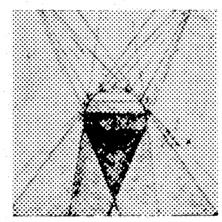
DIMENSIONING (ANSI Y14.5 1994)

Chapter 15



LEARNING OBJECTIVES

Upon completion of this chapter you will be able to:

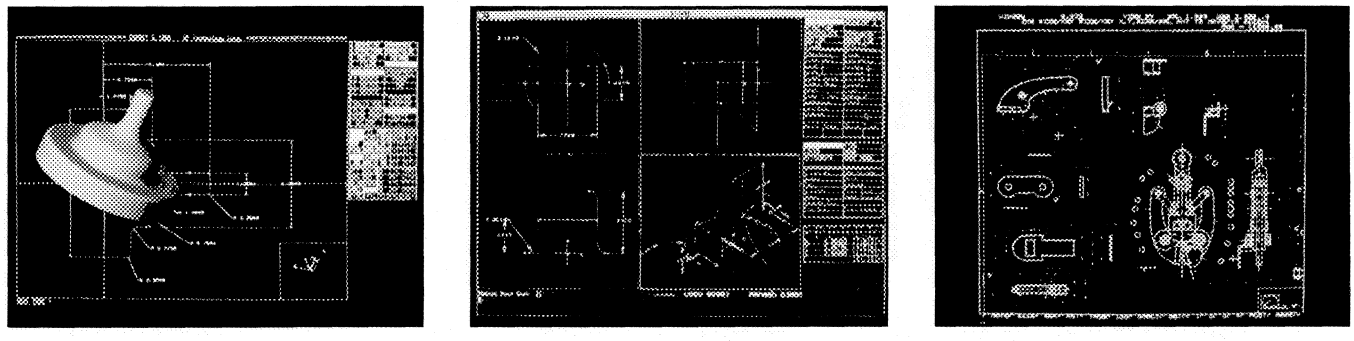
1. Analyze part features in terms of integral geometric shapes to facilitate concise dimensioning within prescribed tolerances.
2. Apply ANSI standards for dimensions and tolerances.
3. Apply angular, callout, overall, limited length, and area dimensions.
4. Dimension and recognize standard symbols for curved features.
5. Define and dimension chamfers, threads, center drills, tapers, knurling, and keyways.
6. Recognize finish marks, general symbols and notes, and ANSI basic surface texture symbols.
7. Apply rectangular continuous coordinate dimensioning and polar coordinate dimensioning.
8. Understand the associative dimensioning capabilities of CAD.

15.1 INTRODUCTION

The ability to analyze a part by recognizing that it is composed of simple geometric shapes enables you to understand what dimensions are required to manufacture that part. Complete or portions of prisms, cylinders, cones, and spheres, alone or in combination, will be common in all design of mechanical parts. Partly for ease of manufacture and partly because of design requirements, these shapes are used throughout engineering design. To dimension, break the mechanical system into these simple shapes. After all, the only real purpose of an engineering drawing is to convey information correctly so that the part can be manufactured correctly from the drawing.

Engineering drawings use dimensions and notes to convey this information. Regardless of whether the part is drawn and dimensioned manually or with a CAD system (Fig. 15.1), knowledge of the methods and practices of dimensioning and tolerancing is essential. The multiview projections of a part graphically represent its shape (*shape description*). However, the drawing must also contain information that specifies size and other requirements.

Drawings are *annotated* with dimensions and notes. Dimensions must be provided between points, lines, or surfaces that are functionally related or to control relationships of other parts. Manufacturing personnel should not have to compute dimensions or guess intent. Each dimension on a drawing has a **tolerance**, implied or specified. The general tolerance given in the title block is called a **general tolerance** or **sheet tolerance** (see Fig. 15.77). Specific tolerances are provided with each appropriate dimension. Together, the views, dimensions, and notes describe the complete shape and size of the part (Fig. 15.2). If any of these are incorrect, the part will be fabricated incorrectly. Therefore, accuracy of views and dimensions is of utmost importance. (Tolerances are discussed in more detail in Chapter 16.)



(a) Solid model dimensioning with a CAD system

(b) Multiview drawing dimensioning with a CAD system

(c) Dimensioning showing assembly and detail drawing of a mechanical part

FIGURE 15.1 Part Design Using CAD

15.2 DIMENSIONING STANDARDS

Uniform practices for stating and interpreting dimensioning and tolerancing requirements were established in **ANSI Y14.5M.** Engineering firms typically have a copy of these standards. This chapter uses both the International System of Units (SI) (metric) and U.S. customary units because SI units are expected someday to replace U.S. customary units on engineering drawings. Figure 15.2 shows a part dimensioned in SI units; Figure 15.3 shows a part dimensioned in U.S. decimal-inch units. Either type of unit can be used, with equal results.

Some of the industry example drawings and problems in the text were completed before 1982 and, therefore, conform to earlier standards. Study them carefully. You will be in contact with older standards in your career since some companies continue to use older practices rather than face the expense of converting to the new standards. *Your instructor will help you to use the most recent standard to complete the exercises in this text. Many instructors have a complete copy of the current ANSI standards in a reference area.*

All drawings must be clear, be laid out well, and contain the required dimensions, text notes, finishes, etc. The advantage of a CAD database over manual drawing is that checker changes and corrections are easily incorporated into the drawing and a new "original" drawing can be plotted. The manual method requires the original drawing to be erased and corrected and requires more time. Another advantage of creating a CAD database for a part is that manufacturing can call up the part and verify the geometry,

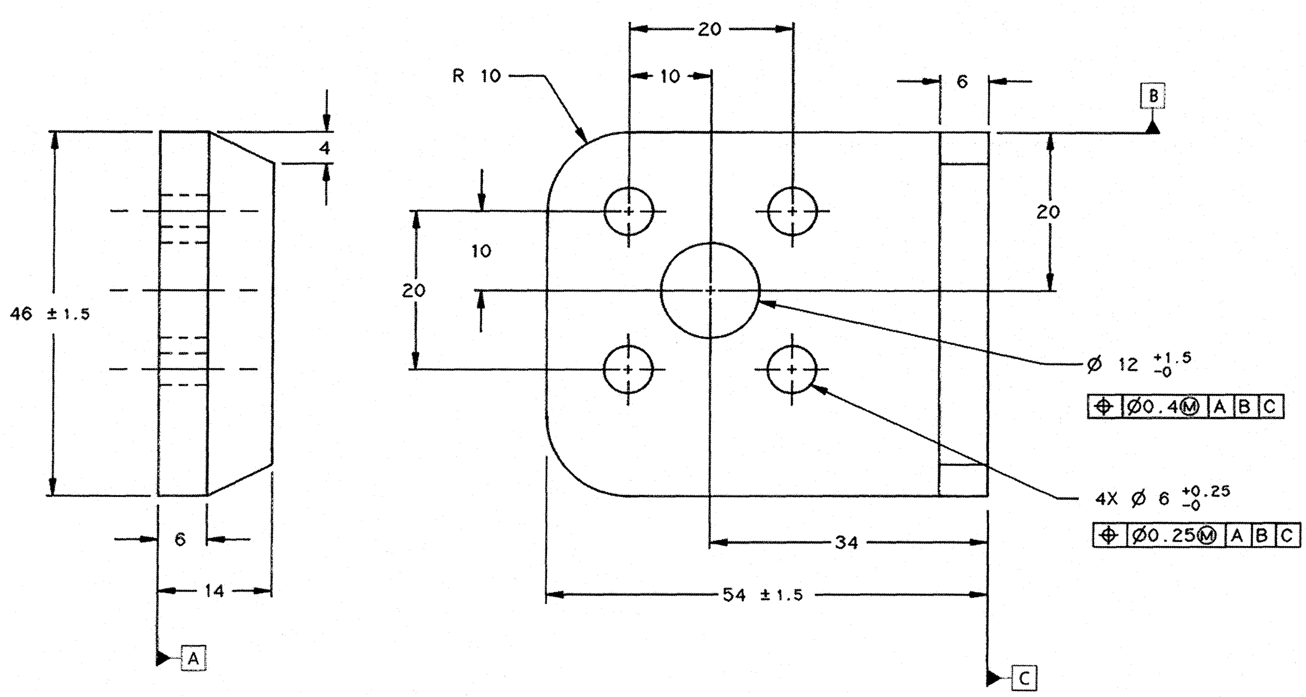


FIGURE 15.2 Mechanical Drawing Using SI Units

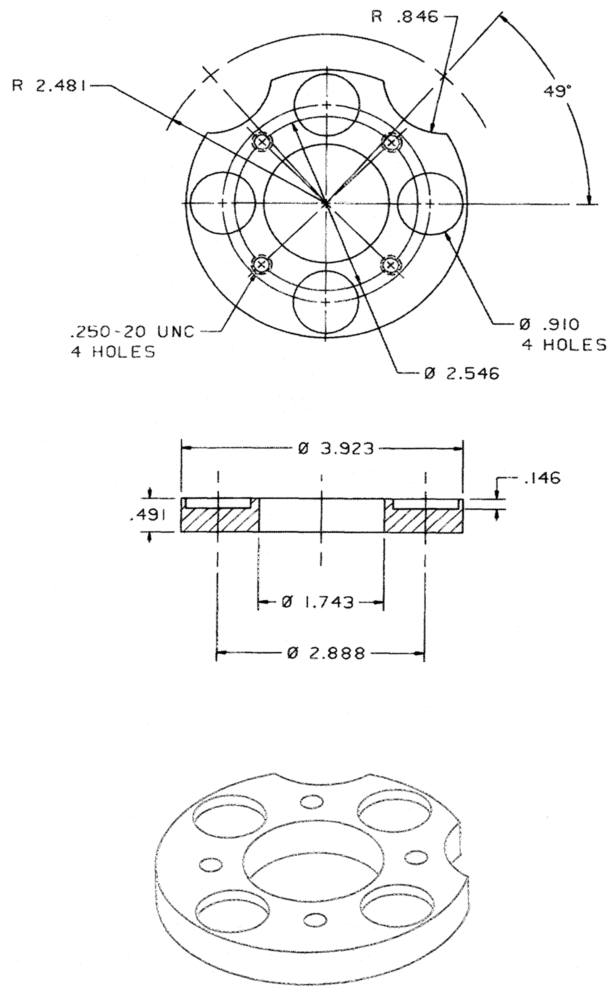


FIGURE 15.3 Mechanical Drawing Using U.S. Decimal Units

ask the system for clarification of a feature, request dimensions not included on the plotted drawing, and even create part programs for CNC machining from the database.

15.2.1 Dimensioning Terms

The following terms are used throughout this chapter.

Dimension A numeric value expressed in appropriate units of measure and indicated on a drawing and in other documents, along with lines, symbols, and notes, to define size or geometric characteristic, or both, of a part or part feature. *Examples:* 12.875 (in.), 25 (mm)

Reference dimension A dimension, usually without tolerance, used for information only. It is considered auxiliary information and does not govern production or inspection operations. A reference dimension either repeats a dimension or size already given or is derived from other values shown on the drawing or related drawings. Reference dimensions are enclosed within parentheses. *Examples:* (23.50), (50)

Datum An exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a part are established.

Feature The general term for a physical portion of a part. *Examples:* a surface, a hole, a slot

Datum feature A geometric feature of a part used to establish a datum. *Examples:* a point, a line, a surface, a hole

Actual size The measured size of the feature.

Limits of size The specified maximum and minimum limits of a feature.

Tolerance The total amount by which a specific dimension is permitted to vary. The tolerance is the difference between the maximum and minimum limits.

15.2.2 Units of Measurement

The SI linear unit commonly shown on engineering drawings is the millimeter. The U.S. customary linear unit on engineering drawings is the decimal-inch. On drawings where all dimensions are either in millimeters or in inches, individual identification of linear units is not required. However, the drawing must contain a note stating:

UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS ARE IN MILLIMETERS (or INCHES).

Dimensions are shown to as many decimal places as accuracy requires. The inch or millimeter symbol is omitted unless the dimension might be misunderstood or where feet and inches are used on construction drawings. With U.S. customary units, fractions and decimals are not mixed on the same drawing. If inch dimensions are shown on a millimeter-dimensioned drawing, the abbreviation "in." must follow the inch values. If millimeter dimensions are shown on an inch-dimensioned drawing, the symbol "mm" must follow the millimeter values.

Angular dimensions are expressed either in decimal parts of a degree or in degrees, minutes, seconds (° = degrees, ′ = minutes, ″ = seconds). If degrees are indicated alone, the numerical value is followed by the symbol °.

15.3 Types of Dimensioning

Decimal dimensioning is used on U.S. drawings, except where certain commercial commodities are identified by standardized nominal designations, such as pipe, steel, and lumber sizes.

Metric Dimensioning (Figs. 15.2 and 15.4):

1. If the dimension is less than 1 mm, a zero precedes the decimal point. *Example:* **.75**
2. If the dimension is a whole number, neither the decimal point nor a zero is shown. *Example:* **12**

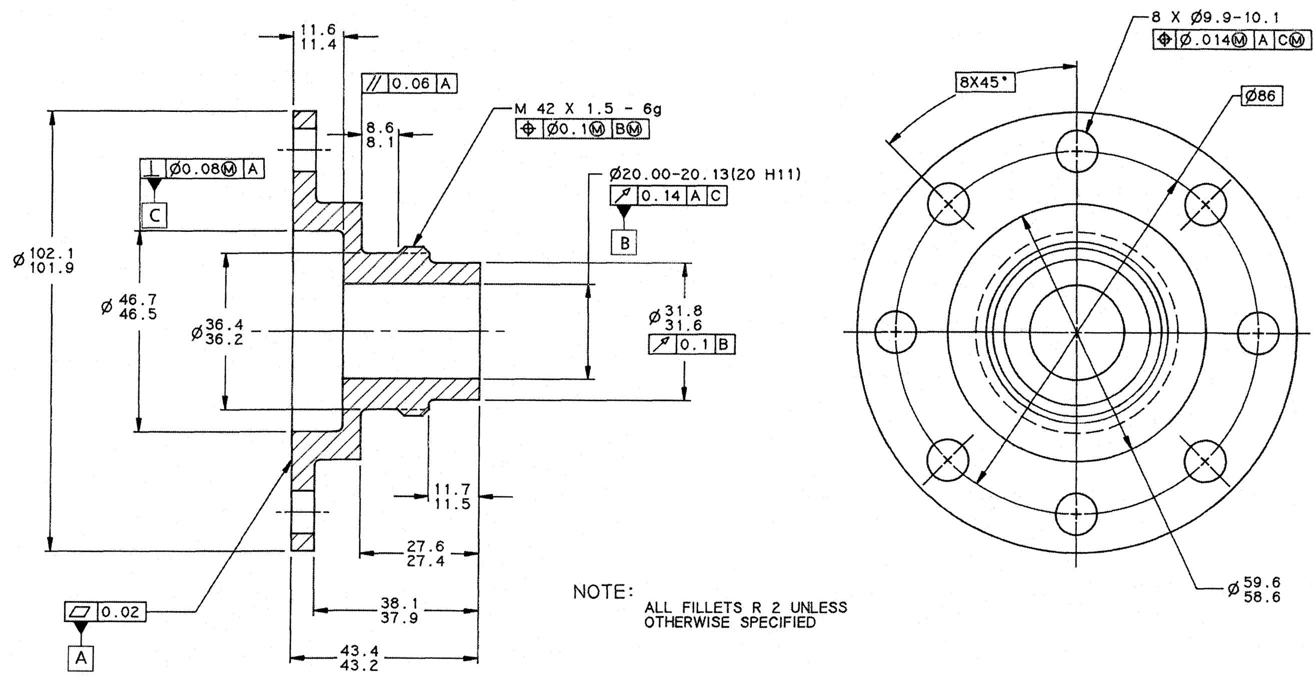


FIGURE 15.4 Geometric Tolerancing and Dimensioning Employed to Dimension a Mechanical Part

3. If the dimension exceeds a whole number by a decimal fraction of 1 mm, the last digit to the right of the decimal point is not followed by a zero. *Example:* **1.5**
4. Neither commas nor spaces are used to separate digits into groups in specifying millimeter dimensions on drawings. *Example:* **2500**

Decimal-Inch Dimensioning (Figs. 15.3 and 15.5):

1. A zero is not used before the decimal point for values less than 1 in. *Example:* **.375**
2. A tolerance is expressed to the same number of decimal places as its dimension. Zeros are added to the right of

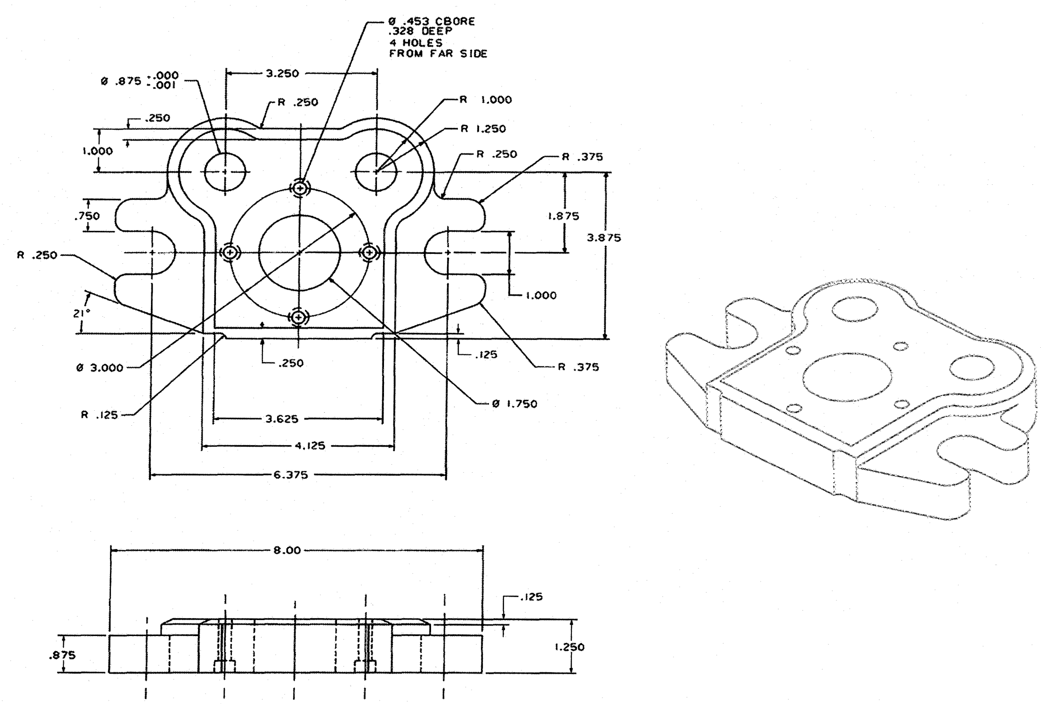


FIGURE 15.5 Mechanical Part Designed and Dimensioned in U.S. Standard Decimal Units

the decimal point where necessary for both the dimension and the tolerance.

Example: **1.001** **1.000**

Decimal Points (SI and U.S. units):

1. Decimal points must be uniform, dense, and large enough to be clearly visible and to meet the reproduction requirements of ANSI Y14.2M. Decimal points are placed in line with the bottom of the associated digits. *Example:* **.875**
2. When a dimension is 1 unit, always add a decimal and a zero. *Example:* **1.0**

15.3.1 Dual Dimensioning

Because many parts designed in the United States are manufactured or traded in foreign countries, some drawings employ **dual dimensioning,** that is, U.S. customary units and metric units. The top measurement, or the first measurement when placed on the same line, is always the unit of measurement used to design the part. *Example:* $\frac{\mathbf{1.00}}{\mathbf{25.4}}$

15.3.2 Dimensioning Numerals

Whole numbers in the inch system are normally shown to at least one decimal place (e.g., **1.0** or **2.0**). This practice prevents dimensions from being "lost" on the drawing, which is common when the number 1 is not accompanied by a decimal point and a zero.

Common-fraction dimensions are seldom used, except on construction drawings. Before the decimal-inch was adopted as a standard, common fractions were employed for subdivisions of an inch. Some companies still use this system. Older drawings and sheet metal drawings also show common fractions.

Using decimals has many advantages. Decimals reduce arithmetic computation time. For example, it can take as much as five times longer to add a series of fractions than a series of decimals. Many decimal measurements must be rounded before they are used.

15.3.3 Rounding Decimal-Inch Measurements

ANSI has a standard method to round decimals. A decimal-inch value may be rounded off to a lesser number of places by the following procedure:

1. If the last digit to be dropped is less than 5, there is no change in the preceding digits.

Examples

.47244	rounds to	.4724
.1562	rounds to	.156
.20312	rounds to	.2031
.35433	rounds to	.3543

2. If the last digit to be dropped is greater than 5, the preceding digit is increased by 1.

Examples

.23437	rounds to	.2344
.55118	rounds to	.5512
.03937	rounds to	.0394
.6406	rounds to	.641

3. If the last digit to be dropped is 5 followed by a zero, round the preceding digit to the nearest even number.

Examples

.98425	rounds to	.9842
.59055	rounds to	.5906
.19685	rounds to	.1968
.4375	rounds to	.438

If precise calculation is required, values should be calculated to two places beyond the desired number of places; rounding should be based on the last two significant digits.

15.3.4 Drawing Scale

Drawings should be drawn to a scale so that they are easy to read and interpret. Scales are constant within a given project for which multiple drawings are needed. Scales are stated in the title block: 1:1 (full scale), 1:2 (half scale), 5:1, 10:1, and so on.

In some cases, such as when a portion of the drawing is enlarged, more than one scale is used on a drawing. In Figure 15.6, **DETAIL B** is **4/1**. The predominant scale is shown in the scale area within the title block.

15.4 DIMENSIONS

Dimensions involve standard elements: dimension lines, extension lines, leaders, arrowheads, and dimension values. The types of dimensions include vertical, horizontal, and aligned linear dimensions, angular dimensions, and callout dimensions using leaders for notes (Fig. 15.7). Figure 15.8 shows typical dimensions with examples in both decimal-inches and millimeters. When a dimension is small, the arrowheads and dimension line can go outside the extension lines, with the value inside. Another method allows the dimension value as well to be placed outside the extension lines.

Any drawing is only as good as its dimensioning. Accurate drawing and correct placement of all dimensioning elements is essential for the engineering drawing to transfer information correctly. *Dimension to get the part made correctly.*

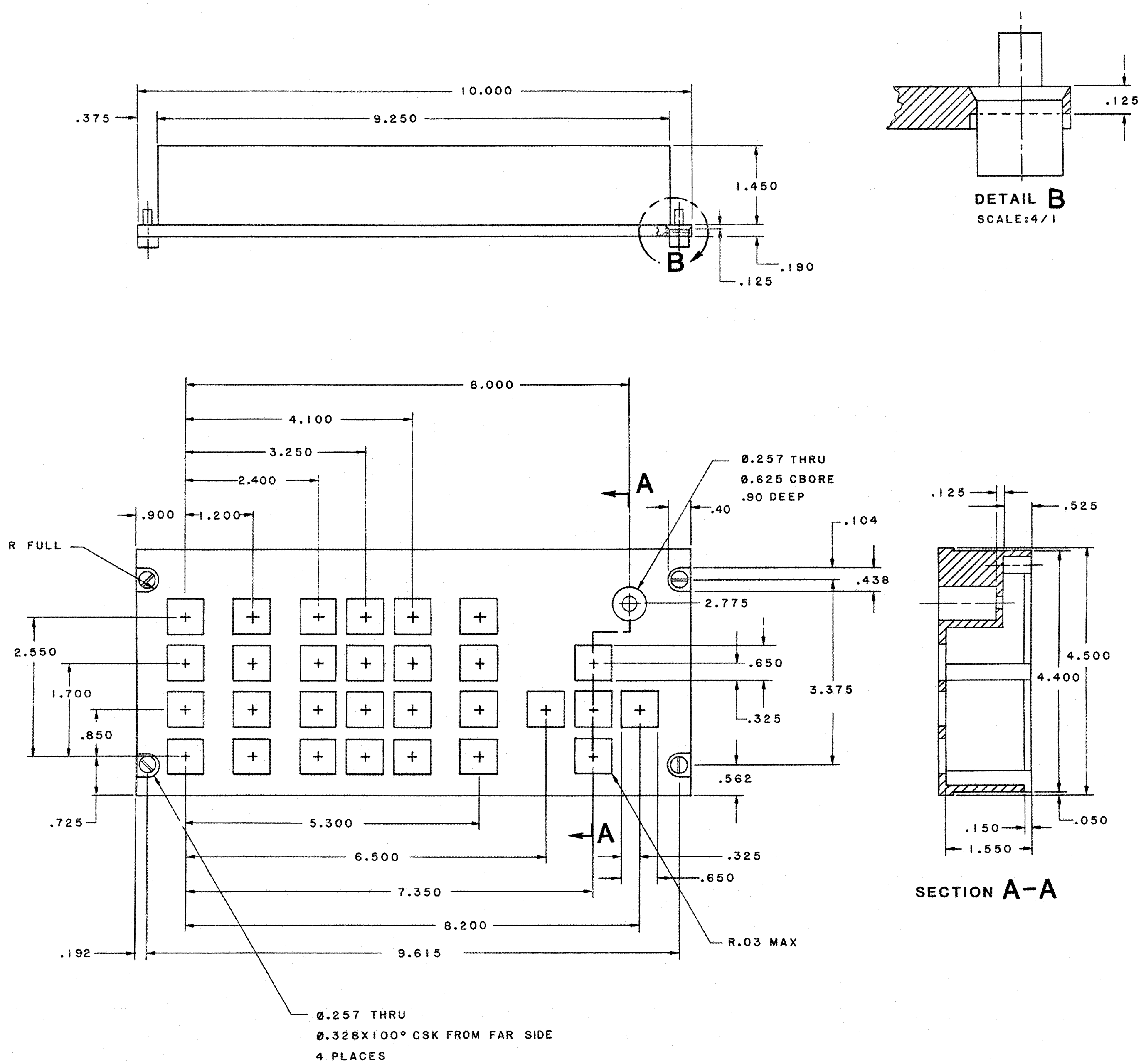


FIGURE 15.6 Panel Detail

15.4.1 Dimension Lines

A **dimension line**, with its arrowheads, shows the direction and extent of a dimension. Numerals indicate the number of units of a measurement. Preferably, dimension lines are broken to insert these numerals, as shown in Figure 15.7. If horizontal dimension lines are not broken, the numerals are placed above and parallel to the dimension lines. *Do not use centerlines, phantom lines, object lines that represent the outline of a part, or a continuation of any of these lines for dimension lines.* A dimension line is used as an extension line only where a simplified method of coordinate dimensioning is employed to define curved outlines.

Avoid crossing dimension lines. Where this is unavoidable, break the dimension lines at the crossing point. The largest dimension always goes on the outside, farthest from the part's outline (Fig. 15.9). Figure 15.9 illustrates a number of rules:

- Cross a dimension line only if it is unavoidable.
- Do not place dimensions within the part outline unless there is no other place to show the dimension properly.
- Place larger dimensions farthest from the part's outline.

Dimensions are usually placed outside the outline of the part. If directness of application makes it desirable or if

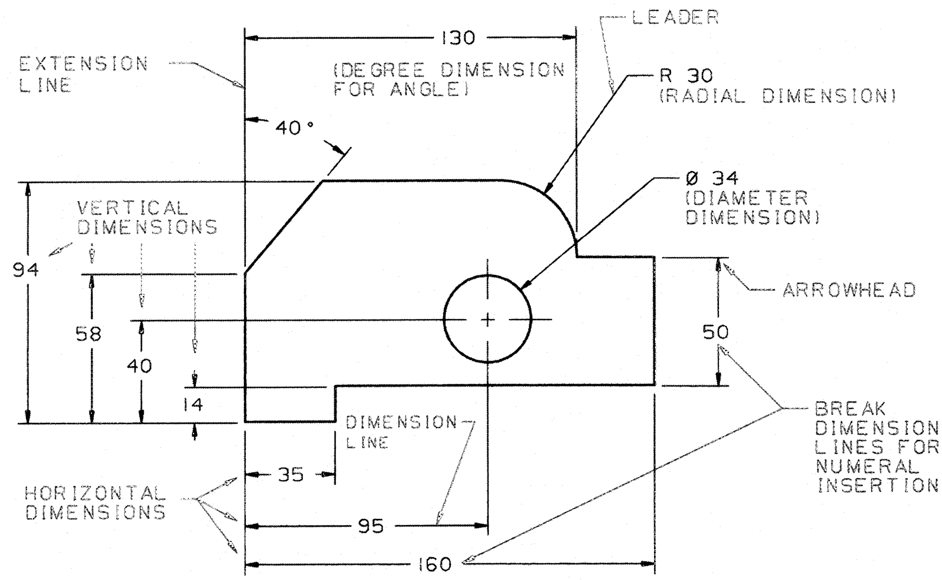
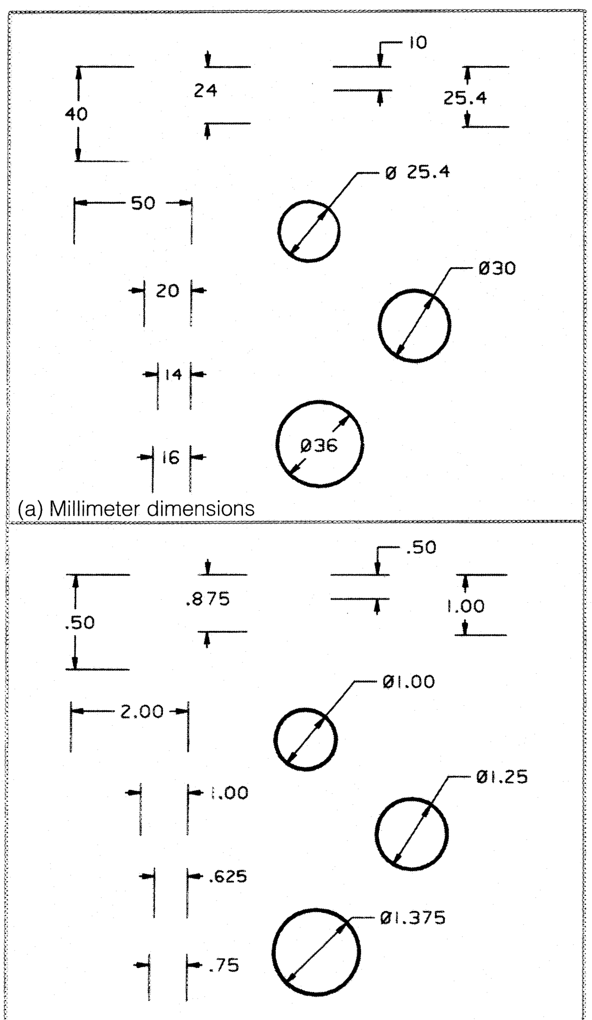


FIGURE 15.7 Dimension Elements



(a) Millimeter dimensions

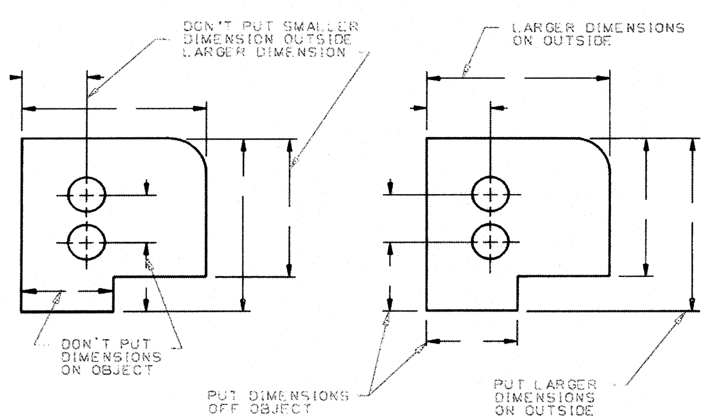
(b) Decimal-inch dimensions

FIGURE 15.8 Dimensions

extension lines or leader lines would be excessively long, dimensions may be placed within the outline of a view. On large, complex drawings, dimensions are sometimes placed within the outline of the part, even when this contradicts the dimensioning rules. If it is necessary to place a dimensioning inside a part's outline that is sectioned, break the section lines around the dimension (Fig. 15.10).

15.4.2 Extension Lines

Extension lines indicate the extension of a surface or point to a location outside the part outline. Extension lines start with a short visible gap from the outline of the part and extend beyond the outermost related dimension line. Extension lines are drawn perpendicular to dimension lines. If space is limited, extension lines may be drawn at an oblique



(a) Incorrect (b) Correct

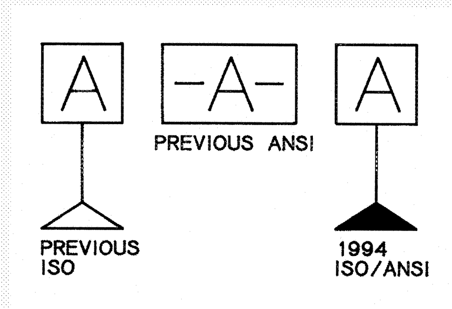
FIGURE 15.9 Dimension and Extension Lines

Focus On . . .

WHY CHANGE ANSI STANDARDS?

Ever since you began your study of technical or engineering drawing, you have been learning how to produce drawings according to standards that have been established by the American National Standards Institute (ANSI). You may not have realized that these established standards change and evolve to match the needs of the ever-changing technology involved in manufacturing and production. At first it might seem odd to you that an established standard could or should change. After all, the very purpose of a standard seems to oppose the idea of changing it. However, standards exist to assist manufacturers and to make the production of parts and assemblies more efficient. If you think carefully about the quick evolution of modern technology and the new worldwide marketing and manufacturing environment, it seems reasonable to expect standards to change.

In 1935 the American Standards Association (the predecessor to ANSI) published the first recognized standard for engineering drawings in the United States—"American Drawing and Drafting Room Practices." The document was eighteen pages long, and the entire subject of tolerancing was covered in two paragraphs. It was clear in that era that the assembly-line manufacturing process created largely by Henry Ford for his Model T replaced forever the old "fit and file to size" craftsman-type manufacturing process. This mass assembly process created a pressing need for different shops to be able to produce the same parts. The exchange of drawings for those parts became critical, and a group was formed to create a standard way to communicate manufacturing and engineering details graphically. It took years for the group to publish the standards document. Whose standard is best is, and will continue to be, a difficult question for any group of individuals charged with creating a standard.



Datum identifying symbols.

World War II provided the motivation to continue to improve engineering drawings and mass production techniques. Scrap rates were too high, and assemblies were hampered by the limitations of the plus/minus tolerance system. It became apparent that geometry and not just variation in size controlled many assemblies. How many times have you drilled a hole with a hand drill and realized that the hole was the correct size but it was of no use to the assembly because the axis of the hole wasn't perpendicular to the right plane? The U.S. Army published an Ordnance Manual on Dimensioning and Tolerancing in 1945 that used symbols to specify form and position tolerances. Unfortunately, the American Standards Association's "American Standard Drawing and Drafting Room Practice," second edition, published in 1946, lacked a comprehensive section on tolerancing. To make matters worse, the Society of Automotive Engineers published its own standards in 1946 and in 1952. After the war, there were *three different*

angle to illustrate clearly where they apply. If oblique lines are involved, the dimension lines are shown in the direction in which they apply (Fig. 15.11).

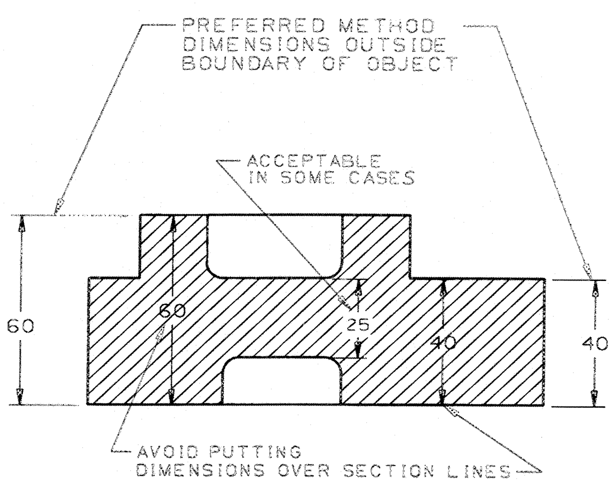


FIGURE 15.10 Dimensions on Section Lining

Extension lines should not cross dimension lines. To minimize such crossings, the shortest dimension line is shown nearest the outline of the part (Fig. 15.12). If

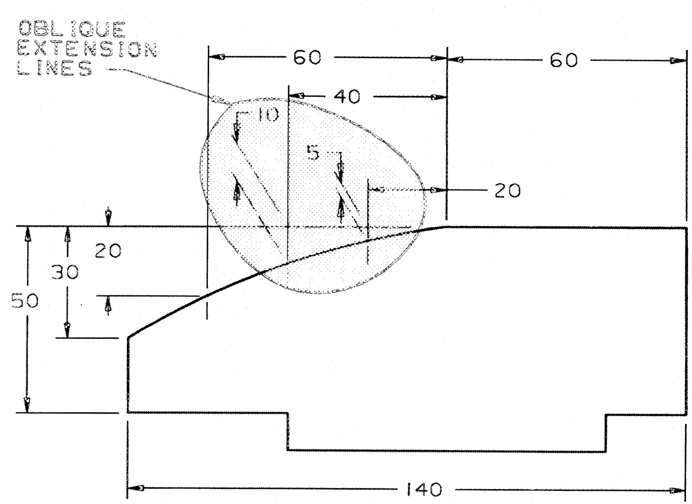
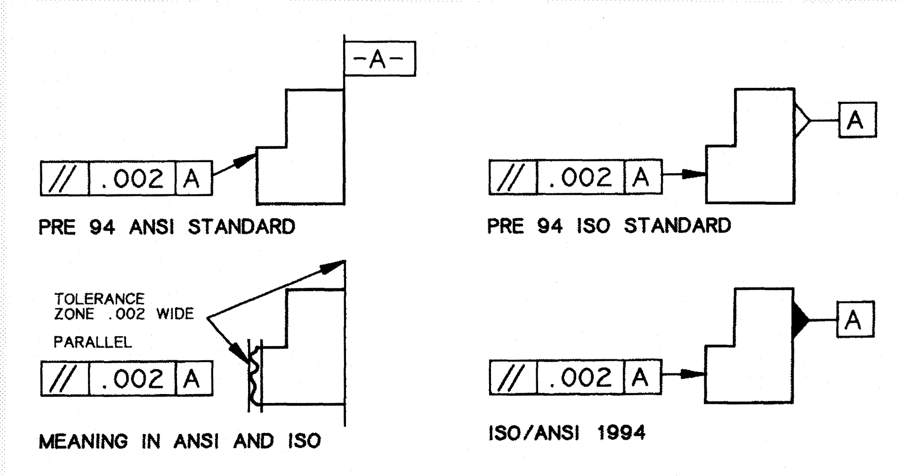


FIGURE 15.11 Oblique Dimensions

Parallelism orientation tolerancing standards.



standards for tolerancing engineering drawings. It wasn't until 1966, after years of debate, that ANSI published the first unified standard on tolerancing and dimensioning—ANSI Y14.5. The standard was updated in 1973 and 1982 to replace notes with symbols for all tolerancing, and the current version was published in 1994. If all that seems a bit complicated, remember that the rest of the world has been developing their own standards and formed the International Organization for Standardization (ISO).

Evolving technology and the need to compete in world-class manufacturing seem to be driving the next round of ANSI standard revisions. CAD and CAM have become key components in manufacturing since the last ANSI Y14.5 revision. Producing a part with a CNC machine becomes easier when the part is dimensioned with that production technique in mind. The use of decimals has replaced fractions for that same reason. Unfortunately, anyone who has used a CAD system will tell you that keeping to ANSI standards has never seemed to worry the makers of CAD systems very much. The outcome of that dilemma has not been resolved. However, competing in world-class manufacturing is very important to all American manufacturers. Parts for any one assembly will more than likely be produced in a variety of shops across the world. To be part of that network, it seems that ANSI standards and ISO standards must be compatible so that engineering drawings convey the same information worldwide. Conveying information quickly and correctly is a must to compete effectively in today's world-class manufacturing environment.

The men and women charged with making ANSI and ISO standards have a difficult and complex job. Each document they produce must contain compromises. As technology and the world environment change, standards must evolve with them. As the world economy becomes more unified, more world-unified standards will certainly follow. The ability to change and adapt seems more important than ever to our continued success.

extension lines must cross other extension lines, dimension lines, or lines depicting features, they are not broken. However, an extension line is broken where it crosses arrowheads or dimension lines close to arrowheads (Fig. 15.12). Note that most CAD systems will not break the extension line for any reason. As a design engineer using a

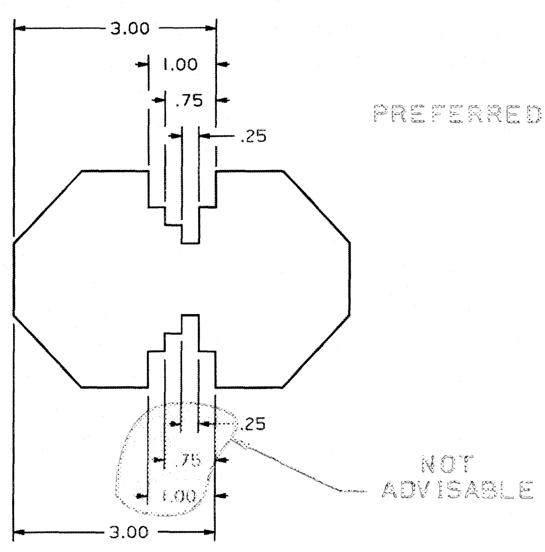


FIGURE 15.12 Breaks in Extension Lines

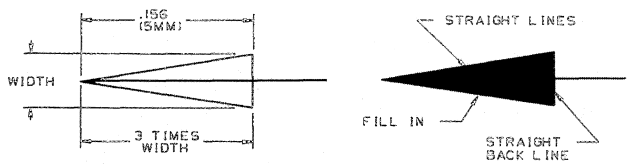


FIGURE 15.13 Arrowheads

CAD system, you will be limited by the capabilities of the system. However, many systems today have sophisticated features that have enhanced abilities to model complex geometries and surfaces.

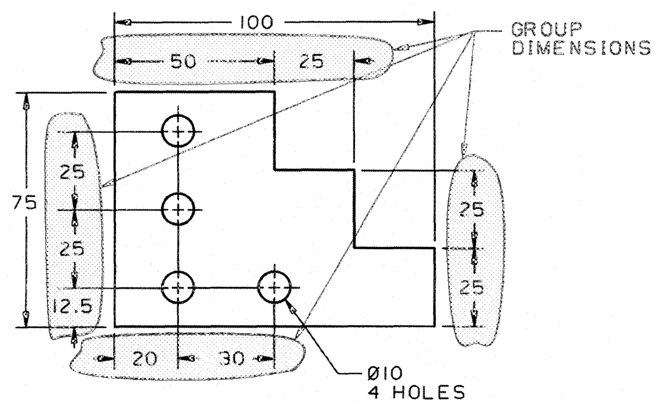


FIGURE 15.14 Grouping Dimensions

15.4.3 Drawing Dimension Arrowheads

The thickness of dimension, leader, or extension lines is normally 0.3 to 0.35 mm. They are the thinnest lines on the drawing (along with section lines) and must be drawn crisp and black. For all lines except construction lines, the thickness may change, but the darkness remains the same.

The arrowhead for dimensions is shown in Figure 15.13. The sides and back of the arrowhead are straight, not curved. An arrowhead is about three times as long as it is wide, with a length approximately equal to the height of the lettering on the drawing. Arrowheads are drawn completely filled. Other types of line terminators used throughout industry include open arrowheads, dots, and slashes. Keep arrowheads consistent and uniform. Avoid large arrowheads that stand out when reading the drawing. With time and practice, your freehand arrowheads will become easy to construct and well formed.

CAD systems provide arrowheads in a variety of sizes, shapes, and types, all of which can be inserted automatically. With a CAD system, you do not actually construct dimensions and arrowheads, but the proper selection of dimensions and their placement is still your responsibility.

15.4.4 Drawing Dimension and Extension Lines

Dimension lines are aligned and grouped for uniform appearance (Fig. 15.14). If there are several parallel dimension lines, the numerals should be staggered for easier reading (Fig. 15.15). Figures 15.15 and 15.16 show staggered spacing for horizontal dimensions; Figure 15.17 shows staggered vertical dimensions.

The minimum distance from the first dimension line to the part outline should be .375 in. (10 mm). The minimum spacing between parallel dimension lines should be .25 in. (6 mm) (Fig. 15.18). In general, .50 in. (12 mm) from the part and .375 in. to .50 in. (10 to 12 mm) between dimensions is suggested for large drawings and those that

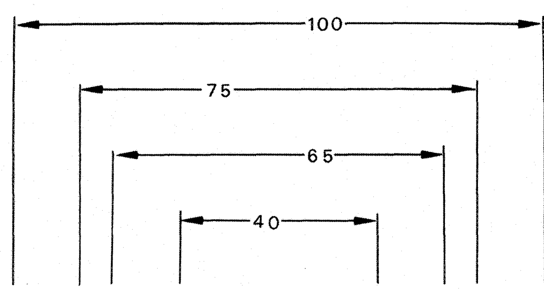


FIGURE 15.15 Staggered Dimensions

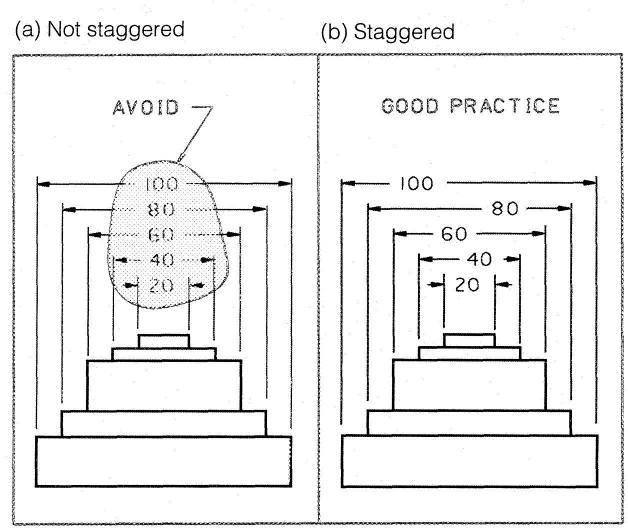


FIGURE 15.16 Horizontal Dimensions

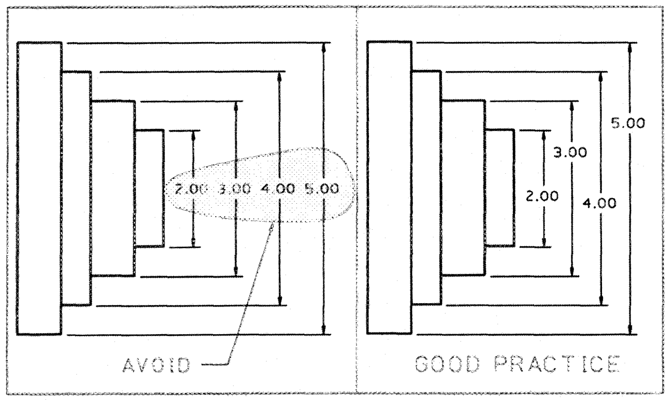


FIGURE 15.17 Vertical Dimensions

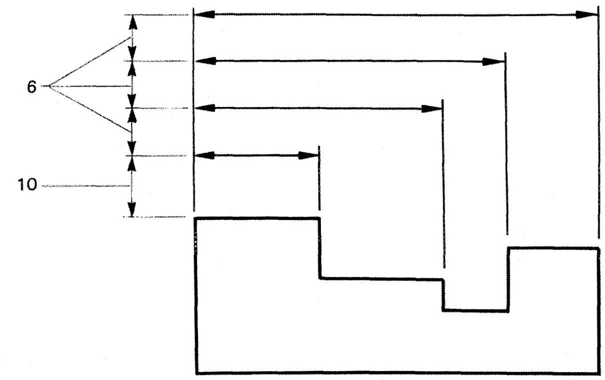


FIGURE 15.18 Setup and Spacing of Dimensions

need to be greatly reduced. These spacings are intended as a guide when dimensioning, not as a rule. If the drawing meets the reproduction requirements of the accepted industry or military reproduction specification, these spacing requirements are not mandatory.

Extension lines should start about .06 in. (1.3 mm) from the part and end approximately .12 in. (2.5 mm) beyond the dimension line and arrowhead (Fig. 15.19). *Centerlines can be used as extension lines but not as dimension lines.*

All holes are dimensioned to their centerlines in two directions, except when the holes are arrayed in a circular pattern (as with a bolt circle). If a point is to be located only by extension lines, the extension lines (from the surfaces) pass through the point (Fig. 15.20).

Extension lines are not drawn to hidden lines on hidden features of the part. However, dimensioning to hidden features is acceptable in some circumstances (when a feature cannot be seen in another view). Another way to avoid dimensioning to hidden lines is to make use of a broken-out section for the hidden features. *Whenever possible, dimension to a visible feature.*

Remember: for almost every dimensioning rule, there is an exception. The rules apply in probably 90% of the situations that you will encounter.

15.4.5 Lettering Dimensions

The preferred heights for lettering dimensions are shown in Figure 15.21. Dimension heights are standardized for each drawing size and reduction requirement. If reduction is not required, .125 inch (3 mm) height for lettering is acceptable.

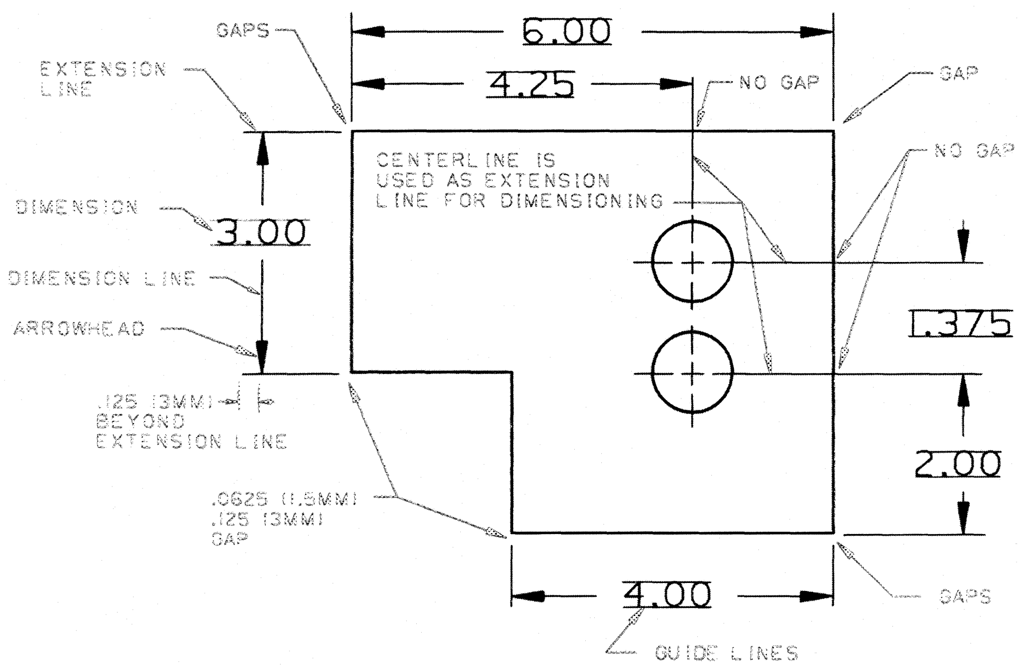


FIGURE 15.19 Gaps and Placement of Extension Lines

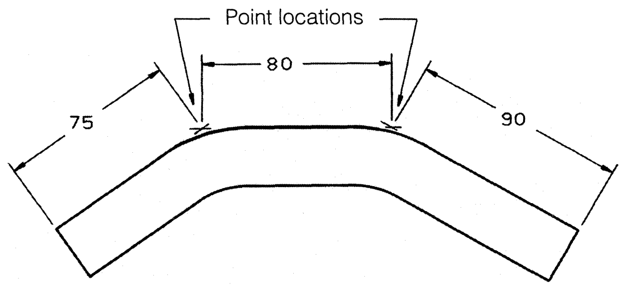


FIGURE 15.20 Point Locations Using Extension Lines

Follow the spacing of lettering in dimensions in Figure 15.21 to complete projects in this text.

Numerals that are placed parallel to dimension lines are called **aligned dimensions** [Fig. 15.22(a)]. Horizontal dimensions are readable from the bottom, vertical dimensions from the right side of the drawing. Point-to-point dimensions of angled edges have the dimensions aligned (parallel) to the edge itself. Aligned dimensions are *not* accepted ANSI practice.

Unidirectional dimensioning [Fig. 15.22(b)] places the dimension text parallel to the bottom of the drawing. This system is preferred, since the drawing can be read and lettered without being turned (Fig. 15.23); that is, it is readable from the bottom of the drawing.

For mechanical drawings, the dimension lines are broken to insert the measurement numerals. Piping, architecture, civil, structural, and other construction drawings do not normally break the dimension line, but instead place numerals above the dimension line.

Regardless of the type of unit or the alignment of the dimension value, use thin, lightly drawn guidelines or a lettering guide. The lettering itself must be crisp, black, and as thick as a hidden or visible object line.

15.4.6 Angular Dimensions

Size and location dimensions may be linear distances or angles. **Angular dimensions** are expressed in degrees, minutes, and seconds or as decimal equivalents of degrees, as in Figure 15.24. For angles of less than 1°, precede the minute mark by 0° (e.g., **0°40′**). For both unidirectional and aligned dimensioning, angular dimensions are placed to be read horizontally between guidelines, with no dash between degrees and minutes.

Angular dimensions should be avoided by locating the endpoints of inclined lines and planes. Because it is easier, quicker, and more reliable, coordinate dimensioning of angled features increases the accuracy during manufacturing.

The dimension line for an angle is drawn as an arc from

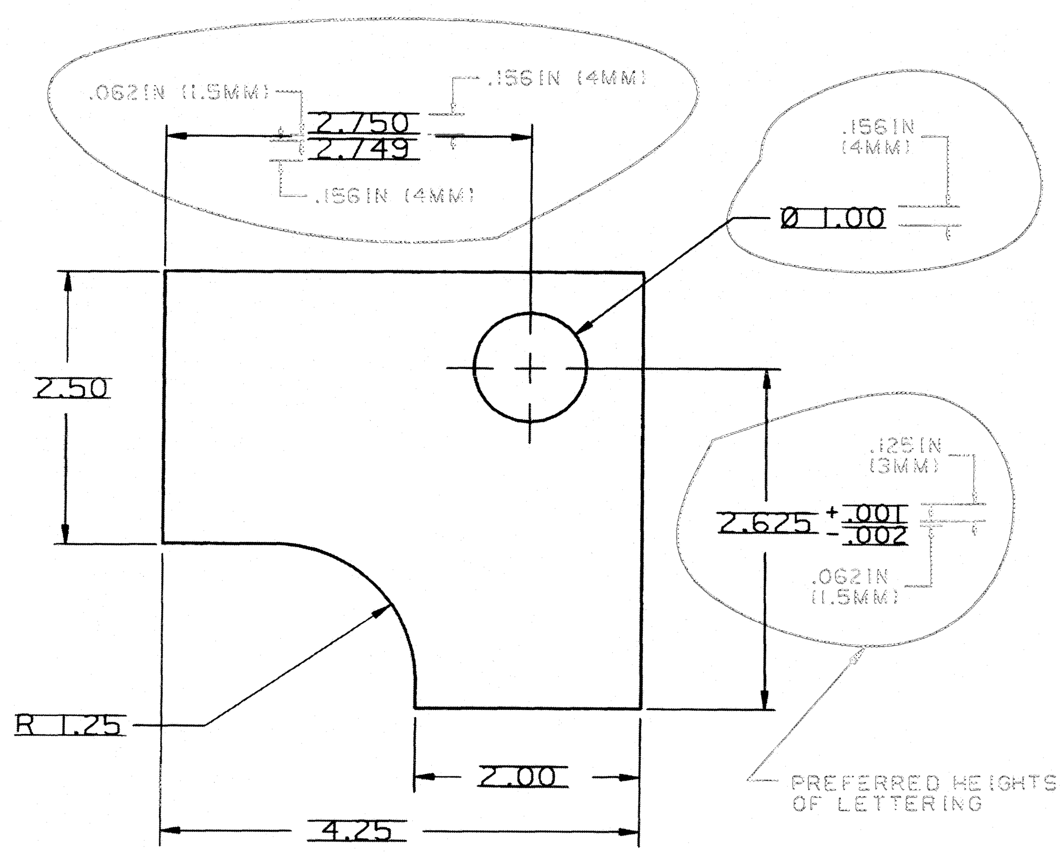


FIGURE 15.21 Preferred Lettering Height for Dimensions

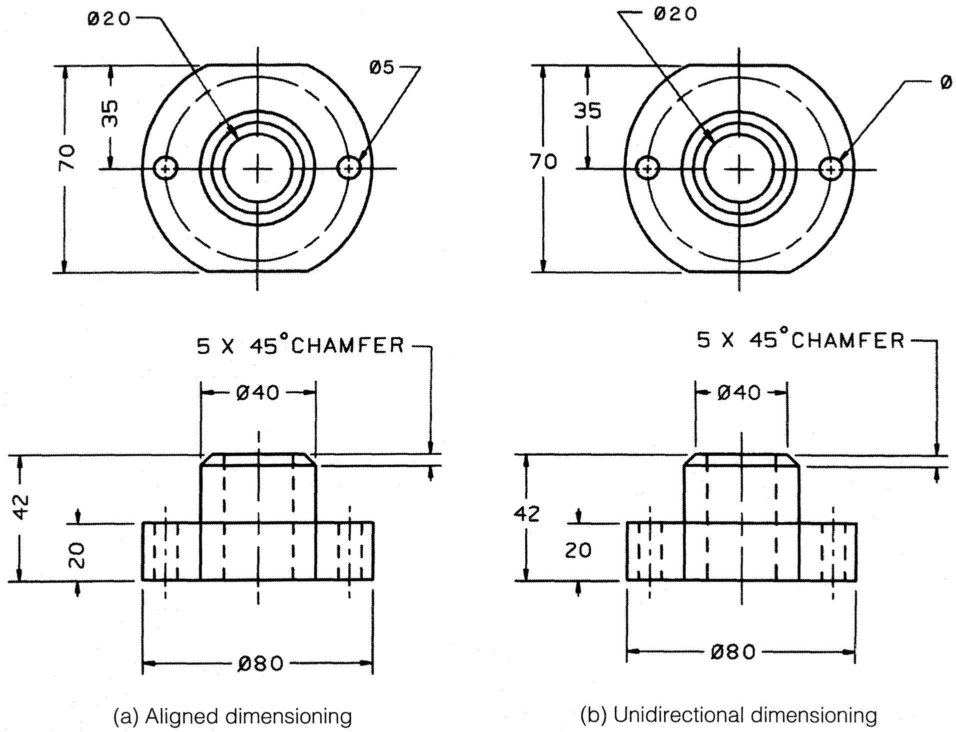


FIGURE 15.22 Dimensioning Methods

a center at the intersection of the sides of the angle. A variety of methods are employed to dimension angles (Fig. 15.24). The arrowheads terminate at the extensions of the two sides, inside or outside the extension lines. The dimension line is an arc with its center at the vertex of the angle being measured, and the angular dimension is placed inside or outside the two controlling extension lines.

Angles are used only where other forms of linear dimensions are unsuitable. In Figure 15.25, two methods of dimensioning a part are illustrated. One method [Fig. 15.25(a)] involves angle dimensions, and the other [Fig. 15.25(b)] the offset method. Because it is easier for the machinist to locate the features of the part with linear measurements, the offset method is preferred. Figure 15.26 shows another example of these methods. Here the parts are different and lend themselves to different dimensioning styles: (a) uses the offset method to locate the holes; (b) has a slot angled to the base. Because the features in Figure 15.26(b) would be difficult to locate with offset dimensions, an angle dimension works best. The features of part (b) are related to the angled surface and, therefore, are located in relation to it when establishing dimensions.

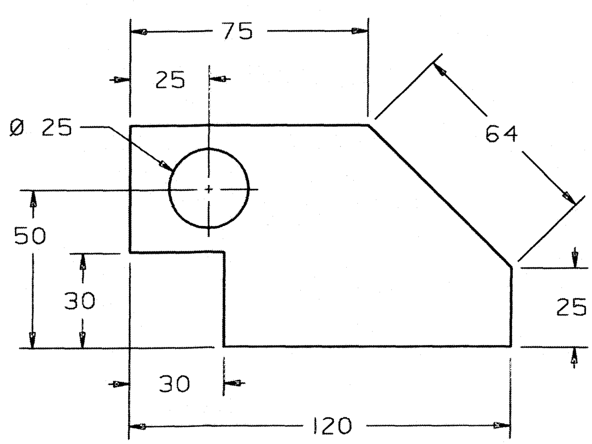


FIGURE 15.23 Unidirectional Dimensions

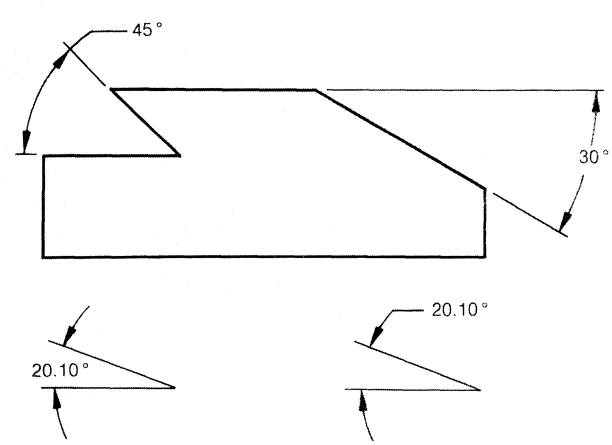


FIGURE 15.24 Dimensioning Angles

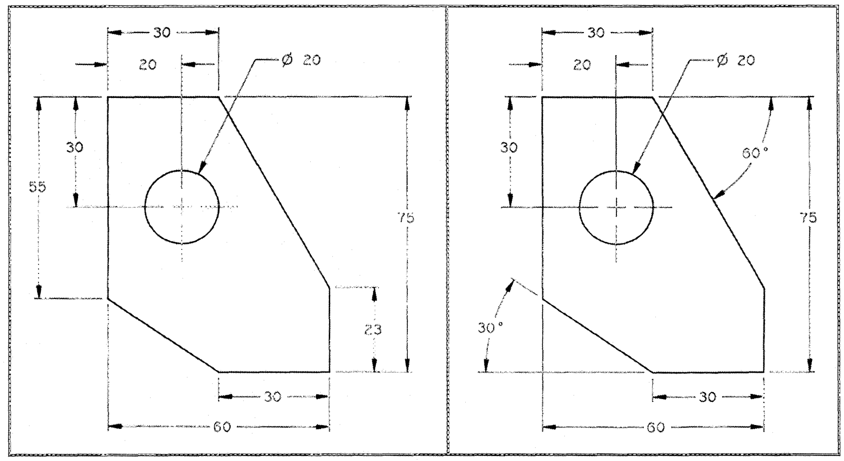
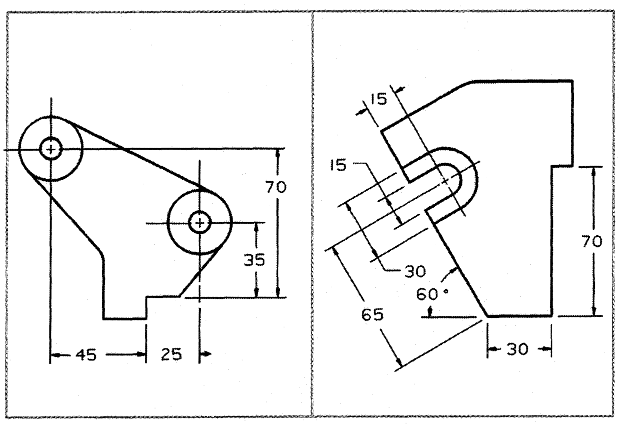


FIGURE 15.25 Dimensioning

(a) Offset (square) dimensioning (b) Angular dimensioning



(a) Offset dimensioning (b) Angular dimensioning

FIGURE 15.26 Dimensioning

15.4.7 Callout Dimensions and Notes Using Leaders

A **leader** directs a dimension, note, or symbol to the intended feature on the drawing (Fig. 15.27). Leaders can point to a curved feature of a part or reference a portion or surface. Most leaders are drawn at 45° to 60° to the horizontal (30° to the vertical) (Fig. 15.28). Leaders terminate in arrowheads. The dimension figure for a callout is placed at the end or the head of a short [6 mm (.25 in.)] horizontal line.

Figure 15.29 shows the three most common uses of a leader: to call out a hole diameter, to call out a radius, and to reference a surface or part with a note. When a leader serves to dimension a circle or arc, it must point to or from (or through) the center of the circle or radius. The arrowhead

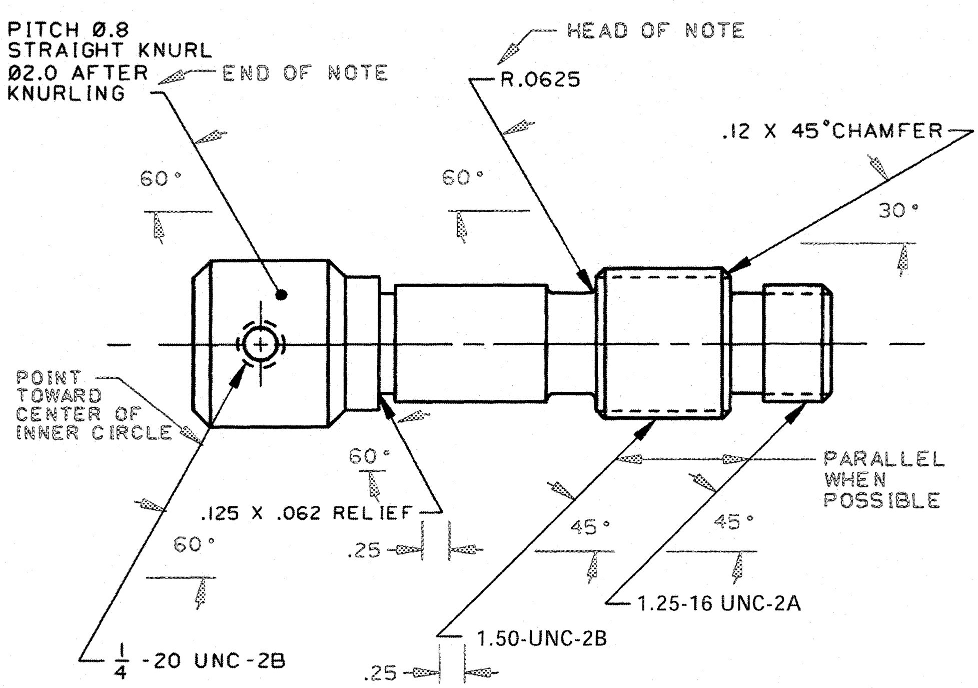


FIGURE 15.27 Leaders on Drawings

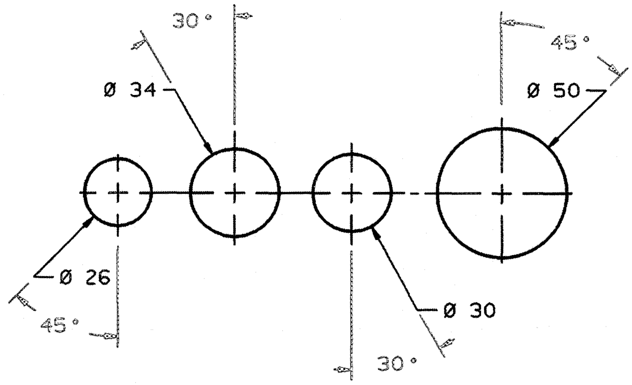


FIGURE 15.28 Leaders for Hole Callouts

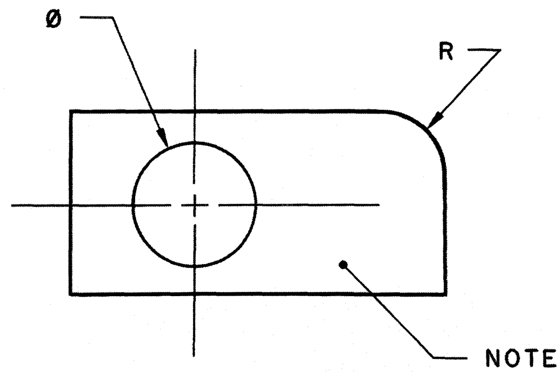


FIGURE 15.29 Three Common Uses of Leaders

points toward the center of the curve (Fig. 15.30). Therefore, *all leaders for radii and diameters are radial.* The arrowhead for a leader terminates at the circumference of the arc or diameter.

The crossing of dimension lines and extension lines by

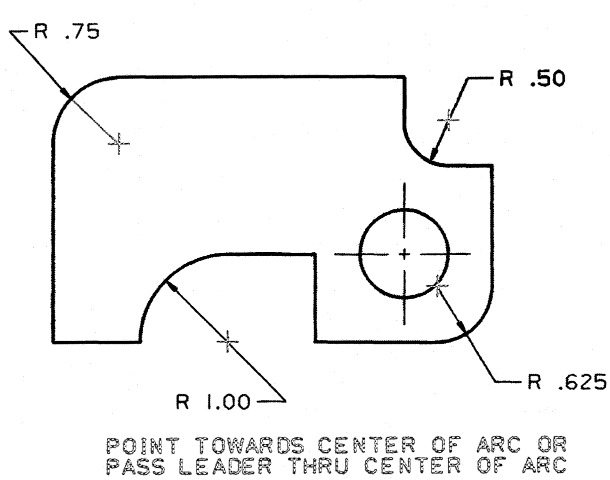


FIGURE 15.30 Dimensioning Fillets and Arcs with Callouts

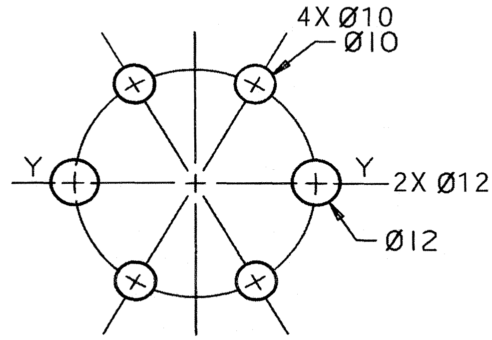


FIGURE 15.31 Minimizing Leaders on Drawings

leaders should be kept to a minimum. The leader line is drawn at a different angle than object lines of the part or section lines on the drawing.

Leaders and their accompanying notes and callouts are kept outside dimension lines and away from the part being dimensioned. Leaders are placed on the drawing after the part is dimensioned. Although leaders can cross object, dimension, and extension lines, *they should never cross other leader lines.*

Leader-directed dimensions are specified individually, for simplicity (Fig. 15.30). If too many leaders impair the legibility of the drawing, letters or symbols are used to identify the features (Fig. 15.31).

15.4.8 Reference, Overall, and Not-to-Scale Dimensions

Reference dimensions are not used for manufacturing or inspection. To identify a reference dimension or reference data on drawings, *enclose the dimension or data within parentheses.*

If an overall dimension is specified, one intermediate dimension is omitted or identified as a reference dimension (Fig. 15.32). When the intermediate dimension is more important than the overall dimension, the overall dimension is identified as a reference dimension.

To indicate that a feature is **not to scale,** the dimension should be underlined with a straight thick line (e.g., **101** in Fig. 15.33), or **NTS** (not to scale) should be added to the dimension.

Only the dimensions required for manufacturing the part are on the drawing. In Figure 15.34, the overdimensioned part is shown on the top of the figure and the correctly dimensioned part below it. Note that the correctly dimensioned example has fewer dimensions and an uncluttered look. The placement of ℄ on the part's centerline means the part is *symmetrical about its centerline.*

15.4.9 Indicating Limited Length or Area

To indicate that a limited length or a limited area of a surface is to receive additional treatment or consideration within limits specified on the drawing, the extent of those limits is

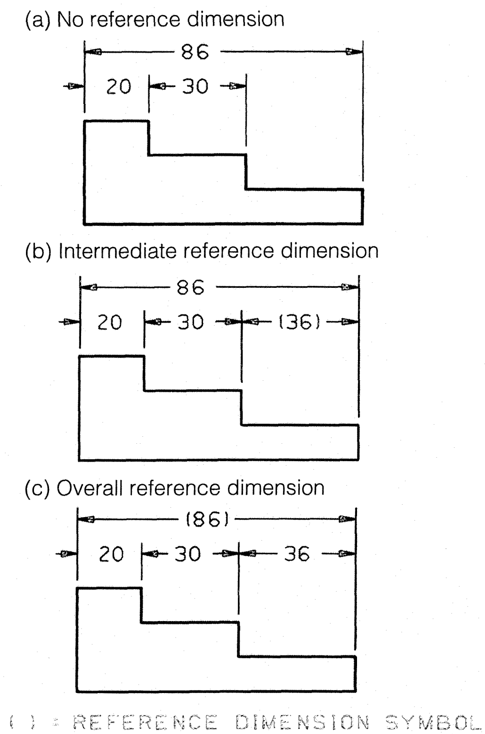


FIGURE 15.32 **Overall and Reference Dimensions**

indicated by a **chain line** (Fig. 15.35). In an appropriate view or section, a chain line is drawn parallel to the surface profile at a short distance from it. Dimensions are added for length and location [Fig. 15.35 (a)]. For a surface of revolution, such as a shaft, the indication is shown on one side only [Fig. 15.35 (a)].

As long as the chain line clearly indicates the location

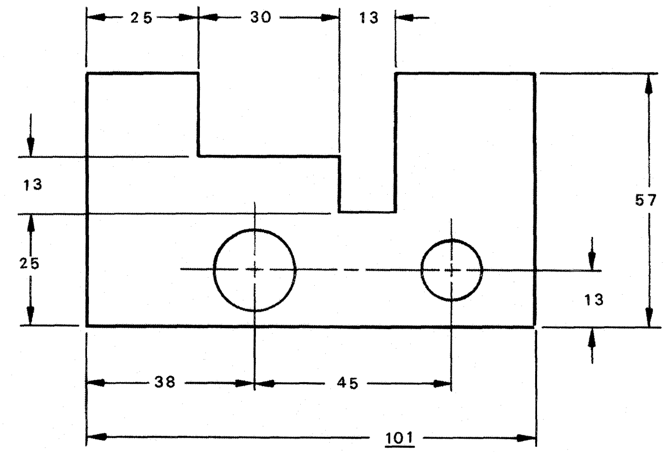


FIGURE 15.33 **Not-to-Scale (NTS) Dimensions**

extent of the *limited length,* dimensions may be omitted [Fig. 15.35(b)]. When the *limited area* is shown on a direct view of the surface, the area is section-lined within the chain line boundary and dimensioned [Fig. 15.35(c)].

15.5 DIMENSIONING CURVED FEATURES

Included in this section are methods of noting and dimensioning curved features, such as radii, diameters, slots, counterdrills, countersinks, spotfaces, and counterbores. ANSI symbology is also covered.

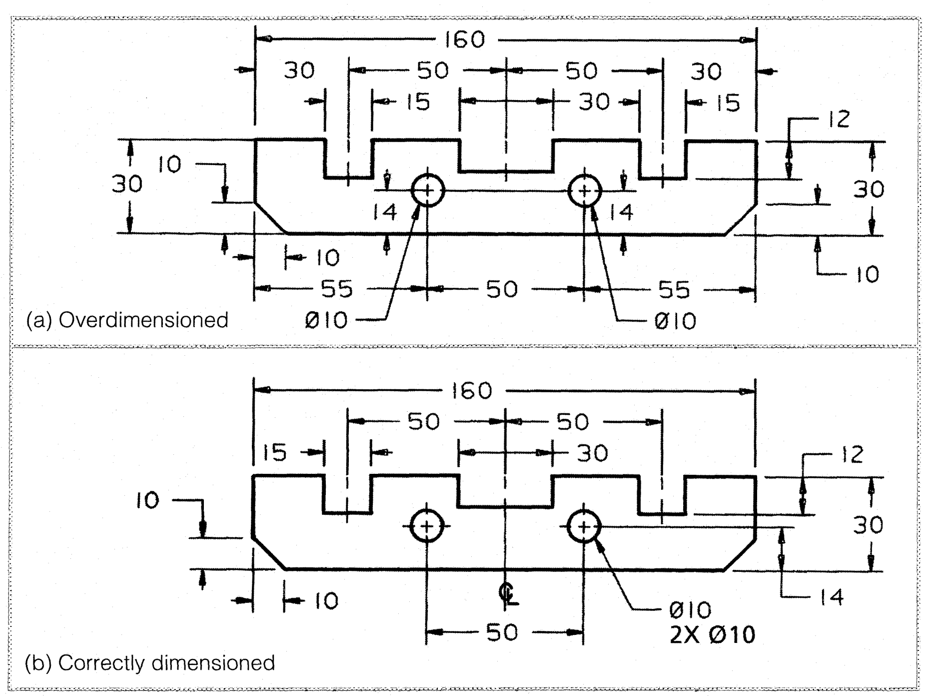


FIGURE 15.34 **Overdimensioning a Part**

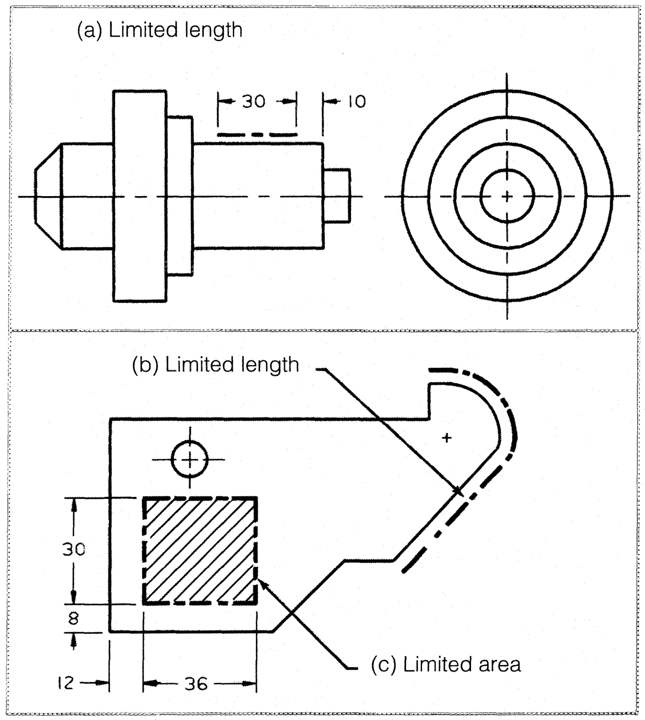


FIGURE 15.35 Limited-Length and Limited-Area Indicators

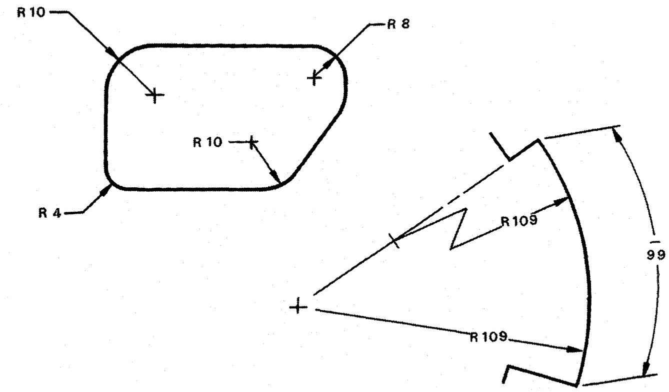


FIGURE 15.36 Dimensioning Radii and Arcs

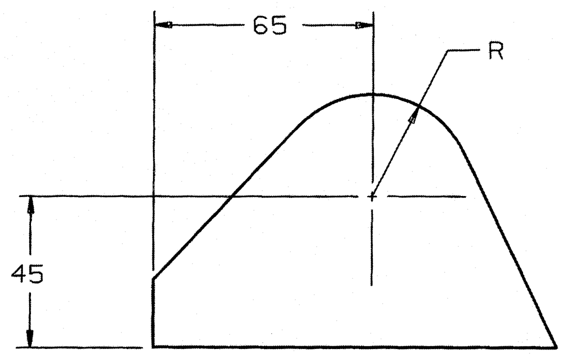


FIGURE 15.37 Radius with a Located Center

15.5.1 Radius Dimensioning

Radius dimensions help to call out slots, curves, arcs, rounds, and fillets. Each radius value on a radius dimension is preceded by the appropriate radius symbol, **R** (Fig. 15.36). A radius dimension line has one arrowhead, which points to the arc from the center. An arrowhead is not used at the radius center. The dimension line for any radius is an angular line extending radially through, from, or toward the center of the feature. Do not use horizontal or vertical lines when dimensioning arcs. Figure 15.36 illustrates the following:

- If the location of the center is important and space permits, draw a dimension line from the radius center, with the arrowhead touching the arc. Place the dimension between the arrowhead and the center.
- When space is limited, extend the dimension line through the radius center.
- When it is inconvenient to place the arrowhead between the radius center and the arc, place it outside the arc, with a leader.
- If the center of a radius is not located dimensionally, no center is indicated.

To locate the center of a radius, draw a small cross at the center. Extension lines and dimension lines can be used to locate the center of an arc (Fig. 15.37). If the location of the center is unimportant, the drawing must show clearly that the arc location is controlled by other dimensioned features, such as tangent surfaces (Fig. 15.38). The center of a fillet or round is not located by dimensions.

Sometimes the center of an arc is moved on a drawing because there is a break or the center lies outside the drawing paper (Figs. 15.36 and 15.39). The new position is on a centerline of the arc, and the newly located "false"

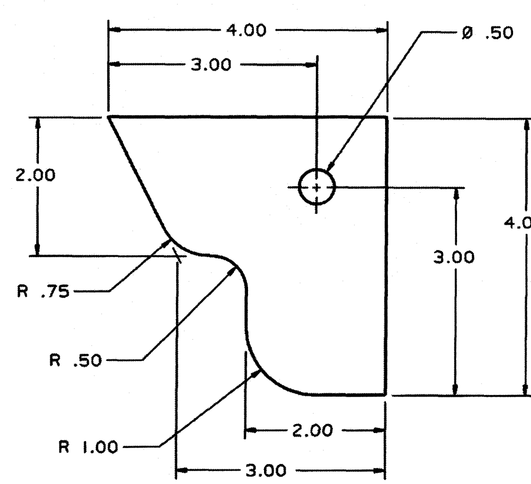


FIGURE 15.38 Radii with Unlocated Centers

Focus On . . .

THE HISTORY OF THE METRIC STANDARD

The metric system is a "standard" system of weights and measures based on the meter, a unit of length, and the kilogram, a unit of mass. But what does "standard" mean?

Noah was told to build his ark 300 cubits long. A cubit was the measured distance from the elbow to the extended finger. Some people have longer arms and fingers than others, so this was an interesting standard unit of length. On average, a cubit is about 18 inches, which means that Noah built an ark about 450 feet long (as big as an ocean liner)!

If you study the history of measurement, you will discover that standards were loosely defined and crudely measured. Many variations were found within a country. Charlemagne, emperor of the Holy Roman Empire from 800–814, used his foot as the standard length measurement. In Europe, the measure used was shorter than the English foot. The Chinese used a measure that was longer. King Henry stated that a yard was the distance from his nose to the outstretched middle finger of his right hand. An inch was the width of three barley corns laying side by side.

It was obvious to many that a "standard" system of measurement was desperately needed. In the 1790s Thomas Jefferson proposed a plan to Congress for the adoption of a standard system. Louis XVI of France tried to persuade the United States and Great Britain to cooperate in setting a standard. Although Great Britain and the United States did not join his effort, many other countries did. The end result of that project was the metric system. The standard was organized when a committee from the French Academy made its report to the National Assembly. It was adopted into French law and, even though other countries adopted it, use of the system spread slowly.

Originally, the meter was one ten-millionth part of the distance from the North Pole to the Equator, passing through Paris. Later, they discovered the measurement was slightly short, so they defined it again. This time it was the distance between two marks on a platinum/iridium bar. The bar became known as the International Prototype Meter and was placed in the Bureau of Archives in Paris. A meter was later redefined as 1,650,763.73 wavelengths of the orange-red line of krypton 86. The International Bureau of Weights and Measures was formed in 1875 (Paris). The copies of the standards owned by the United States are housed in the National Institute of Standards and Technology, NIST, in Washington D.C. (formerly the National Bureau of Standards).

The metric system has been universally accepted—except in the United States and parts of the British Commonwealth. Its use was legalized in the United States in 1866. In 1975, the U.S. Congress passed a bill allowing for voluntary conversion to the metric system. A special board, the U.S. Metric Board, was formed to implement this program.

The use of the metric system in the United States has increased consistently over the years. The automobile industry has been one proponent of this conversion. With cars assembled from parts manufactured all over the world, it seems very reasonable to agree on one standard measurement. How-

center leads to a *staggered dimension.* The portion of the dimension line touching the arc is a radial line drawn from the true center, whereas the staggered dimension is drawn parallel to the first radial line. When the radius dimension line is foreshortened and the center is located by coordinate dimensions, the dimension line locating the center is foreshortened as well.

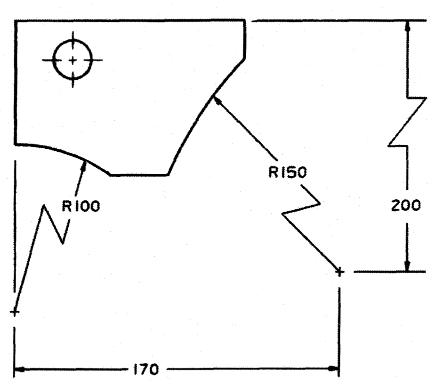


FIGURE 15.39 Foreshortened Radii Dimensions

When a radius is dimensioned in a view that does not show the true shape of the radius, **TRUE R** is added before the radius dimension. A true-shape view of the radius is shown and dimensioned whenever possible.

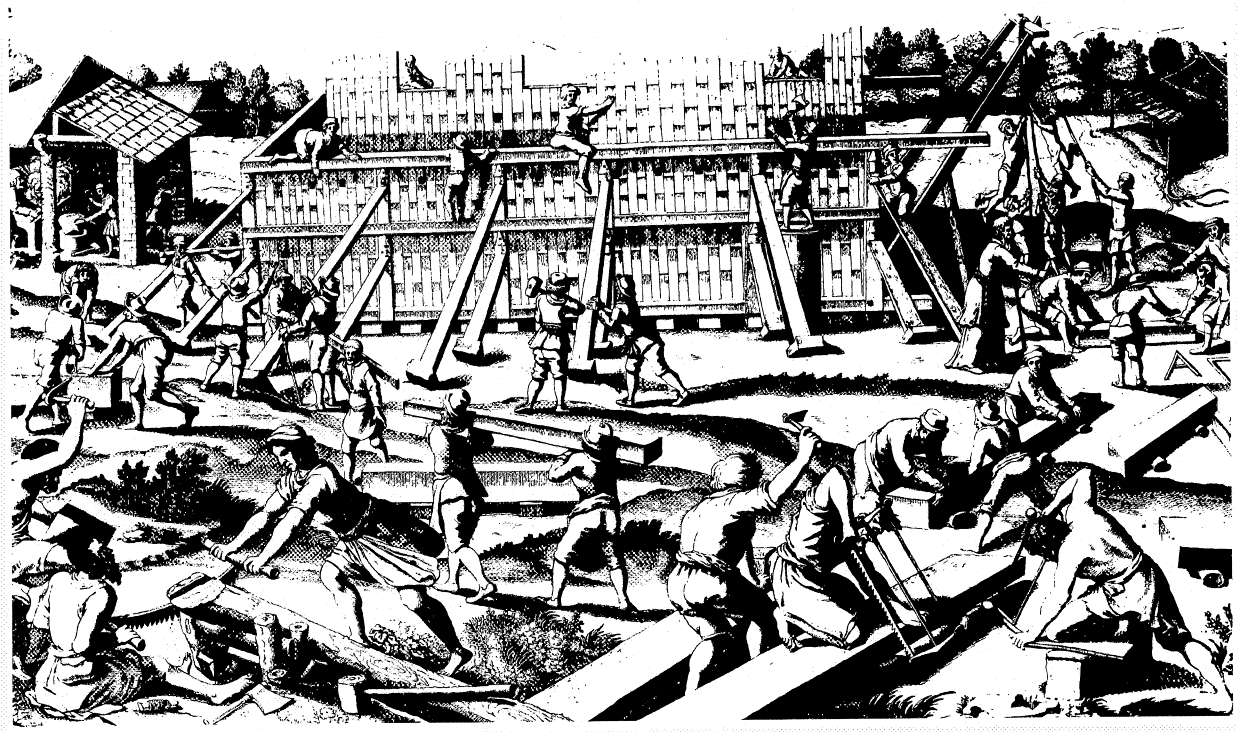
Instead of dimensioning each radius when a part has a number of radii of the same dimension, a note such as the following may be used:

ALL RADII .75 UNLESS OTHERWISE NOTED

A **spherical surface** for a solid part is dimensioned with a radius dimension preceded by the symbol **SR** (Fig. 15.40).

15.5.2 Detailing Chords, Arcs, and Rounded Ends

An angle measurement is the most common way to dimension arcs and chords (Fig. 15.41). The arc dimension with the arc symbol and the chord dimension are used in

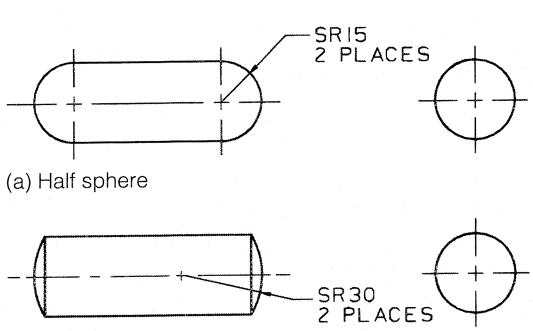


The cubit, an ancient unit of measure.

ever, there is an investment in the "English" system in the United States. The "English" system is now called the U.S. Customary System. New computer numerically controlled (CNC) machines can be used to cut a millimeter or an inch because of the way their motors are controlled. It seems reasonable to expect metric units to replace U.S. customary units in the United States and become the universal "standard." When is another question.

applications such as nipple placement on large pressure vessels in piping design.

Overall dimensions are required for parts having rounded ends (Fig. 15.42). For the fully rounded ends of Figure 15.42(a), the radii are indicated but not dimensioned. For parts with partially rounded ends, the radii are dimensioned [Fig. 15.42(b)]. If corners are rounded, dimensions define the edges, and the arcs are tangent to the edge lines (Fig. 15.43). Radii dimensions use leaders to point to the arc. *The*



(a) Half sphere

(b) Partial sphere

FIGURE 15.40 Spherical Radius Dimensions

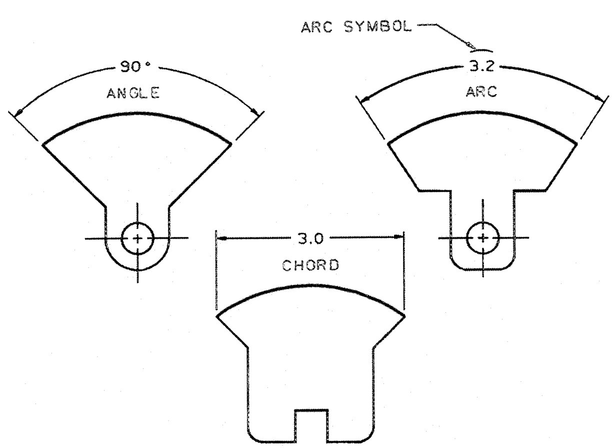


FIGURE 15.41 Dimensioning Angles, Arcs, and Chords

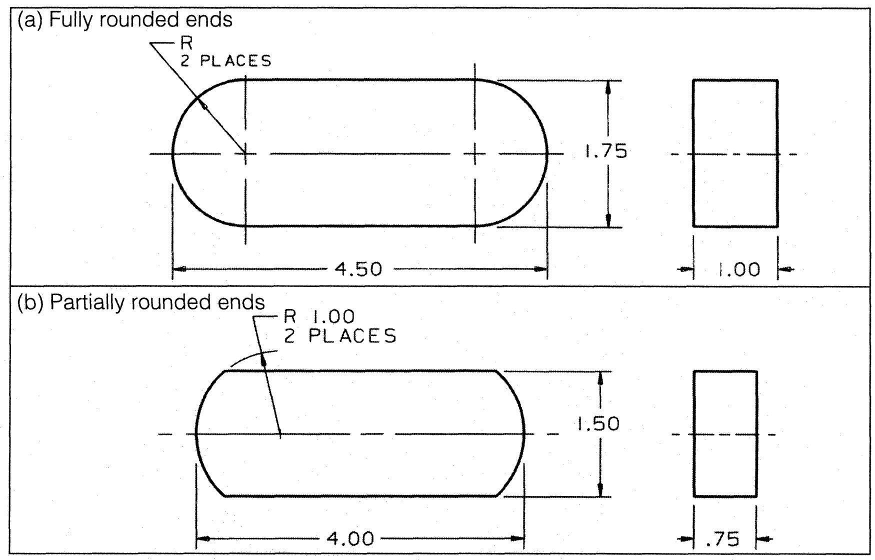


FIGURE 15.42 **Dimensioning Rounded Ends**

leader "aims" at the center point of the arc. It is acceptable to cross one extension or one dimension line.

A curved outline composed of two or more arcs is dimensioned by giving the radii of all arcs and locating the necessary centers with coordinate dimensions (Fig. 15.44). Other radii are located on the basis of their points of tangency.

Regardless of the arc requirements, the dimensions for a part are shown in a view that best displays the features. In Figure 15.45, the dimensions for the part's angles and radii are dimensioned in the only view that shows them accurately; the true shape is shown in the front view.

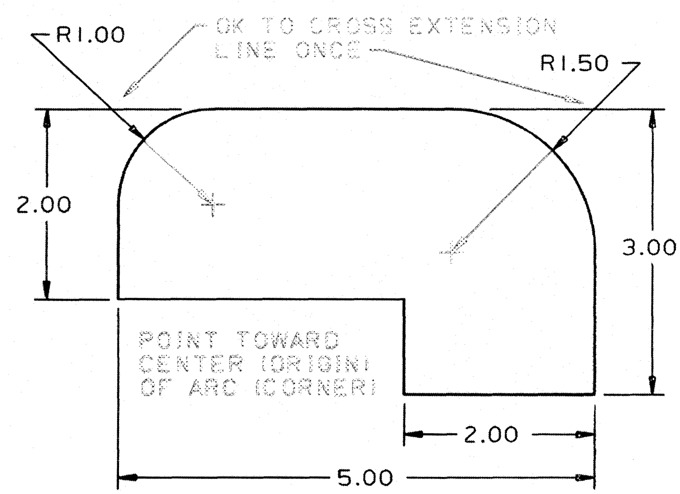


FIGURE 15.43 **Dimensioning Rounded Corners**

15.5.3 Irregular Outlines

Irregular outlines are dimensioned in Figures 15.46 and 15.47. Circular or noncircular objects are dimensioned through rectangular coordinates or an offset method. Coordinates are dimensioned from base or datum lines. If many coordinates are required to define an outline, the vertical and horizontal coordinate dimensions can be given in a table.

15.5.4 Symmetrical Outlines

When only part of a symmetrical outline can be conveniently shown, the **symmetrical outline** is dimensioned on one side of its *centerline of symmetry* (Fig. 15.47). Symmetry is indicated by applying symbols for part symmetry to the centerline. Notice that the dimension in Figure 15.47 uses *dimension lines as extension lines.* This is one of the few situations where this practice is allowed.

15.5.5 Diameter Dimensions

All diameter dimensions are preceded by the international symbol for diameter: a circle drawn the same size as the

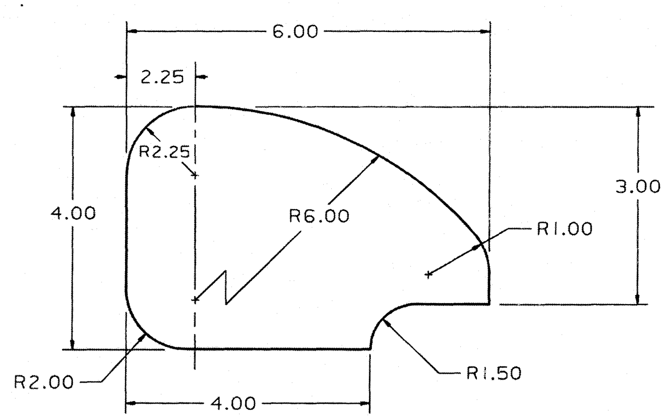


FIGURE 15.44 **Dimensioning Circular Arc Outlines**

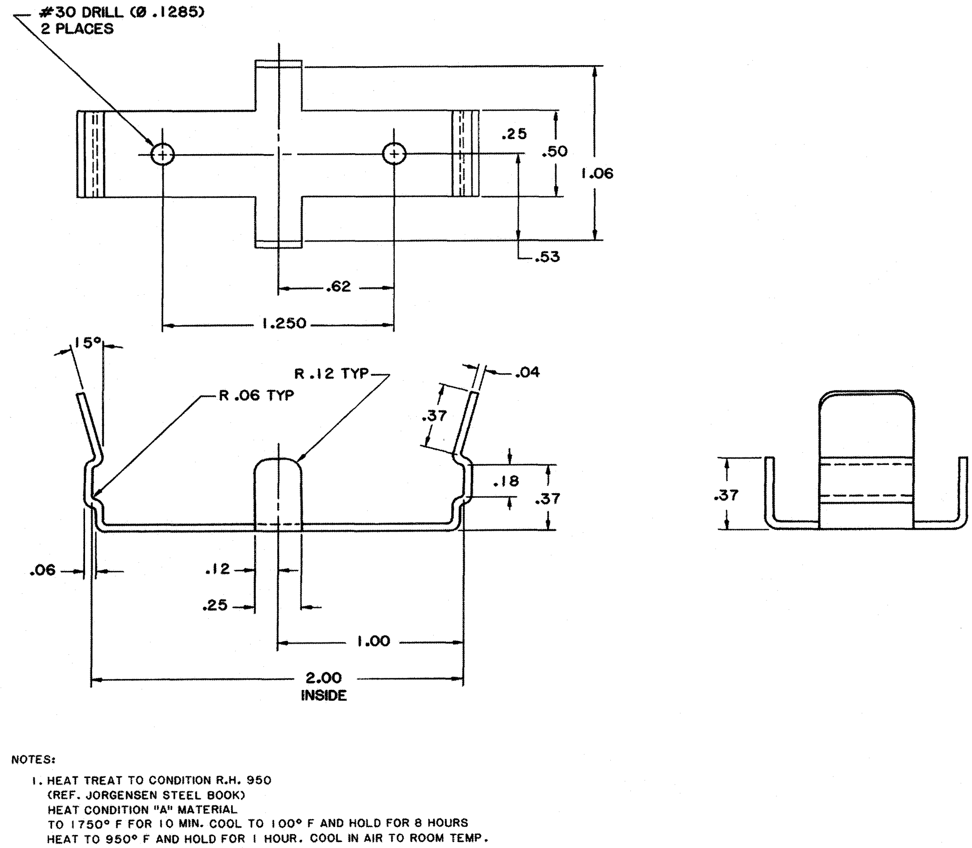


FIGURE 15.45 Detail of CRT Holder

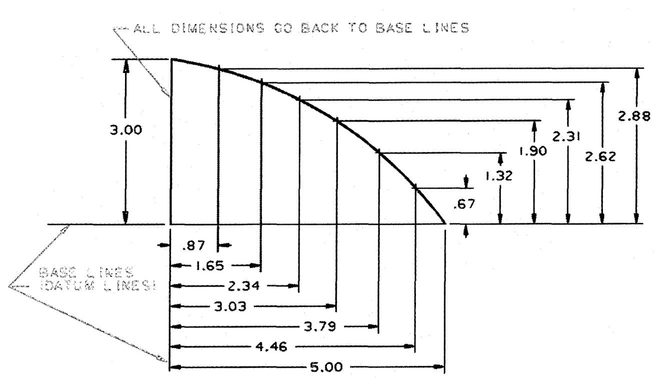


FIGURE 15.46 Coordinate Dimensioning of Curved Outlines

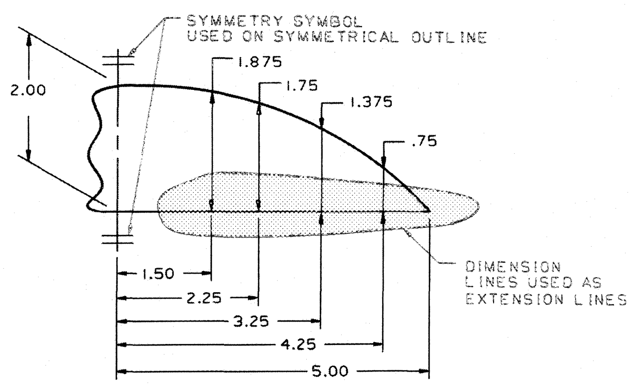


FIGURE 15.47 Dimensioning Symmetrical Outlines Using Dimension Lines as Extension Lines

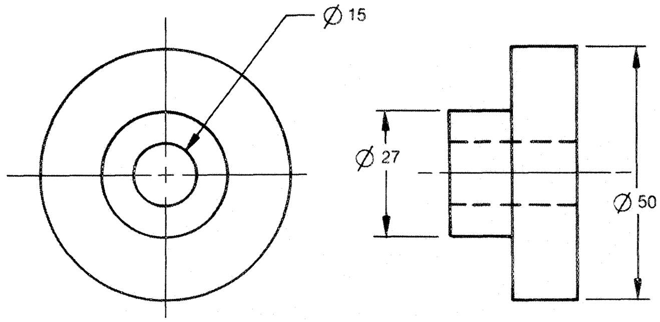
numerals, with a 60° slanted line passing through its center (∅). On some older U.S. standard unit drawings, the size of the diameter is called out with the abbreviation **DIA** after the numerals (e.g., **.375 DIA**). Some of the drawings in this text reflect this older practice.

The diameter symbol precedes all diameter values (Fig. 15.48). When the diameters of a number of concentric cylindrical features are specified, the diameter is dimensioned in a longitudinal view (Fig. 15.48).

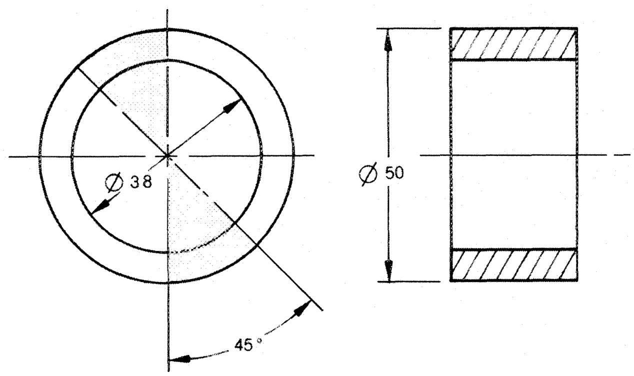
When a hole cannot be adequately called out where it shows as a circle, the hole is dimensioned in a side view or a section view. The depth of the hole, if not included in a note, can be dimensioned in the longitudinal view. The dimensions of a very large hole are shown by drawing the dimension line at an angle through the diameter [Figure 15.48 (lower left)]. For aligned dimensions placed inside the circular form, the area within the section should be avoided when the dimension runs through the diameter.

Holes should be called out with a leader and a note. The leader points toward the center of the circle. *Solid round shapes* are dimensioned on the noncircular view.

Figure 15.49 shows an alternative way to dimension a



(a) Diameter callouts



(b) Diameter dimensions and area to avoid

FIGURE 15.48 **Dimensioning Diameters**

cylindrical part. The first method [Fig. 15.49(a)] shows two views of the part, with standard dimensioning callouts for the hole diameters. The second method, (b), gives only the longitudinal edge view of the part and uses the **DIA** callout for all diameter dimensions.

You May Complete Exercises 15.1 Through 15.4 at This Time

15.5.6 Hole Depths and Diameter Dimensions

Holes dimensions are shown by pointing to the diameter with a leader and giving a note containing size and type. If the depth of the hole is not obvious or not dimensioned, the word **THRU**, implying drill through, follows the size specification.

A **blind hole** does not go through the part. The depth dimension of a blind hole is the depth of the full diameter from the surface of the part. If a blind hole is also counterbored or counterdrilled, the depth dimension is still taken from the outer surface.

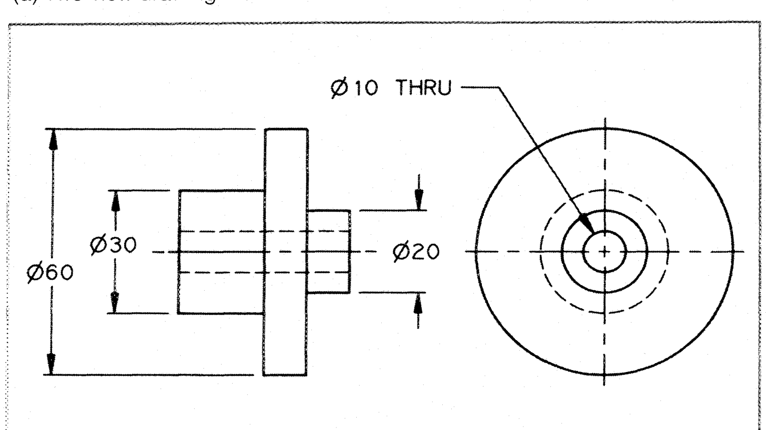
A number of methods are used to call out hole diameter and depth:

Fraction-Inch		Decimal-Inch		Decimal-Inch (symbology)
$\frac{1}{2}$ DIA THRU	or	.50 DIA THRU	or	∅ .50 THRU
$\frac{1}{2}$ DIA	or	.50 DIA		∅ .50
$\frac{3}{4}$ DEEP	or	.750 DEEP	or	↧ .750

15.5.7 Dimensioning Slotted Holes

Figure 15.50 shows three methods for dimensioning slots. In Figure 15.50(a) the slot's centerlines are located between centers. A dimension from the edge of the part or other controlling feature is given as well. The slot width is given as an **R** (radius) pointing to the end of the slot arc. The **R** is accompanied by the note **2 PLACES**. ANSI calls for **R** for a radius; however, many companies still use **RAD**. Figure 15.50(b) shows a leader and a note stating the outside dimensions of the slot (**20 × 60**). An **R** callout is also included. The slot also can be located from the part's edges [Fig. 15.50(c)], or its centerlines can be located from two

(a) Two-view drawing with circular view



(b) One-view drawing without circular view

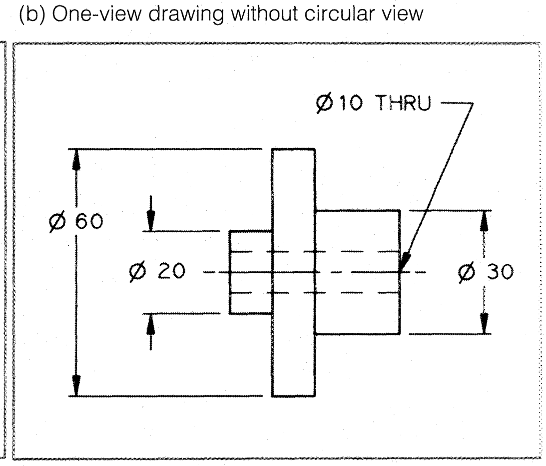
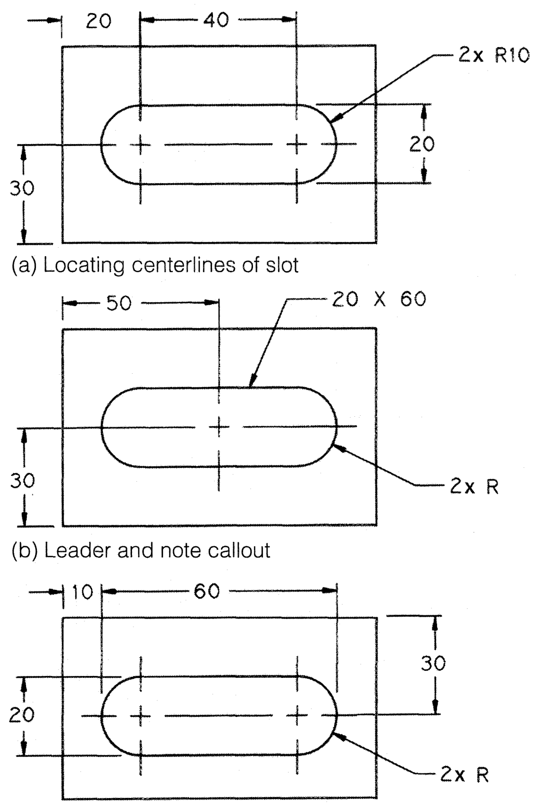


FIGURE 15.49 **Dimensioning Diameters on Drawings Without Circular Views**



(a) Locating centerlines of slot

(b) Leader and note callout

(c) Overall slot dimension

FIGURE 15.50 Dimensioning Slots

controlling edges. Figure 15.50(c) shows the dimensions of the slot on the view; an **R** callout is also given.

The choice of methods for dimensioning slots is determined by the design and the required slot tolerance. If something fits into the slot, accurate tolerance and dimensions are required. The methods in Figure 15.50(a) and (c) are recommended when accuracy is important. The method in Figure 15.50(a) is good for milled slots; the method in Figure 15.50(c) is good for punched forms.

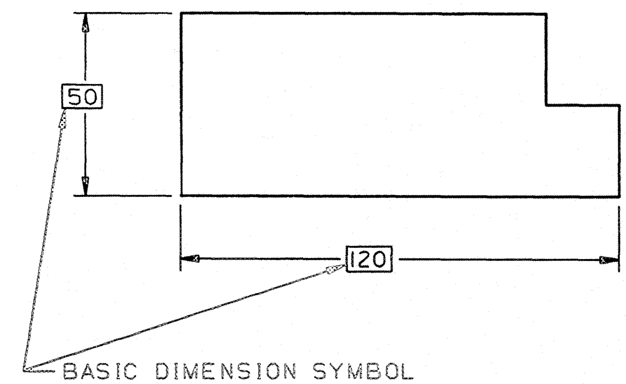


FIGURE 15.52 Basic Dimensioning Symbol

15.6 Dimensioning Features with Symbols

Geometric characteristics and other dimensional requirements can be established via standard **symbols** instead of traditional terms and abbreviations. These symbols must conform to ANSI Y14.2M (for symbols denoting geometric characteristics see Fig. 15.51, and for symbols identifying a basic dimension see Fig. 15.52). *A basic dimension is a numerical value describing the theoretically exact size, profile orientation, or location of a feature or datum target.* Basic dimensions are the basis from which permissible variations are established by tolerances on other dimensions or in notes.

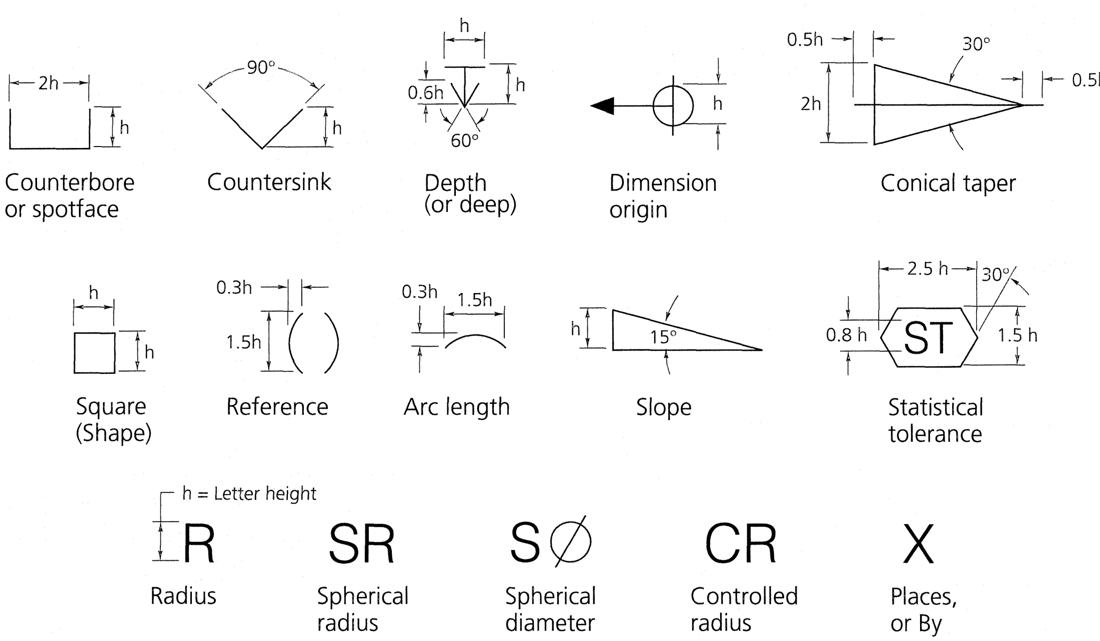


FIGURE 15.51 Form and Proportion of Dimensioning Symbols

Term	Symbol
AT MAXIMUM MATERIAL CONDITION	Ⓜ
AT LEAST MATERIAL CONDITION	Ⓛ
PROJECTED TOLERANCE ZONE	Ⓟ
DIAMETER	∅
SPHERICAL DIAMETER	S∅
RADIUS	R
SPHERICAL RADIUS	SR
REFERENCE	( )
ARC LENGTH	⌒

FIGURE 15.53 **Modifying Symbols for Dimensions**

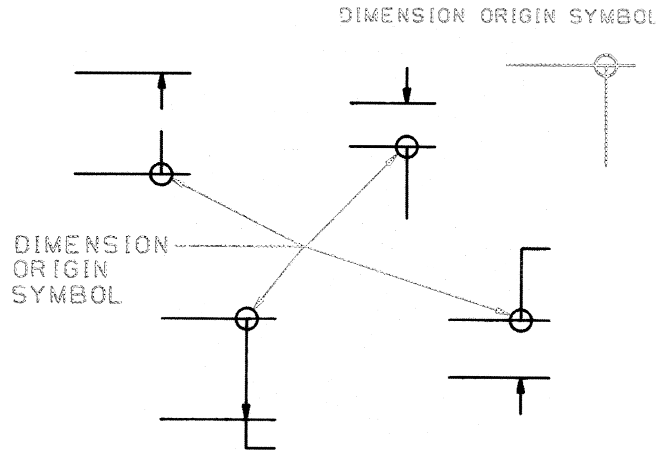


FIGURE 15.55 **Dimension Origin Symbol**

The symbols for indicating diameter, spherical diameter, radius, and spherical radius are shown in Figure 15.53. Symbols precede the value of a dimension or tolerance given as diameter or radius. Reference dimensions (or reference data) are enclosed within parentheses. Symbology designates a variety of geometric features and dimensions, including the following.

- The symbol to indicate a linear dimension is an arc length measured on a curved outline (Fig. 15.53). The symbol is placed above the dimension (Fig. 15.41).
- To indicate a single dimension applied to a square shape, the "square" symbol precedes the dimension (Fig. 15.54).
- The symbol to indicate a toleranced dimension between two features originates from one of these features (Fig. 15.55).
- The depth of a hole (Fig. 15.56), a counterbore (Fig. 15.57), a spotface (Fig. 15.58), a countersink (Fig. 15.59), or a counterdrill (Fig. 15.59) can be given symbolically. To indicate where a dimension applies to the depth of a feature, the depth symbol precedes the dimension.

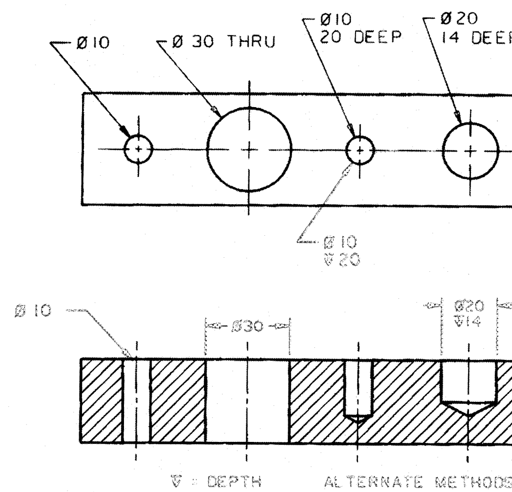
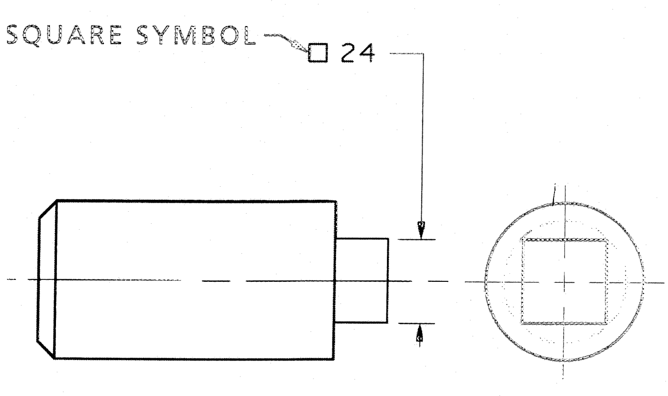


FIGURE 15.56 **Using Symbols to Dimension Holes**



IGURE 15.54 **Square Dimensioning Symbol**

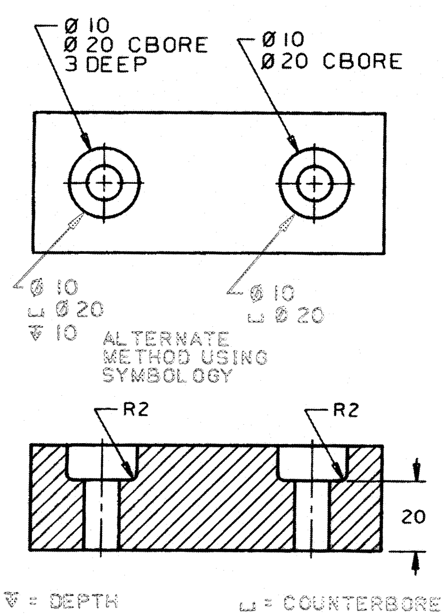


FIGURE 15.57 **Dimensioning Counterbores**

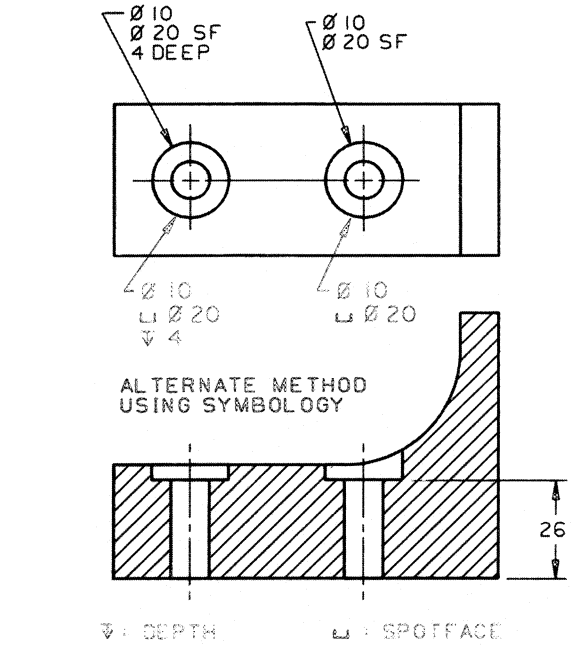


FIGURE 15.58 Dimensioning Spotfaces

15.6.1 Countersunk and Counterdrilled Holes

Countersinking (**CSK**) ensures that flathead screws are flush with the surface of the part (Fig. 15.59). The diameter and included angle of the countersink are specified. The flathead screw requires a conical seat, usually specified by the included angle and the diameter at the large end.

For **counterdrilled holes** (**CDRILL**), the diameter and depth of the counterdrill are given. Specifying the included angle of the counterdrill is optional (Fig. 15.59, lower right). The depth dimension is the depth of the full diameter of the counterdrill measured from the outer surface of the part. Symbology can also be used on these features. A counterdrilled hole differs from a counterbored hole in that the bottom of the counterdrilled hole is conical. Counterdrilled holes are created with a step drill or with two drills of different diameters.

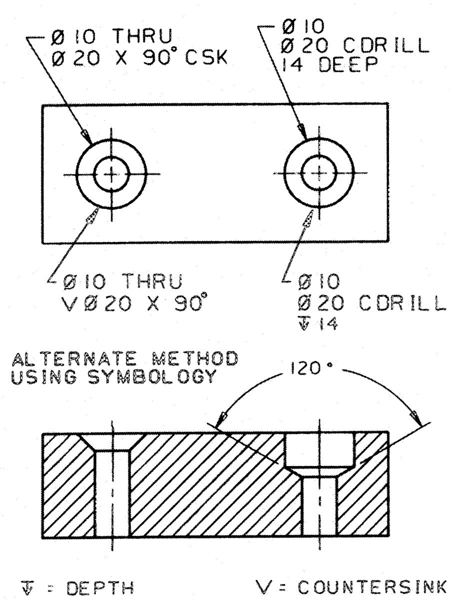


FIGURE 15.59 Dimensioning Countersinks and Counterdrills

15.6.2 Spotfaced Holes

A **spotface** (**SF**) (Fig. 15.58) is a method of cleaning up and squaring a rough surface, such as on a cast metal part. Material is removed so a screw head will seat flush against the surface. Its depth is usually not dimensioned.

The diameter of the spotfaced area is specified by the diameter symbol and a value. When a depth is required, either the depth or the remaining thickness of material may be specified. A spotface sometimes is specified by a note. If no depth or remaining thickness of material is specified, the spotfacing is the minimum depth necessary to clean up the surface. Figure 15.58 shows both methods for calling out the spotface.

15.6.3 Counterbored Holes

Counterbored holes are used extensively for socket-head screws so that the head of the screw is flush with or below the surface of the part. A **counterbore** (**CBORE**) is an enlarged hole, piloted from a smaller hole to maintain concentricity. Counterbored holes are machined to a square seat at a specified depth (Fig. 15.57). The depth is called out within the hole note as the distance from the upper surface (beginning surface) to the bottom of the counterbore. Either the symbol for the counterbore or the note **CBORE** is used. The depth symbol or the note **DEEP** can also be used. When the thickness of the remaining material has significance, it, rather than the depth, is dimensioned (Fig. 15.57). Figure 15.60 shows a simple mechanical part dimensioned with symbols.

You May Complete Exercises 15.5 Through 15.8 at This Time

15.7 DIMENSIONING SPECIAL FEATURES

Chamfers, threads, centerdrills, tapers, knurling, keyways, and other *geometric features* require specific, standardized dimensioning. These dimensions are based on the method used to machine them or on a standard purchased part mated with the feature.

15.7.1 Threads

Thread callouts are found in almost every mechanical drawing. Figure 15.4 shows a part designed with metric units. The callout **M 42 × 1.5 - 6g** specifies a metric thread.

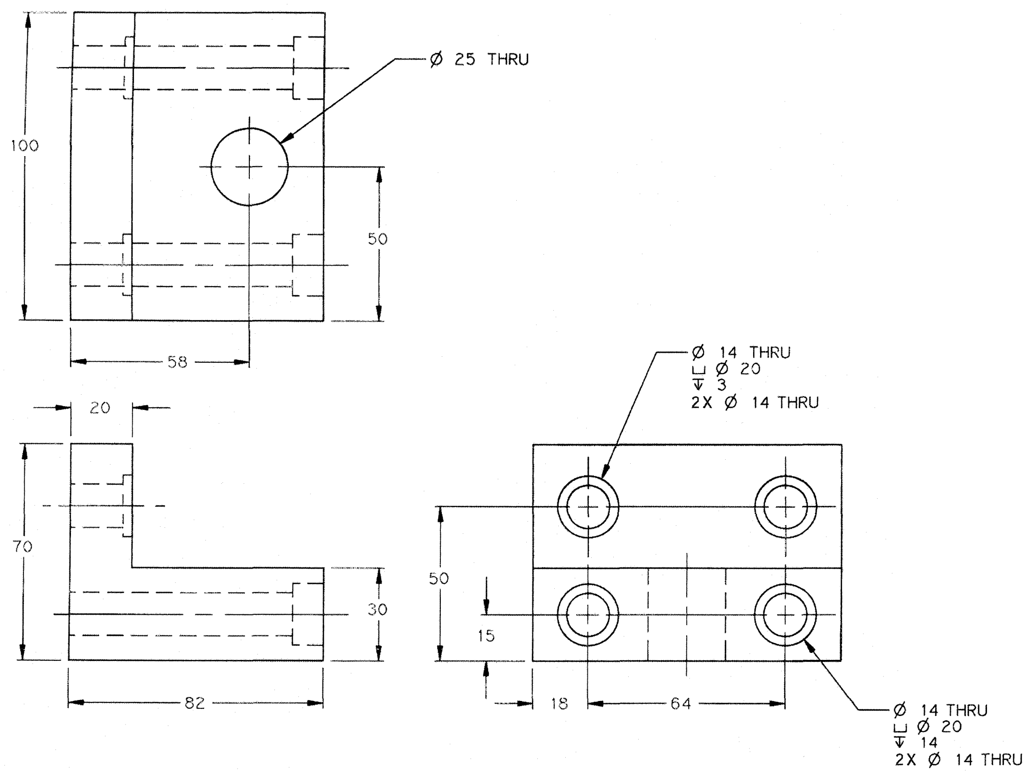


FIGURE 15.60 Part Dimensioned Using Dimensioning Symbols

Figure 15.3 shows an example of a unified thread callout on a decimal-inch drawing (**.250-20 UNC**).

Nonmetric threads are classified according to the number of threads applied to a specific diameter. Unified (**UN**) thread is the standard type of thread for the United States. To specify screw threads, the nominal major diameter is given first, followed by the number of threads per inch and the series designation. Finally, the class of fit between male and female threads is given, followed by an **A** for male threads or a **B** for female threads. For tapped holes, the complete note contains the tap drill diameter and the depth of the hole, followed by the thread specification and the length of the tapped threads. All threads are assumed to be right hand unless left hand is specified by **LH** following the class. A few examples of screw thread notations follow.

Decimal-Inch

- **.190-32 UNF-2A** or **#10-32 UNF-2A**
- **.250-20 UNC-2B** or **$\frac{1}{4}$-20 UNC-2B**
- **2.000-16 UN-2A**
- **2.500-10 UNS-2B**

Metric

- **M6 × 1-4h6h**
- **M16 × L4 -P2-4h6h**

The thread type and size are given on the drawing, and the machinist chooses the correct drill diameter. Specifying the drill and tapping requirements requires the diameter and the depth. In this case, the tap drill size, its depth, the thread specification, and the depth of threads are provided, as in the examples that follow.

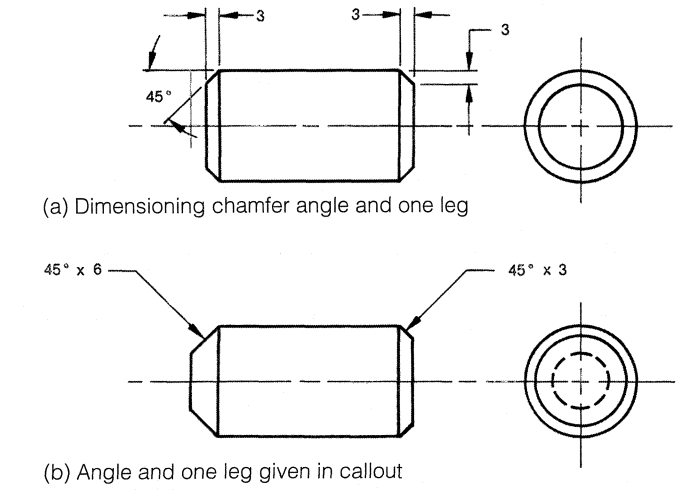
∅.312 1.25 DEEP
.375-16 UNC-2B, .88 DEEP or **$\frac{3}{8}$-16 UNC-2B**
3 HOLES

∅ .422 1.25 DEEP
.500-13 UNC-2B LH, 1.12 DEEP or **$\frac{1}{2}$-13 UNC-2B LH**
2 HOLES

More information on methods of specifying and dimensioning screw threads is found in ANSI Y14.6.

15.7.2 Chamfers

Manual and automated assembly techniques both benefit from tapered features to help the parts engage. **Chamfers** are specified by dimensions or notes. It is not necessary to use the word **CHAMFER** when the meaning is obvious. If the chamfer is other than 45°, dimensions show the direction of the slope. Figure 15.61 shows methods for dimensioning external chamfers. You can show chamfer dimensions by the chamfer angle and one leg, by dimensioning both legs, or by pointing to the chamfer and giving the angle and one leg as a callout. Internal dimensions for chamfers (Fig. 15.62) are dimensioned by the included angle and the largest diameter. The metric method of dimensioning chamfers is also shown on this figure. For inch-unit drawings, the angle is sometimes given second and the leg first, for example, **.25 × 45°**.



(a) Dimensioning chamfer angle and one leg

(b) Angle and one leg given in callout

FIGURE 15.61 Dimensioning Chamfers

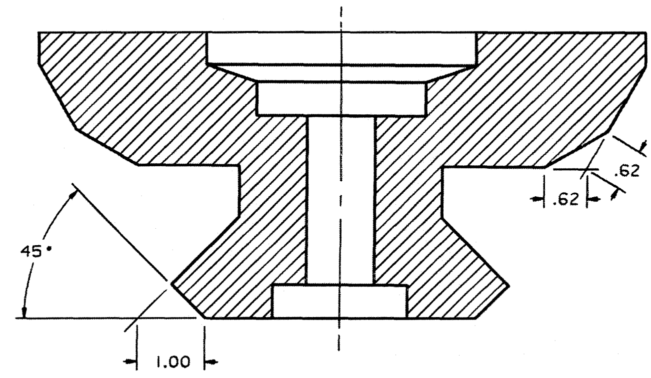


FIGURE 15.63 Dimensioning Chamfers Between Surfaces Not at 90°

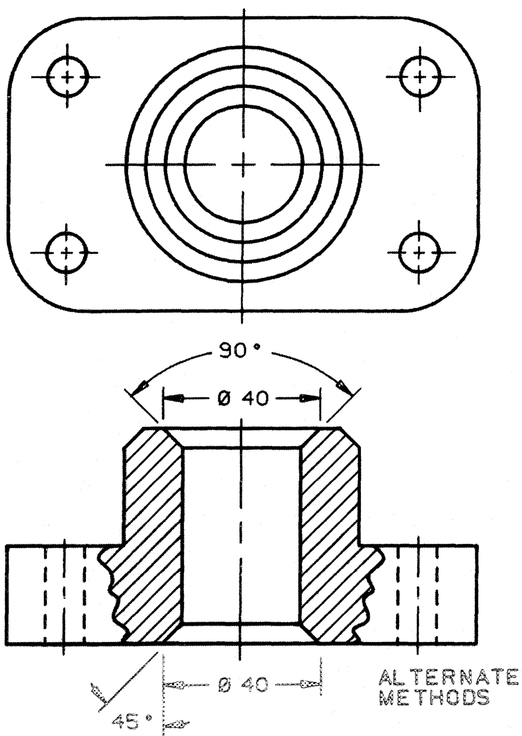


FIGURE 15.62 Internal Chamfers

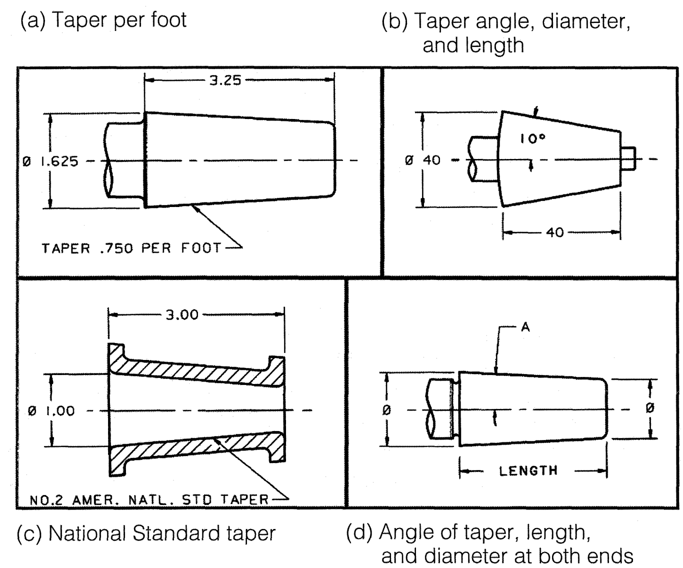


FIGURE 15.64 Dimensioning Tapers

This method is being replaced by ANSI Y14.5M. If chamfers are required for surfaces intersecting at other than right angles, the methods shown in Figure 15.63 apply.

15.7.3 Taper Dimensioning

Tapers are used on machines to align and hold machined parts that require simple and speedy assembly and disassembly. A round taper has a uniform increase in the diameter on a round part for a given length measured parallel to the axis of the workpiece (conical). Internal or external tapers are noted by taper per foot (TPF), taper per inch (TPI), or degrees. TPF or TPI refers to the difference in diameters within 1 foot or 1 inch [(Fig. 15.64(a)]. The difference is measured in inches. The *angles of taper* refer to the inclined angles with the part's centerline (axis) [Figure 15.64(b)].

In Figure 15.64(a), the taper per foot, the length of the part, and the large diameter are given. In Figure 15.64(b), the diameter, the length, and the angle are given. In Figure 15.64(c), the length, the diameter, and the note **NO. 2 AMER. NATL. STD TAPER** are given. In the last example [Fig. 15.64(d)], the two diameters, the length, and the angle are given.

In Figure 15.65(a), the internal taper is designated through a note (**TAPER 1.75:12 ON DIA FIT TO GAGE**), the gage diameter (**1.00 GAGE**), locating dimension (**1.50**), angle (**4°10′**), and the part length (**3.50**). Figure 15.65(b) uses the same method on an external taper. Figure 15.66 expresses the taper ratio (**10:1**) and gives the gage location and gage diameter.

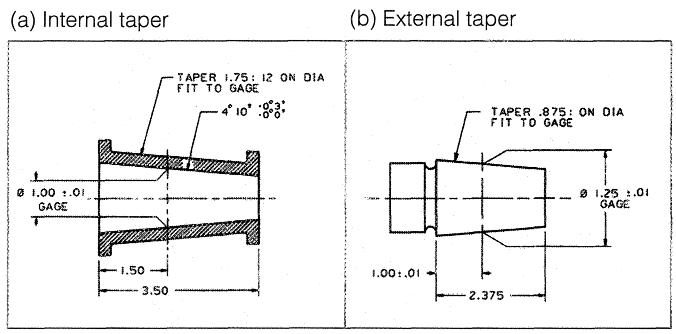


FIGURE 15.65 Taper Dimensioning

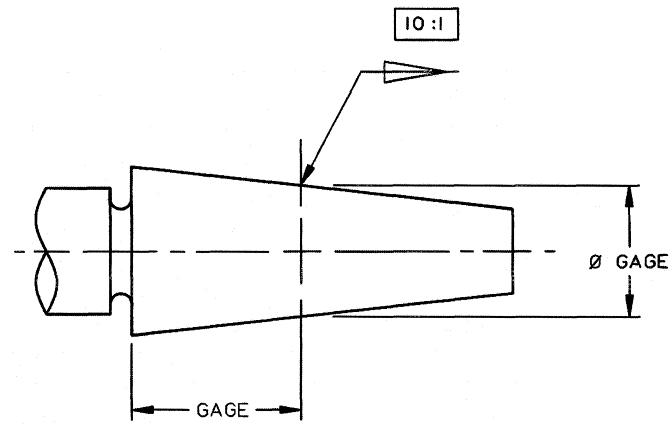


FIGURE 15.66 Dimensioning a Taper with a Ratio

15.7.4 Center Drill Dimensioning

When a part is held and turned between the centers of a lathe, a *center hole* is required on each end of the cylindrical workpiece. The center hole has a 60° angle that conforms to the center and a smaller drilled hole to clear the center's point. The center hole is made with a combination drill and countersink called a **center drill** (Fig. 15.67). In this figure, the A dimension is the workpiece diameter, the B dimension is the body diameter of the center drill, the C dimension is for the diameter of the drilled center hole, and the D dimension is the depth. The drill tip is drawn at 120°.

15.7.5 Keys and Keyseats

A **key** as a demountable machinery part. When assembled into keyseats, a key provides a positive means for transmitting torque between a shaft and a hub. A **keyseat** is an axially located rectangular groove in a shaft or hub. Keyseats are dimensioned by width, depth, location, and, if required, length (Fig. 15.68). The depth is dimensioned from the opposite side of the shaft or hole.

15.7.6 Knurling

A **knurl** is a machined rough geometrical surface on a round metal part. Knurling improves the grip or helps press-fit the knurled part into a hole in a mating part. Knurling is also done for appearance.

Knurling is specified in terms of type, pitch, and diameter before and after knurling. When diameter control is not required, the diameter after knurling is omitted. If only a portion of a feature is to be knurled, axial dimensioning is necessary. Knurling can be either diamond patterned or straight patterned and fine, medium, or coarse. Knurling is specified by a note that includes the type of knurl required,

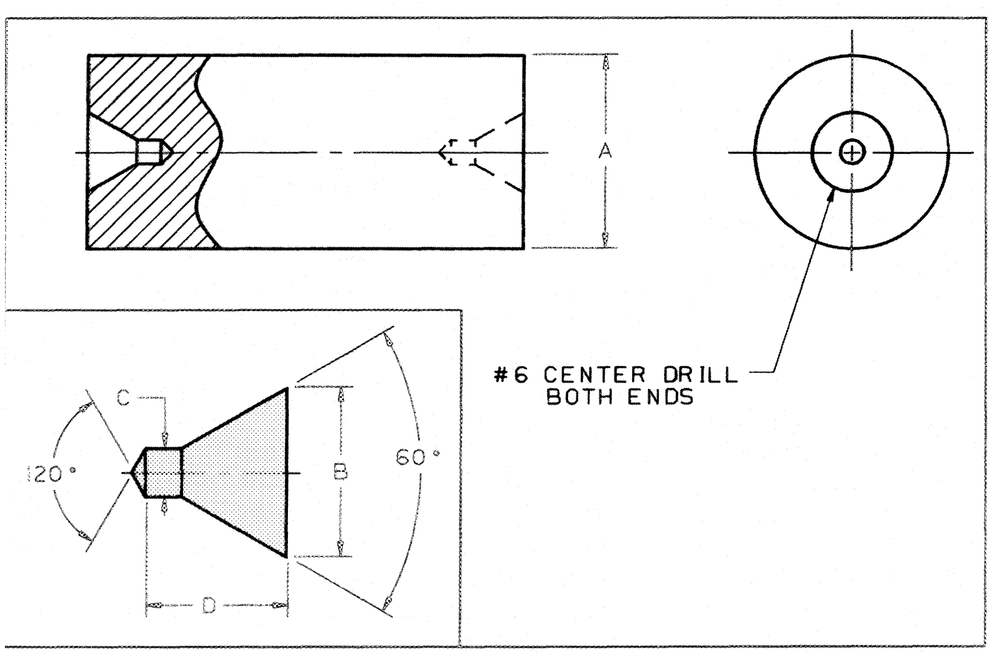


FIGURE 15.67 Center Holes and Center Drills

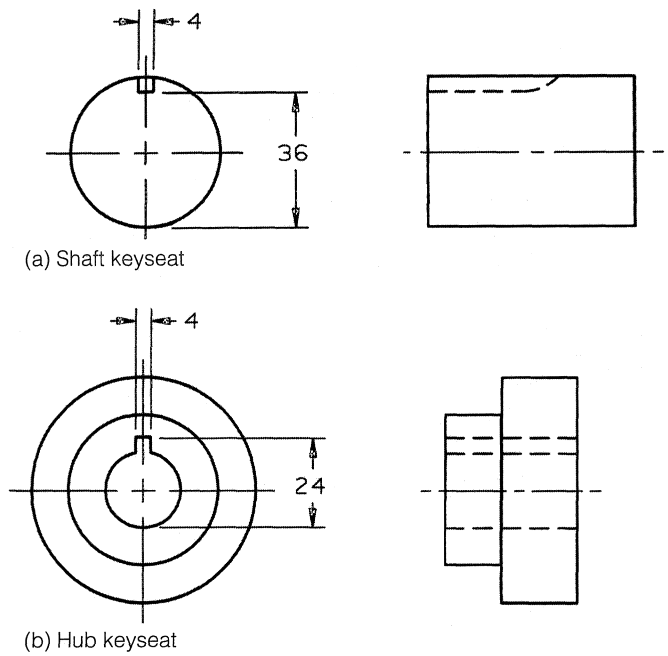


FIGURE 15.68 Dimensioning Keyseats

the pitch, the toleranced diameter of the feature prior to knurling, and the minimum acceptable diameter after knurling (Fig. 15.69).

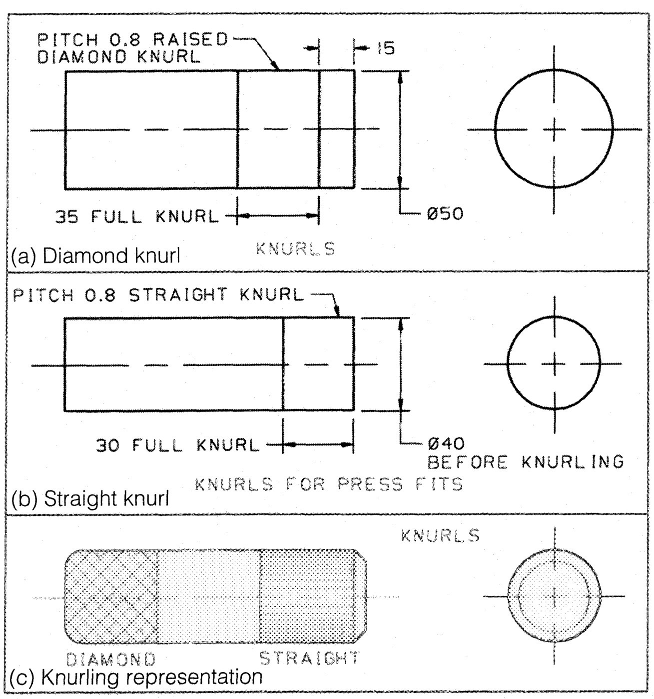


FIGURE 15.69 Dimensioning Knurling

15.8 LOCATING FEATURES ON A DRAWING

The location of holes, slots, and machined features on a part is very important. During dimensioning, *consider how the part is to be machined.* Understanding the detailing and dimensioning process begins with analyzing the part geometrically.

If a CAD system was used for the design of the part, the engineer need only call up the 3D design and place it in the views necessary to describe the part with dimensions and notes.

15.8.1 Geometric Analysis of a Part

Figure 15.70 shows simple geometric shapes and the dimensions required to describe the shapes. The machinist should have all necessary dimensions. *A drawing is never "scaled" to find a location or size that should have been described by a dimension.*

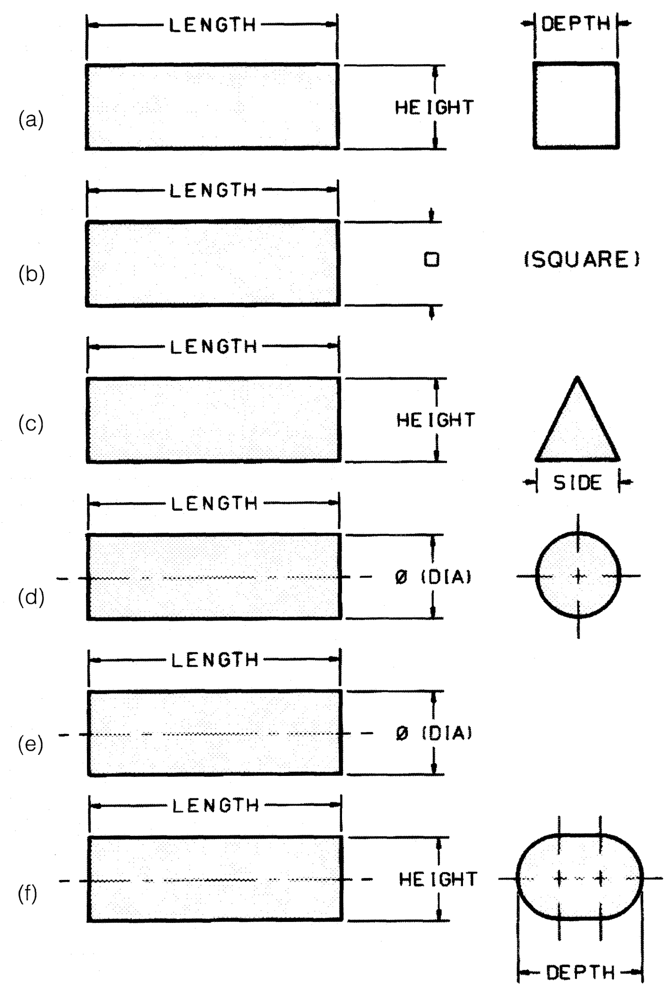


FIGURE 15.70 Dimensions for Common Geometric Shapes

In Figure 15.71, a simple clamp is shown in three views with appropriate dimensioning. It is composed entirely of rectangular prisms. Each of these prisms must be sufficiently dimensioned so a machinist can make the part. *The three most important dimensions are also required: height, width, and depth.* The 20 × 20 cutout in the top view, in Figure 15.71, is an example of a negative area. When a mating part must fit into the area, the negative area must be dimensioned.

The part in Figure 15.72 is composed entirely of cylindrical shapes. In fact, only the ends are flat planar surfaces. Notes that the holes are removed negative cylinders.

Size dimensions for a part are given to establish the shape itself. The diameter of the cylinder is a size dimension. **Location dimensions** position a geometric shape in space. In Figure 15.73, the .34 dimension establishing the location of the hole in the front view is an example of a location dimension. The location of a shape and the shape's size are

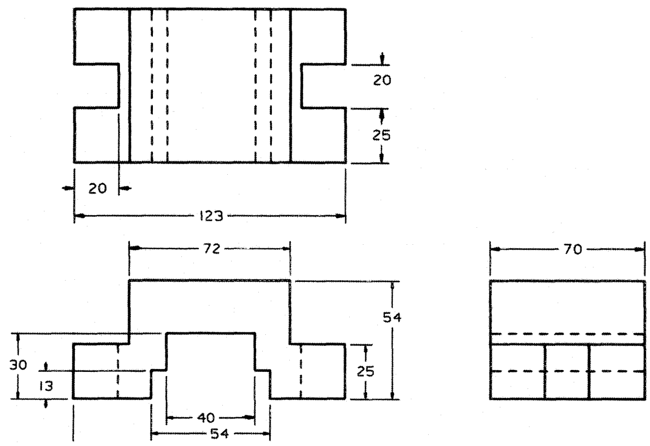


FIGURE 15.71 Dimensioned Clamp

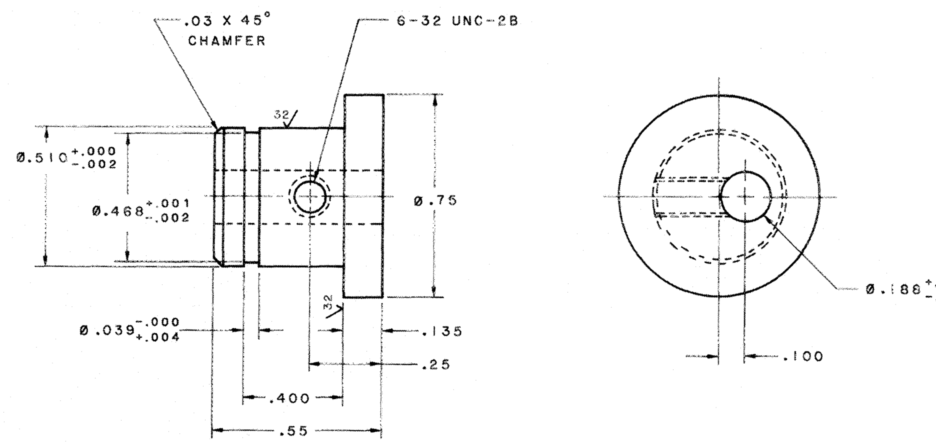


FIGURE 15.72 Cap Detail

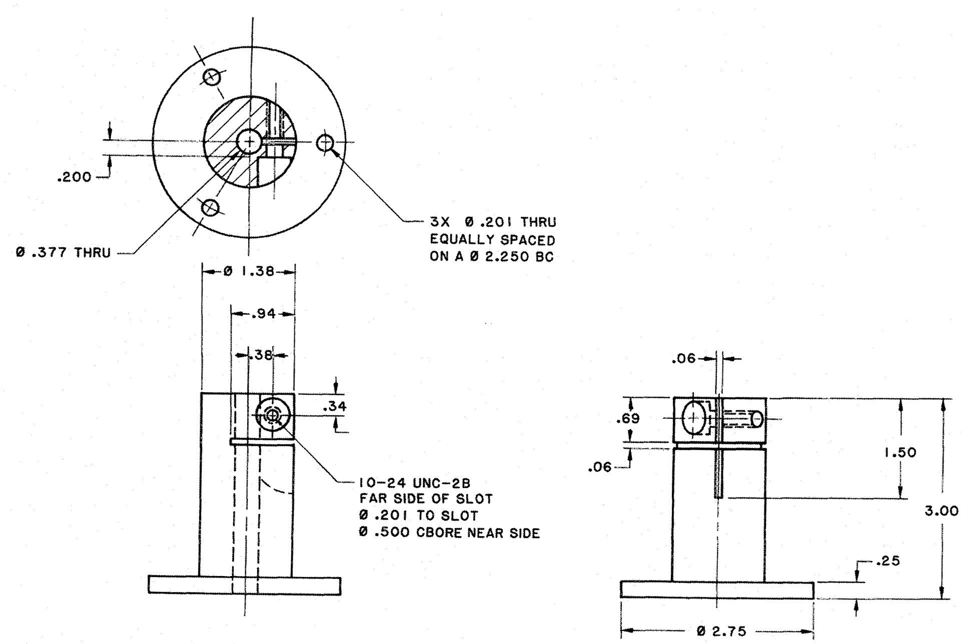


FIGURE 15.73 Pedestal Detail

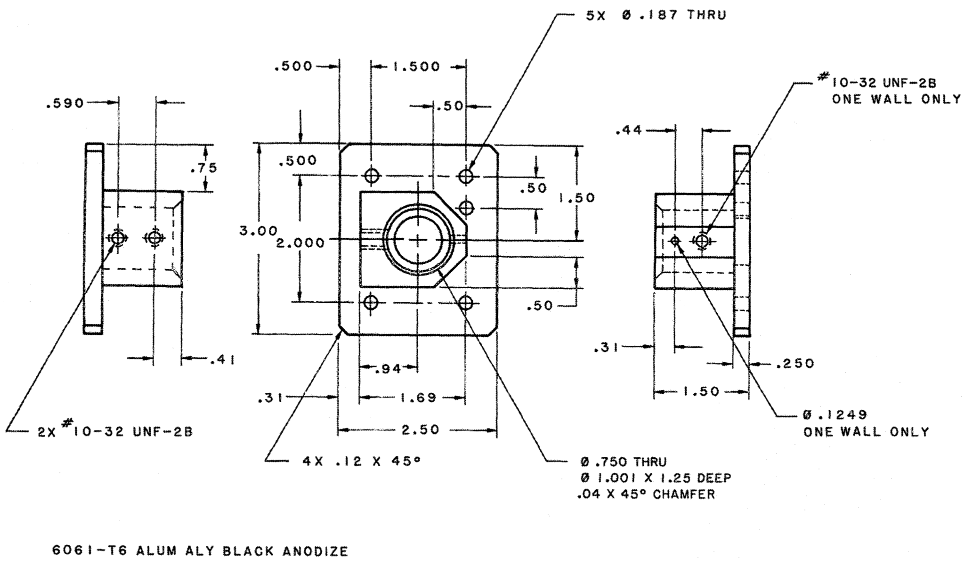


FIGURE 15.74 Base Mounting Detail

equally important. Size and location dimensions must be complete to avoid misunderstandings. On the other hand, it is important not to overdimension, that is, to give two dimensions that locate the same feature. The details of locating holes about a center and locating holes on a part are given in section 15.8.5. The details covering datum dimensions are given in section 15.10.1.

15.8.2 Mating Parts and Dimensions

In Figure 15.74, the part has features that obviously relate to a mating piece. The base of the part attaches to a mating part using the .187 clearance holes. A .750 diameter chamfered hole runs through the part. A shaft or other cylindrical item is to be inserted here during assembly and will be held in place by one or more screws entering the side of the part. In most cases, the part to be detailed will be accompanied by a description and/or an illustration of the assembly. *From the assembly drawing, the use, location, orientation in space, and mating pieces are readily identified.*

The assembly of the slide plate in Figure 15.75(a) was designed on a 3D CAD system. Each piece is shown separately in Figure 15.75(b), (c), and (d). An assembly is provided in Figure 15.75(e). Because the slide must fit into the slot in the base, the slide and base have **related dimensions.** The slide's bottom portion is mated with the top of the base. The mating dimension for the slide is slightly smaller than that for the base because it must fit inside the slot. The designer establishes the *clearance fit.*

When dimensioning parts that must mate with other parts in an assembly, the related surfaces on each part should be dimensioned. Mating dimensions are also necessary to establish hole patterns used to secure one part to another.

15.8.3 Finish Marks and Machined Surfaces

Rough stock shapes, castings, and forgings have rough and unmachined surface textures. **Machined surfaces** are established on the drawing through finish marks. A **finish mark** is a symbol that tells the machinist the machining requirements for a surface. Machined surfaces must be established from a rough surface, in any direction (top, front, side). Many features on a casting or forging must be machined, because these processes produce a part with every required geometric shape, but the surfaces are rough. All other machined surfaces or holes are established from that first machined surface. The symbol can be the traditional finish mark [Figure 15.76(a)], the general symbol [Figure 15.76(b)], or the ANSI-recognized **basic surface texture symbol** [Figure 15.76(c)].

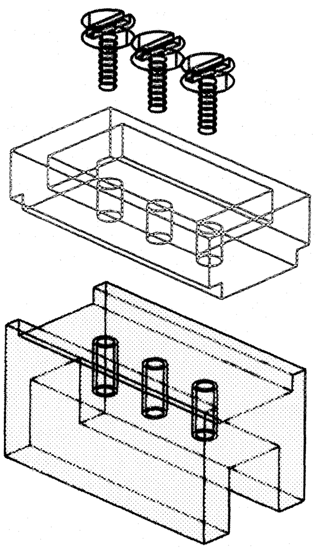
The placements and measurements for constructing the three types of symbols are shown in Figure 15.76. Symbol templates are available for quick, easy, and accurate insertion of symbols on a drawing.

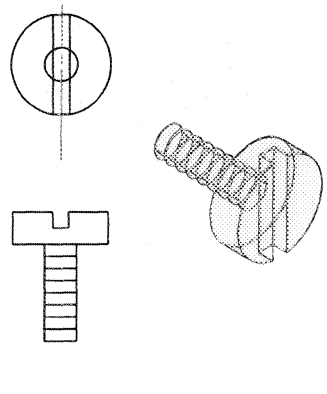
The general symbol establishes the surface to be machined without providing any details as to the quality or type of surface. The basic surface texture symbol, on the other hand, establishes a surface to be machined or how it can be altered to provide specifications for the lay, roughness, and waviness of a surface. This symbol is used whenever there is a need to control the surface irregularities of a part (Fig. 15.77).

You May Complete Exercises 15.9 Through 15.12 at This Time

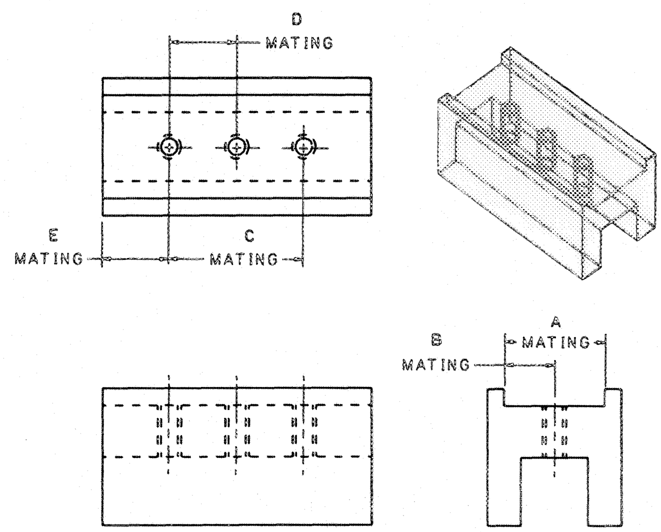
FIGURE 15.75 Mechanical Assembly

(a) Exploded view of assembly

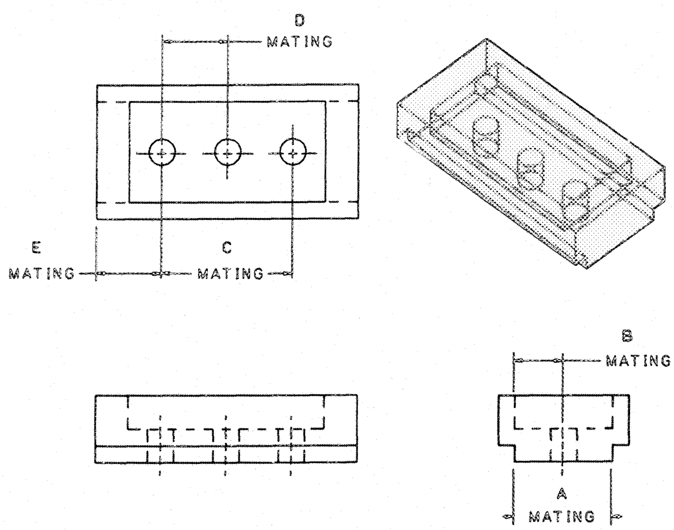




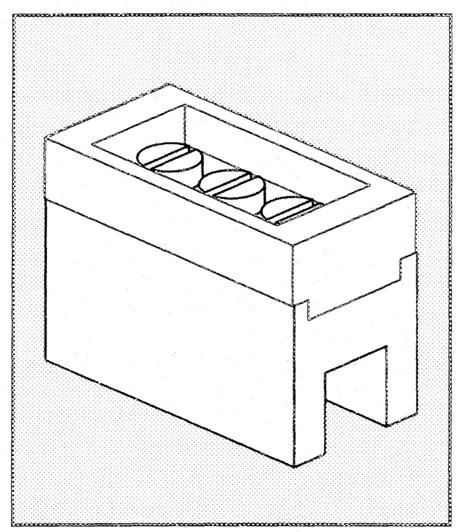
(b) Bolt for assembly



(c) Base detail of assembly showing mating dimensions



(d) Slide plate of assembly showing mating dimensions



(e) Assembly created by inserting parts together

15.8.4 Locating Holes and Features on a View

Because machined surfaces are used to establish machined features such as holes and slots, it is important to locate them by dimensions from prominent features or surfaces (Fig. 15.78). The bracket arm is a cast part and has a number of machined surfaces and holes. Because the central hole is obviously the most important, all machined holes and slots are located from it. In the front view, the bottom surface was used to locate each of the height dimensions.

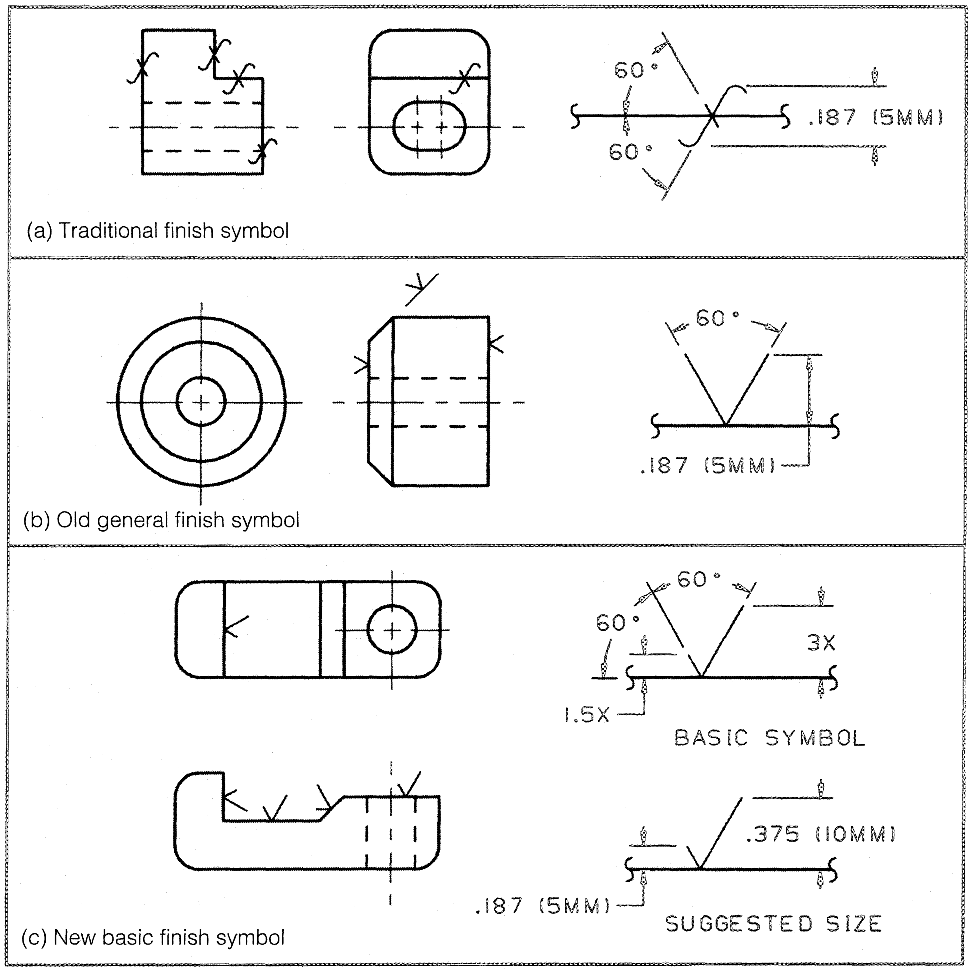
A general rule for dimensioning is to dimension from a rough surface to a finished surface once in each direction and between rough surfaces for all other nonmachined surfaces in each direction. Dimensions are given between all other machined surfaces and the first finished surface in each direction. Features such as holes and slots are dimensioned between each other and back to the prominent finished surface in each direction.

When the true shape of a feature or surface does not appear in one of the six standard views, an auxiliary view of that surface and feature should be projected to locate it properly (Fig. 15.79).

15.8.5 Locating Holes on a Part

Size dimensions for a part's features are established first, followed by the location of features such as holes. Holes are located from a machined surface. In most cases, holes are established in patterns and dimensioned accordingly. Figure 15.80 shows a detail of a connector. Because the part is thin (.25), only one view is needed. Each of the holes is located from the lower left corner. The 0,0 position (origin) is used to establish control surfaces from which all dimensions are taken. This part has dimensions without dimension lines. This is called **rectangular coordinate dimensioning with-**

FIGURE 15.76 Finish Marks



out dimension lines.

Features that lie about a common center, such as slots and holes (Fig. 15.81), can be dimensioned with angular dimensions, (a), or with a note such as "equally spaced," (b). **Offset dimensioning** takes each locating dimension along the axis of the part from its centerline. This method is preferred because it is easier to set up a machine to locate the holes for machining with rectangular coordinates.

Because each of the holes dimensioned in Figures 15.81 and 15.82 is taken from the center of the part, they are all **dimensioned from a finished surface.** Whenever possible, location dimensions to machined features are taken from a finished surface (Fig. 15.2).

Hole patterns, a set of holes related to another mating part, are established by locating the center hole (if there is one), or by locating the same hole in both directions, in which case dimensions between the holes are given.

15.9 NOTES ON DRAWINGS

Notes can be either **local notes,** as in the callout of a hole, or **general notes** (Fig. 15.45). General notes are placed outside the geometry and beyond dimensions. Notes are one of the last items to be placed on a drawing.

General notes are located, according to ANSI standards, in the upper left-hand corner or the lower left-hand corner of the drawing, as in Figure 15.77, where the following note appears:

NOTE:
ALL FILLETS AND ROUNDS
TO BE R .250 UNLESS
OTHERWISE SPECIFIED

Drawings completed to older ANSI standards have notes above the title block on the right side of the drawing. Some of the examples in the text reflect this older standard, as will many drawings encountered on the job. Many companies apply their own in-house standards that may deviate from accepted ANSI standards.

Here is an example of typical general notes:

NOTES:

1. MATL: .093 THK. ALUMINUM-5052.
2. FINISH: CLEAR ANODIZE–FRONT & REAR PANELS BRUSHED.
3. OPTIONAL RELIEF FOR BREAK.
4. MIN. BEND RAD. TYP. (4) PLACES.
5. SILK-SCREEN PER DWG. 18014-201

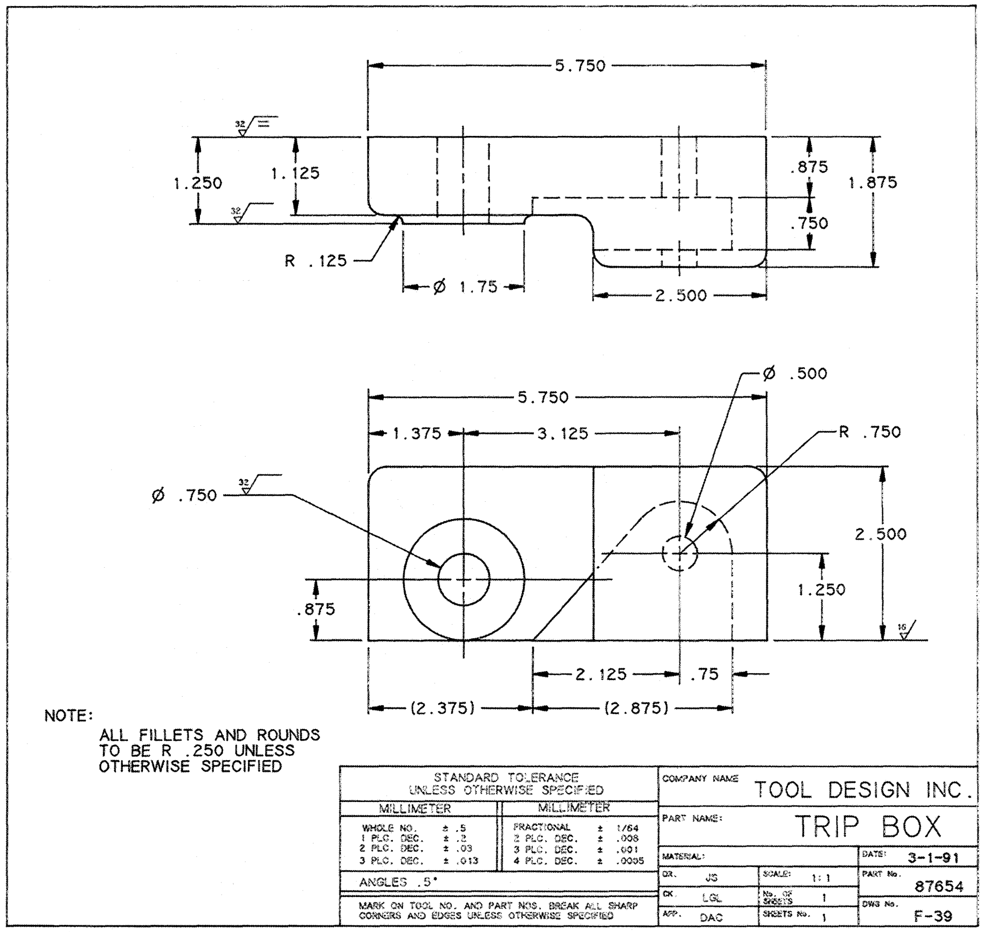


FIGURE 15.77 Finish Marks Used on Mechanical Detail

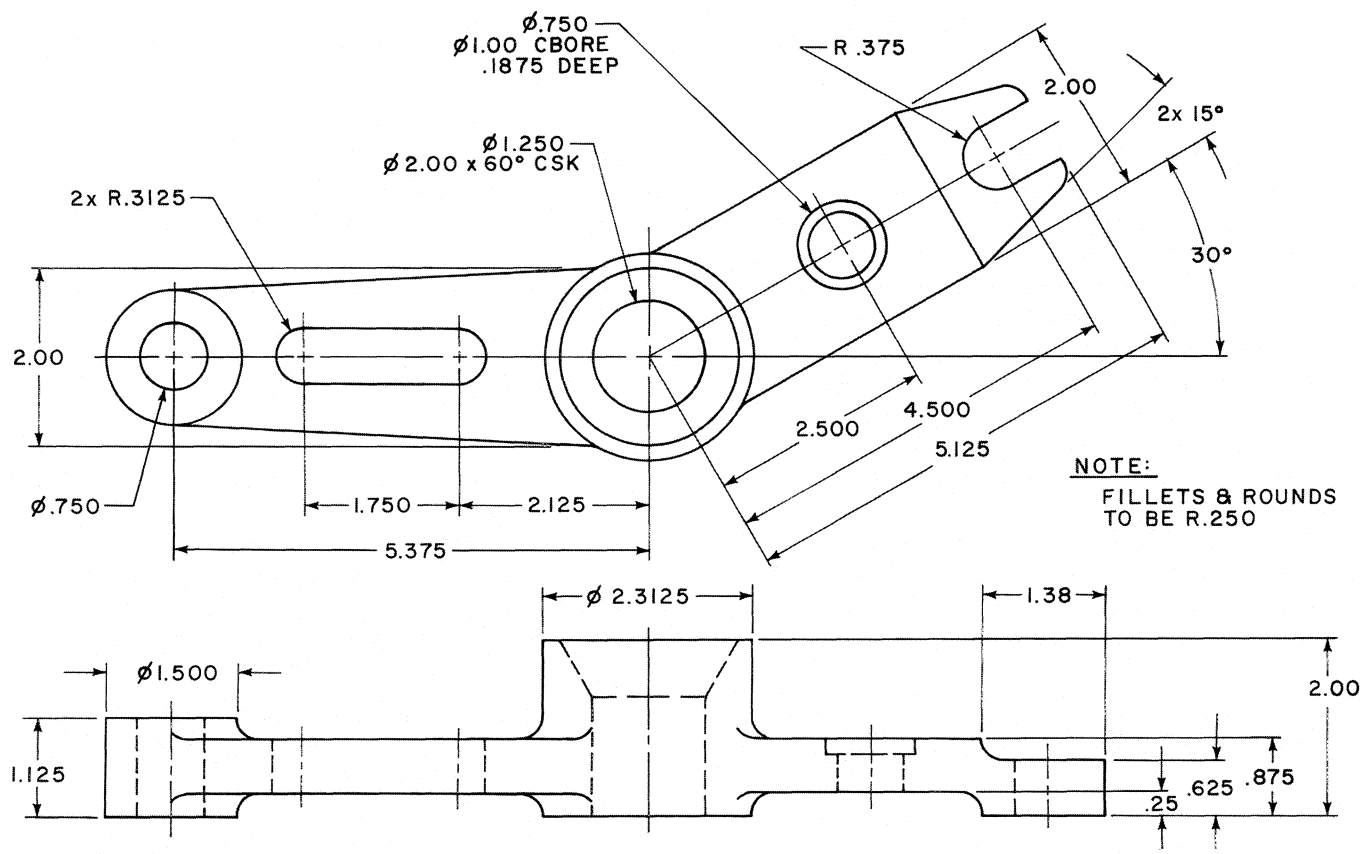


FIGURE 15.78 Dimensioning Machined Features

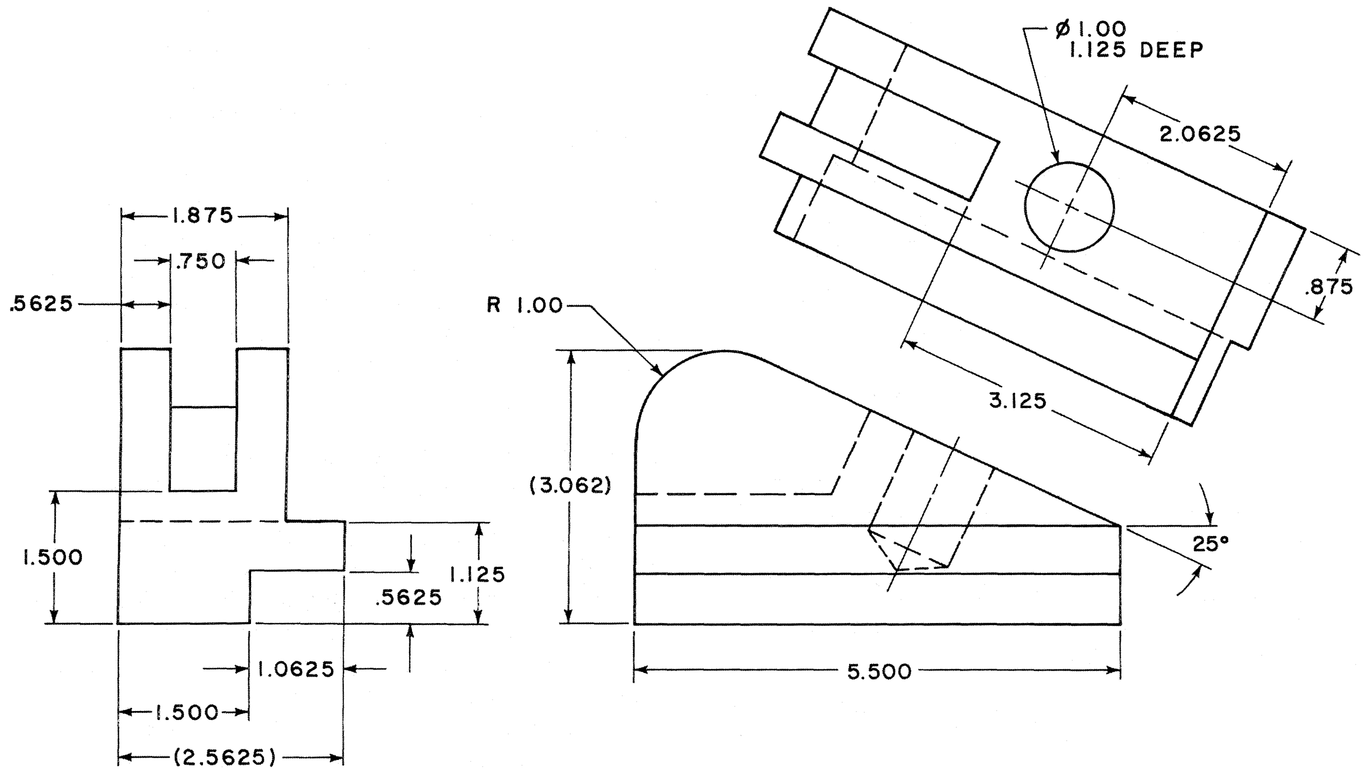


FIGURE 15.79 Anchor Detail

A variety of abbreviations appear in notes. Abbreviations should conform to ANSI Y1.1. See Appendix B for common abbreviations on drawings. Keep your use of abbreviations to a minimum.

Notes are lettered in uppercase except when they are long and detailed; here it is acceptable to use upper- and lowercase. This is the only place on mechanical drawings where lowercase lettering is permitted. The width of notes should be limited to the width of the parts list or of the revision block on the drawing.

.25 ALY ALUM ANODIZE BLACK

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

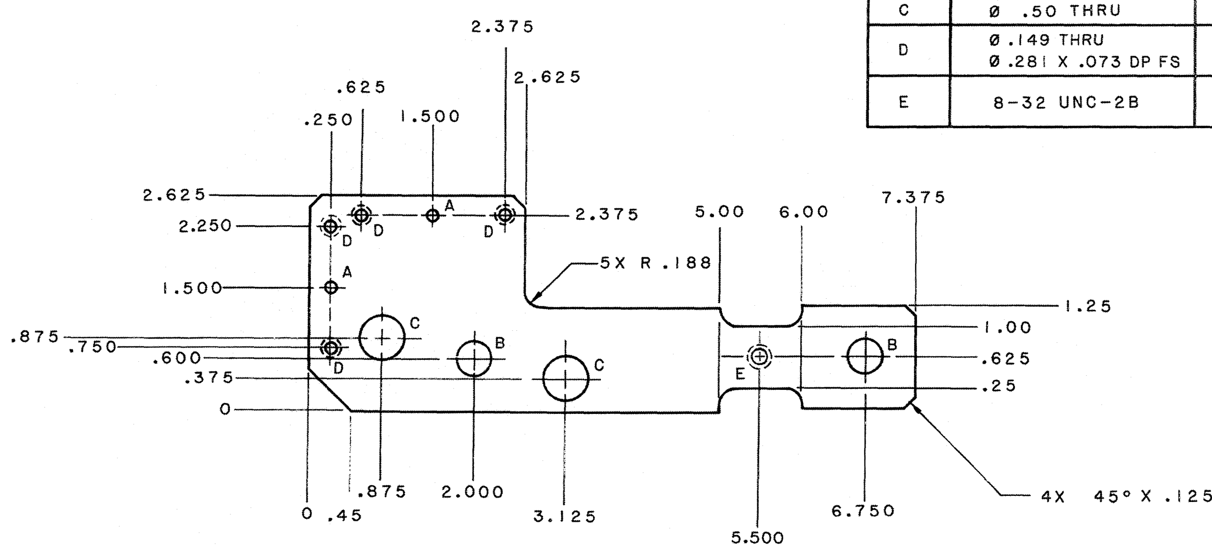


FIGURE 15.80 Dimensioned Part

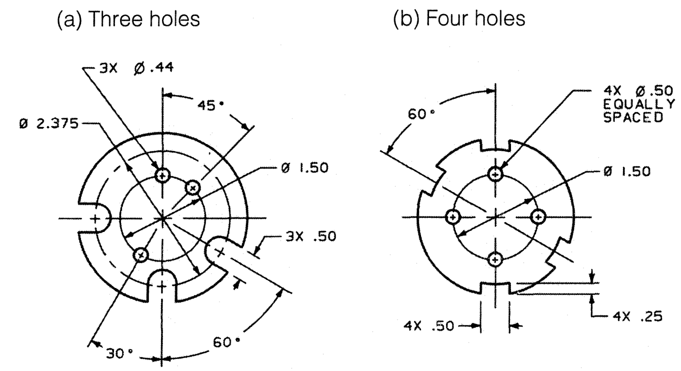


FIGURE 15.81 Dimensioning Repetitive Features

Notes that apply to a view can be placed under the view. **Local notes** are placed away from the view outline (Fig. 15.83):

6-32 UNC-2B	4-40 UNC-2B	.250-20 UNC-2B
.50 DEEP	.25 DEEP	2 PLACES
2 PLACES	8 PLACES	

If a drawing is large and complicated, local notes are allowed within the view. However, avoid this practice whenever possible.

15.10 LOCATION OF FEATURES AND DIMENSIONING METHODS

To design a part, you must know its manufacturing method. If the part to be manufactured does not require close tolerancing, simple dimensioning is suitable. If the manufacturing method is automated, coordinate dimensioning is needed. Design for manufacturability (DFM) is an important factor in creating clear, precise, manufacturable, and successful parts and products.

Rectangular coordinate dimensions locate features accurately with respect to one another and, as a group or individually, from a datum or origin (Fig. 15.82). The features that establish the datum must be identified clearly on the drawing. Coordinate dimensioning is the most frequent method of dimensioning because of automation in the machine tool and manufacturing areas.

15.10.1 Rectangular Coordinate Dimensioning

Rectangular coordinate dimensioning locates features by dimensioning from two or three *mutually perpendicular planes*. The cylindrical part in Figure 15.84 is dimensioned with geometric tolerancing. The three mutually perpendicular planes are established from the center and the bottom. This type of dimensioning either establishes **datum lines** (**X**, **Y**, and **Z** coordinate lines from which all dimensions are taken) or uses the centerlines of a symmetrical or circular shape. In Figure 15.85(a), the **X** and **Y** coordinates serve as datum lines. Here, all dimensions are positioned rectangularly from the datum lines/baselines. The part is located in quadrant I so that all values for dimensions are positive. In Figure 15.85(b), the circular part and its hole pattern are dimensioned from the center of the piece. All dimensions are drawn perpendicular to the part's center.

The four quadrants of the rectangular coordinate system are shown in Figure 15.86. The rectangular coordinate system has two perpendicular axes, **X** and **Y.** The plane

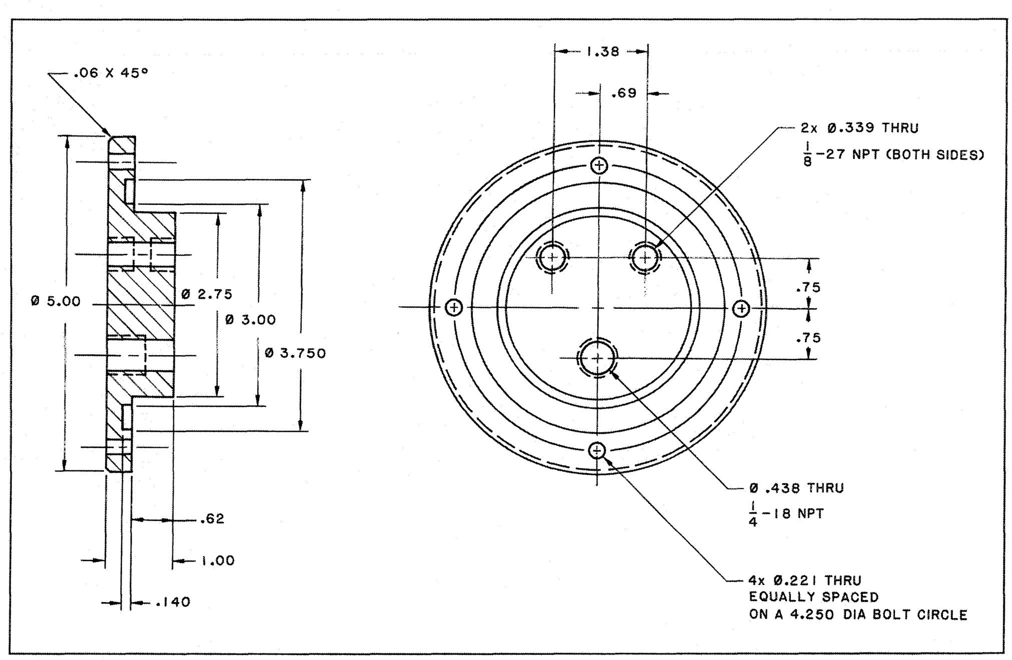


FIGURE 15.82 Square Dimensioning Holes

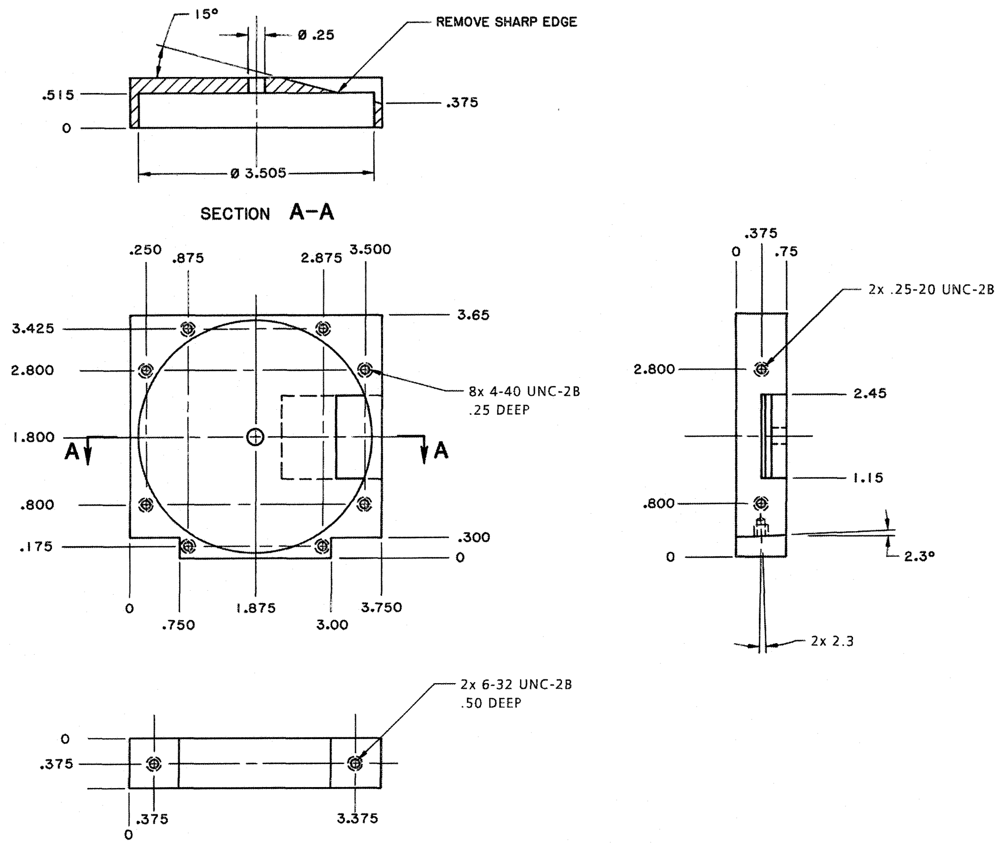


FIGURE 15.83 Part Detailed Using Dimensions Without Dimension Lines

formed by the **X** and **Y** axes establishes the origin of the **Z** axis (Fig. 15.87). The intersection of the three axes is the origin, which has a numerical value of zero (**X0,Y0,Z0**). The three reference planes can locate the part, as in Figure 15.88. The **X** and **Y** axes may also be established from the part, as in Figure 15.89.

Figures 15.90 and 15.91 use the rectangular coordinate method for dimensioning. Holes and curved features are dimensioned by locating center points from datum lines/baselines, indicated by zero coordinates. Dimensions are established so as to have values that can be entered easily when programming the part during CNC machining. This part also uses a hole chart. All holes are through the workpiece, so **X** and **Y** dimensions are required.

The parts in Figures 15.90 and 15.91 are dimensioned from datum lines. The dimension lines are eliminated, and only measurements and extension lines are shown. This is called **rectangular coordinate dimensioning without dimension lines** (ordinate method). **Ordinate dimensioning** is one of the easiest and clearest ways to dimension a part. Dimensions are shown on extension lines without dimension lines or arrowheads. The base (datum) lines are indicated as zero coordinates, or labeled as **X, Y,** and **Z.**

15.10.2 Polar Coordinate Dimensioning

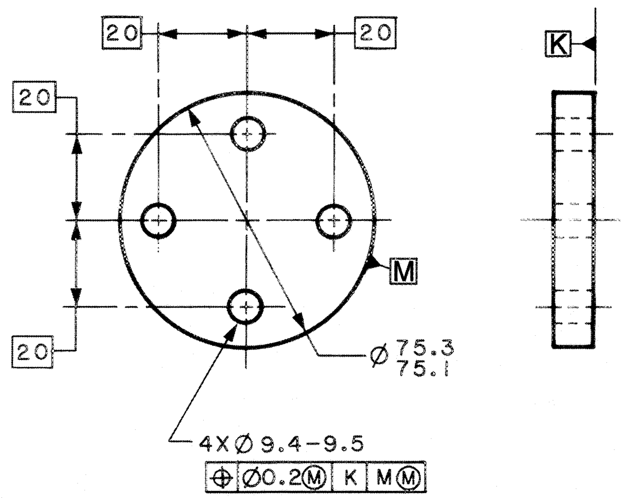
When **polar coordinate dimensioning** is used to locate features, a linear and an angular dimension specify a distance from a fixed point at an angular direction from two or three mutually perpendicular planes. The fixed point is the intersection of these planes (Fig. 15.92). The holes are established with a radial value (**R2 .62**) and angles for each hole. The 0,0 position is in the lower left; the radial value and the angles are established from a location hole.

If a CAD system is used to detail the part, polar coordinates should be given in decimal units and in decimal degrees.

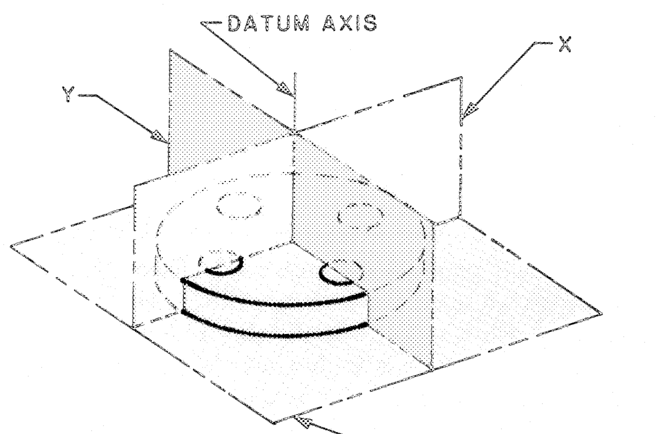
15.10.3 Datums and Tolerances

Datum points, lines, or **surfaces** are features that are assumed to be exact. They are baselines or references for locating other features of the part. A feature selected as a datum must be easily accessible and clearly identified. In many cases, datums are established as the far left surface and bottom surface in a view (Figs. 15.90 and 15.91). An artificial datum like a construction hole or a line edge is sometimes machined in a part for manufacturing and checking only. In Figure 15.93, the part is symmetrical about its vertical centerline. In this example, all dimensions are established from the center hole. The lower left corner of the part is a large curve, which makes it inappropriate for establishing dimensions.

A datum surface must be more accurate than any location measured from that datum. It may be necessary to specify **form tolerances** for the datum surface to ensure that locations can be established accurately. Mating parts use the same feature surface. When parts must match or mate, the related hole centers serve as the datum.

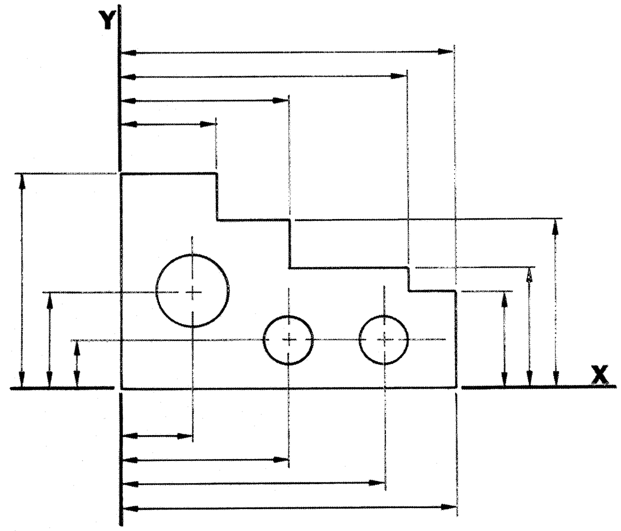


(a) X and Y coordinates used as datums

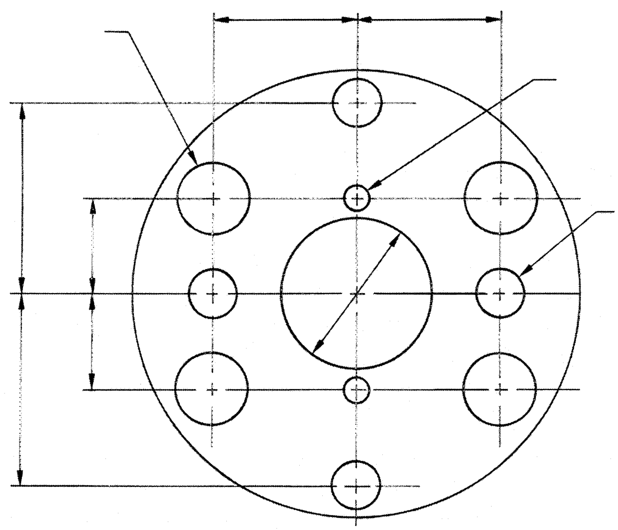


(b) Datum planes

FIGURE 15.84 Part with Cylindrical Datum Feature



(a) Datum line dimensioning



(b) Rectangular coordinate dimensioning

FIGURE 15.85 Dimensioning Methods

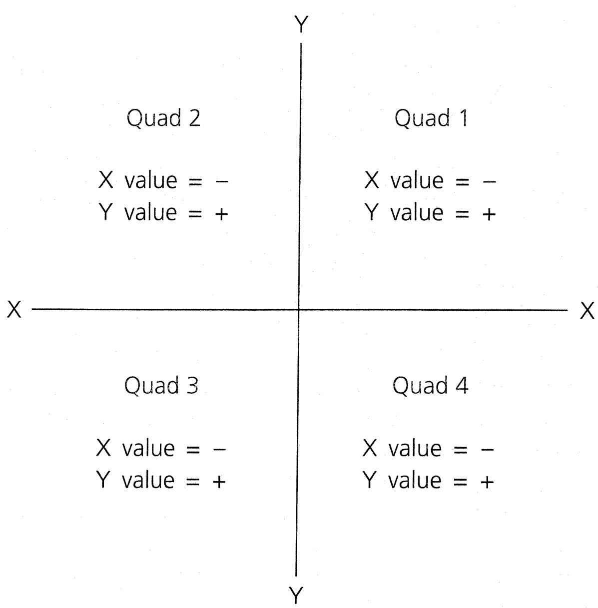


FIGURE 15.86 Quadrants

Dimensioning from a common base or datum reduces the overall accumulation of the tolerance. But the tolerance on the distance between any two features, located with respect to a datum and not with respect to one another, is equal to the sum of their tolerances. Therefore, if it is important to control two features closely, the dimension is given directly from a datum or a baseline.

15.10.4 Hole Charts and Tabular Dimensioning

Tabular dimensioning is a type of rectangular coordinate dimensioning in which dimensions from mutually perpendicular datums are listed in a table on the drawing (Fig. 15.94). This method is good for parts that require locating a large number of similarly shaped features, such as multiple

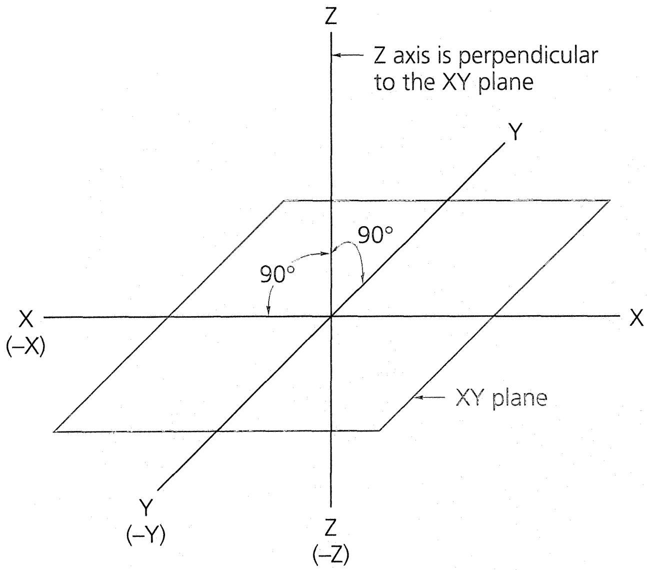


FIGURE 15.87 X, Y, Z Axes

holes, slots, or hole patterns. The information is listed in tables. For automated tooling and programming CNC machines, providing **X, Y** dimensions and **Z** depths is the best method. Hole sizes are also given on a hole chart.

For complicated parts, or a parts with a multitude of holes in one or more surfaces, **hole charts** simplify the drawing. In hole charts, the surface of hole entry and each hole are identified on the drawing. In Figure 15.94, the hole chart lists the **X** and **Y** position of each hole with the depth (**Z**).

The surfaces of hole entry are identified with the names of the principal views. The order of these views for hole charts is:

1. Top
2. Front
3. Right
4. Left
5. Bottom
6. Rear
7. Auxiliary (if used)

The hole chart shows the surface of entry of each hole, the symbol number that identifies each hole, and the number of times each hole is used in this surface. It also gives the complete specification for each hole. Identical holes in a surface are shown by a single symbol number or letter. Hole charts are used for sheet metal details and drilling drawings for printed circuit boards. On parts with very complex hole patterns, the locating dimensions for the holes are shown in the chart as the **X** and **Y** positions in each view; this is called **rectangular dimensioning in tabular form** (Fig. 15.94).

In **X** and **Y** coordinate dimensioning, each hole has a separate identifying symbol. You should group holes by giving diameters the same size and the same letter symbols or by numbering them consecutively. All holes are listed in the hole chart. Holes are normally listed alphabetically, starting from the largest with the letter A. In another method of labeling holes for tabular dimensioning, each hole is numbered consecutively from number 1, without regard to size.

When the hole is completely through the part, **THRU** is

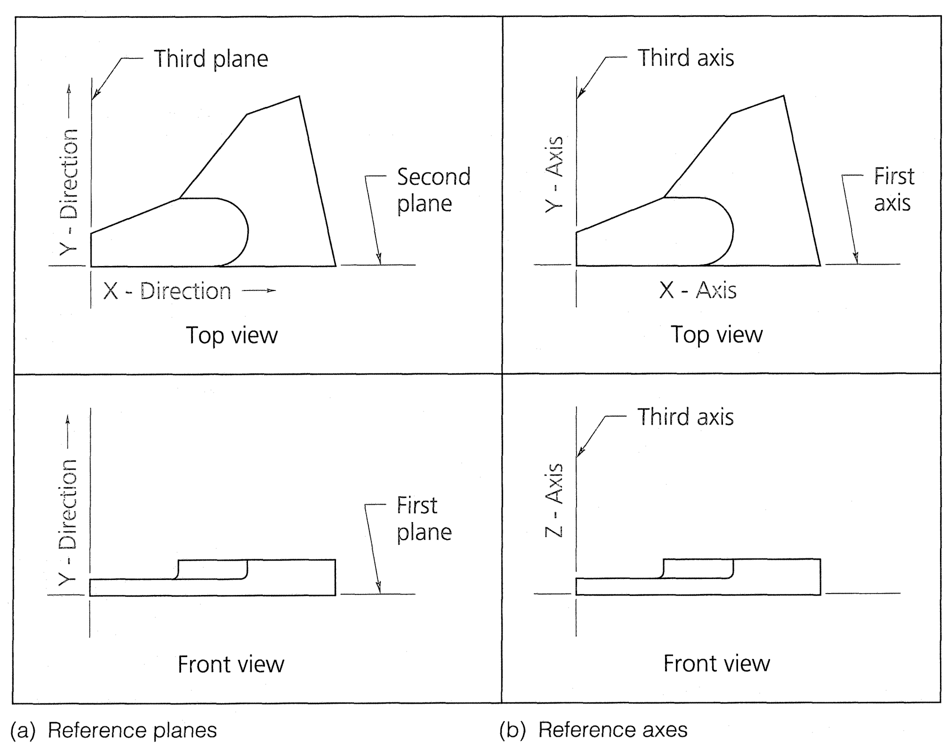
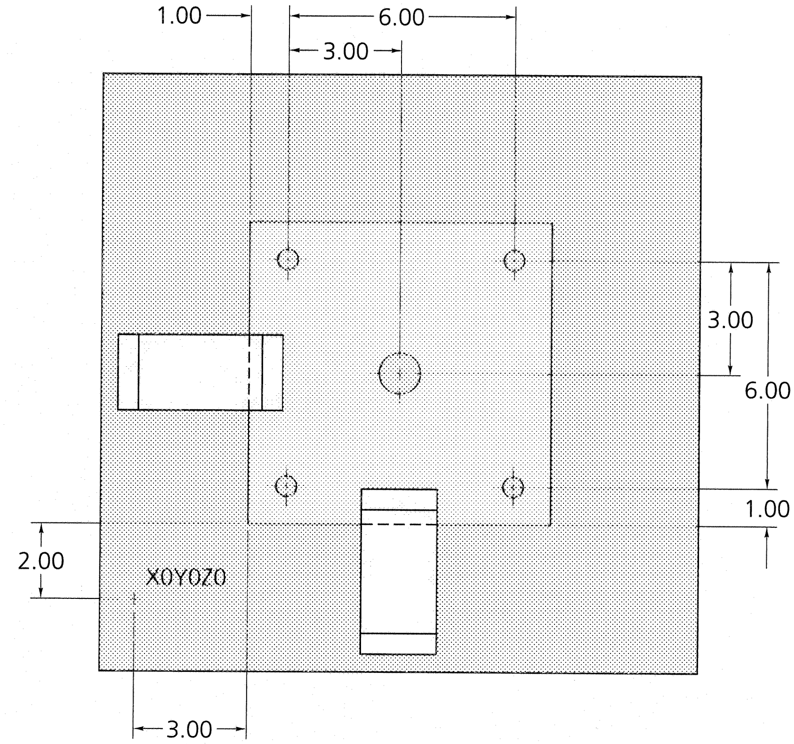


FIGURE 15.88 Reference Surfaces

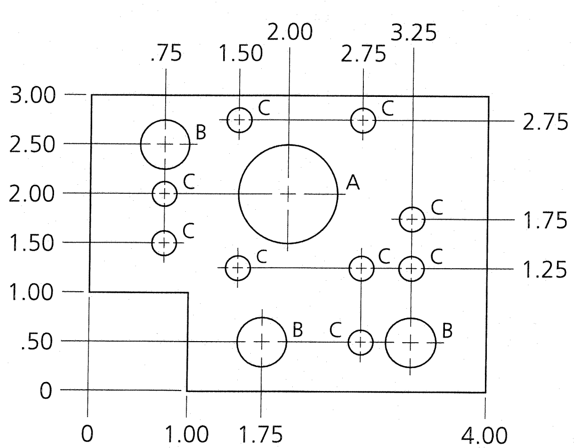


IGURE 15.89 0,0,0 Position Established from he Workpiece. Worktable and clamps hown with part.

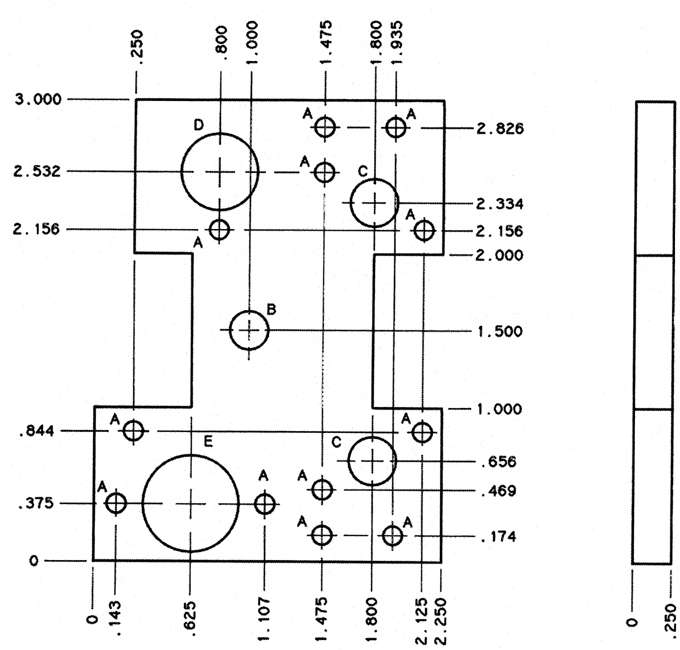
ised as the **Z** dimension. If more than one surface is to have ioles called out, **X** and **Y** axes are established for each urface or view. The depth is specified for each hole, and the iew is noted in the hole chart.

Tabular dimensioning is also found in many catalogs, vhere a standard part has varying dimensions for size and ength. Bolts, screws, keys, pipe fittings, valves, and other

Size symbol	A	B	C
Hole dia.	1	.5	.25



GURE 15.90 Hole Charts and Coordinate mensioning



HOLE	DESCRIPTION	QTY
A	.125 DIA	12
B	.250 DIA	1
C	.312 DIA	2
D	.500 DIA	1
E	.625 DIA	1

NOTE: ALL HOLES THRU
.25 ALY ALUM 6061-T6 ANODIZE BLACK

FIGURE 15.91 Hole Charts and Coordinate Dimensioning

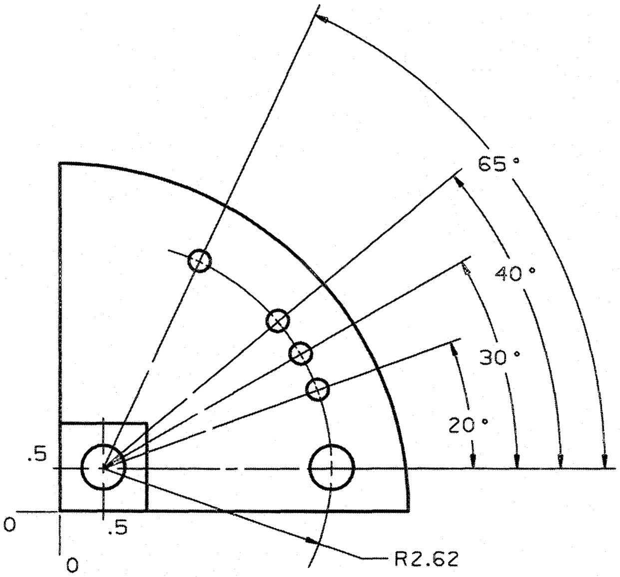
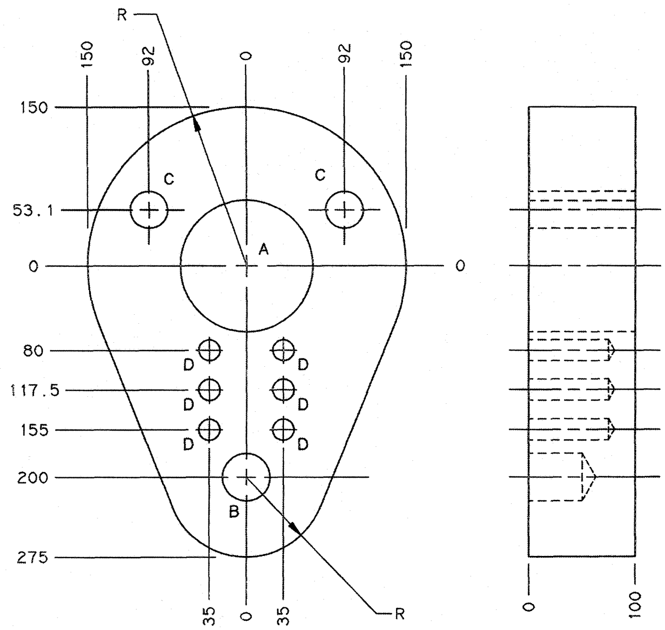


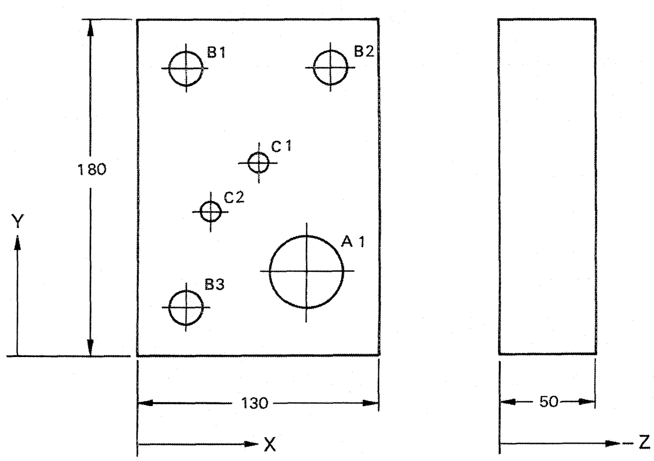
FIGURE 15.92 Polar Coordinate Dimensioning



HOLE	SIZE	DEPTH	QUANTITY
A	125	THRU	1
B	45	50	1
C	35	THRU	2
D	20	75	6

FIGURE 15.93 0,0 Position Established at Center of Part

HOLE	FROM	X	Y	−Z
A1	X,Y	90	44	10
B1	"	26	150	30
B2	"		150	30
B3	"	26	26	30
C1	"	64	100	40
C2	"	40	76	40



HOLE	DESC	QTY
A	Ø 40	1
B	Ø 20	3
C	Ø 10	2

FIGURE 15.94 Rectangular Coordinate Dimensioning in Tabular Form

standard items have dimensions in tabular form (see Appendix C).

15.10.5 Repetitive Features or Dimensions

Repetitive features (Fig. 15.95) or dimensions are specified with an × following a numeral to indicate the "number of times" or "places" that a feature is required. Features like holes and slots, which are repeated in a series or pattern, are specified by giving the required number of features and an × followed by the size dimension of the feature. A space is placed between the × and the dimension (Fig. 15.95).

If it is difficult to distinguish between the dimension and the number of spaces, one space is dimensioned and identified as a reference (**1.00** in Figure 15.95). *Reference dimensions* are enclosed in parentheses.

The part in Figure 15.96 has repetitive features (holes) that are not equally spaced. Angle dimensions in degrees locate each hole from the vertical or horizontal centerline. A note gives the size and number of holes.

Equal spacing of features in a series or pattern may be specified by giving the required number of spaces and an ×, followed by the applicable dimension (Figs. 15.97 and 15.98). A space is inserted between the × and the dimension. In Figure 15.97 the part has five holes, each with a diameter of 14 mm, equally spaced at 15°. The dimension

Applying Parametric Design . . .

FEATURE-BASED MODELING AND DIMENSIONING

Models can be dimensioned automatically in Draw mode (Fig. A). Pro/ENGINEER displays **dimensions** in a view based on how the part was modeled (Fig. B). The dimension type is selected before the options for showing the dimensions on the drawing. Linear dimensions (Fig. C) and ordinate dimensions (Fig. D) are two of the options.

After a part's features are sketched (Fig. E), aligned, and dimensioned, you modify the dimension values to be the exact sizes required on the design (Fig. F). The dimensioning scheme, the controlling features, the parent–child relationships, and the datums (Fig. G) used to define and control the part features are determined as you design and model on the system. When detailing, you simply ask the system to display views needed to describe the part (Fig. H) and then to display the dimensions for modeling the part. These are the same dimensions as in the part design. You cannot underdimension or overdimension the part because the system displays exactly what is required to model the part. Pro/ENGINEER will not duplicate dimensions on a drawing. If a dimension is shown in one view, it will not be shown in another view. The dimension, however, can be switched to the other view through detailing options.

How to Display Dimensions on a Drawing

1. Choose **Show** from the DETAIL menu and **Dimension** orientation **Ref Dim** from the DETAIL ITEM menu (Fig. I).
2. Choose the dimension type from the LIN ORD menu.
3. Choose one of the following options from the SHOW ITEM menu:

- **Show All** Shows all the dimensions for an object. If an assembly drawing, shows assembly dimensions and also previously erased component dimensions; if a part, shows all feature dimensions.
- **By View** Shows all the dimensions associated with a selected view. Select the view(s) you would like dimensioned.
- **By Feature** Shows all the dimensions associated with a particular feature in the appropriate views. Selects feature to

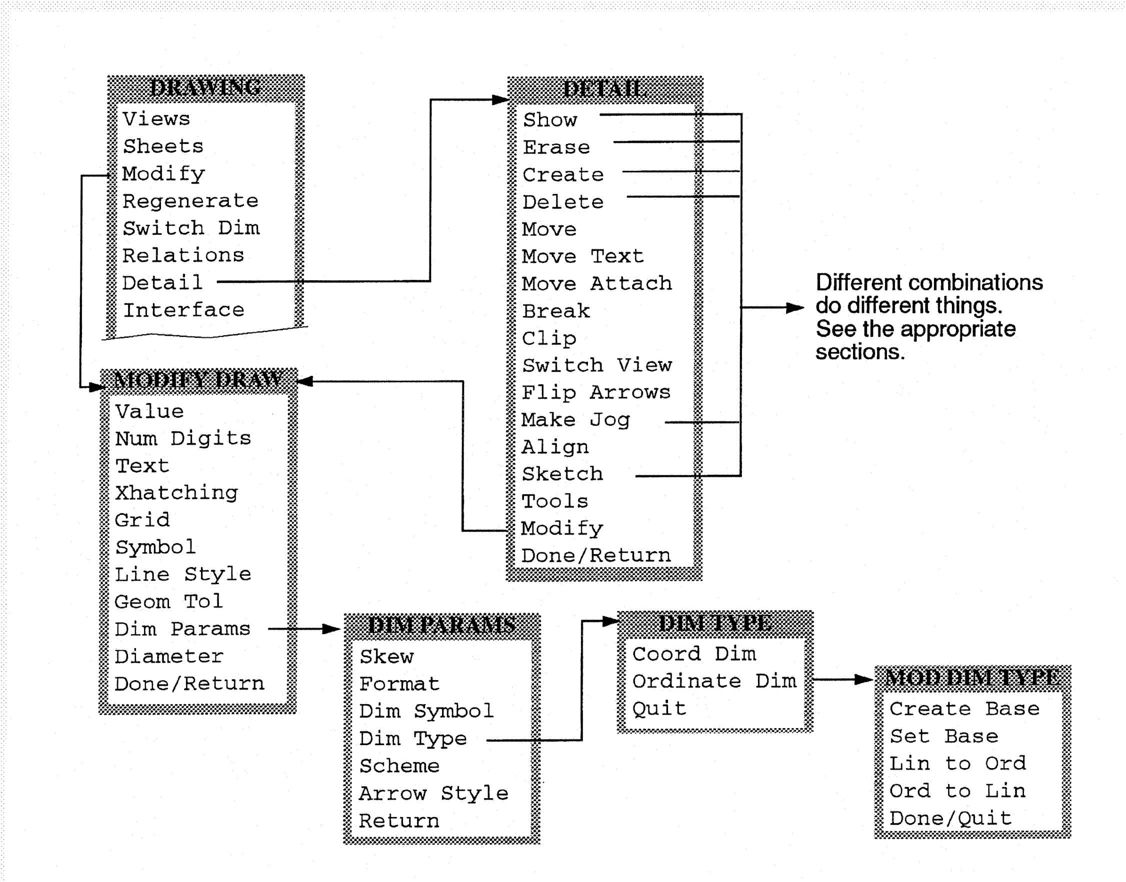


FIGURE A Menu Structure for Dimensioning

FIGURE B Dimensioned Part

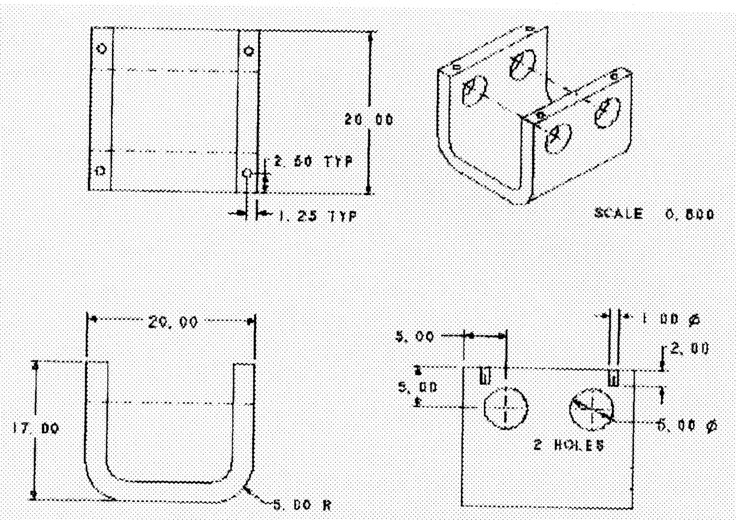


FIGURE C Linear Dimensioning

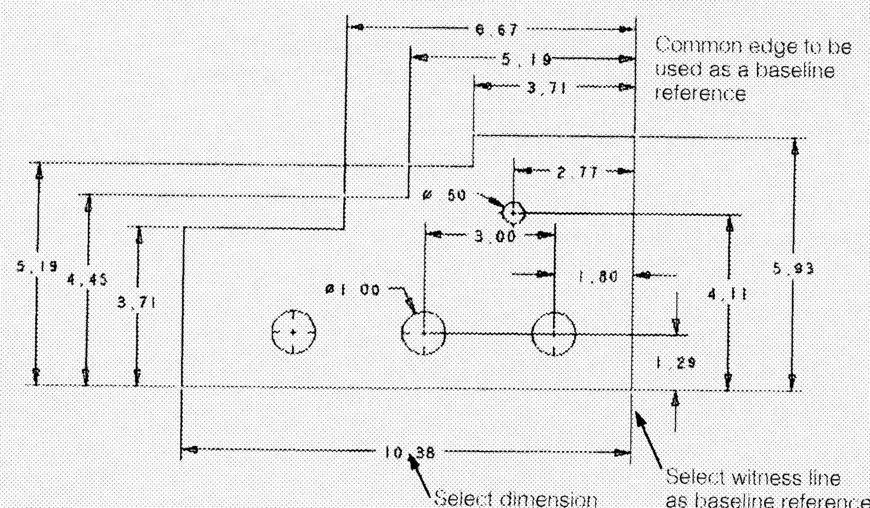
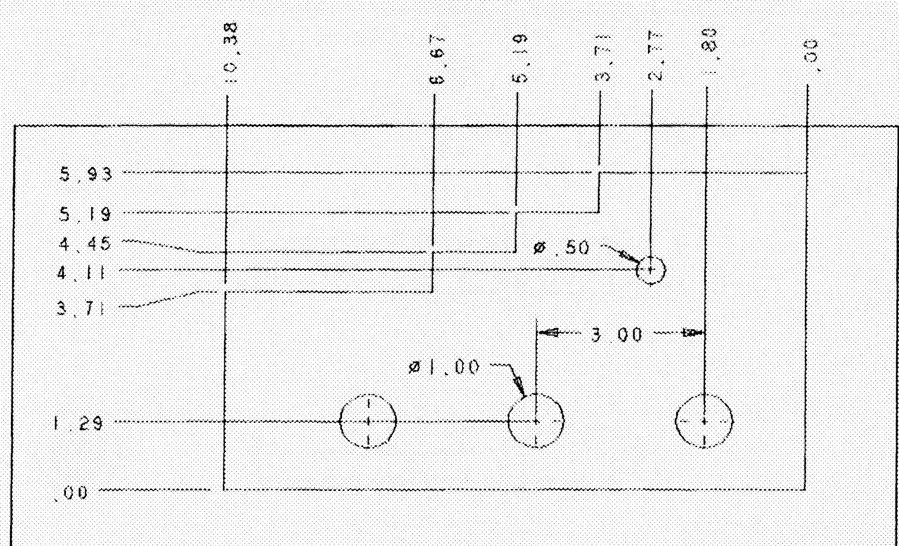


FIGURE D Ordinate Dimensioning



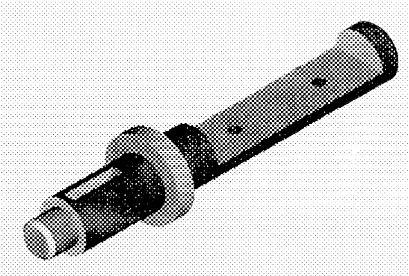


FIGURE E Shaded Shaft

be dimensioned. In models with a lot of features and dimensions, this is the easiest way to work.

- **Feat & View** Shows all dimensions for a single feature in a single view. Select a feature in the view where the dimensions are to be displayed.

The **Clean Dims** options in the DRAWING menu allows you to distribute standard and ordinate dimensions with equidistant spacing along witness lines, displaying them in a more orderly and readable fashion (Fig. J).

To Clean Up the Dimension Display:

1. Select **Clean Dims** from the DRAWING menu.
2. Enter the offset value for the first dimension line (the one closest to the model).
3. Enter the distance between all other dimensions.
4. Select the view to be cleaned up by picking on the model.
5. Dimensions pertaining to the selected view are displayed with the specified spacing. In the event that the new display of dimensions is unsatisfactory, Pro/ENGINEER will ask if you would like to move the dimensions back to their previous display.
6. Select another view, to clean dimensions with the same offsets, or press the middle mouse button to quit the process.

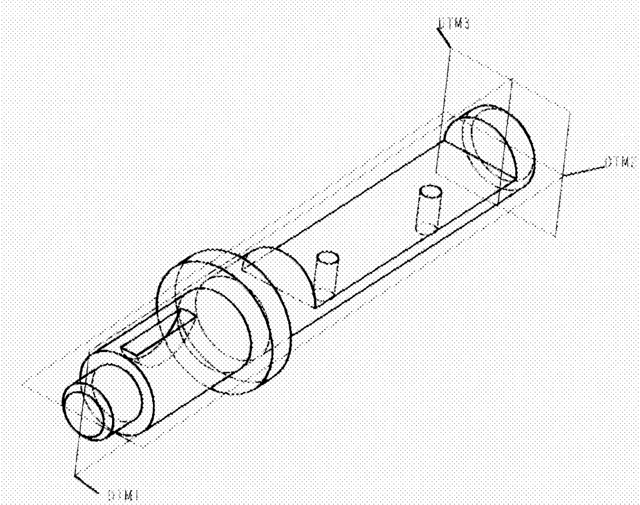


FIGURE G Datums Used to Create, Align, and Dimension the Shaft

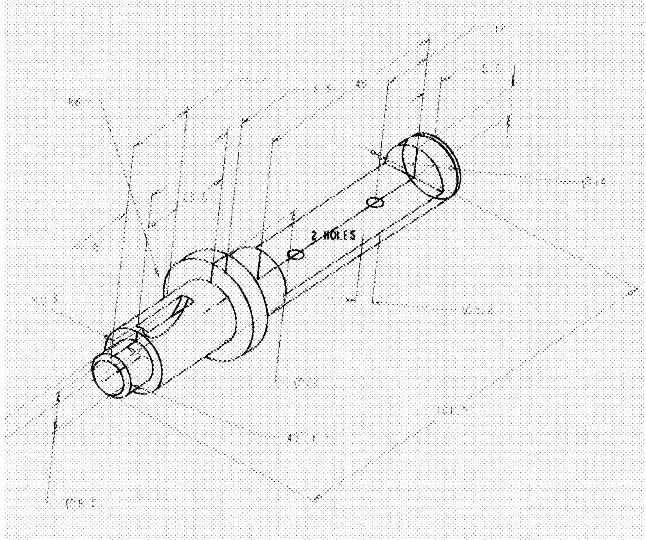


FIGURE F Pictorial View of Shaft with Dimensions Displayed

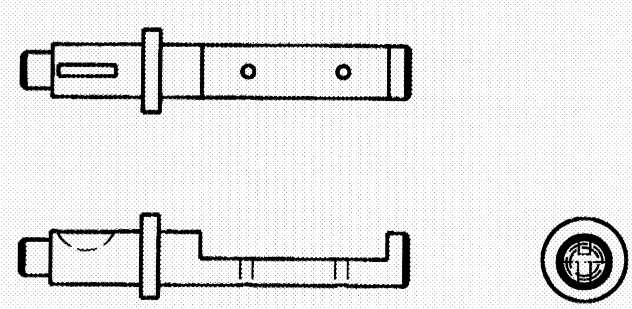


FIGURE H Views of the Shaft Displayed on a Drawing

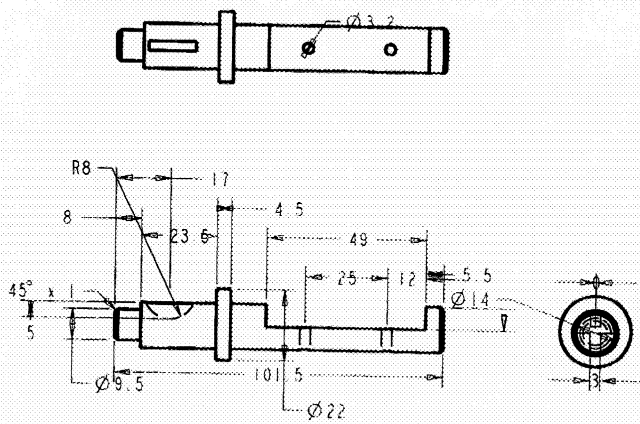


FIGURE I Dimensions Displayed with **Show**

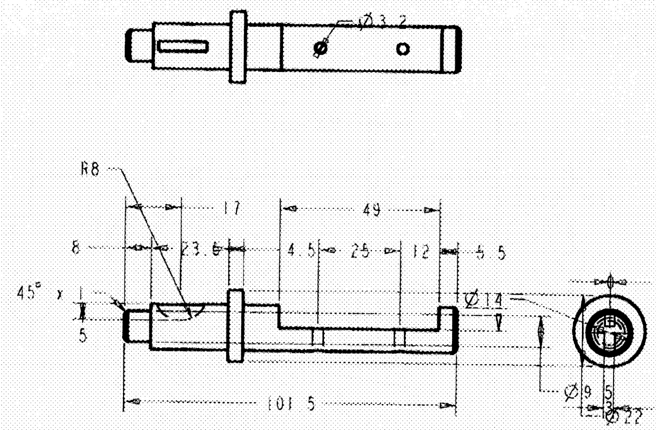


FIGURE J Dimensions After the **Clean Dim** Command

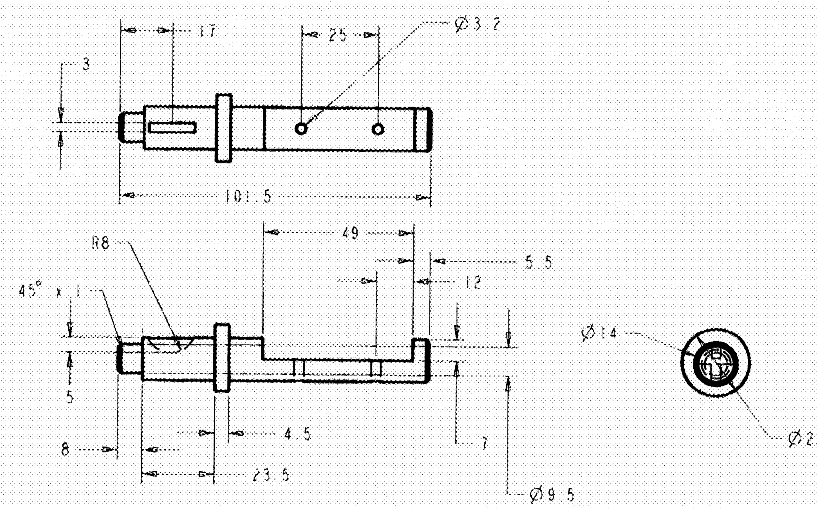


FIGURE K Repositioned Dimensions Using **Move, Move Attach, Move Text,** and **Flip Arrows**

The cleaned dimensions are usually not in the best positions for each dimension and note. After cleaning, the next step normally is to move and reposition the dimensions to create an ANSI-standard drawing with correct dimensioning standards (Fig. K).

Dimensions can be removed from the display by erasing them. **Erasing dimensions** does not delete them from the model (regular dimensions cannot be deleted, but reference dimensions can). Dimensions that have been erased can be redisplayed with the option **Show**.

To Erase a Dimension from the Drawing:

1. Choose **Erase** from the DETAIL menu and **Dimension** or **Ref Dim** from the lower portion of the ERASE ITEM menu.
2. Choose one of the following options from the ERASE ITEM menu.

- **By Feature** Erases all the dimensions associated with a particular feature. Select the feature.
- **By View** Erases all the dimensions associated with a selected view. Select the view(s).
- **Erase All** Erases all the dimensions in the drawing.
- **One Item** Erases the dimension selected.

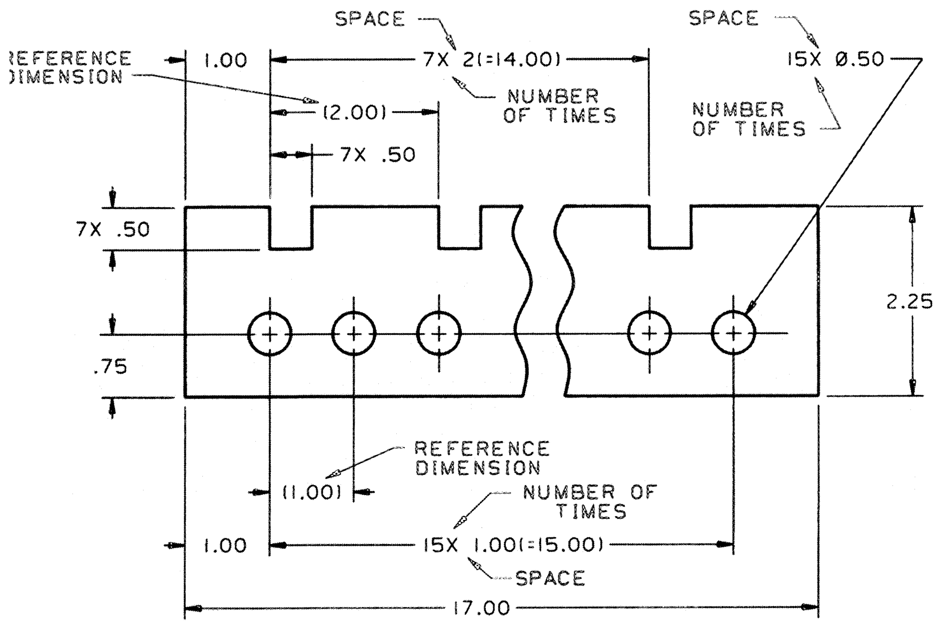
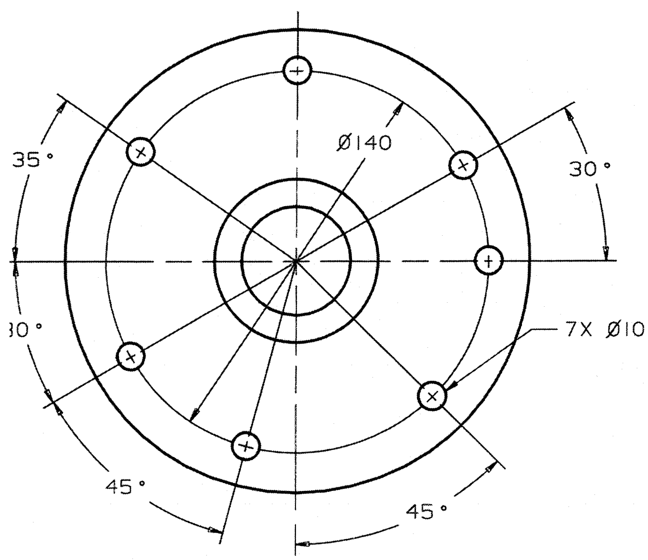


FIGURE 15.95 Repetitive Feature Dimensioning



IGURE 15.96 Dimensioning Repetitive Holes on a ommon Center

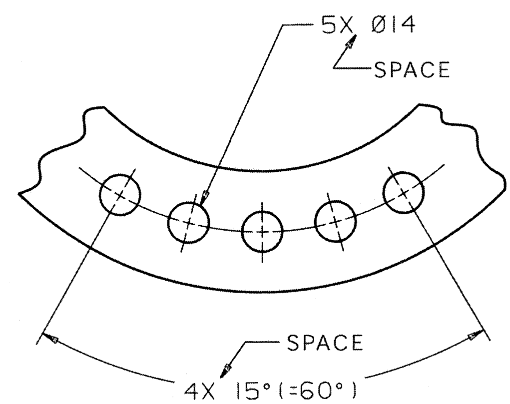


FIGURE 15.97 Repetitive Dimensions

ɔr spacing the holes gives the number of spaces (**4**), the egrees between each hole (**15**), and the total degrees (**60**).

The notation **×** may also be used to indicate "by" between ɔordinate dimensions. An example is when dimensioning a ıamfer: **50 × 45°** or **.250 × 30°**. *The* **×** *is preceded and ıllowed by a space.* If both these practices ("by" and "number [features") are used on the same drawing, ensure that each sage is clear by providing proper spaces.

ou May Complete Exercises 15.13 Through 5.16 at This Time

15.11 DIMENSIONING WITH A CAD SYSTEM

One of the most important aspects of the introduction of the computer into the design process is that the modeled 3D part is associative to the dimensions, which means that the engineer or designer inserts the dimensions with regard to placement but the system puts the proper dimension value on the drawing based on the size and location when the geometry was created.

The elements that make up the geometry of the part are established mathematically. Therefore, it is impossible to put the wrong-size dimension value on the drawing. Others utilizing the same database can verify (**LIST**) each feature as to size and location. This is impossible when a manual

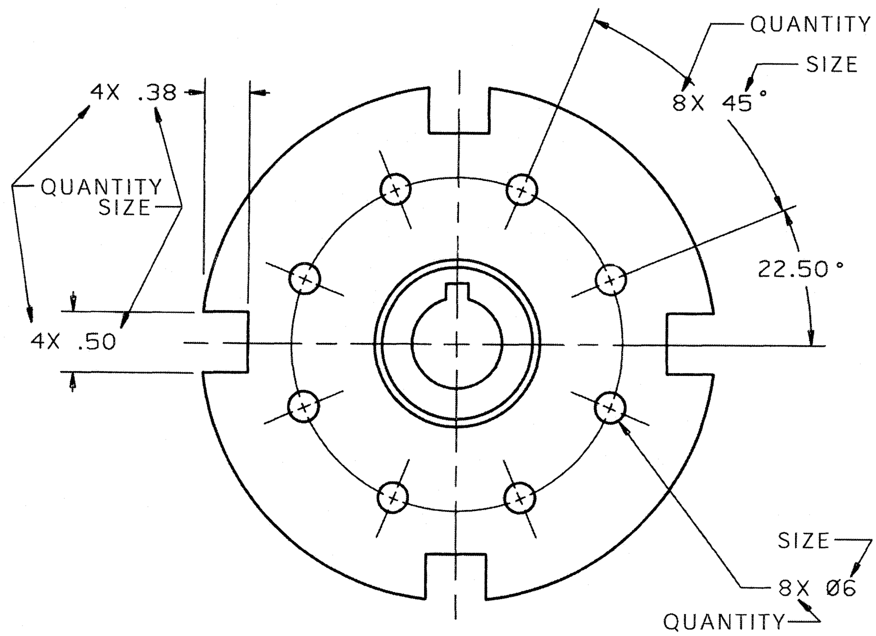


FIGURE 15.98 Repetitive Features and Dimensioning

drawing is produced. If a CAD-generated part is dimensioned insufficiently, manufacturing can activate the part and request the information from the system via an **ID, LIST,** or **MEASURE** command (AutoCAD).

In most engineering applications, a precise drawing plotted to scale is not sufficient to convey all of the desired information. Annotations must be added to show the lengths of features, the distances between features, or the angles between features. Until all CAD databases can be transmitted to the machine tool area for postprocessing and for driving a numerical control machine, dimensioning will be required. The machine tool area and the manufacturing department will be able to use the graphically created database directly to machine and manufacture the part. Many companies already follow this method, and a fully integrated CAD/CAM system is the goal of most manufacturers. Until this technology is in place throughout industry, the designer or engineer will need to define the drawing with dimensions based on the most recent ANSI standards. Many CAD systems provide ANSI standards for geometric tolerancing and dimensioning, though the quality of dimensioning packages differs widely between systems. The designer must still determine the proper dimensioning requirements and decide on their placement on the drawing.

Dimensioning in CAD involves adding annotations to a drawing. Dimensioning also refers to the annotations themselves. CAD systems provide a variety of dimensioning options. The commands and procedures for dimensioning differ among systems. All systems provide quick and easy insertion of dimensions, but you still need to know why certain things are dimensioned and where to place the dimensions. CAD systems automate the process, but the knowledge of ANSI or other standard dimensioning and tolerancing specifications must be mastered along with how to enter the commands for a specific system.

The designer selects dimension options that include decimal or fractional representation, U.S. or SI units of measure, bilateral or unilateral tolerances, feature control symbols, and datums. Dimensions are easily inserted on a drawing by identifying the two locations to be measured and dimensioned. A third location places the dimensioning text and associated values. The system automatically inserts extension lines, dimension lines, leader lines, dimension arrowheads, and dimensioning text at the location indicated. Linear, rotated, diameter, radial, angular, and ordinate dimensioning are available. A variety of arrowhead lengths and types can also be selected (Fig. 15.99). Dimension text, notes, and labels are easily inserted on detail drawings.

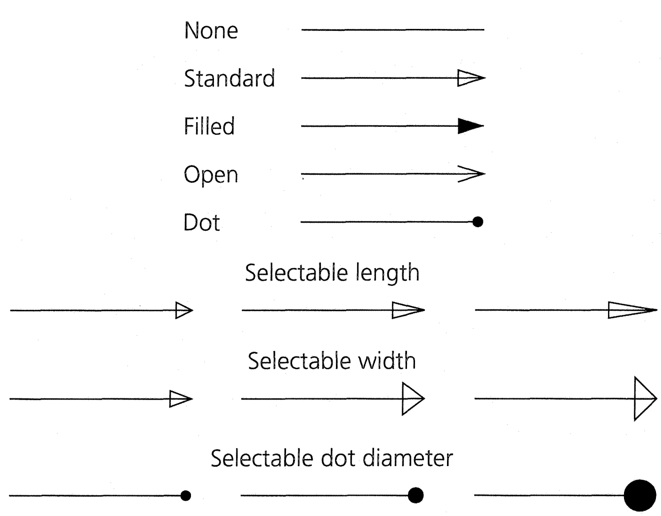


FIGURE 15.99 Arrowhead Variations

You can determine what type of text font (**STYLE**) to elect and its height, width, slant, spacing, case, and justifi-ation. **Labels** offer the same variety of characteristics, and nclude a leader line that is attached automatically to the ssociated label. The angle of the leader line can be set by he designer.

Dimensioning is a drawing activity rather than a model-ng activity when creating the part geometry; in other words, : is 2D, not 3D. The geometry of the part can be 3D or 2D epending on the system's capabilities. Dimensions, like ext, are for information only. They do not exist as actual art entities or features. Therefore, all dimensions are 2D. he same procedure is used to detail a part on a 2D system s on a 3D system.

Before proceeding with the details of dimensioning with AD, a few terms, system capabilities, and options must be xplained. Each system has certain differences that deter-nine exactly where dimensions can be placed. Sometimes it s necessary to work around these limitations.

.5.11.1 Dimension Line

. **dimension line** is a line with arrows at each end, drawn at ne angle at which the dimension has been measured. The imension text is situated along this line, usually dividing it nto two lines. Usually, the dimension line is inside the neasured area. Sometimes, however, it does not fit. If it oesn't, two short lines are drawn outside the measured rea, with the arrows pointing inward. The option for arrows nside or arrows outside is available on most CAD systems. he dimension line is established on the drawing by select-ng the distance away from the part, normally the third election in the command. The first two selections locate the eature ends or two positions to be dimensioned. For ngular dimensioning, the dimension line is actually an arc.

5.11.2 Extension Lines

' the dimension line is drawn outside the part being neasured, straight **extension lines,** sometimes called *witness nes,* are drawn from a feature of the part, perpendicular to ne dimension line. Extension lines are used only in linear nd angular dimensioning. When not needed, one or both of nem can be left off via a suppression capability available on nost systems.

5.11.3 Dimension Text

imension text is a **text string** that specifies the actual neasurement. Most CAD systems provide methods to use ne measurement computed automatically by the system, upply different text, or suppress the text entirely. If you tilize the default text, the system can be instructed to ppend plus or minus tolerances to it automatically.

The dimension text is drawn in the currently selected text nt (style). The default text format is governed by the efault units. Defaults are embedded in the software when it is installed. Defaults can be changed before or during part creation on most systems.

15.11.4 Leaders

For some dimensioning, notes, and other annotations, the text is not placed next to the part it describes. In such cases, it is customary to place the text nearby and to draw a **leader line** from the text to the part. For instance, when diameter or radius dimensioning is desired, but the arc or circle is too small for the dimension text to fit inside, a leader can be drawn from the text to the arc or circle.

15.11.5 Center Mark and Centerline

A **center mark** is a small cross marking the center of a circle or arc. **Centerlines** are broken lines crossing at the center and intersecting the circumference of the circle or arc at its quadrant points. A center mark and centerlines are needed for all circular dimensions. CAD systems know where the center of each circle or arc is located. Inserting a point or center mark at the origin of an arc or circle involves telling the system to put it at the center of the selected arc, fillet, or circle. For many systems, drawing a centerline involves one command that inserts two perpendicular centerlines, with their short dashes crossing at the center of the curved entity.

15.11.6 Dimensions and Scaling

An engineer can change the CRT display size of the part for dimensioning purposes. For example, if the part is increased in size for visual clarification, dimensions inserted by the designer will reflect the true size of the part and not the new CRT display scale. In other words, the display size may be set at 2 to 1 but the dimensions will always be 1 to 1. By zooming in on a portion of a drawing, it is easier to place dimensions, especially in complicated or cluttered portions of a complex part.

15.11.7 Layer Separation of Dimensions on CAD Systems

A **layering scheme** is a means to separate logical groups or types of entities. Each part created on the CAD system has multiple layers associated with it. Direct access is available to any of these layers and to the specific entity or information contained on it. These layers should be thought of as transparent sheets on which the drafter places specific types of information. Any number of these layers can be displayed at one time. For example, text can be placed on one layer, all bolt holes on another layer, and all centerlines on the third layer. Any combination of these layers can be displayed. Some systems can assign automatically specific types of items to be placed on selected layers determined by the engineer. *Dimensions should always be assigned a layer of their own.* Figure 15.100 shows an example of placing dimensions

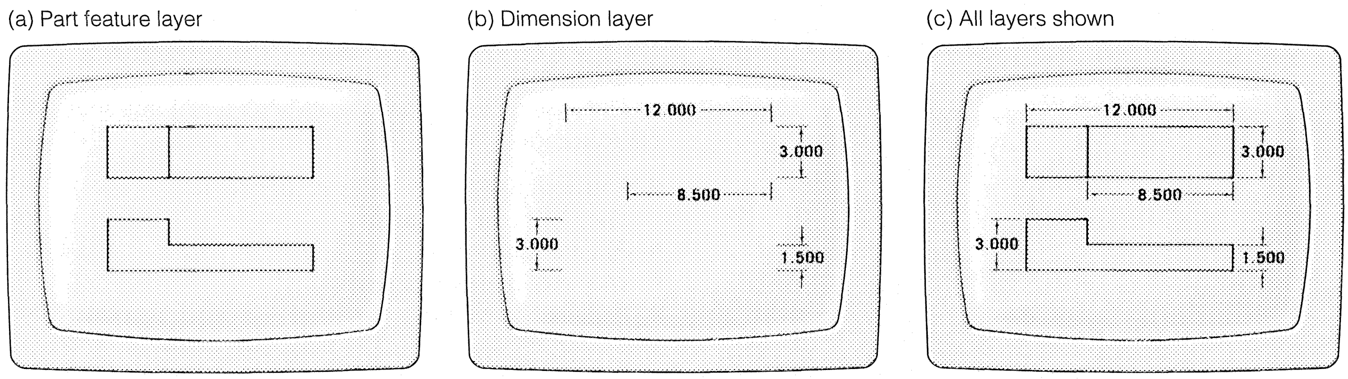


FIGURE 15.100 Dimensioning and Layers

on a separate layer from the geometry of the part. The dimensions or the geometry can be viewed separately [(a) and (b)] or together [(c)].

15.11.8 Color and Linetype

CAD systems with color capabilities provide a palette of colors. *Colors* make it easy to identify and distinguish different kinds of information on the drawing. Many engineers associate a certain color with a specific layer. Layers, entities, or information such as dimensions or notes can all be color coded.

With layers, components of a drawing are easily grouped. A layer or a set of layers holds the items related to a particular aspect of the drawing. Visibility, color, and linetype are easily controlled. Most companies develop a *standard layering scheme,* with assigned layers for construction lines, dimensions, and the part itself, each with its own associated color and *linetype* (font). The following tables show possible layering schemes:

Font, Color, Pen Size, and Layer Scheme

Layer	Color Number	Use	Linetype (Font)	Pen Size (mm)
1	4 (Yellow)	Part geometry	Continuous	0.7
5	7 (Magenta)	Hidden geometry	Dashed	0.5
10	6 (Gray)	Dimensions	Continuous	0.35
15	3 (Cyan)	Centerlines	Centerline	0.35
20	8 (Red)	Labels	Continuous	0.5
30	12 (Blue)	Border and title block	Continuous	0.9

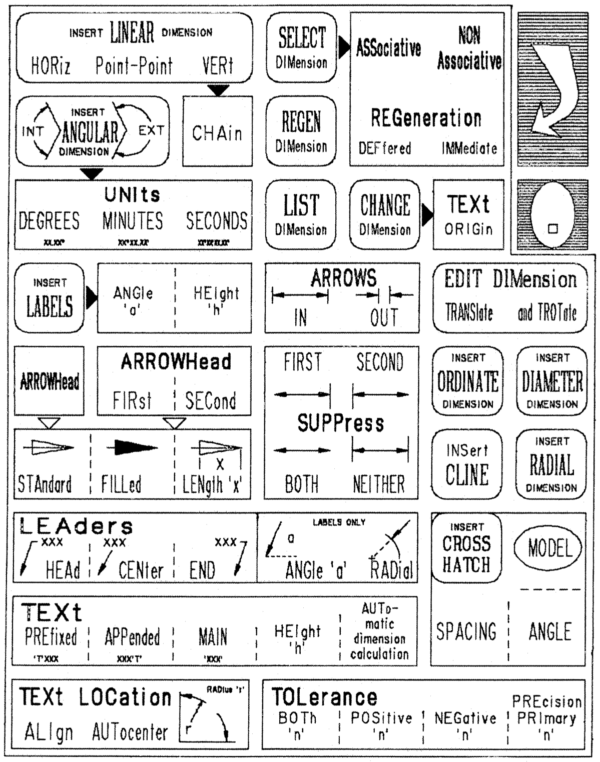
A layering scheme makes it easy to keep track of where different information resides. Layering can also serve to sort graphic information temporarily. For example, layers can help separate geometry in a congested area of the part. By turning off the display of unnecessary information, such as text and dimensions, computer processing takes less time. Placing geometry on one layer and dimensions on another allows for viewing of the model (part) with or without dimensions.

Industry Layering Scheme

Layer Number	Contents
0	Layer Index/Table of Contents
1-50	Construction Geometry and Drawing Formats [contains all construction items used in the design of your part(s), e.g., lines, points, arcs, and the different types of drawing formats you may use]
51-120	Manufacturing (contains manufacturing information, e.g., NC and tool path information, jigs, fixtures, and tooling)
121-145	Dimensions, Text, Labels (contains all dimension information, text, and labels)
146-175	Illustrations [contains technical illustrations using the part(s) as its source—may be presented in 3D view with hidden lines removed]
176-200	Analysis (Engineering), FEM, Physical Properties (contains different kinds of analytical information concerning the part's structure, content, and properties)
201-254	Construction Aids, Miscellaneous Information (contains construction items that help design and dimension a part, as well as any miscellaneous information connected with the part design)

15.12 DIMENSIONING COMMANDS

When dimensioning a drawing, the **DIMENSION** command (**DIM** in AutoCAD) is entered from the keyboard or picked from a screen or from a dimensioning toolbar. All systems provide screen or tablet menus for quick and efficient dimensioning. Figure 15.101 shows a tablet menu with areas for selecting dimension parameters, changing existing dimensions, adding text, suppressing extension lines, altering the arrows and leader style or location, and specifying the type of dimension and its tolerance. Most systems also have screen menus devoted entirely to dimensioning commands. Selecting standards as defaults, inserting dimen-



IGURE 15.101 Dimensioning Menu

sions, and changing existing dimensions can be accomplished with pull-down menus, dialog boxes, and other dynamic menu capabilities.

Dimensioning commands can be grouped into four general categories:

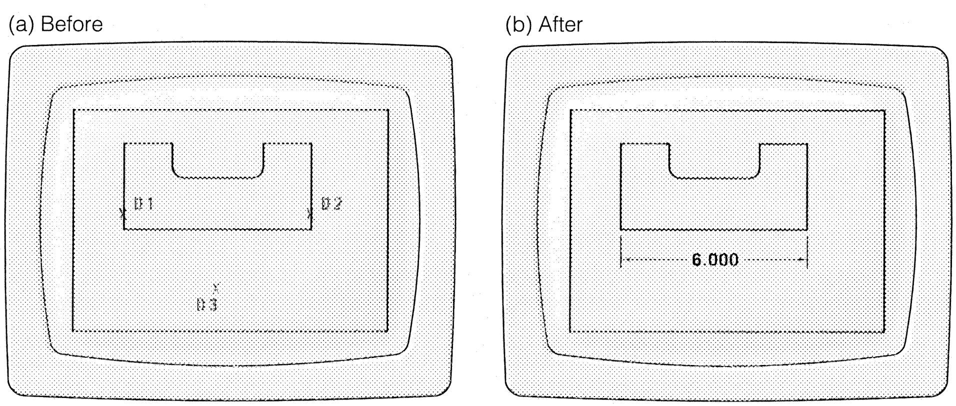
- Linear dimensioning (vertical, horizontal, point-to-point, ordinate)
- Angular dimensioning
- Diameter dimensioning
- Radius dimensioning

15.12.1 Linear Dimensioning Commands

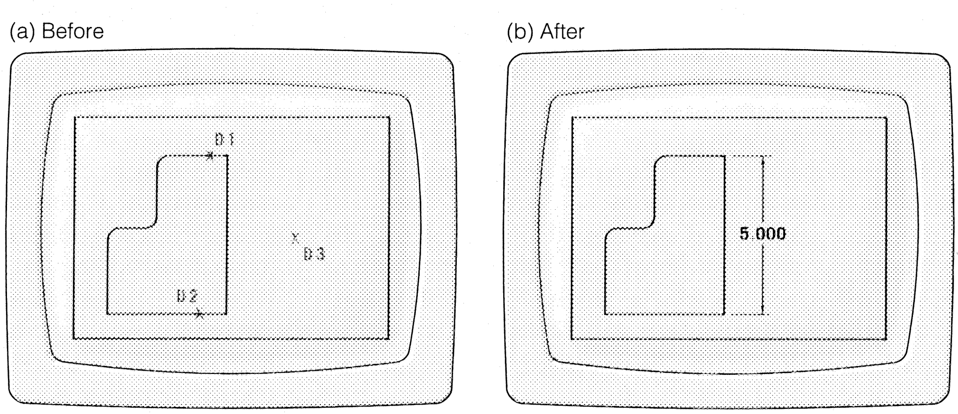
The following gives a general overview of linear dimension capabilities.

Horizontal generates a horizontal dimension line (Fig. 15.102). After the dimension command is entered, the drafter selects the endpoints of the horizontal distance to be dimensioned (**D1** and **D2**) and then the location of the dimension line and text (**D3**). In AutoCAD R13, linear dimensioning is used for both horizontal and vertical lines.

Vertical generates a vertical dimension line (Fig. 15.103) The first, second, and third selections accomplish the same results as in horizontal dimensioning.



IGURE 15.102 Placing a Iorizontal Dimension



IGURE 15.103 Placing a ertical Dimension

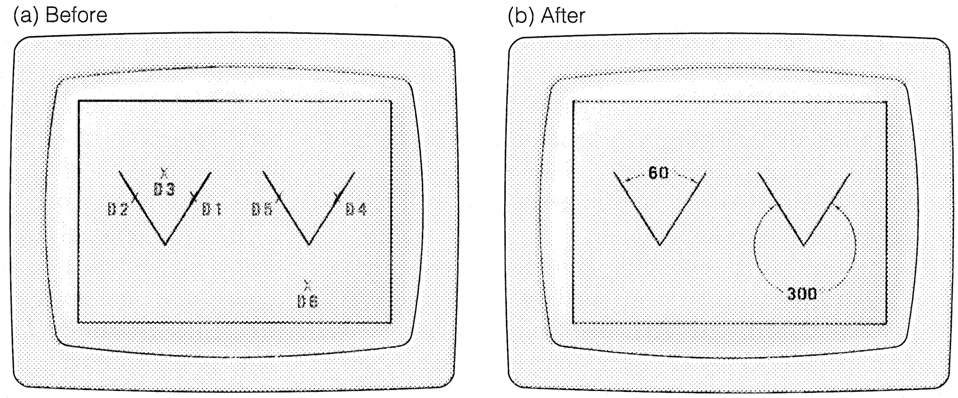


FIGURE 15.104 Placing Angle Dimensions

Aligned (or rotated) generates a linear dimension, with the dimension line drawn parallel to an angled linear entity or surface edge or rotated to a specific angle.

Ordinate allows the selection of two perpendicular datum lines or surface edges, and generates dimensions using rectangular coordinate dimensions without dimension lines.

15.12.2 Angular Dimensioning

Angular dimensioning generates an arc to show the angle between two nonparallel lines. The angle can be either inside or outside; Figure 15.104 shows examples of both inside (60°) and outside (300°) angular dimensioning. In both cases, the engineer enters the command, identifies the two lines to be measured, and places the dimension with the third selected location.

15.12.3 Diameter and Radius Dimensioning

When dimensioning the diameter of a circle, only two selections are required. After the command is entered, the designer identifies the circle to be dimensioned, then locates the dimension and the end of the leader with the second selection (Fig. 15.105).

Radius dimensioning is the same as **diameter dimensioning,** except the system measures and then dimensions the radius of an arc (Fig. 15.105). Remember, circles are dimensioned by giving a diameter dimension, not a radius dimension. Radius dimensions are to be provided for fillets, arcs, and slots.

15.13 CAD AND DIMENSION STANDARDS

Some CAD systems are designed to use dimensions that adhere to American National Standards Institute (ANSI) standards. Some systems provide the option to choose Japanese Industrial Standards (JIS) or the International Organization for Standardization (ISO) standards, which differ somewhat from the ANSI conventions. The selection of a drafting standard and unit of measurement should be made *before* the project is started, though many CAD systems allow you to reset standards, units, tolerances, etc., and to update the entire project automatically. AutoCAD uses **SETVAR** and the **UNITS** command to establish defaults for a project.

15.13.1 U.S. versus Metric Units

You can choose either SI or U.S. customary units of measurement for your design. Both of these standards can have particular units specified for dimensions: inches, feet, yards,

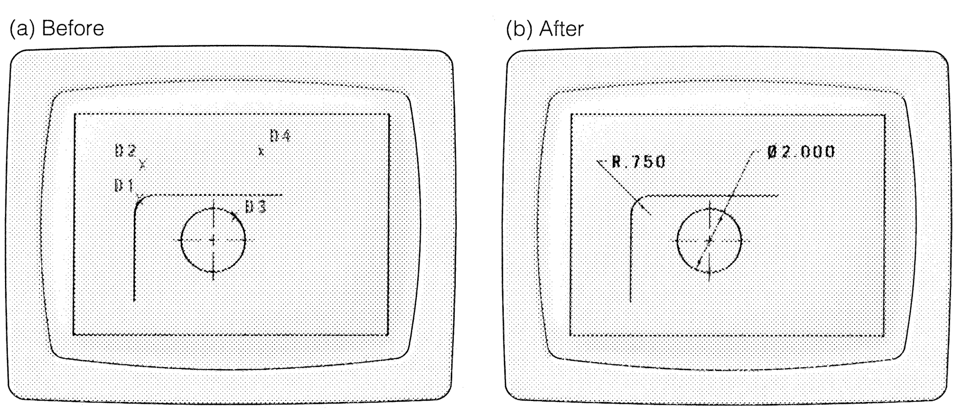


FIGURE 15.105 Placing Radius and Diameter Dimensions

or miles for U.S. standard units of measurements, or centimeters, millimeters, or kilometers for SI (metric) units.

Some systems allow the automatic placement of U.S. units on one layer and SI units on another. In addition, the option for dual dimensioning may be available. The designer dimensions the part once, and the system automatically places the specified primary unit together with the secondary unit in one dimension.

Designs created in one unit of measurement can be converted to other units by changing the setup units (**UNITS** command in **DDIM**) and then updating the file with the **UPDATE** command when using an AutoCAD system. **UPDATE** is reached through the **DIM** command. This capability makes dual dimensioning obsolete. There are several powerful features in AutoCAD R13 for modifying the style of the dimensions. **MODIFY** can update a particular dimension only if that is desired.

15.14 MECHANICAL DESIGN AND DIMENSIONING WITH CAD

Mechanical-design CAD programs can add dimensional information, notes, and labels to your drawings. You also can manipulate drawings of the model for aesthetic reasons or for visual clarification. These manipulation features include choosing a variety of line patterns, removing hidden lines, defining any type or number of views, inserting dual dimensions, defining standards, sectioning, and crosshatching. Dimensional information is associative.

15.14.1 Associativity

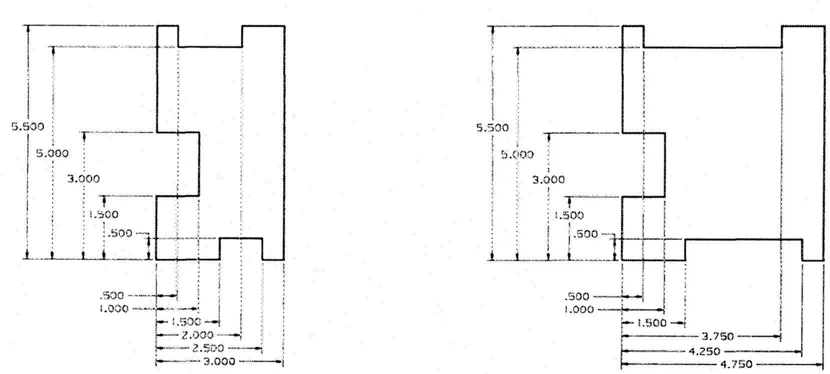
Associativity means that if the geometry is changed or modified in the model, its dimensions will be updated automatically to reflect those changes on detail drawings. Associativity also refers to the ability to carry out design change specifications all the way to the manufacture of the product; e.g., changes to the database could automatically update CNC files and quality-control features as well as inventory and other items attached to the database. Both 2D and 3D systems can be associative. But a 3D system is associative in all of its 3D views, whereas a 2D system is associative in only one view at a time.

It is not unusual for design changes to be made after a part is dimensioned. If the model has modifications, existing dimensions are updated automatically by the system to reflect those changes. The design in Figure 15.106(a) is the original dimensioned part. After design modifications to the part, the dimensions were automatically updated by the system [Fig. 15.106(b)].

15.15 PREPARING THE DRAWING: 2D AND 3D EXAMPLES

Figure 15.107 shows the step-by-step procedures for detailing the one-view drawing of a hold-down plate. The series starts with the part geometry already created [Fig. 15.107(a)]. This could have been done on a 2D or 3D system. The **ZOOM WINDOW** command can enlarge the area where a dimension is to be inserted on the part. Figure 15.107(b) shows the before-and-after sequence of the command for drawing a horizontal dimension on the lower portion of the part. Not all of the part's dimensioning is described in detail; only one example of each of the basic types is shown.

A vertical dimension is inserted between the center of the hole and the bottom of the plate [Fig. 15.107(c)]. Normally, you would have had to tell the system to lock onto the endpoint of the line and the center of the circle. Figure 15.107(d) shows an alternative to an angle dimension. Here, a **rotated (aligned)** linear dimension is inserted to measure

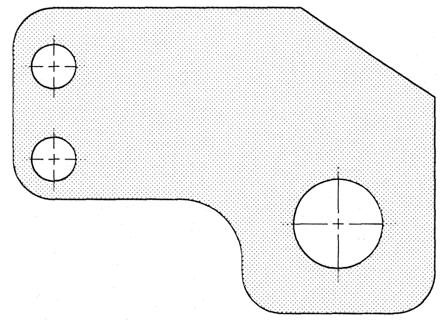


(a) The original part geometry that was created and later dimensioned

(b) The original part after design modifications. The dimensions were updated automatically when modifications were made to the design.

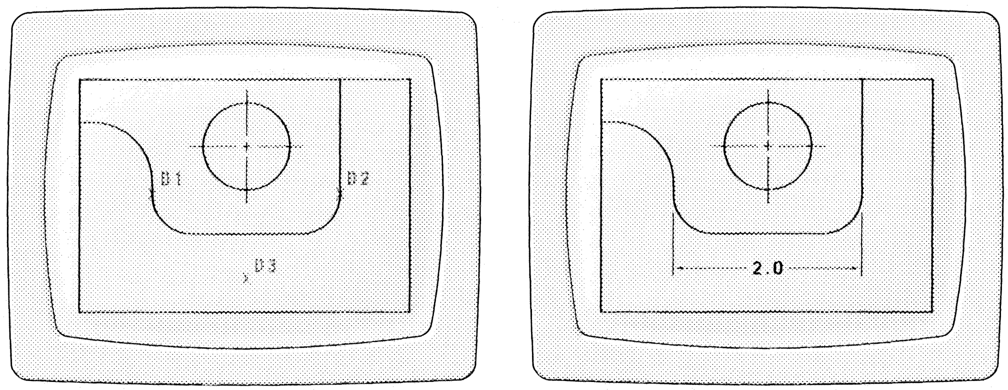
FIGURE 15.106 Associativity

FIGURE 15.107
Dimensioning a One-View Drawing



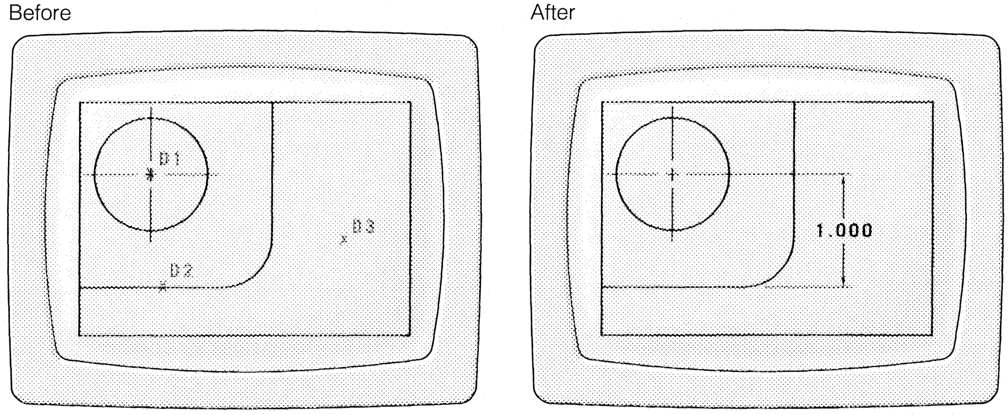
(a) Single-view drawing of a hold-down piece

Before After



(b) Placing a horizontal dimension via the **ZOOM** command to enlarge the area to be dimensioned and then placing the dimension

Before After



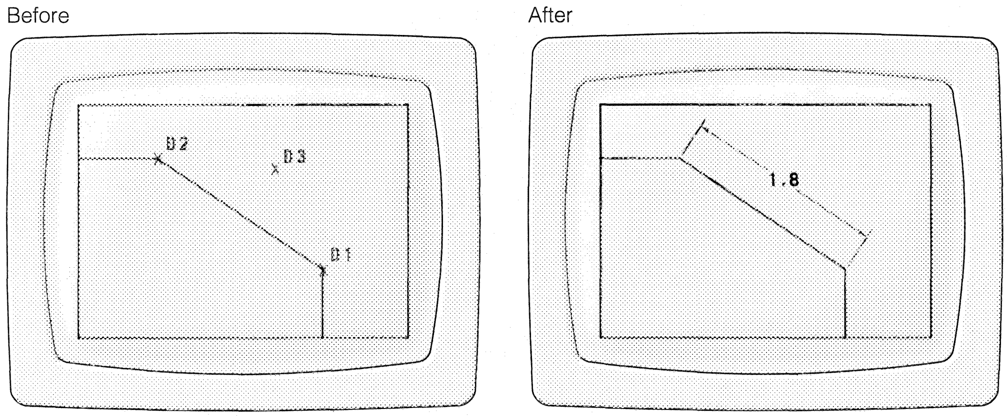
(c) Dimensioning the vertical distance between the hole and the hold-down plate's lower edge

the distance of the angled cut. The **DIMENSION ANGLE** command is used [Fig. 15.107(e)] to show the angle of the cut, instead of an aligned measurement. The diameter of the two small holes is dimensioned next [Fig. 15.107(f)]. The large fillet is then dimensioned with a radius [Fig. 15.107(g)]. Last, the notes are added with a text insertion command [Fig. 15.107(h)]. Figure 15.107(i) shows the completed part with all dimensions placed on the drawing. Figure 15.107 was created with Personal Designer.

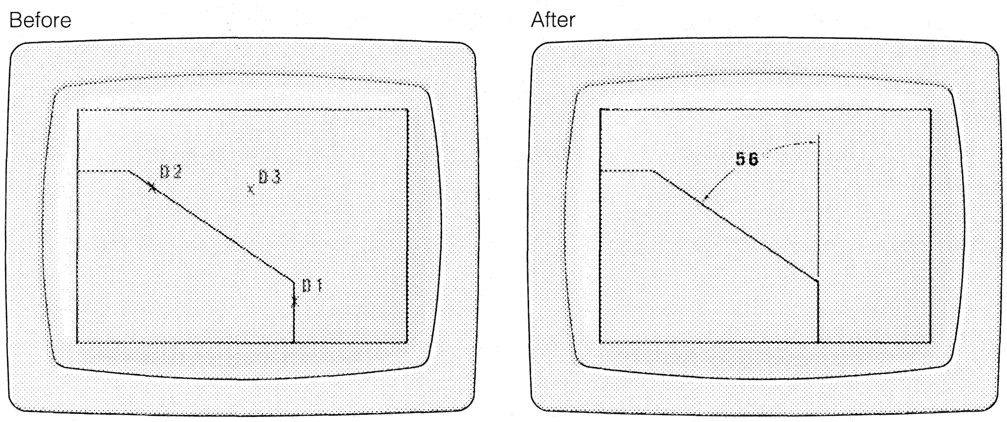
Computervision software was used to create the breaker shown as a 3D model in Figure 15.108(a). The engineer activates the part file and places it in appropriate views [Fig. 15.108(b)]. The engineer then changes the appearance of the model to conform to standard drafting conventions. For example, the hidden lines are changed to dashed lines and the centerlines are added. Last, the part is dimensioned [Fig. 15.108(c)].

Remember, regardless of the method used to draw the part, manual or CAD, the standards and rules of dimensioning apply to every drawing.

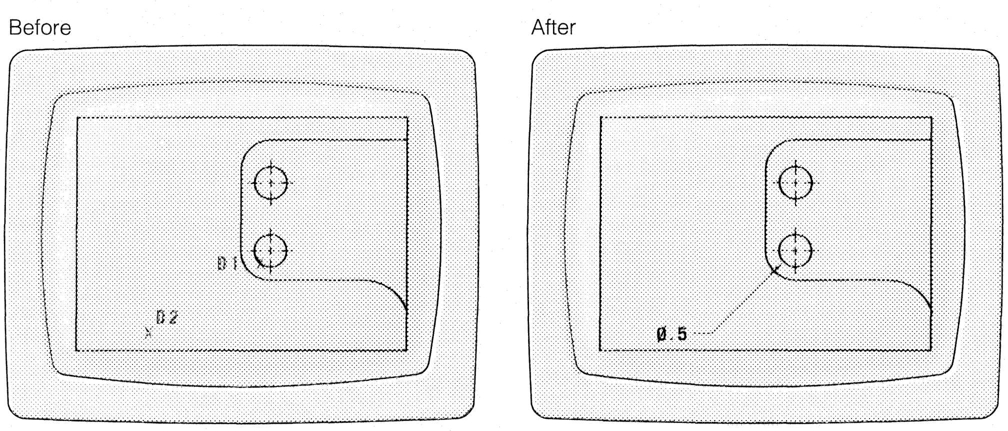
FIGURE 15.107
Dimensioning a One-View Drawing—*Continued*



(d) Dimensioning the angled cut with the **POINT TO POINT** linear dimensioning command



(e) Dimensioning the angled cut using the **DIMENSION ANGLE** command



(f) Dimensioning the small holes

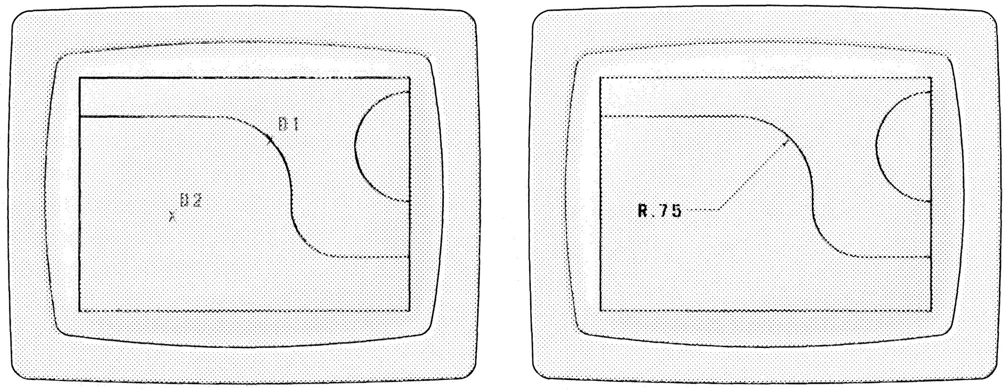
15.16 BASIC DIMENSIONING RULES AND DRAWING CHECKLIST

The following list is provided as a guide to dimensioning any drawing and to check a drawing after it is dimensioned. This list is by no means exhaustive.

1. Give the dimensions that will be used to fabricate the part in the shop or in CNC programming.
2. Make all figures totally legible—a misread dimension can result in an error in fabrication.
3. To help legibility, do not crowd dimensions around the part. Allow space for the dimensions in their proper location by planning for dimensions at the layout stage of the project.

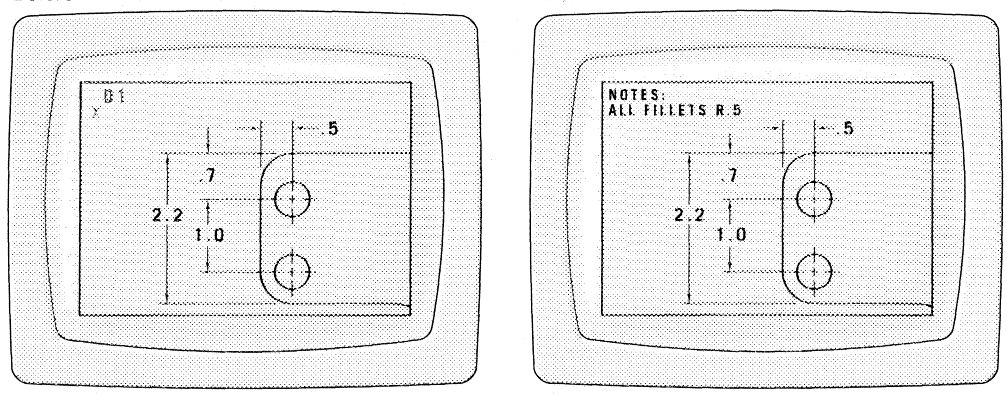
FIGURE 15.107
Dimensioning a One-View Drawing—*Continued*

Before After

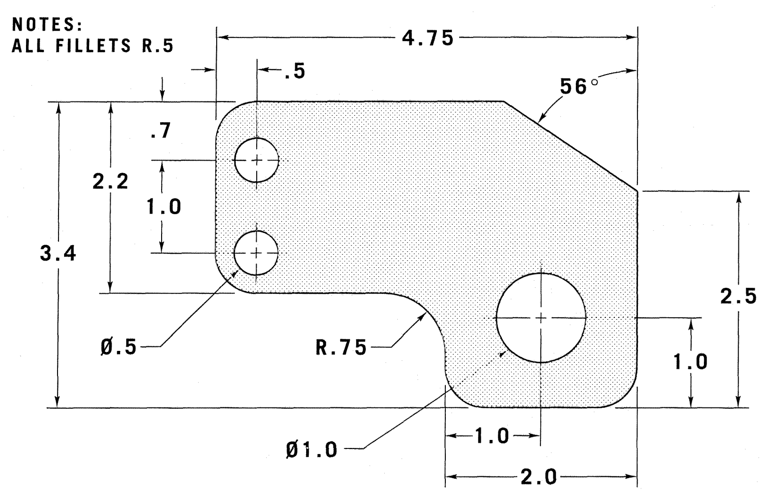


(g) Dimensioning the large radius

Before After

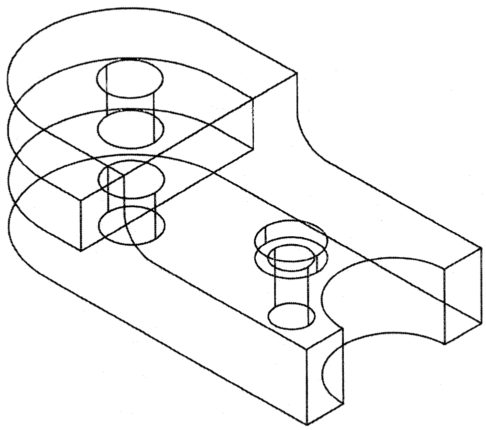


(h) Inserting notes

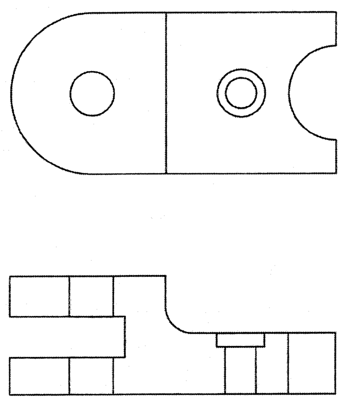


(i) Completed detail of the hold-down plate

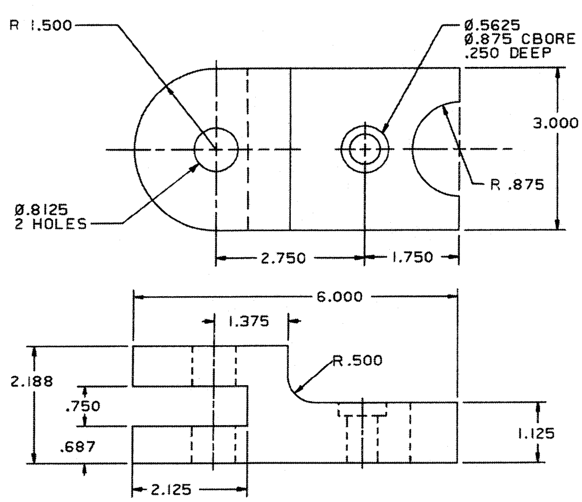
4. Do not dimension on the part unless it is absolutely unavoidable. Use extension lines, and whenever possible keep figures off the views. Place dimensions outside a view.
5. Use proper lettering technique (with guidelines) for all lettering.
6. Dimension the views that show the characteristic shape and prominent features of each portion of the part.
7. Place numbers for dimension values so that they can be read from the bottom of the drawing (aligned), unless the drawing is for one of the construction engineering fields.



(a) 3D wireframe of the breaker



(b) Front and top views



(c) Completed detail

FIGURE 15.108 Dimensioning a 3D Part

8. Do not use a part line as a dimension line. Object lines are used as extension lines only if unavoidable.
9. Locate dimension lines so they do not cross extension lines, by placing the largest dimensions outside of smaller dimensions.
10. Never cross two dimension lines.
11. Place parallel dimensions equally spaced and the numerals staggered, to avoid confusion on the drawing.
12. Give locating dimensions to the centers of circles that represent holes, cylindrical features, bosses, and slots.
13. Group related dimensions on the view where the contour of a feature is prominent.
14. Arrange a series of dimensions in a continuous line, i.e., chain dimensions.
15. Dimension from a machined (finished) surface, a centerline, or a datum (base) line that is easily established during manufacturing.
16. Do not repeat dimensions of the same feature on a drawing (double dimension).
17. Make dimensioning complete so that it is not necessary for manufacturing or inspection to add or subtract to obtain a needed dimension to scale the drawing manually.
18. Provide the diameter of a circle, never the radius.
19. Dimension as required by the production method. Parts with radial ends will have diameters and center-to-center dimensions.
20. Dimension to limit the tolerance buildup and to maintain ease of manufacture.
21. Dimension so that mating parts will fit in the worst case of tolerance buildup on the part.
22. When all dimensions are in inches, generally omit the inch symbol, except for construction drawings.
23. Provide the radius of an arc, and place the abbreviation R before the dimension.
24. If possible, avoid dimensioning to a hidden line.
25. Avoid dimensioning on sectioned areas of the part.
26. Use a note to establish repetitious features of a part, e.g., fillets with the same radius value.

QUIZ

True or False

1. Dimensioning is not as important as a graphically correct drawing.
2. Holes should be called out with a note giving the radius of the hole and its depth.
3. Center drills are used to hold a workpiece between centers on a lathe.
4. Dual dimensioning is used on most drawings in the United States.
5. Simplified methods for showing threads should be used on metric drawings only.
6. The diameter symbol always follows the size dimension.
7. Symbols can be used when calling out counterbores, spotfaces, and counterdrills.
8. Leaders are always drawn radially from a curved feature when placing a local note.

Fill in the Blanks

9. Fractions are used on __________ , __________ , and __________ drawings in the United States.
10. The ability of the system to update dimensions automatically after design changes have been made is called __________ .
11. Angles can be called out as ________ angles, or ________ , and __________ .
12. SR is used to define __________ and __________ .
13. Counterbores can be specified by the symbol or ________ .
14. __________ dimensions are enclosed within parentheses.
15. Chamfers can be specified by __________ or __________ .
16. There are two types of knurling: __________ and __________ patterned.

Answer the Following

17. Describe the difference between the dimensioning process of a 3D CAD system and that of a 2D CAD system.
18. What are the four types of linear dimensions? Describe the process of placing each on a part.
19. Describe four methods for calling out a taper.
20. Describe the process of geometric breakdown of a part.
21. Why are mating parts and mating dimensions important when dimensioning a part?
22. What is the difference between a radial and a diameter dimension, and when should each be used?
23. What is a finish mark, and why is it important when dimensioning a part?
24. Describe what notes are used for on a drawing. What is a local note, and what is a general note?

PROBLEMS

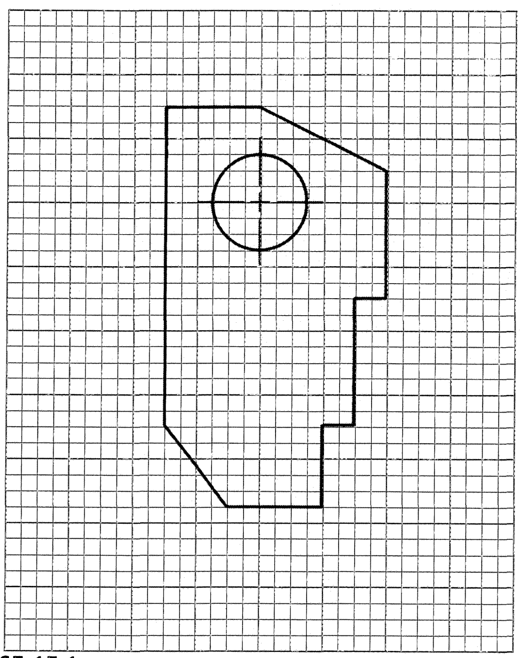
Your instructor can assign any of the figures presented in this chapter as problems. For every problem, redraw the part and dimension using the most recent ANSI Y14.5M standards. Other industry drawings and problems at the end of each chapter throughout the text can also be assigned by the instructor.

EXERCISES

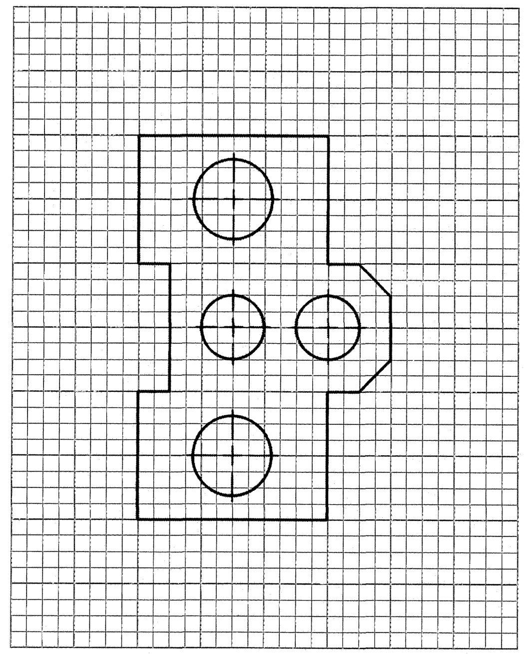
Exercises may be assigned as sketching, instrument, or digitizing projects. Transfer the given information to an "A"-size sheet of .25 in. grid paper. Complete all views and solve for proper visibility, including centerlines, object lines, and hidden lines. Exercises that are not assigned by the instructor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

After Reading the Chapter Through Section 15.5.5 You May Complete the Following Four Exercises

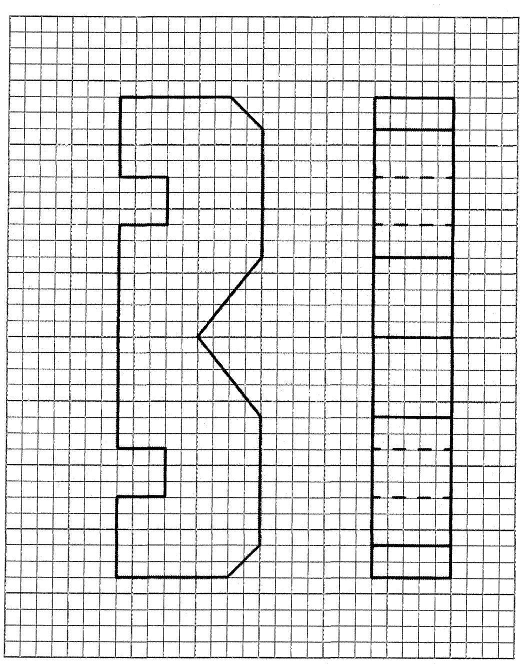
Exercise 15.1 Dimension the .25 in. thick one-view part completely.
Exercise 15.2 Dimension the .125 in. thick aluminum plate completely.
Exercise 15.3 Dimension the two-view part as needed.
Exercise 15.4 Show the proper placement of all dimensions on appropriate views.



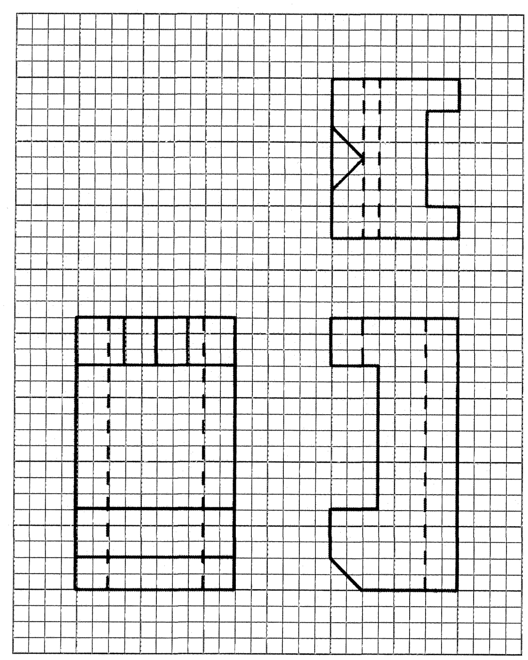
EXERCISE 15.1



EXERCISE 15.2



EXERCISE 15.3



EXERCISE 15.4

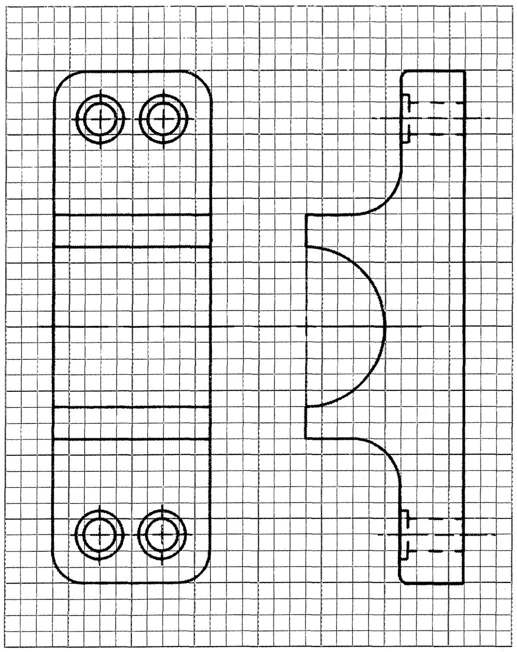
After Reading the Chapter Through Section 15.6.3 You May Complete the Following Four Exercises

Exercise 15.5 Completely dimension the part as required. Use symbology to call out the spotfaced holes. Place finish marks on the machined faces, and dimension accordingly.

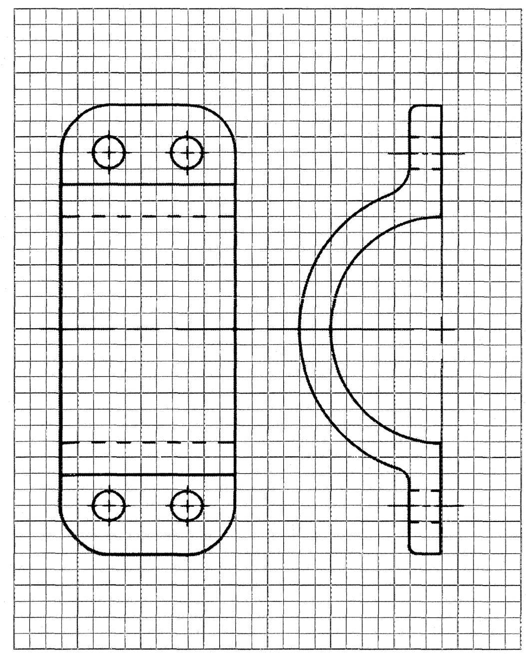
Exercise 15.6 Dimension the part completely. The bottom surface is machined.

Exercise 15.7 Dimension the two-view part.

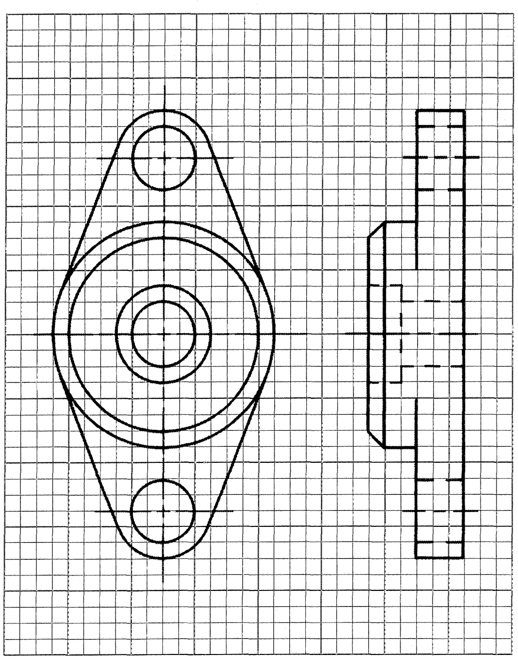
Exercise 15.8 Dimension the three views of the part. The bottom surface, the left side surface, and the boss are the only finished surfaces. Add appropriate fillets and rounds for the cast surfaces (top and around the boss). Place basic finish marks on machined surfaces.



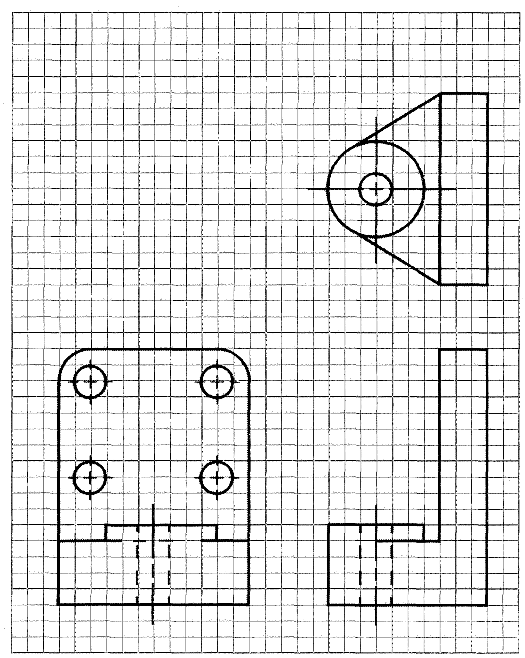
EXERCISE 15.5



EXERCISE 15.6



EXERCISE 15.7



EXERCISE 15.8

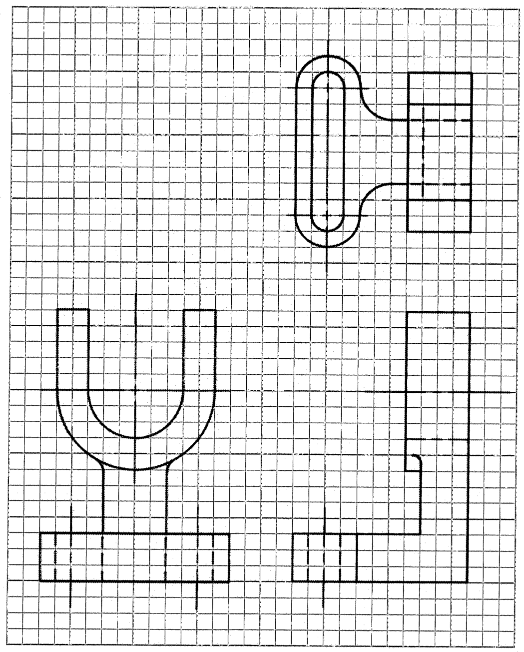
After Reading the Chapter Through Section 15.8.3 You May Complete the Following Four Exercises

Exercise 15.9 Dimension the cast part completely. Add appropriate fillets and rounds to cast surfaces. Place finish marks on machine surfaces. The left surface and bottom surface are machined, along with the upper **U**-shaped surface. All other surfaces are cast.

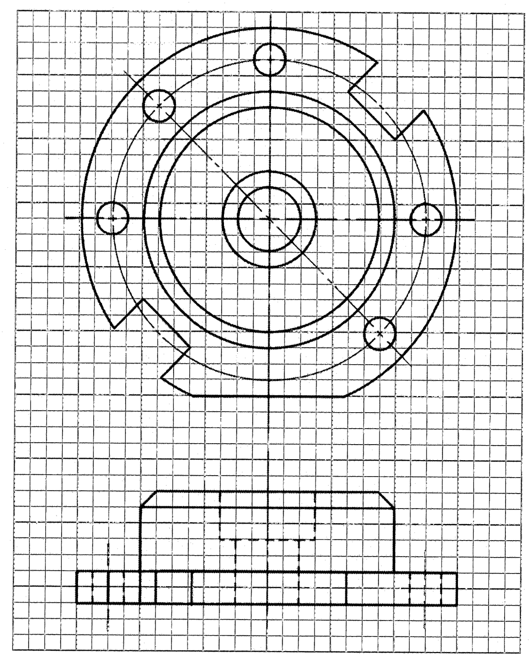
Exercise 15.10 Because of space limitations, dimension only the hole pattern and holes (call out the bolt circle), the slots, and the chamfer. Make sure all views are visually correct.

Exercise 15.11 Dimension the part completely. Complete the views for proper visibility. Call out the knurling with a note. The small hole goes through to the center hole only.

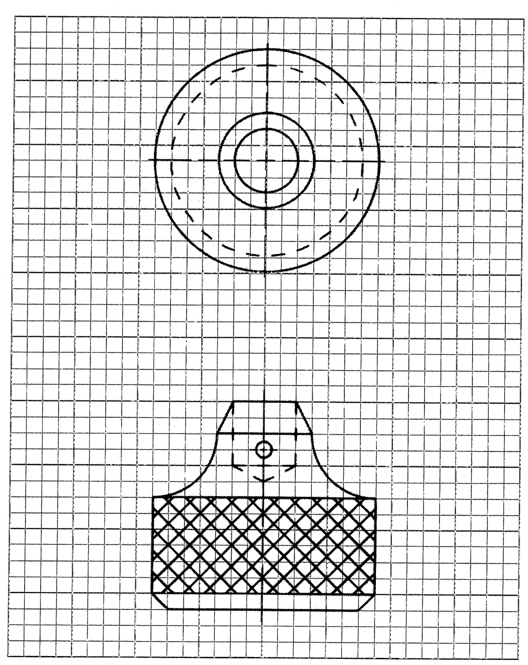
Exercise 15.12 Because of limited space, dimension only the hole pattern, the slot, and the counterdrilled holes. Use symbology to call out the holes.



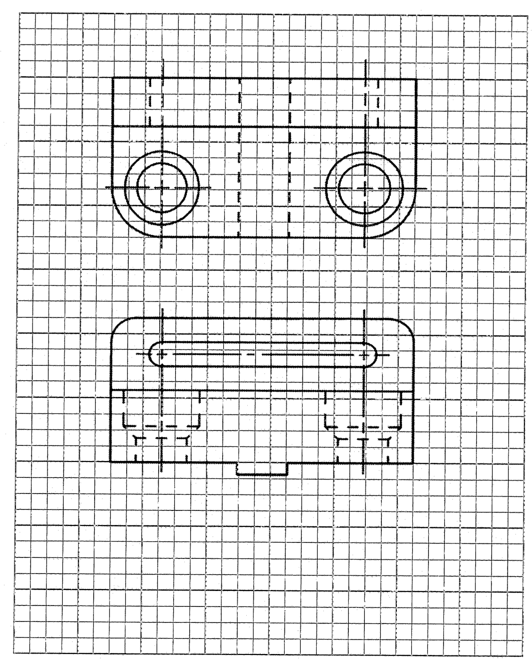
EXERCISE 15.9



EXERCISE 15.10



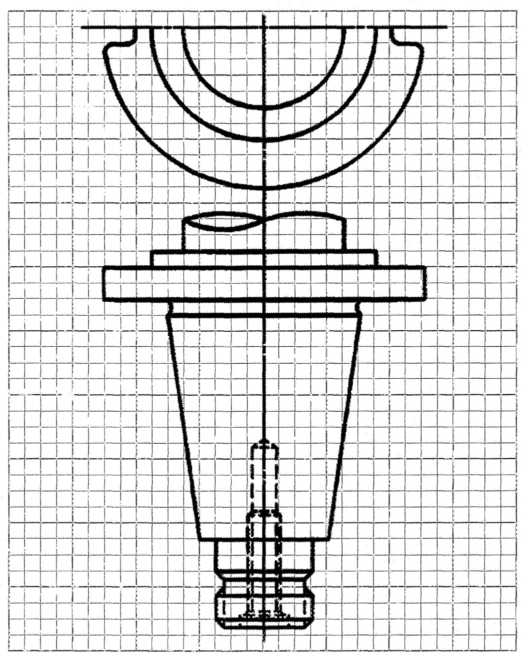
EXERCISE 15.11



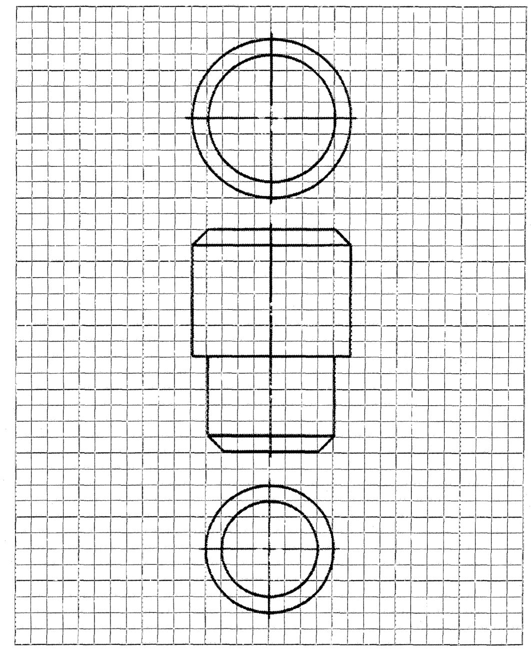
EXERCISE 15.12

After Reading the Chapter Through Section 15.10.5 You May Complete the Following Four Exercises

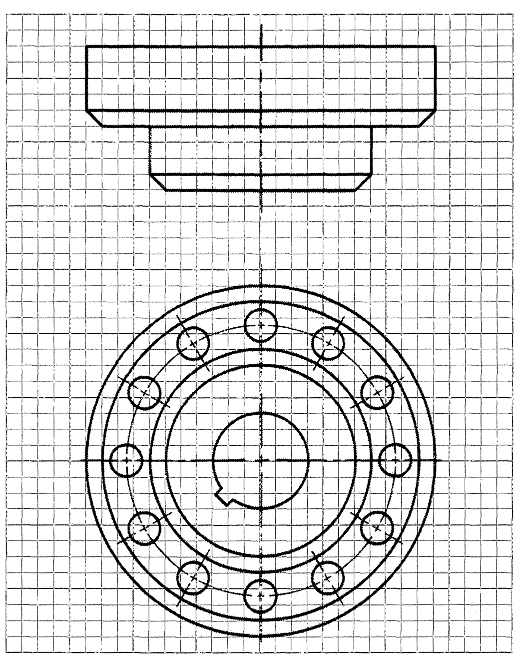
Exercise 15.13 Dimension the taper with a callout. Because of space limitations, dimension only the lateral length dimensions, not the diameter dimensions of the rest of the part.
Exercise 15.14 Place a #4 center hole for a center drill on both ends of the workpiece. Dimension the whole part, and call out the center drill with a note. Complete the views.
Exercise 15.15 Complete the side view. Dimension the hole pattern, the hole sizes, and the keyway. Look up the proper keyway size and type for the given shaft diameter.
Exercise 15.16 Complete the side view of the part. Dimension the holes with a callout using symbology. Call out the keyway based on the shaft diameter. Dimension only one "ear" of the part.



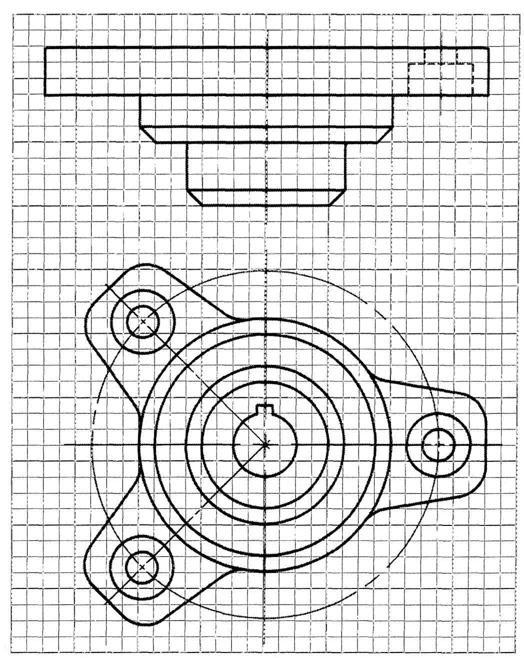
EXERCISE 15.13



EXERCISE 15.14



EXERCISE 15.15



EXERCISE 15.16