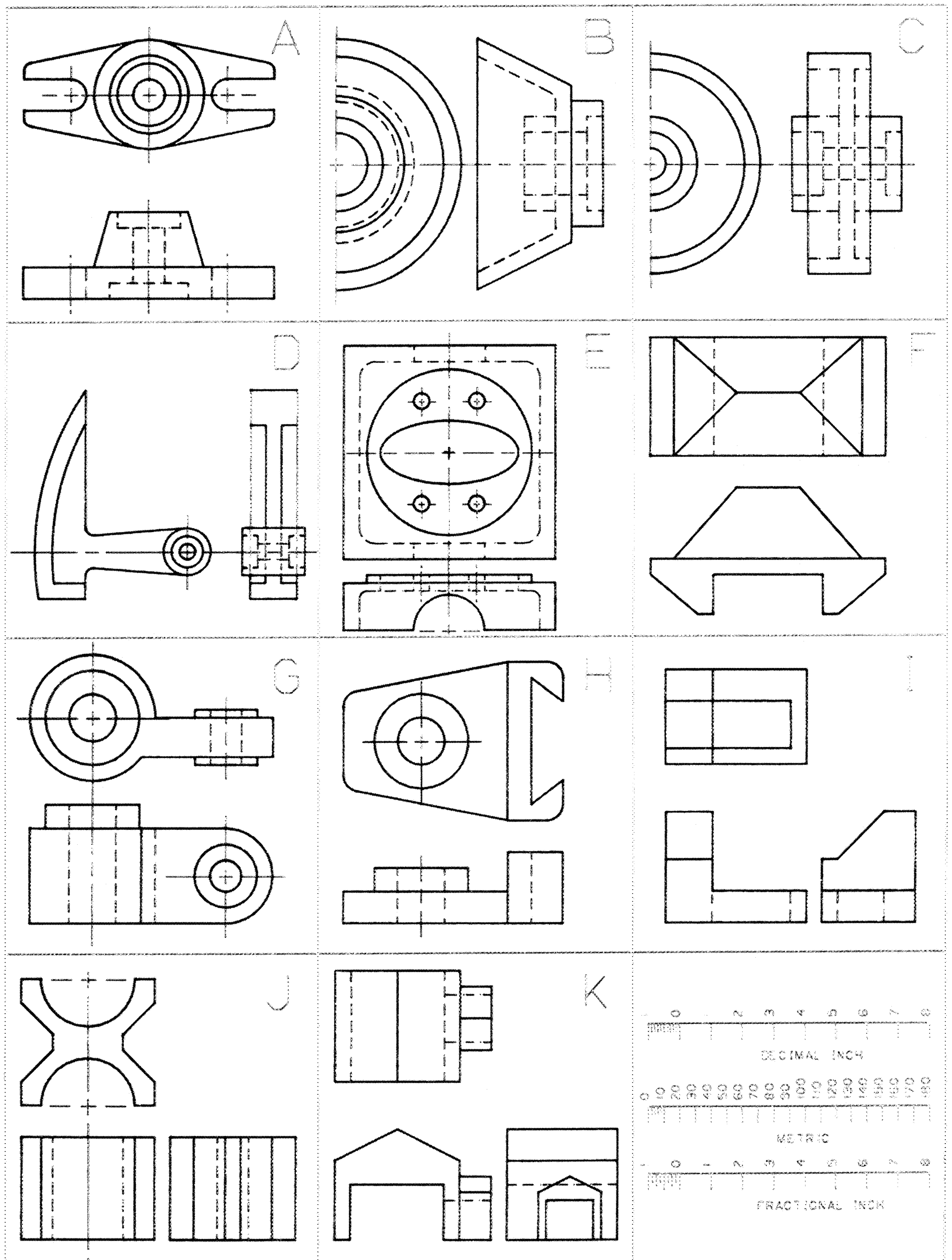
EXERCISE 13.6



EXERCISE 13.8

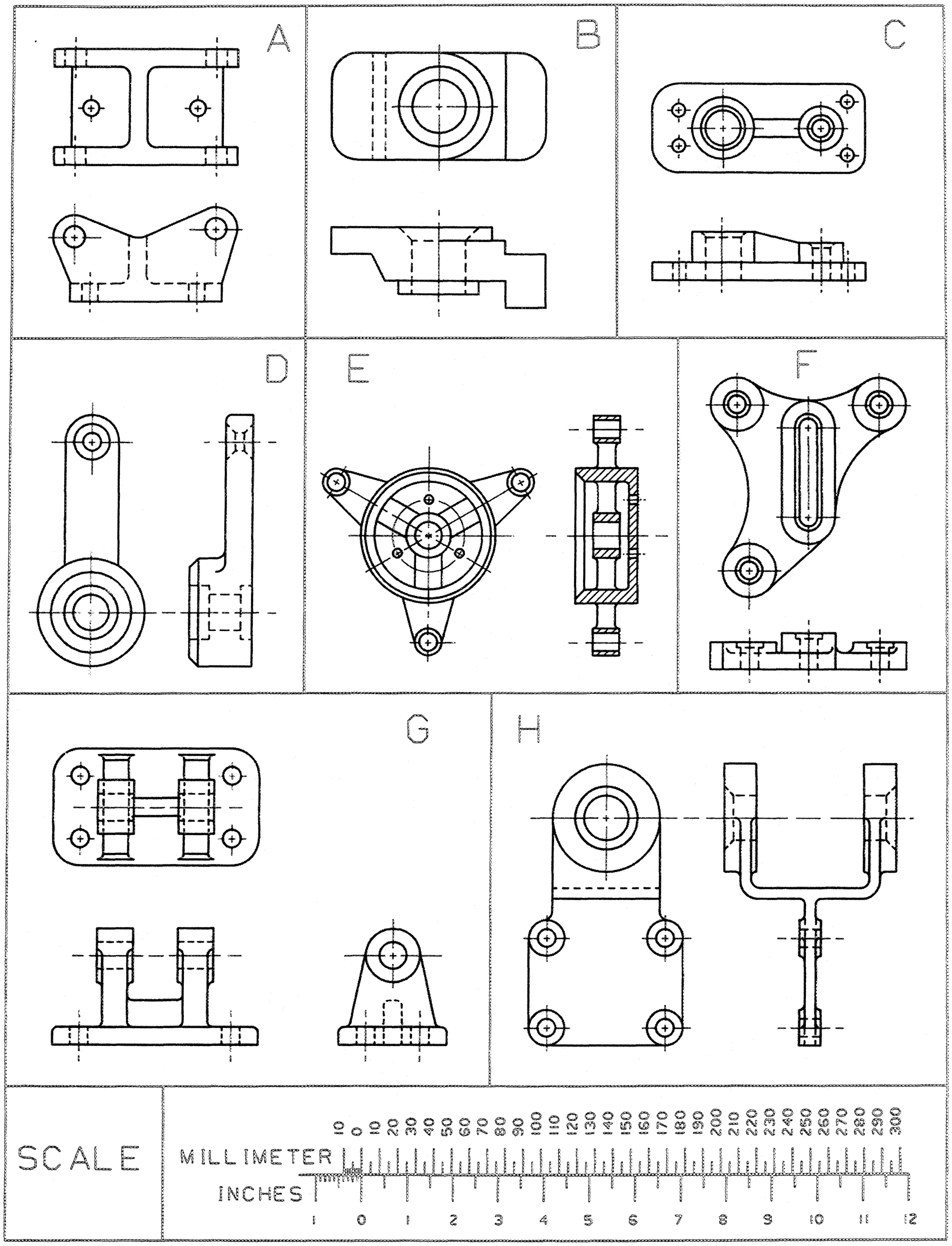
PROBLEMS

Problems 13.1(A) Through (K) These problems can be assigned as any of the three major types of pictorial projection or as 3D CAD models. Unless assigned as a specific type of projection, the problems could be used to test students' understanding of the suitability of projection types for a particular problem. The instructor can allow the student to determine the projection method based on the part's features and pictorial requirements as described by the instructor.

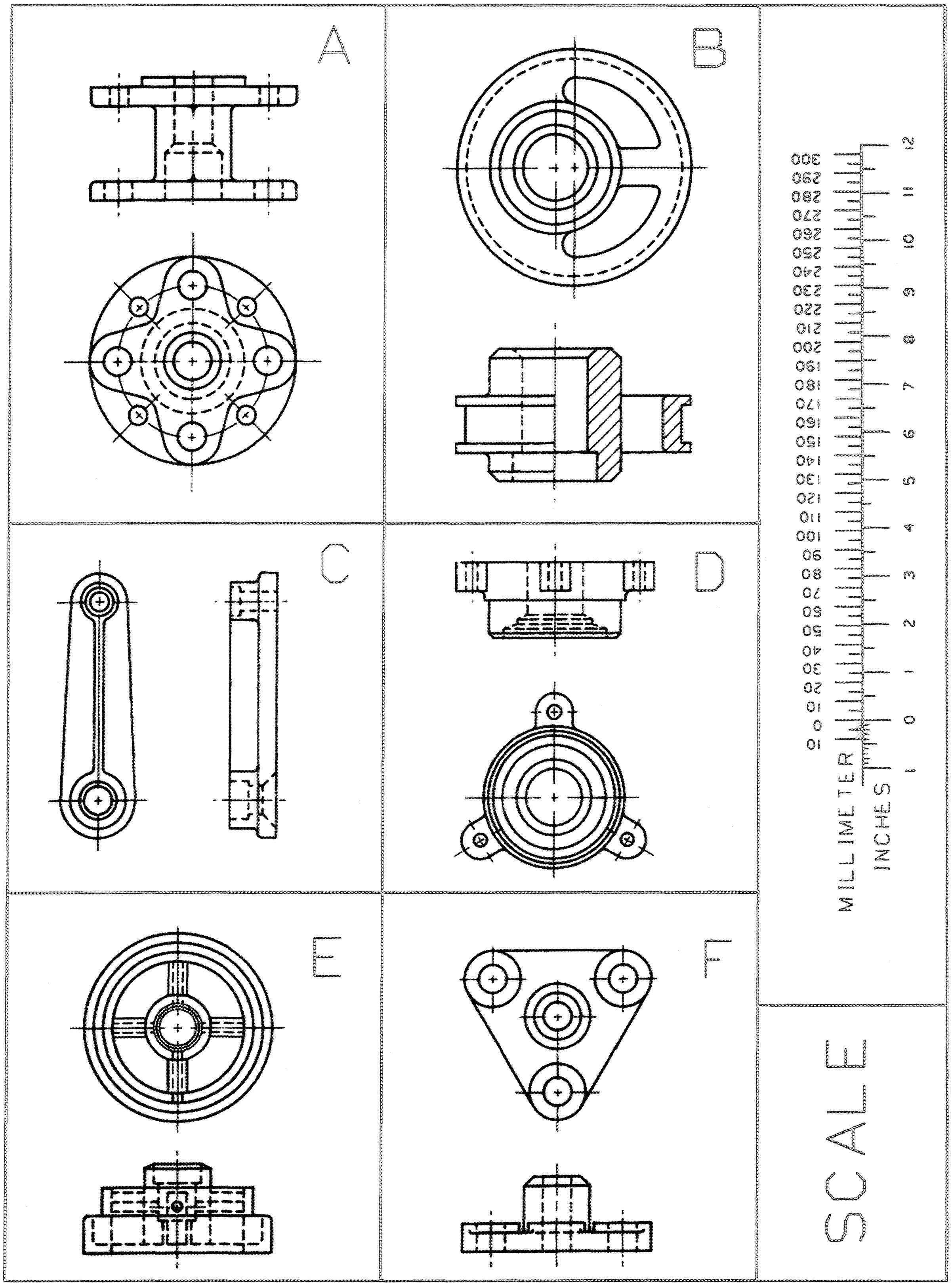


Problems 13.2 through 13.4 were created in metric units, and the scale provided is to be used when taking the part from the text and transferring it to the drawing board (or computer). The decimal scale can also be used if the problem as assigned will be done in decimal-inch units.

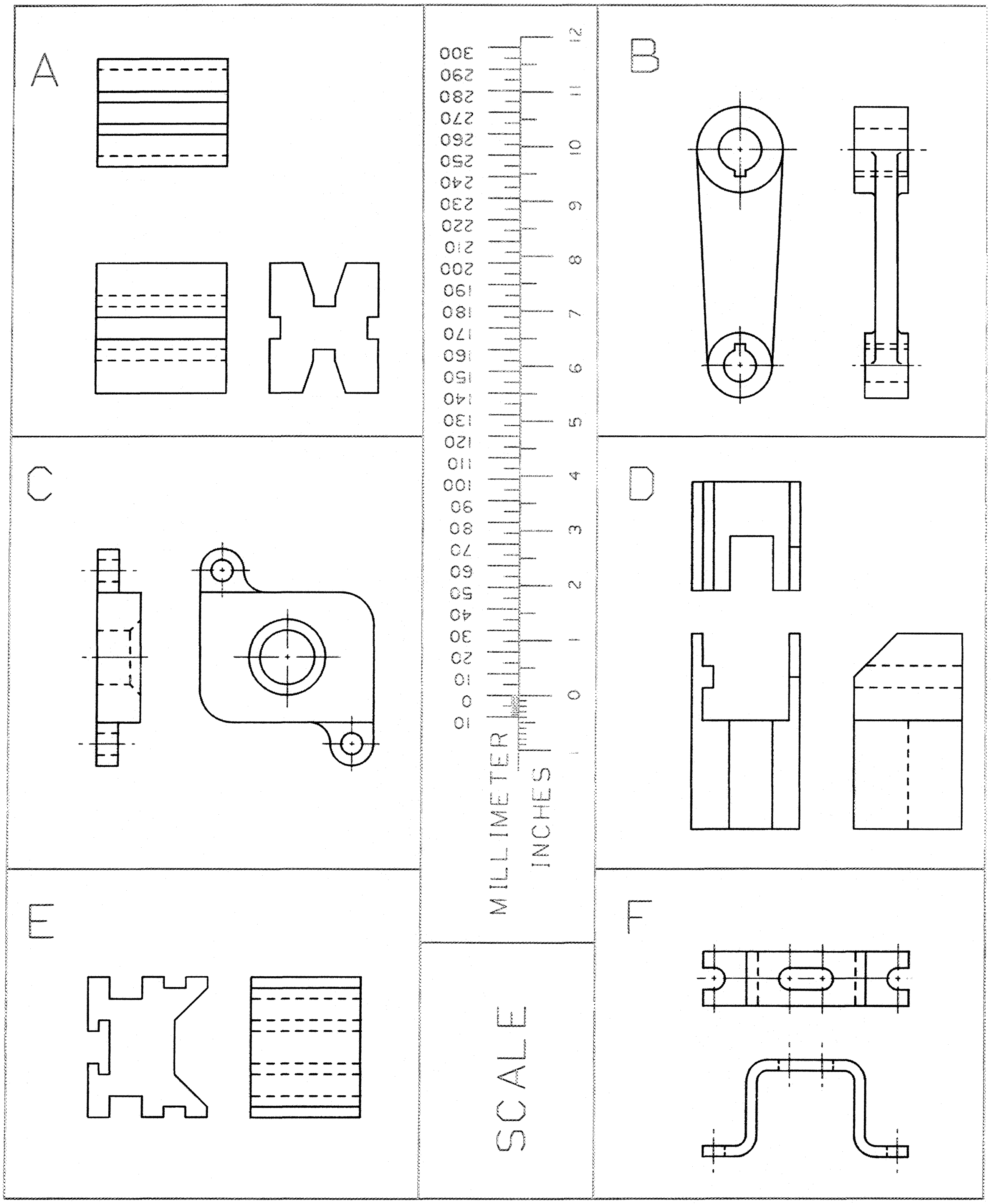
Problems 13.2(A) Through (H) These problems are meant for isometric projection but can be drawn as any type of pictorial projection, as assigned.



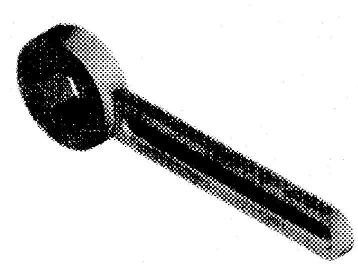
Problems 13.3(A) Through (F) These problems are designed as oblique projects, although the instructor can assign other methods of projection.



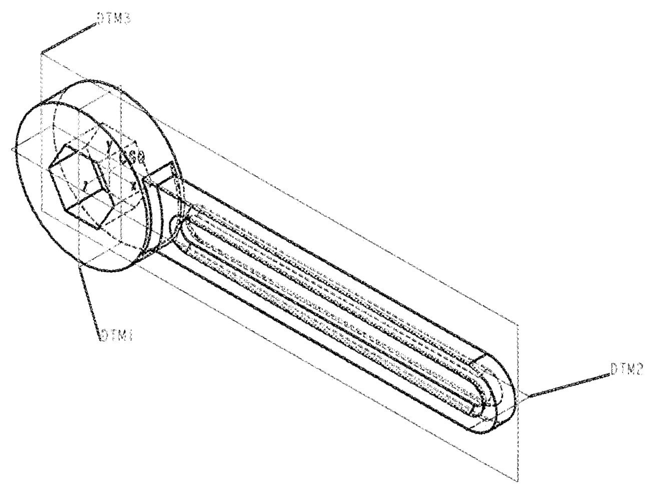
Problems 13.4(A) Through (F) These problems can be used for any method of pictorial projection but were intended as perspective projection projects.



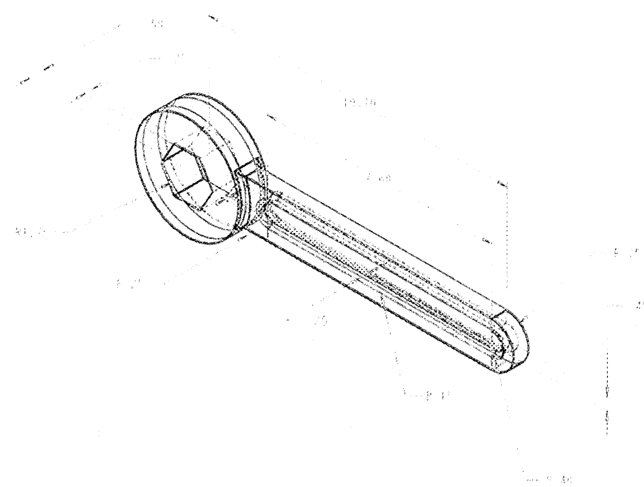
Problem 13.5 Model the wrench in 3D. Display and plot the part as a shaded model (a), as a 3D model with hidden lines (b), as a dimensioned pictorial (c), and as a detail (d).



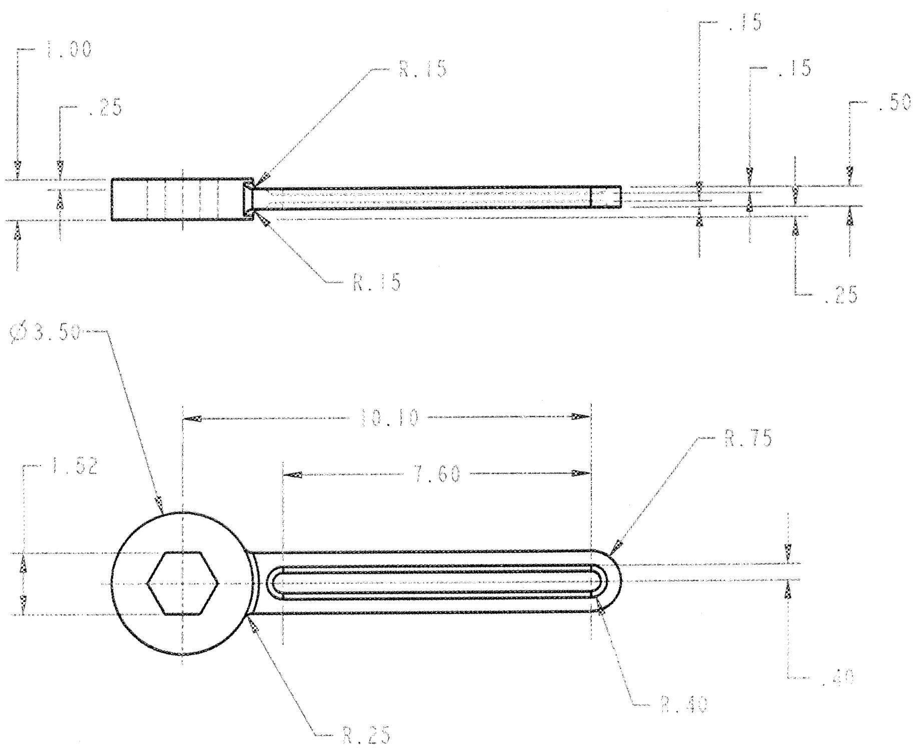
(a)



(b)



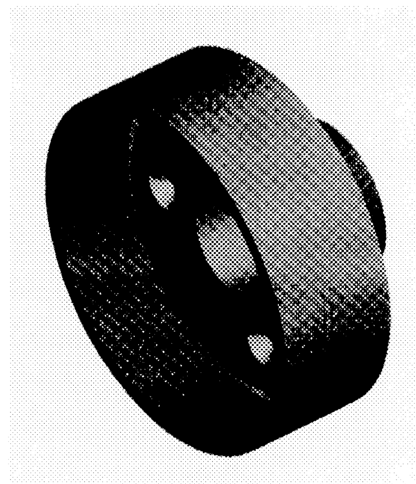
(c)



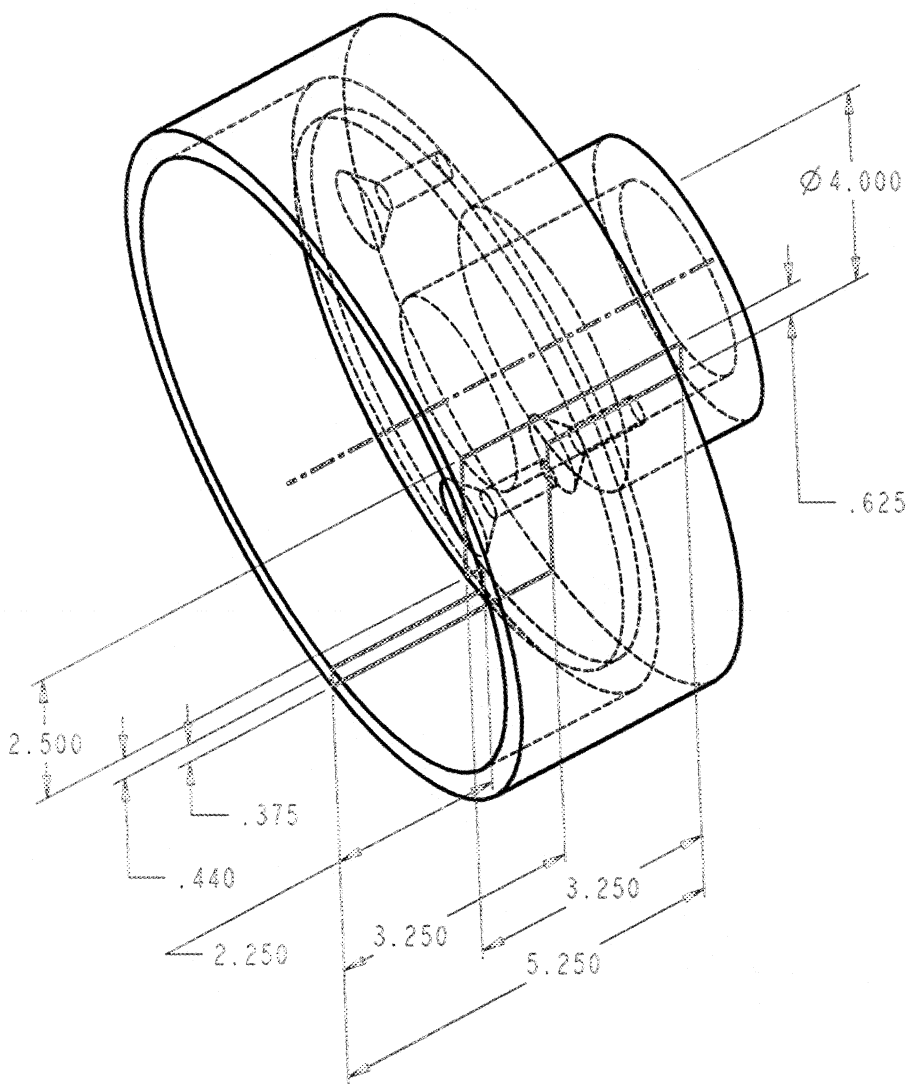
(d)

PROBLEM 13.5

Problem 13.6 Model the part in 3D. Display and plot the part as a shaded model (a), as a dimensioned pictorial (b), and as a dimensioned detail (c).

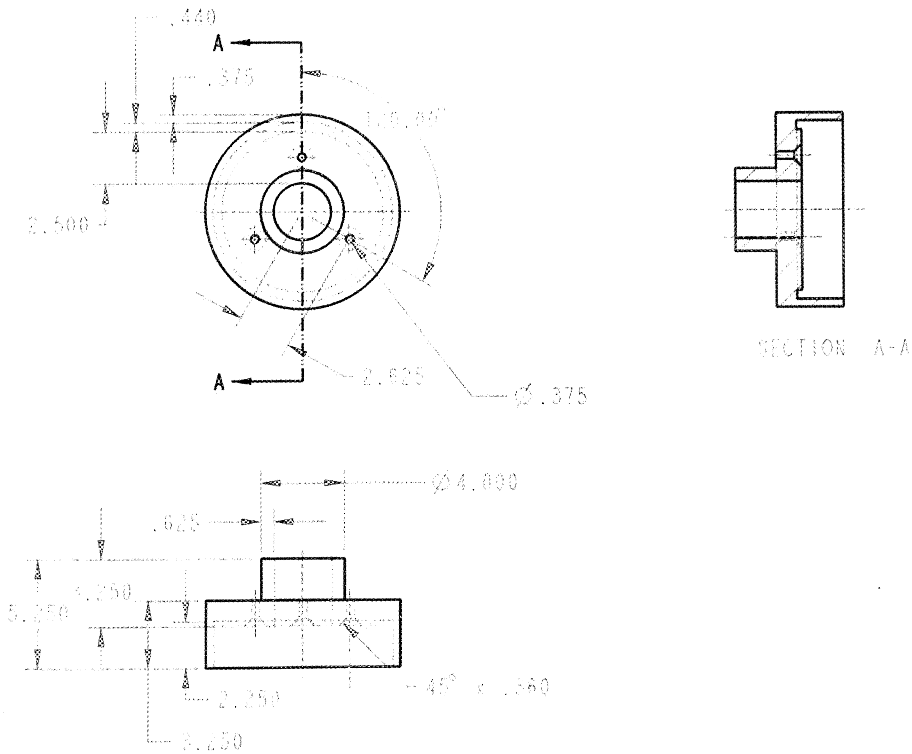


(a)



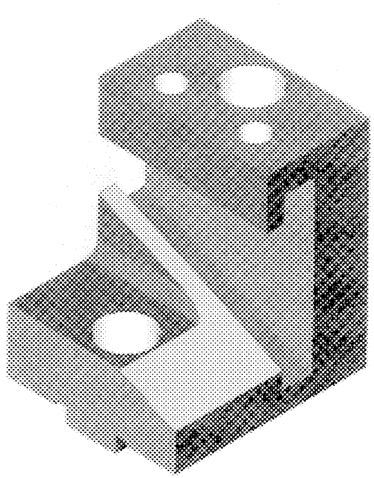
ɔ)

ROBLEM 13.6

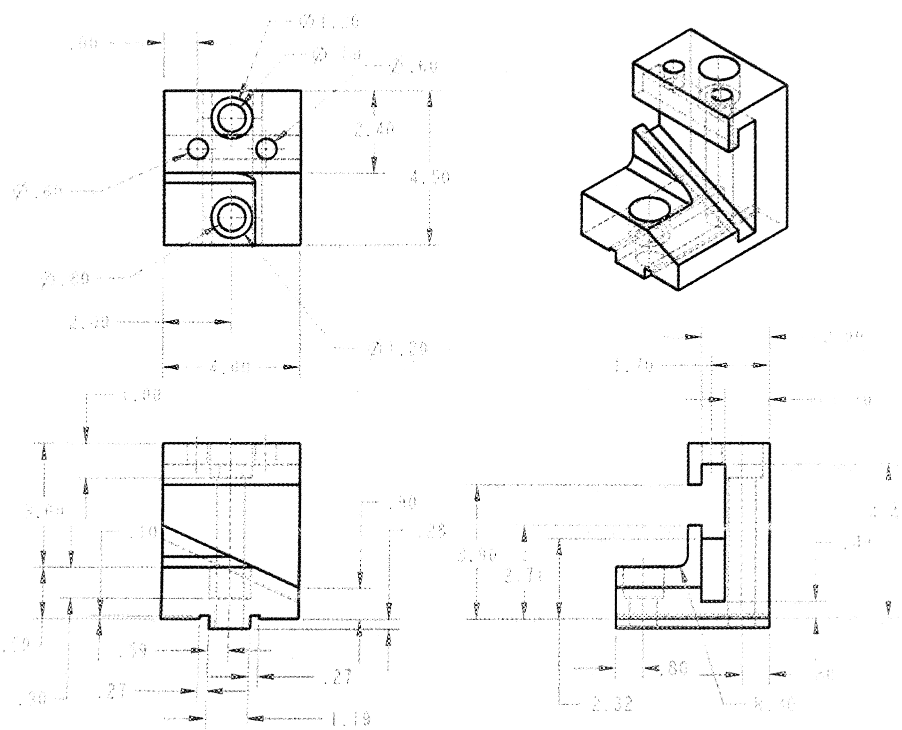


(c)

Problem 13.7 Model the part in 3D. Display and plot the part as a shaded model (a) and as a dimensioned pictorial (b).



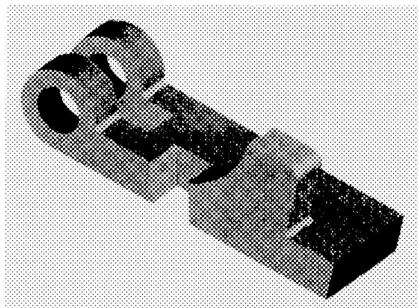
(a)



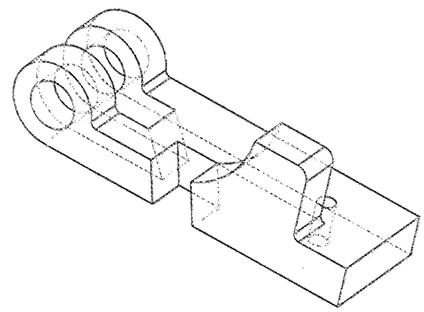
(b)

PROBLEM 13.7

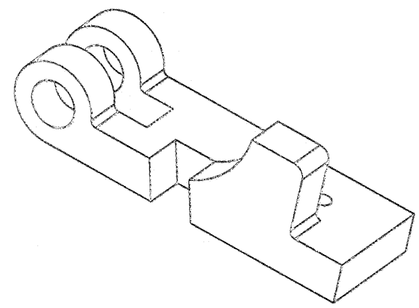
Problem 13.8 Model the part in 3D. Display and plot the part as a shaded model (a), as a 3D model with hidden lines (b), as a 3D model with without hidden lines (c), and as a dimensioned detail (d).



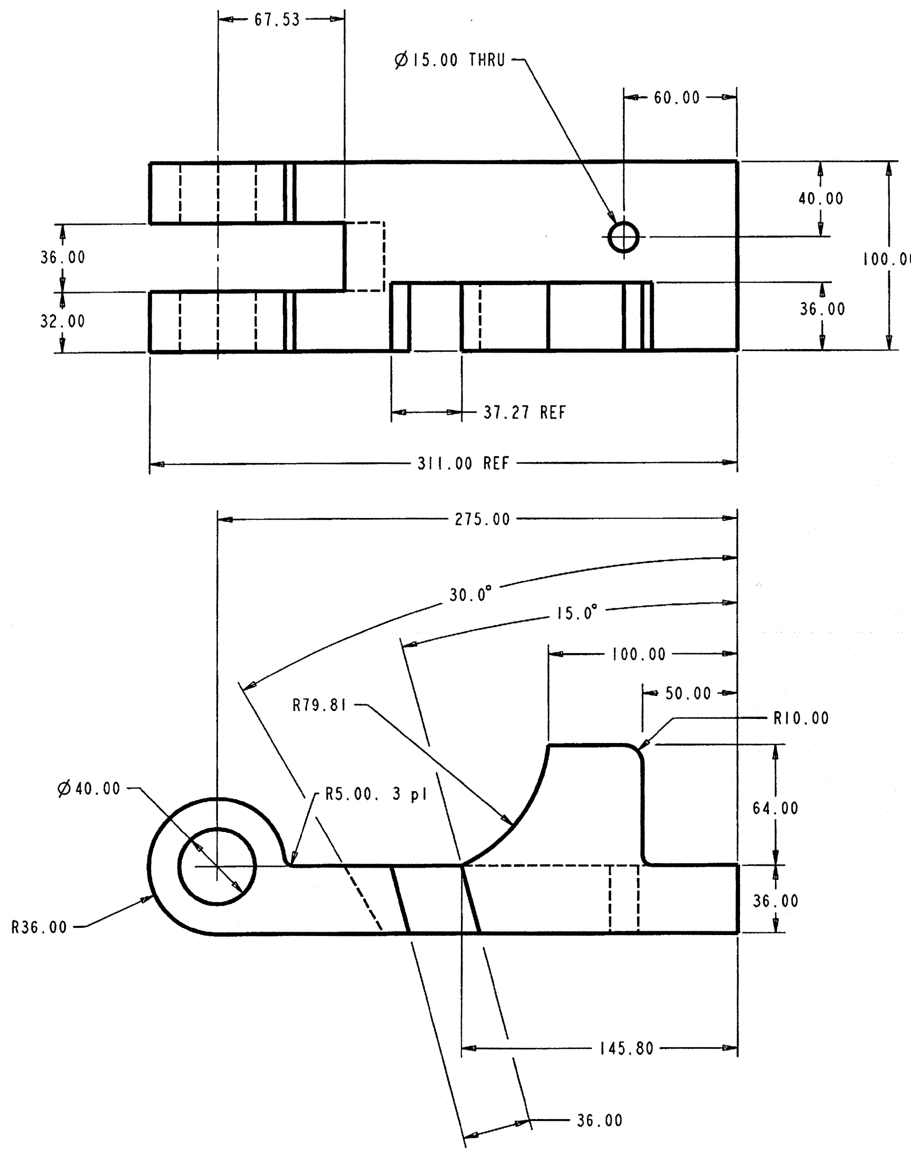
(a)



(b)



(c)



d)

PROBLEM 13.8