

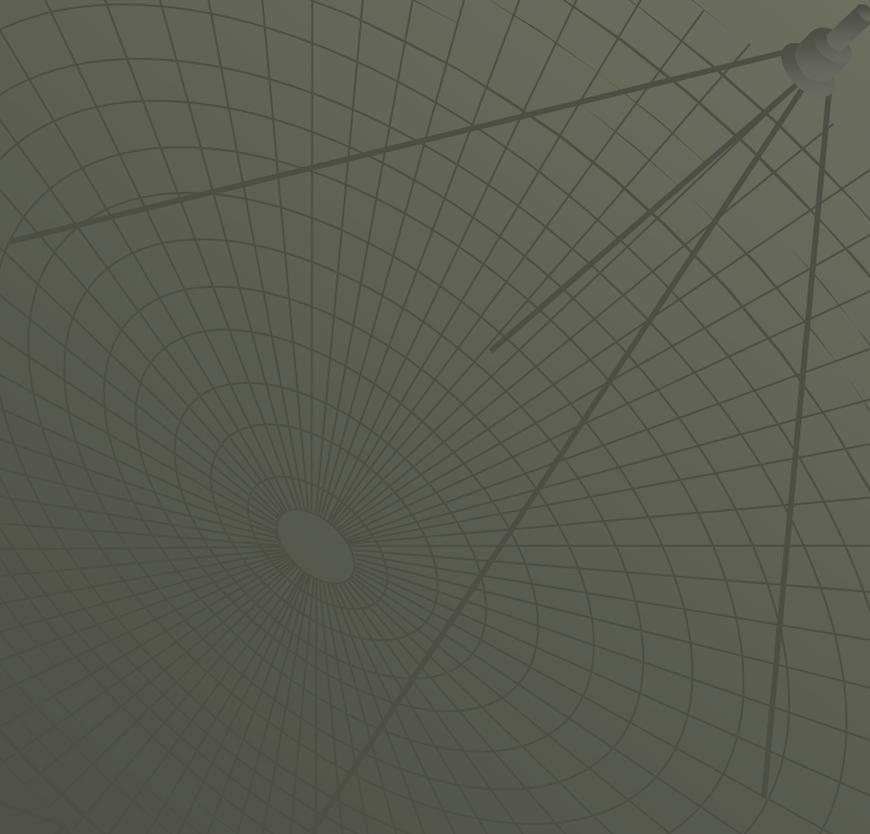
DIMENSIONING

Introduction

- Rule of Thumb
- meter and yard
- International System of Units (SI system) → ISO standards
- American National Standards Institutes (ANSI) → ANSI standards

9.2 Size Description

The need for **interchangeability** of parts is the basis for modern **dimensioning** (see ANSI / ASME Y14.5M-1994).



9.3 Scale of Drawing

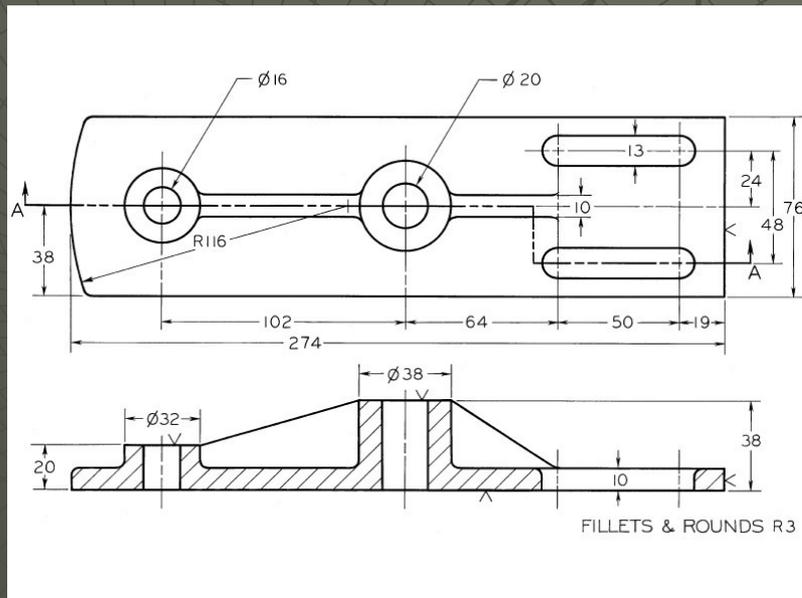
Drawings are usually made to scale, which is indicated in the title block. If any part of the drawing is **not to scale** (NTS), then it is clearly mentioned in the drawing.



9-4 Learning to Dimensioning

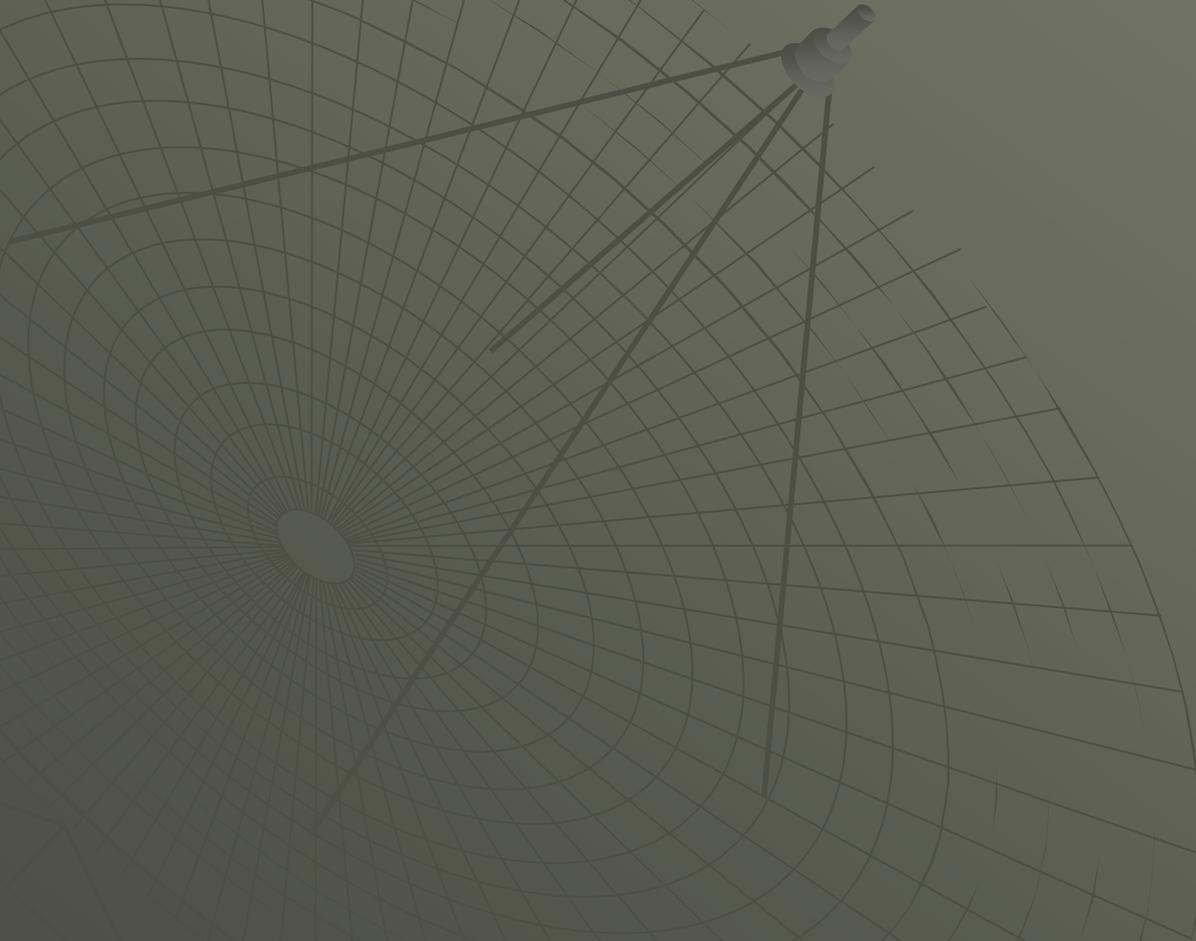
The ability to dimension properly requires the following:

1. **Technique of dimensioning:** the standard for appearance of lines, the spacing of dimensions, the size of arrowheads, and so on allow others to interpret your drawing. A typical dimensioned drawing is shown below. Note the strong contrast between the visible lines of object and the thin lines used for dimensions.



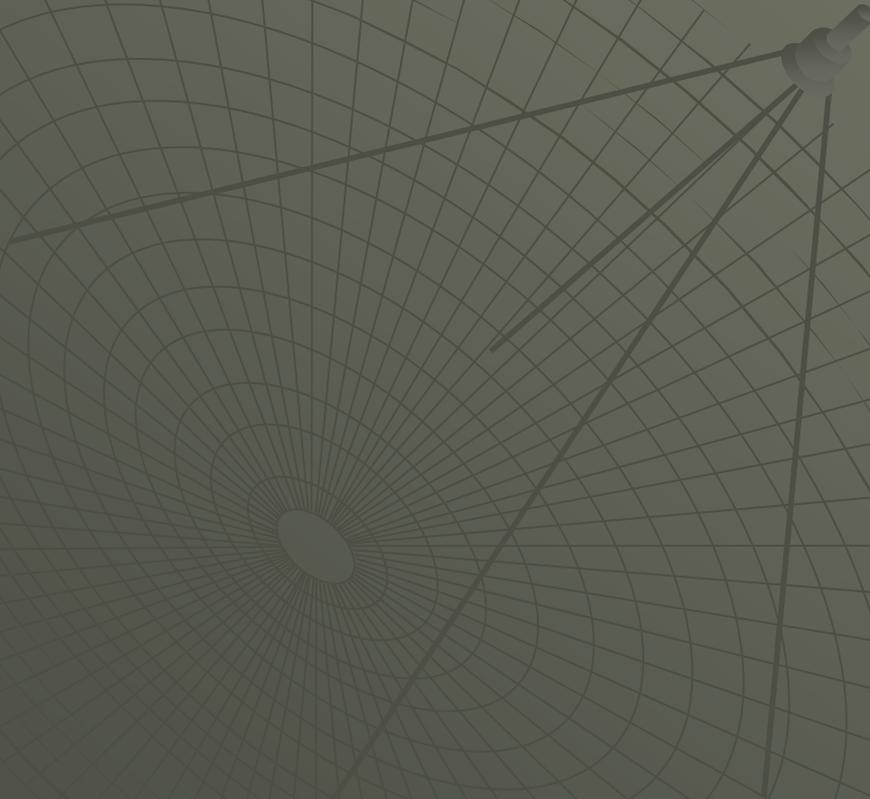
(cont)

2. *Placement of dimensions*: Use logical placement for dimensions.
3. *Choice of dimensions*. The dimensions chosen by designer how the design is manufactured.



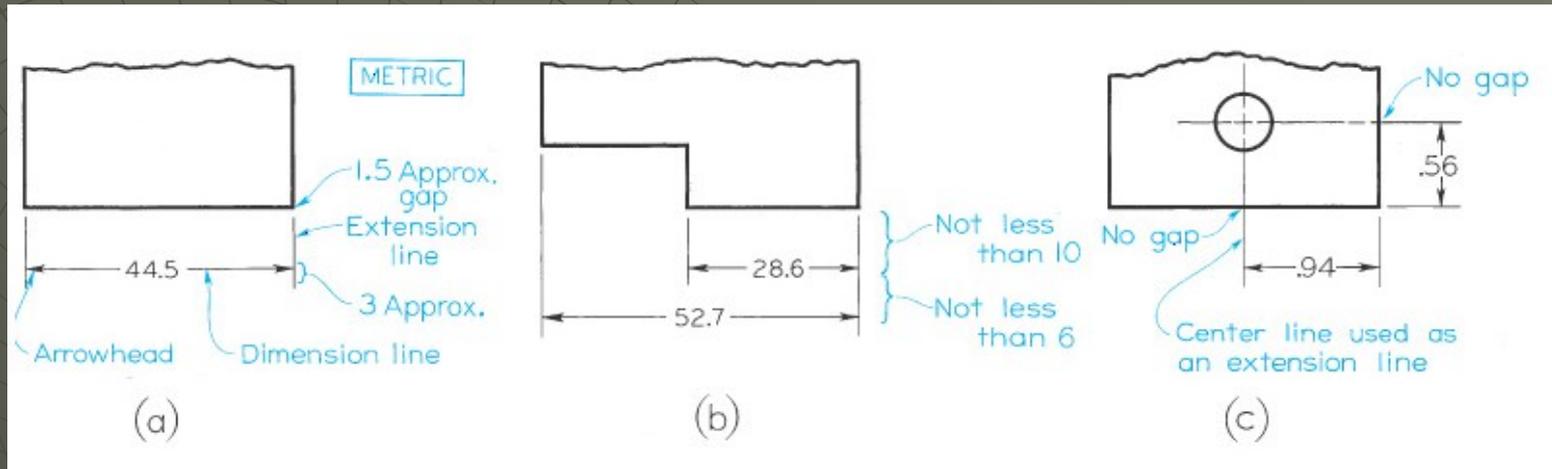
9.5 Tolerance

Tolerance is the total amount that the feature on the actual part is allowed to vary from the specified dimension. This will be discussed in the next chapter in details.



9-6 Lines Used in Dimensioning

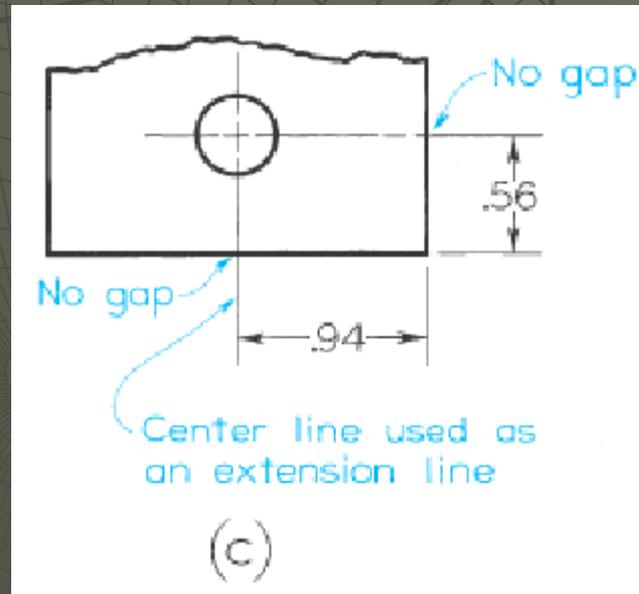
A **dimension line** is a thin, dark, solid line terminated by arrowheads, indicating the direction and extent of a dimension, as shown in Fig. (a)



As shown in Fig. (b), the dimension line nearest the object outline should be spaced at 10mm away. All other parallel dimension lines should be 6 mm apart, and more if space is available.

(cont)

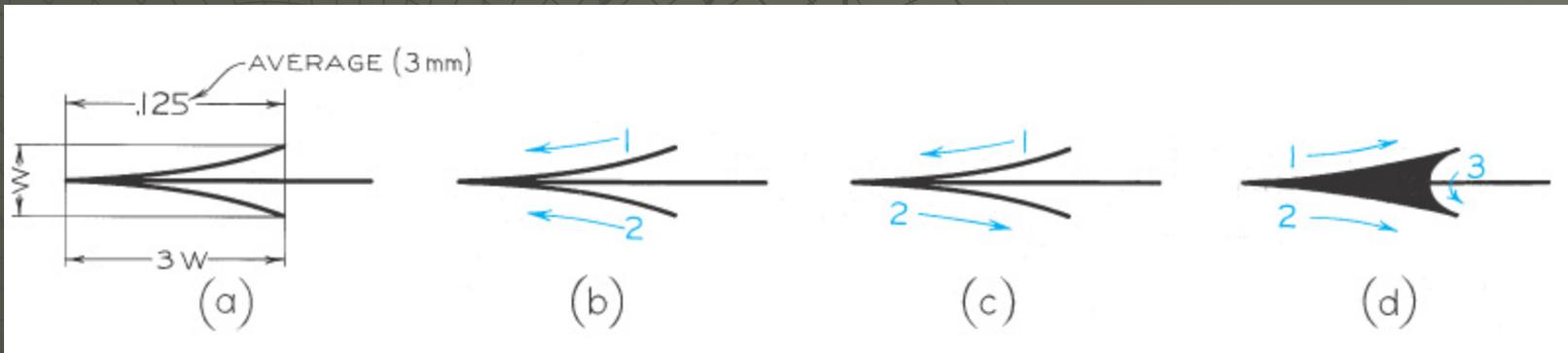
An extension line is a thin, dark, solid line that extends from a point on the drawing to which a dimension refers. The dimension line meets the extension lines at right angles, except in special cases. A gap of about 1.5 mm should be left where the extension line would join the object outline. The extension line should extend about 3 mm beyond the outermost arrowhead.



9-7 Arrowheads

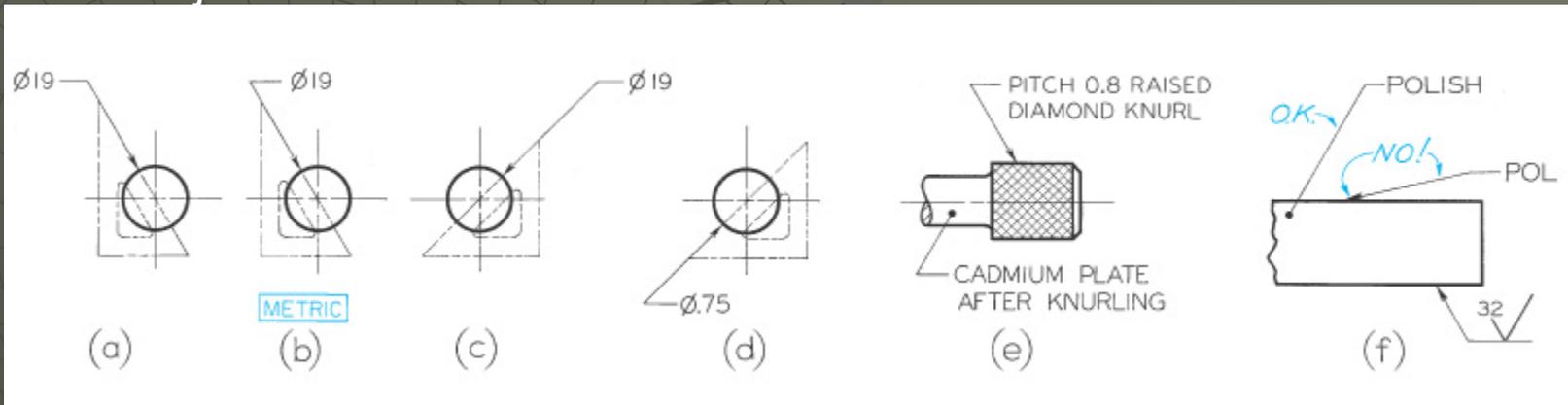
Arrowheads, shown below, indicate the extent of dimensions. They should be uniform in size and style throughout the drawing, not varied according to the size of the drawing or the length of dimensions.

Sketch arrowheads freehand so that the length and width have a ratio of 3:1.



9-8 Leaders

A **leader** is a thin, solid line directing attention to a note or dimension and starting with an arrowhead or dot. Use an arrowhead to start the leader when you can point to a line in the drawing, such as the edge of a hole; use a dot to start the leader when locating something within the outline of the object.

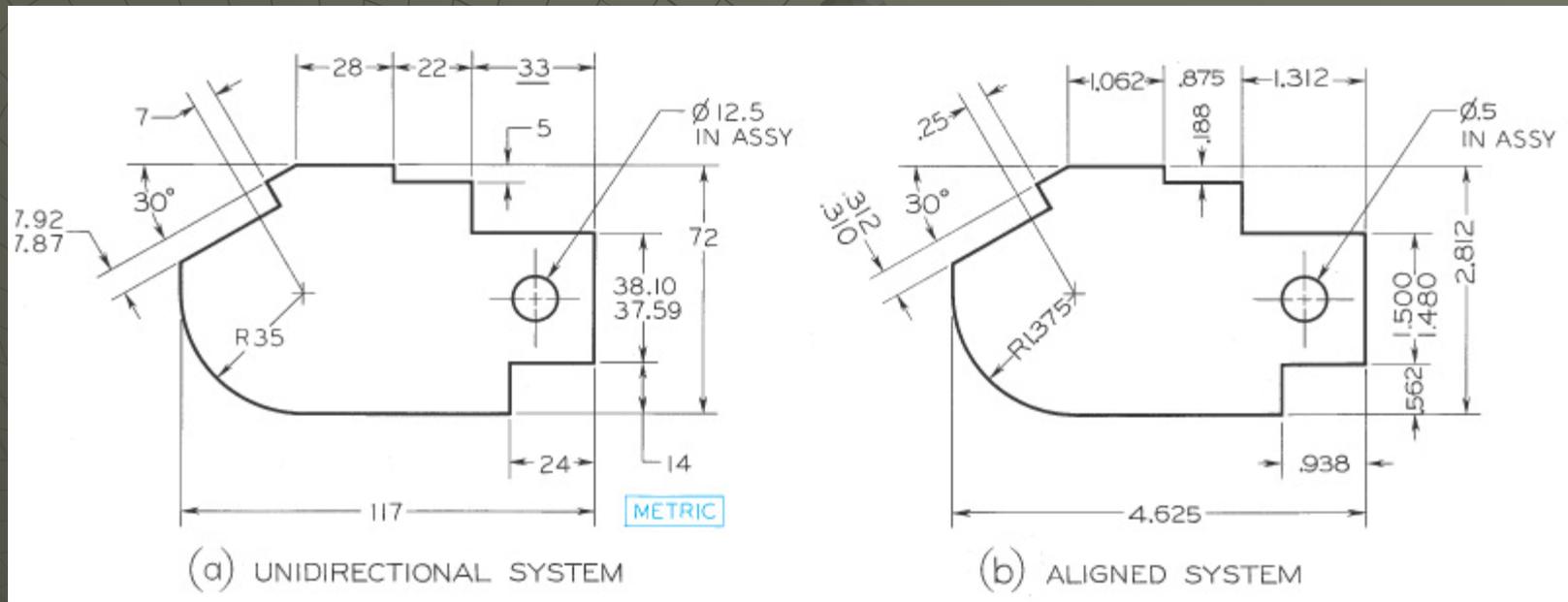


For the best appearance, make leaders:

- Near each other and parallel
- Across as few lines as possible

9-9 Direction of Dimension Figures

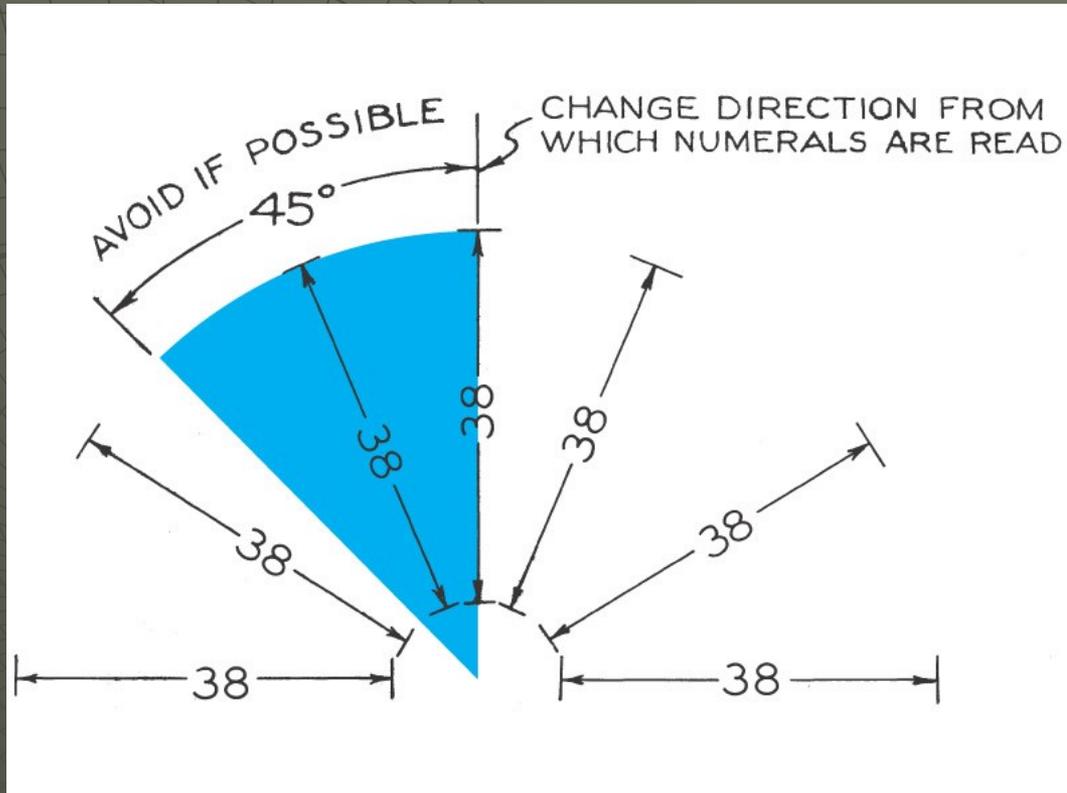
Figure below shows the two systems of reading direction for dimension values. In the preferred **unidirectional system**, approved by ANSI, all dimension figures and Notes are lettered horizontally and are read from the bottom of the sheet.



In the **aligned system**, all dimension figures are aligned with dimension lines so that they may be read from the bottom or right side of the sheet.

(cont)

Dimension lines in aligned system should not run in the directions included in the shaded area of figure below if avoidable.



9-10 Fractional, Decimal, and Metric Dimensions

Drawing may be dimensioned entirely with whole numbers and common fractions, or entirely with decimals, or with a combination of the two. However, more recent practice is to use the decimal-inch system and the metric system as recommended by ANSI.

Millimeters and inches in decimal form can be added, subtracted, multiplied, and divided more easily than can fractions.

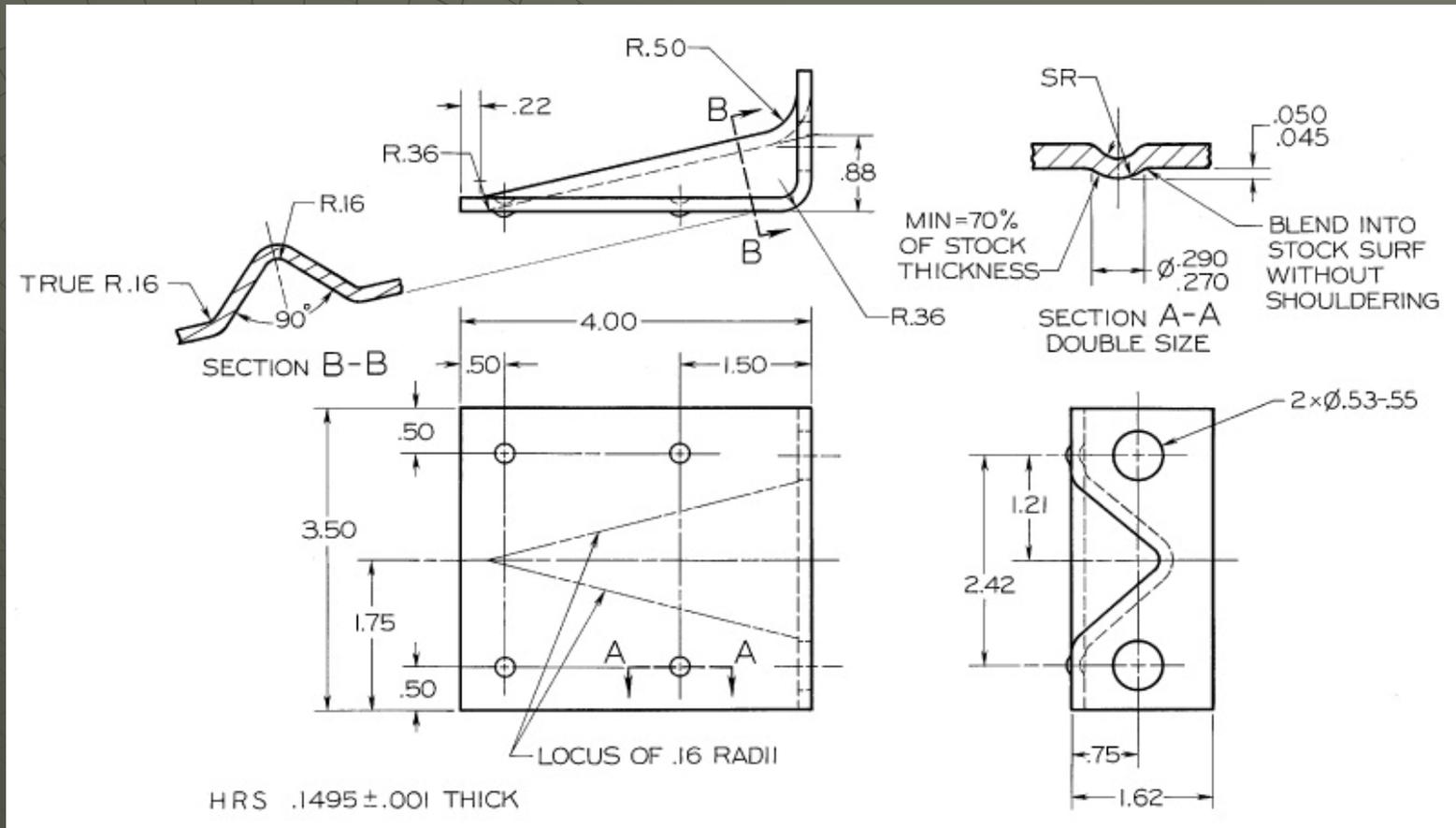
9-11 Decimal Systems

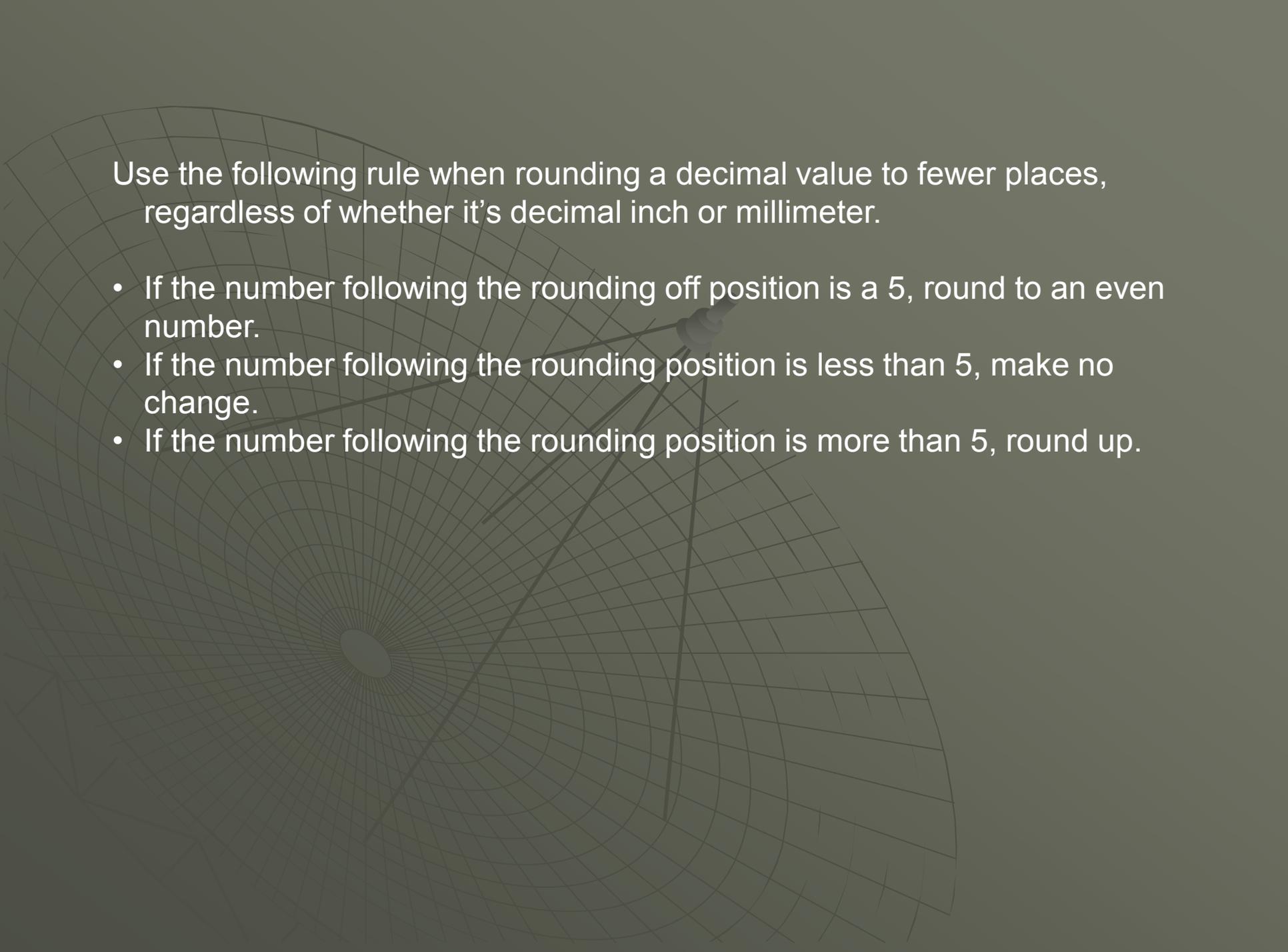
Complete decimal dimensioning uses decimals for all dimensions except where certain commodities, such as pipe and lumber, are identified by standardize nominal designations. In these systems, 2-place inch and 1-place millimeter decimals are used when a common fraction has been regarded as sufficiently accurate.

Combination dimensioning uses decimals for all dimensions except for the nominal sizes of parts or features, such as bolts, screw threads, keyseats, or other items that use standard fractional designations.

(cont)

A typical example of the use of the complete decimal-inch system is shown below.



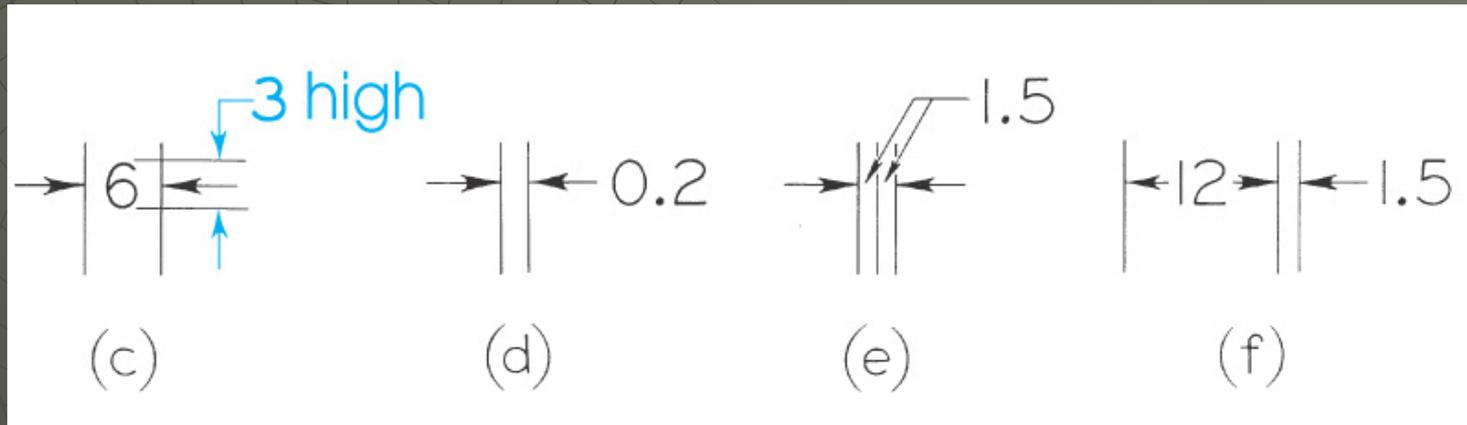


Use the following rule when rounding a decimal value to fewer places, regardless of whether it's decimal inch or millimeter.

- If the number following the rounding off position is a 5, round to an even number.
- If the number following the rounding position is less than 5, make no change.
- If the number following the rounding position is more than 5, round up.

9-2 Dimension Values

Use the methods shown in figure below, when there is not enough room for both the figure and dimension line inside the extension lines.

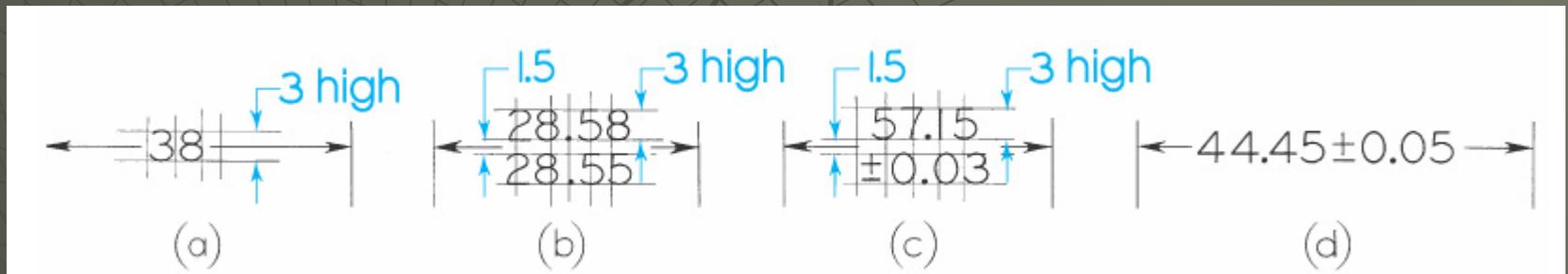


Make the decimal points bold, allowing ample space. Where the metric dimension is a whole number, do not show either zero or a decimal point. Where the metric dimension is less than 1mm, a zero precedes the decimal point.

(cont)

Where the dimension exceeds a whole number by a fraction of 1mm, the last digit to the right of decimal point is not followed by a zero except when expressing tolerance.

Figure below from (a) to (d) shows examples of correct metric dimension values.



(cont)

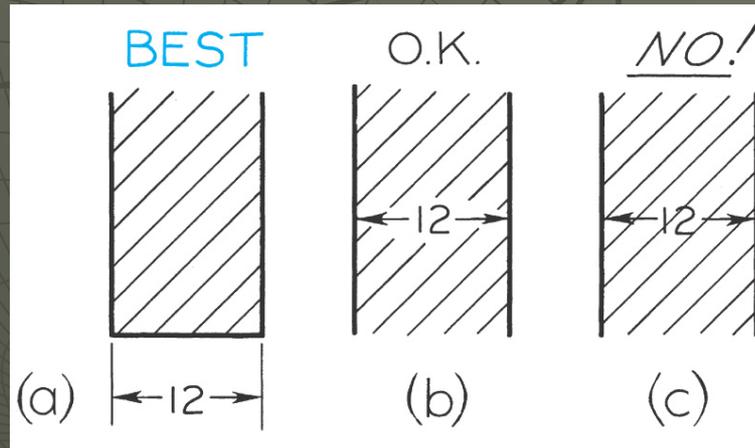
Where the decimal-inch is used on drawing, a zero is not used before the decimal point of values less than 1 inch. The decimal-inch dimension is expressed to the same number of decimal places as its tolerance.

Zeros are added to the right of the decimal point as necessary. Correct decimal dimension values are shown below.



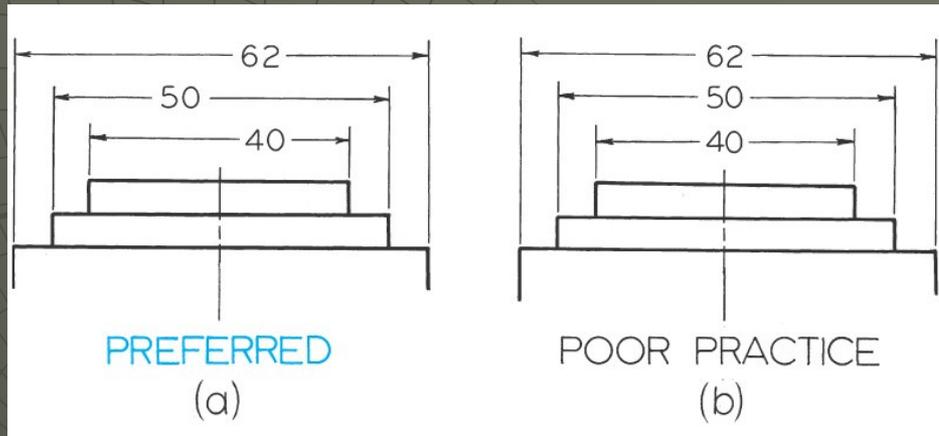
(cont)

Never letter a dimension value over any line on the drawing; if necessary, break the line. Place dimension values outside sectioned is possible. When a dimension must be placed on a sectioned area, leave an opening in the section lining for the dimension figure.



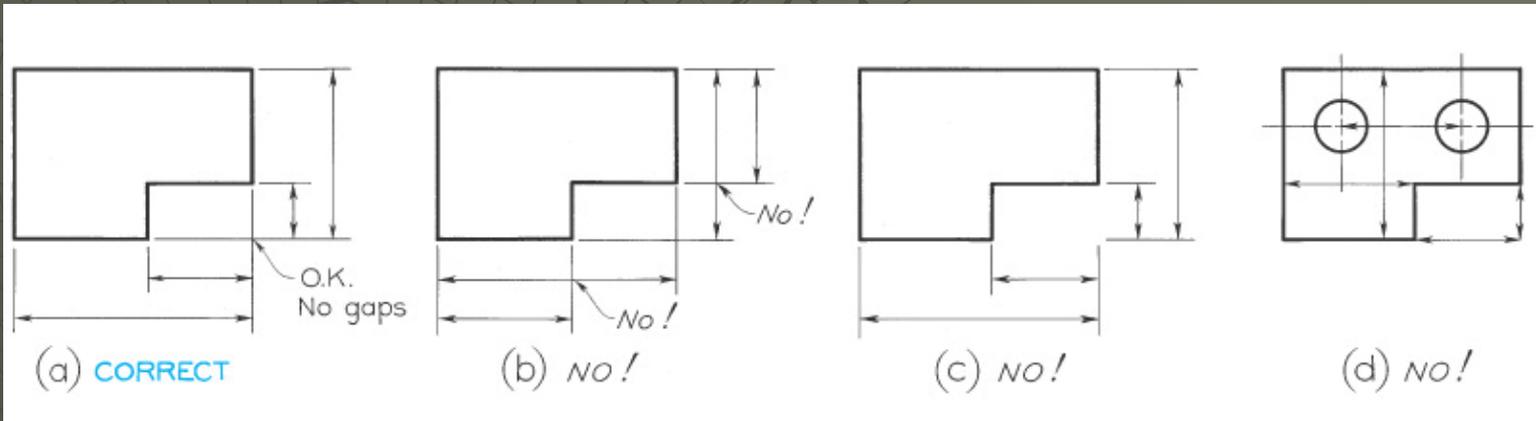
(cont)

In a group of parallel dimension lines, the numerals should be staggered, as show below (a), and not stacked up one above the other, as in (b)



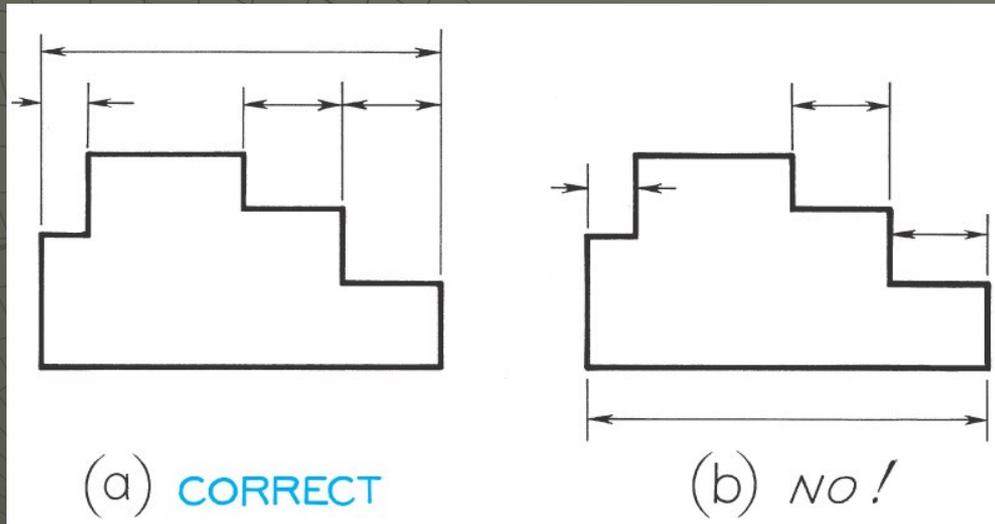
9-14 Placement of Dimension and Extension Lines

The correct placement of dimension lines and extension lines is shown in (a). Dimension lines should not cross extension lines as in (b). It is perfectly OK for extension lines to cross one another, but they should not be shortened like those shown in (c). Dimension lines should not coincide with or continue any line of drawing, as in (d).



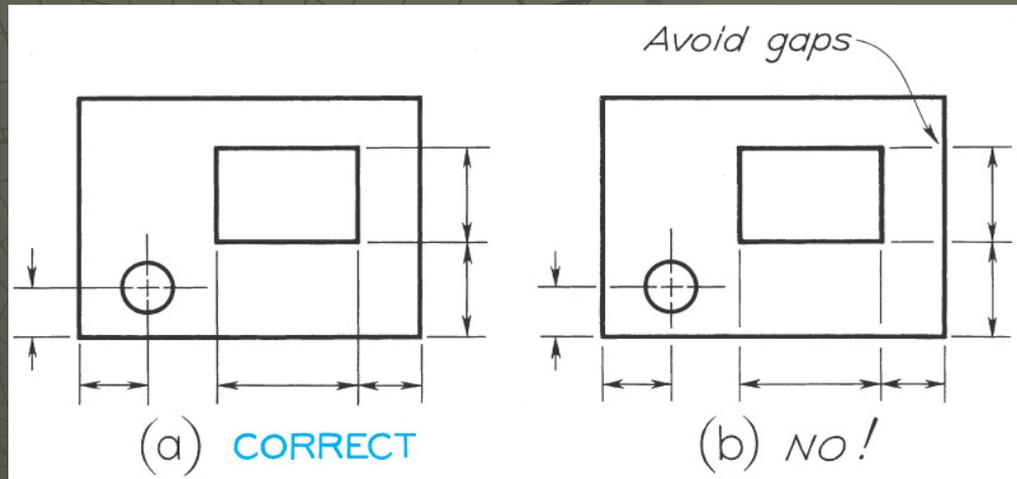
(cont)

Dimension should be line up and grouped together as much as possible, as shown below.



(cont)

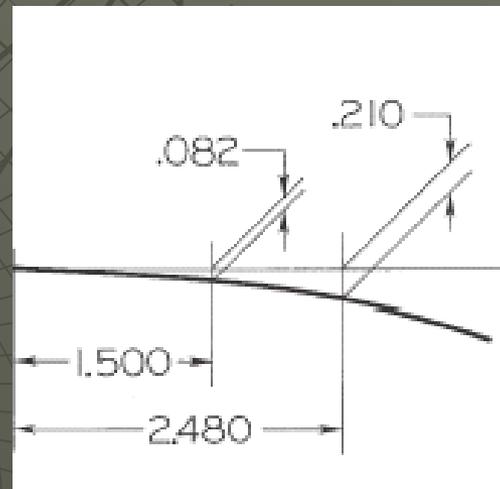
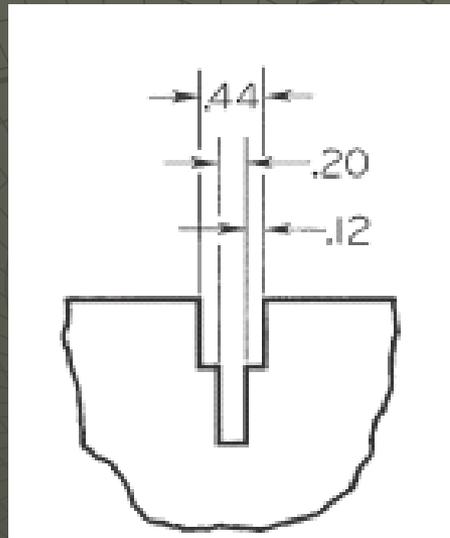
Extension lines and centerlines can cross visible lines of the object to locate dimensions of interior features. Do not leave a gap in either line when crossing object lines, as shown in (b).



(cont)

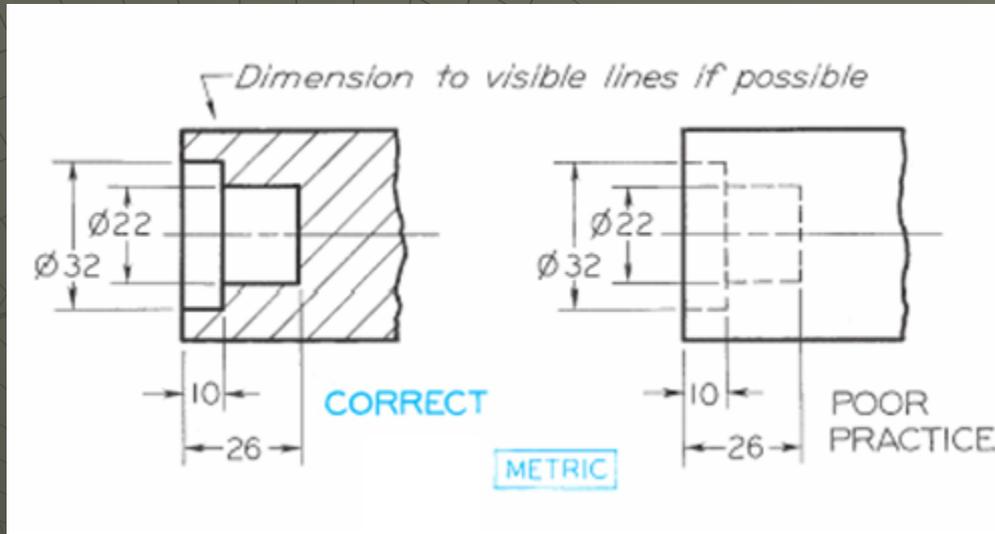
To fit dimensions into a crowded area, you may leave gaps in extension lines near arrowheads so the dimensions show clearly, as example below.

Dimension lines are usually drawn at right angles to extension lines, unless showing them otherwise improves clarity, as shown below.



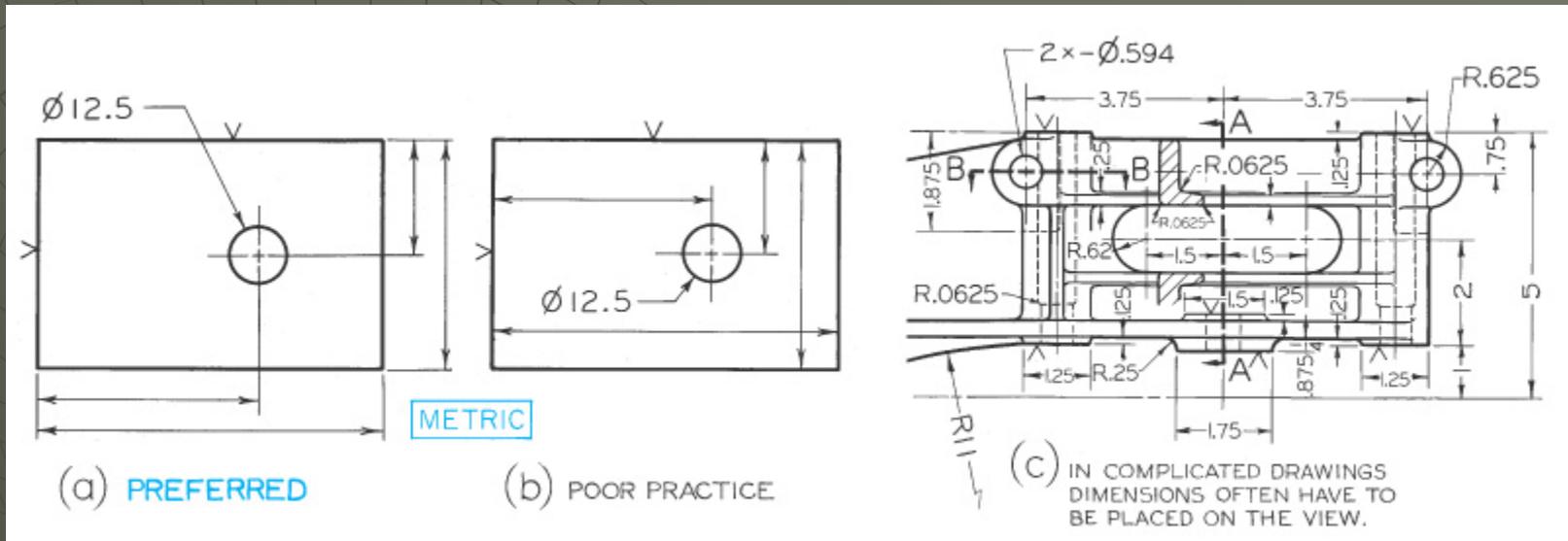
(cont)

Avoid dimensioning to hidden lines.



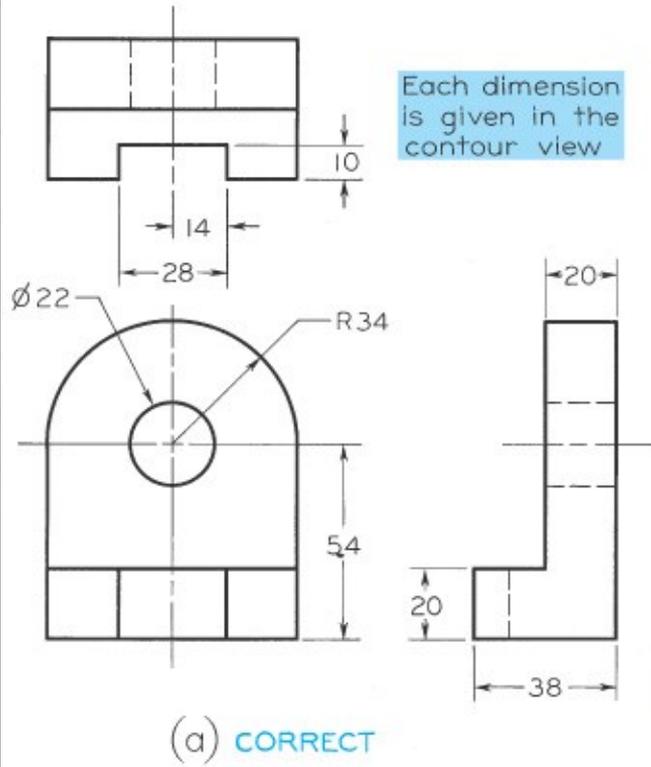
(cont)

Dimensions should not be placed on a view unless doing so prompts the clarity of the drawing, as shown below.

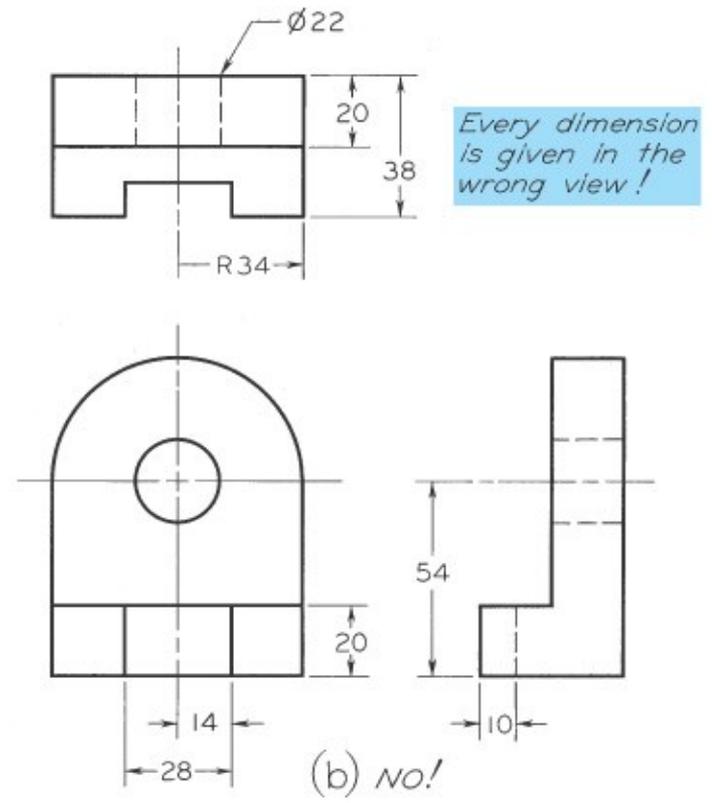


When a dimension must be placed in a hatched area or on the view, leave an opening in the hatching or breaks in the lines for the dimension values, as shown in (b) and (c).

(cont)

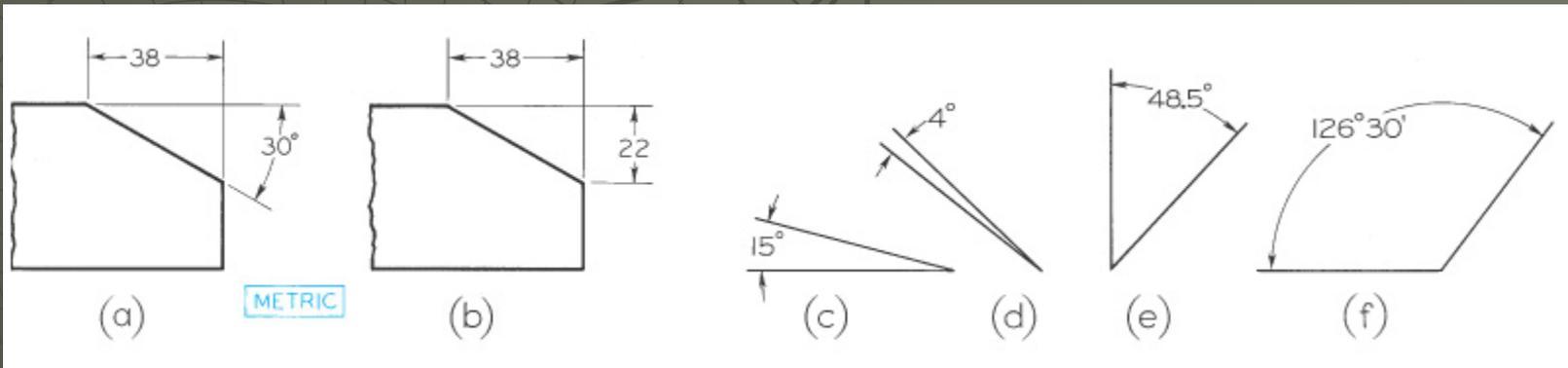


METRIC



9-15 Dimension Angles

We should dimension angles by specifying the angle in degrees and a linear dimension as shown in (a). We can also give coordinate dimensions for two legs of a right triangle, as shown in (b). Methods of indicating angles are shown on figures.



9-16 Dimension Arcs

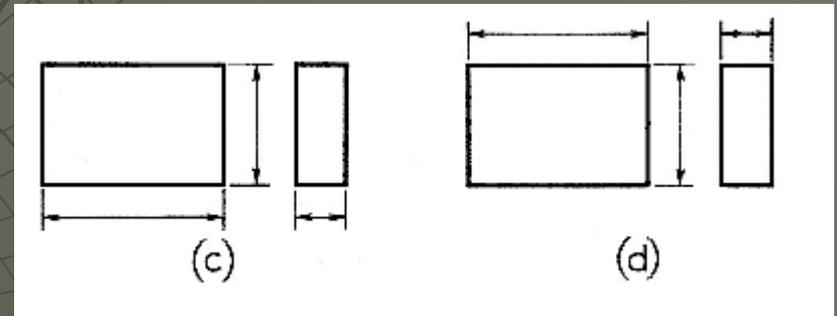
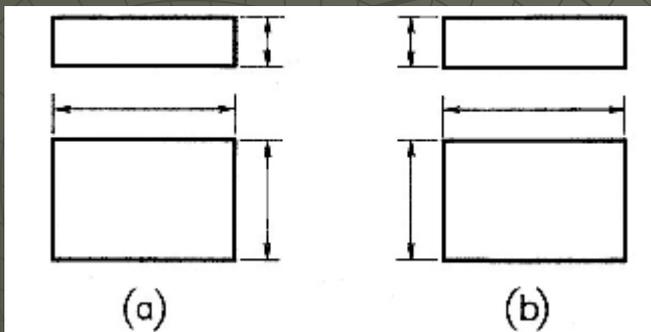
A circular arc is dimensioned in the view where you see its true shape by giving the values for its radius preceded by the abbreviation R.

9-17 Fillets and Rounds

Individual fillets and rounds are dimensioned like other arcs. If there are only a few and they are obviously the same size, giving one typical radius is preferred. However, fillets and rounds are often numerous on a drawing, and they are usually are some standard size, such as metric R3 and R6, or R.125 and R.25 when decimal-inch.

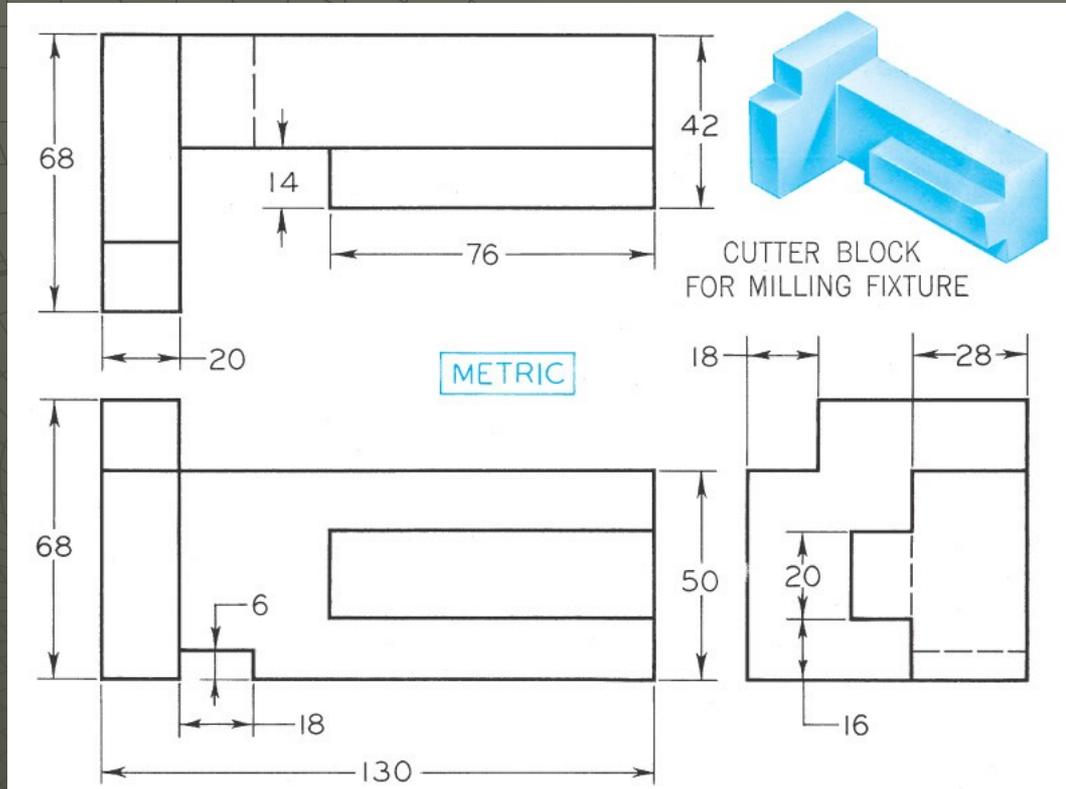
9-19 Size Dimensions: Prisms

The right rectangular prism is probably the most common geometry shape. Front and top views are dimensioned as (a) and (b) below. The height and width are usually given in the front view, and depth in the top view. Front and side views should be dimensioned as (c) and (d) below.



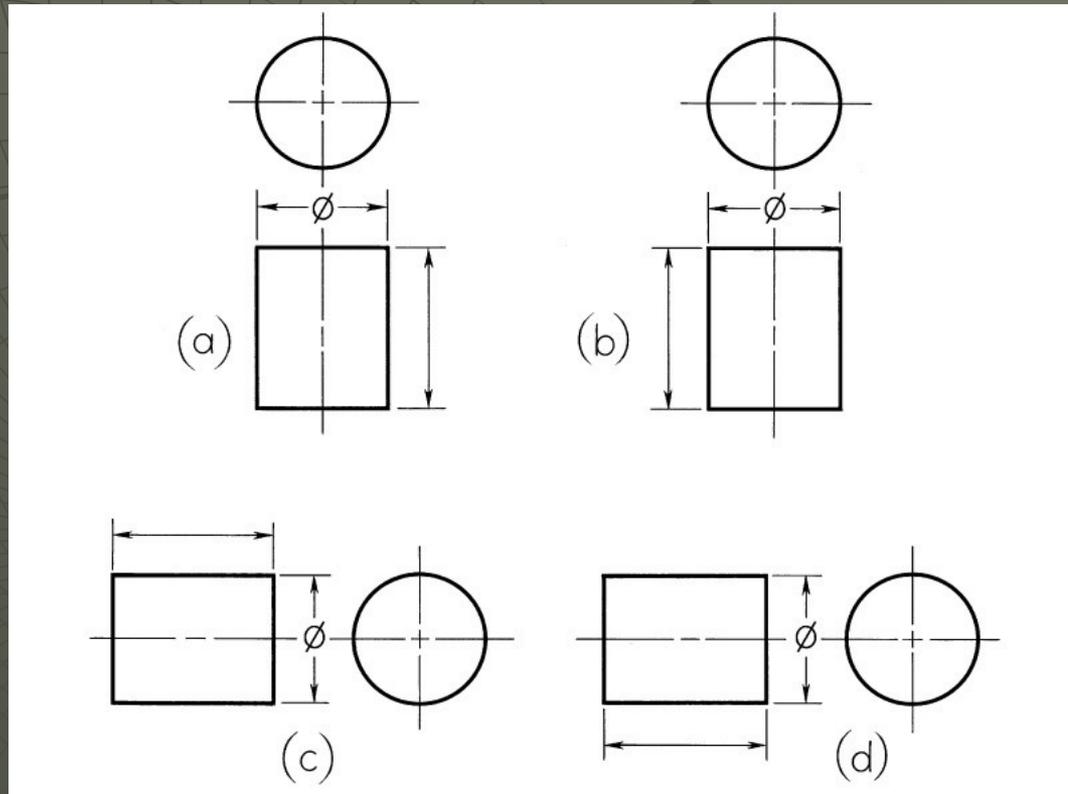
(cont)

An example of size dimensions for a machine part made entirely of rectangular prisms



9-20 Size Dimensions: Cylinders

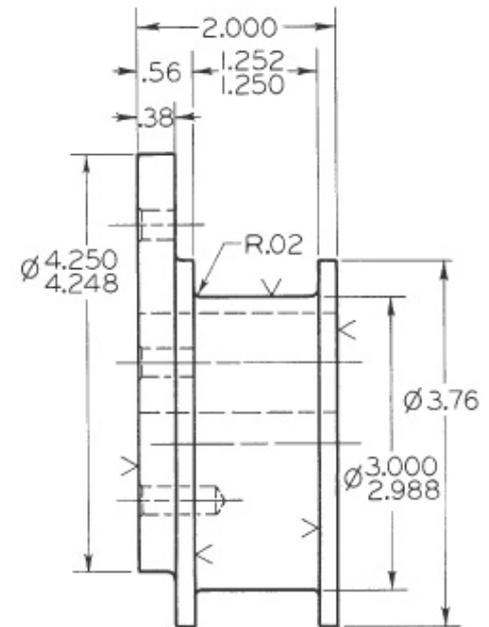
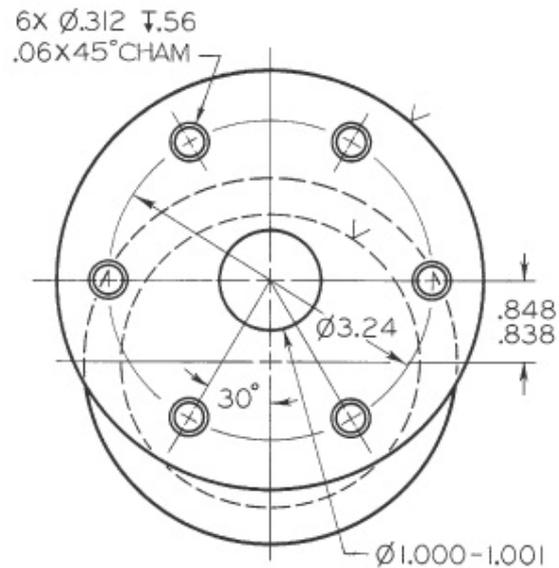
The right cylinder is the next most common geometric shape and its common seen as a shaft or a hole. Cylinders are usually dimensioned by giving the diameter and length where the cylinders appear as rectangle.



(cont)

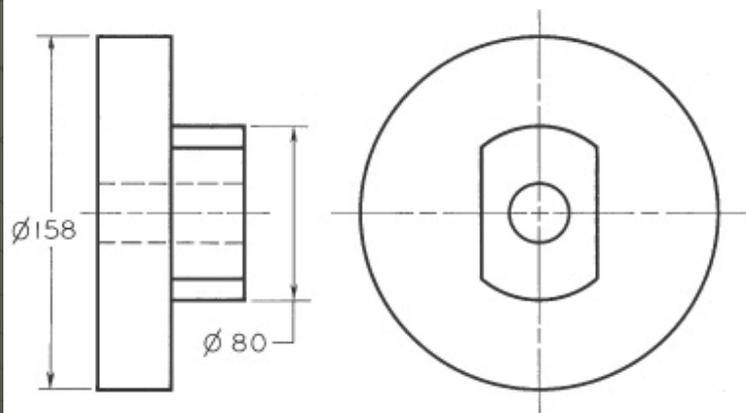
An example of size dimensions for a machine part that is composed of cylindrical shapes

ECENTRIC
FOR
CANNING MACHINE



(cont)

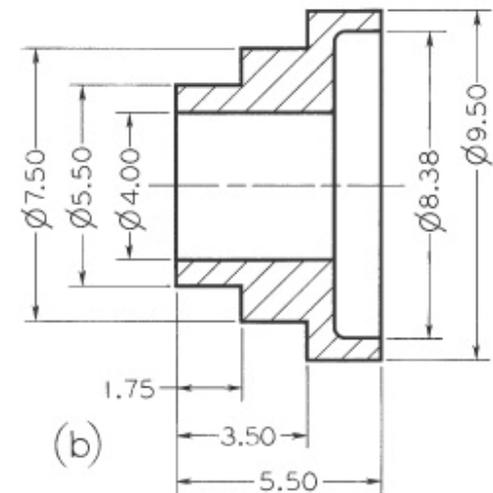
Use of Φ for diameter in dimensioning cylinders



(a)

METRIC

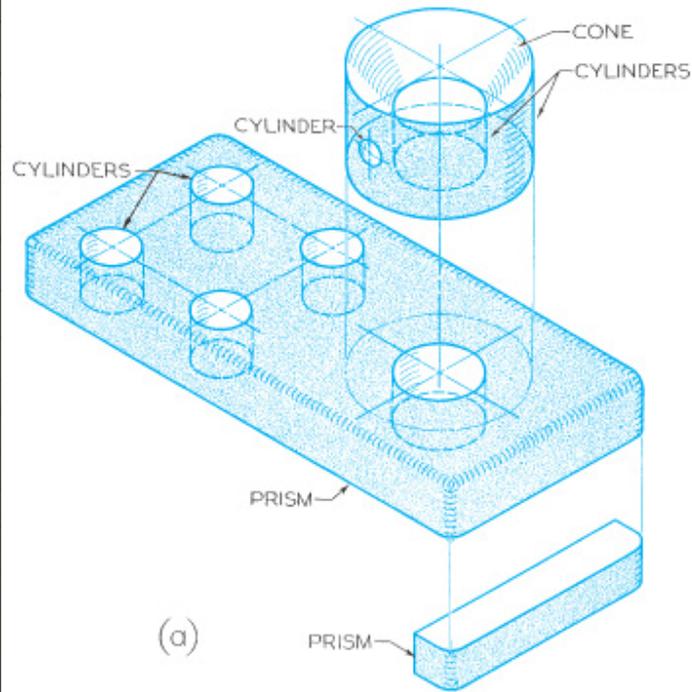
USE OF " Φ " TO INDICATE CIRCULAR SHAPE



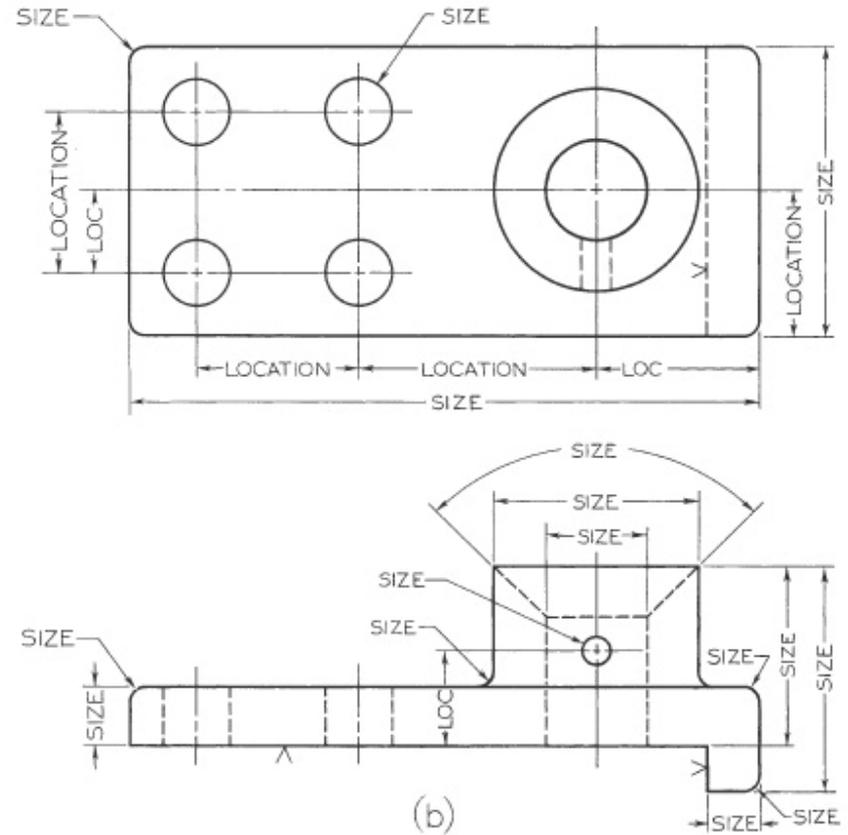
(b)

USE OF " Φ " TO OMIT CIRCULAR VIEW

Step by Step 9.1



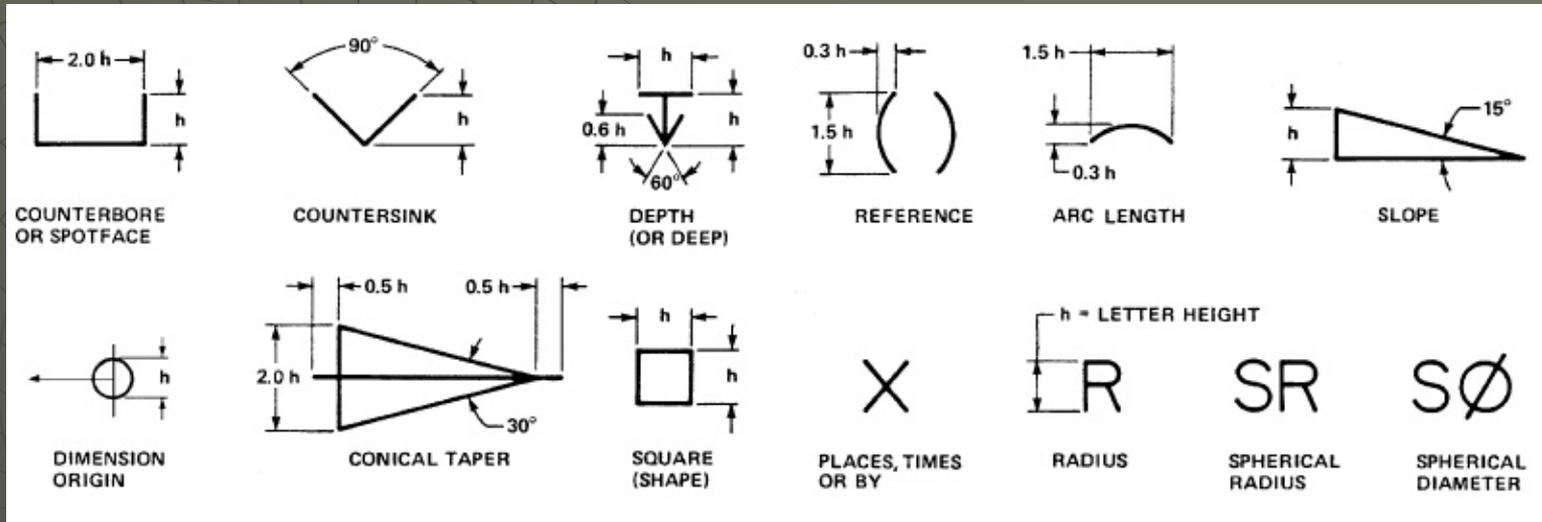
(a)



(b)

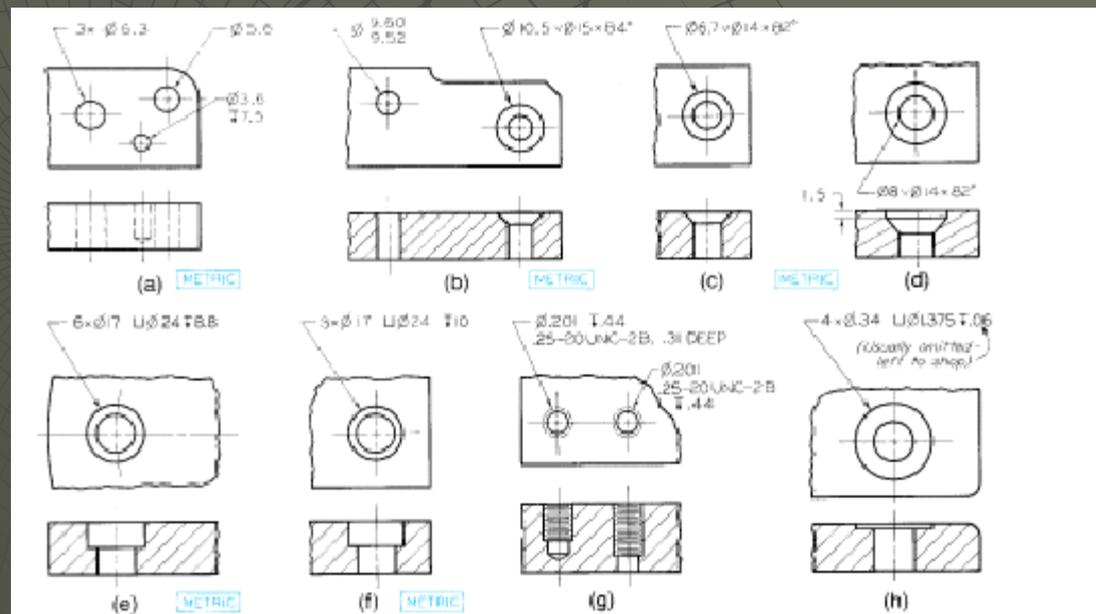
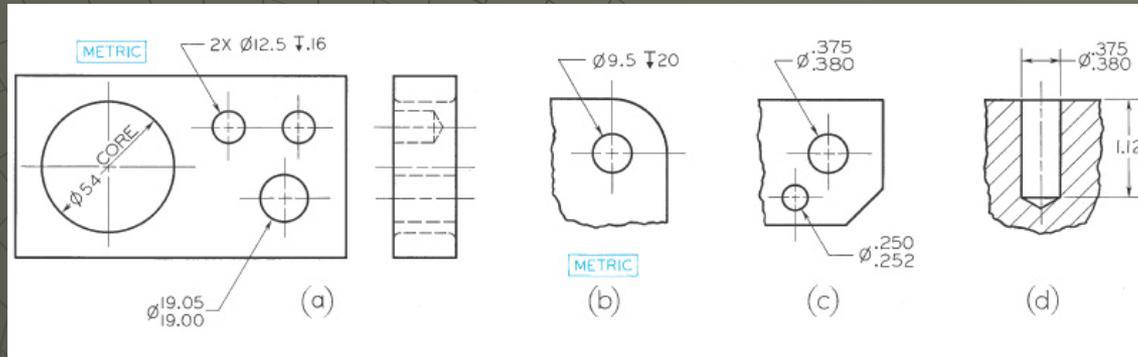
9-21 Size Dimensions: Holes

Figure below shows standard symbols used in dimensioning.



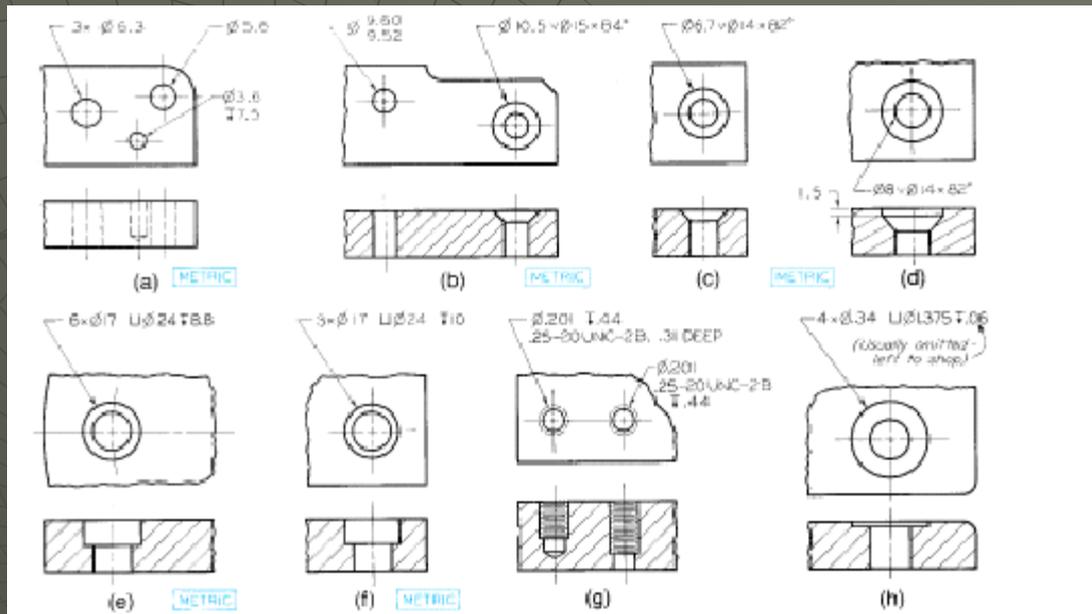
(cont)

For example, countersunk, counterbored, and tapped holes are usually specified by standard symbols or abbreviations, as shown below.



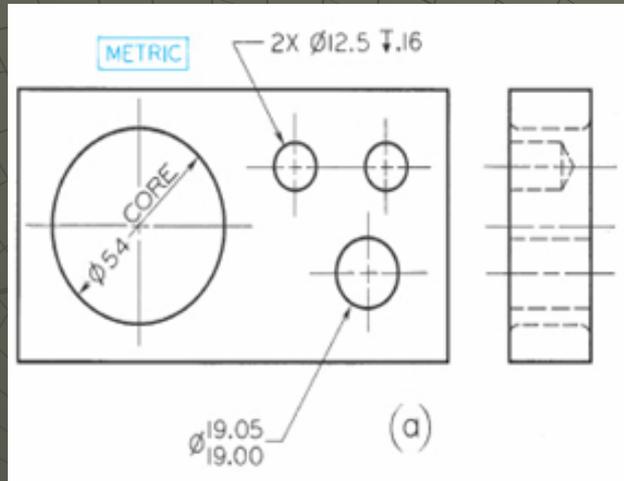
(cont)

When the circular views of the holes has 2 or more concentric circles, as for countersunk, counterbored, and tapped holes, the arrowhead should touch the outer circle.



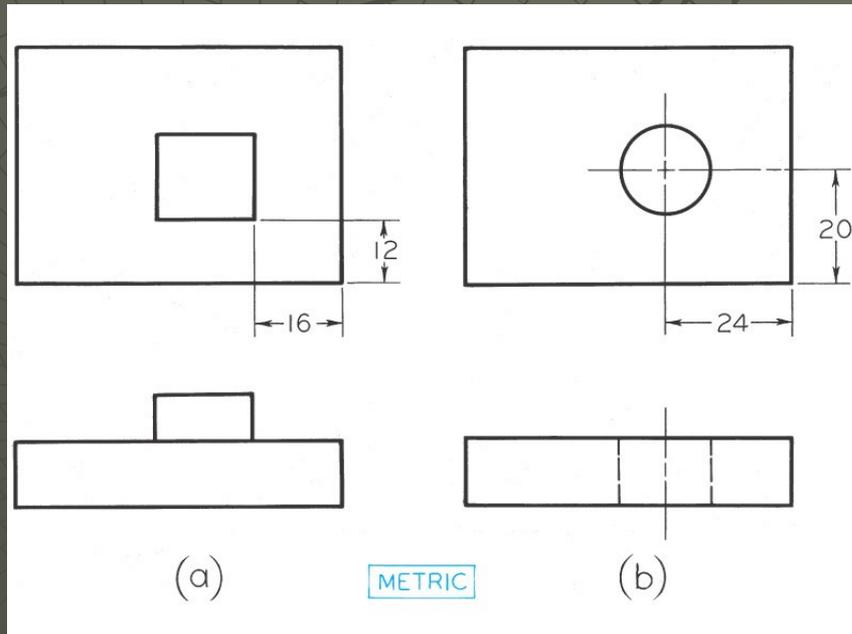
(cont)

Two or more holes can be dimensioned by a single note and by specifying the numbers of holes, as shown below.



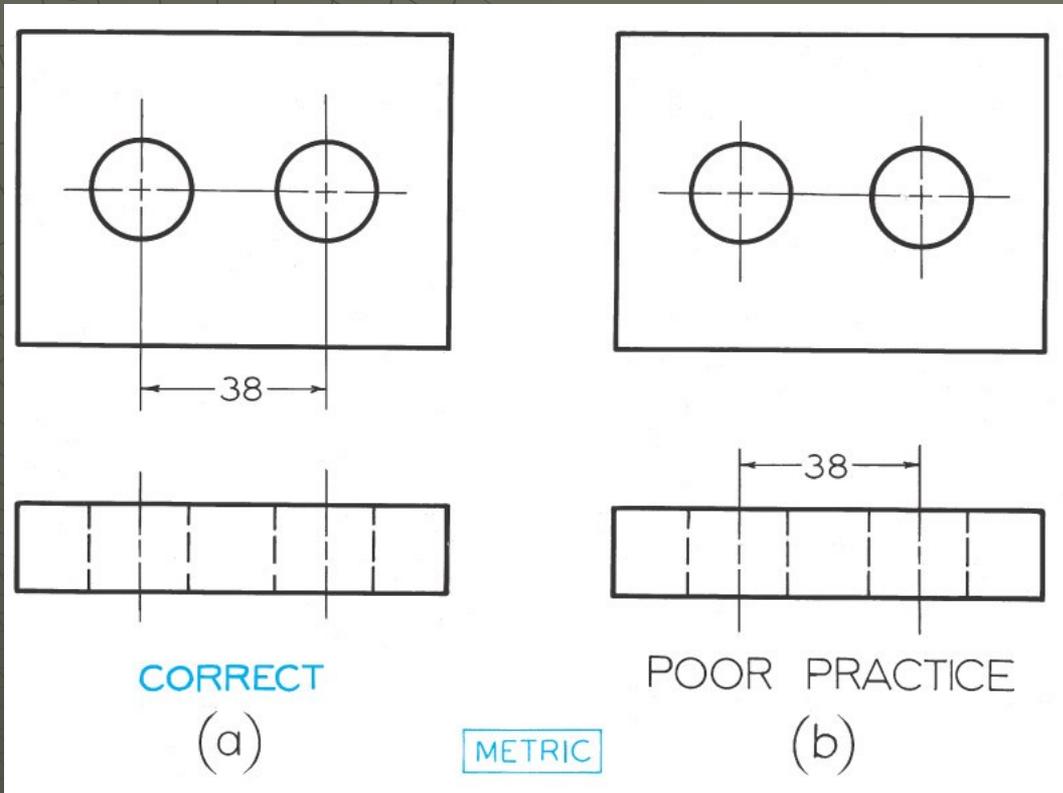
9-22 Location Dimensions

(a) After we have specified the sizes of the geometric shapes composing the structure, give location dimensions to show the relative positions of these geometric shapes. The figure (a) shows rectangular shapes located by their faces. In (b), cylindrical or conical holes or bosses, or other symmetrical shapes, are located by their centerlines.



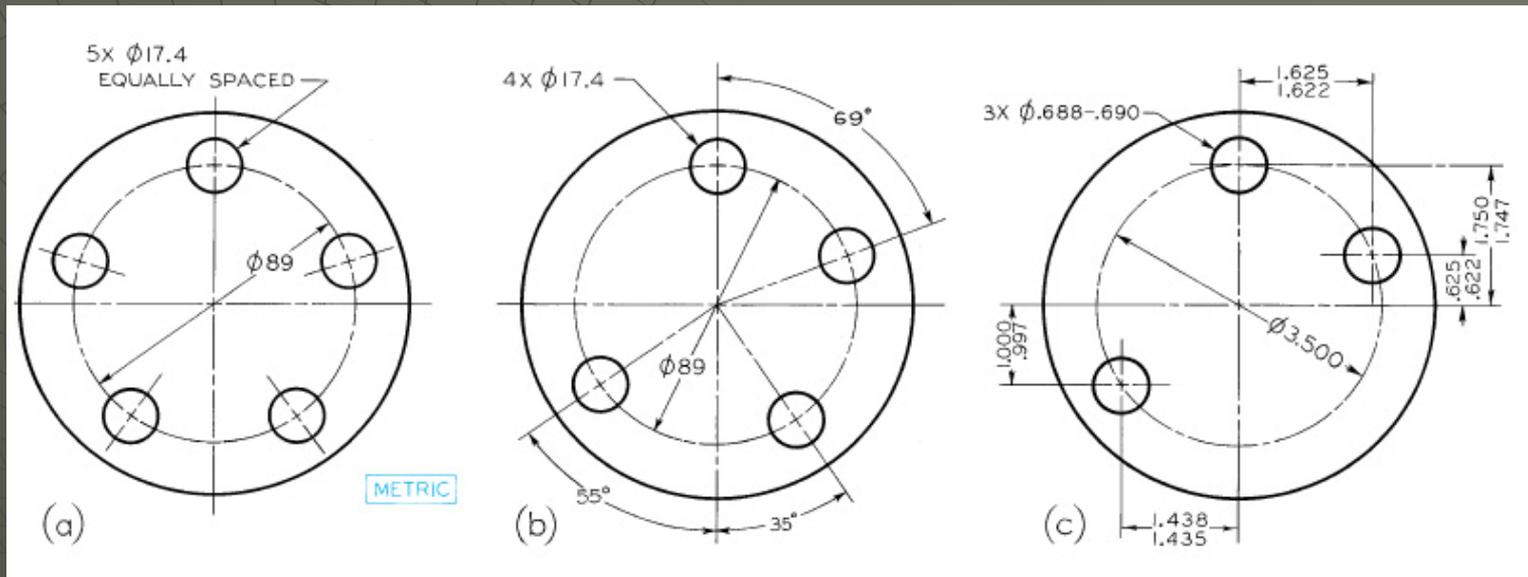
(cont)

Location dimensions for holes are preferably given where holes appear circular, as shown below.



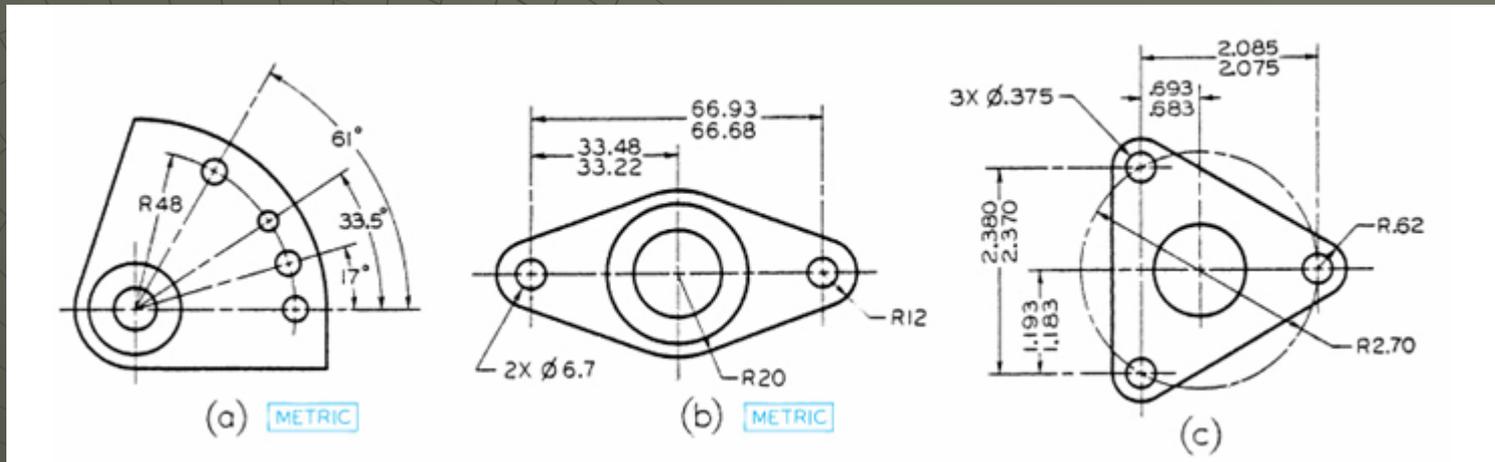
(cont)

Unequally spaced holes are located by means of the bolt circle diameters plus angular measurements with reference to only one of the centerlines. Examples are shown below.



Where greater accuracy is required, coordinate dimensions should be given, as shown in (c). In this case, the diameter of the bolt circle is enclosed in parentheses to indicate that it is to be used only as a *reference dimension*.

When several nonprecision holes are located on a common arc, they are dimensioned by giving the radius and the angular measurements from a baseline, as shown in (a). In this case the baseline is the horizontal centerline.

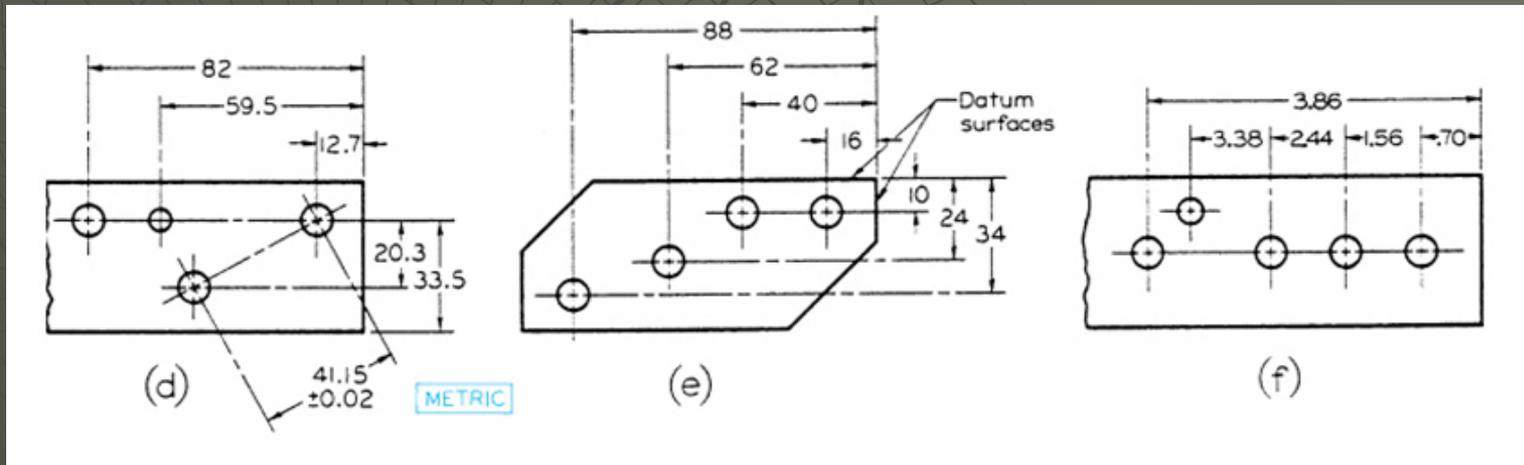


In (b), the 3 holes are on a common centerline. One dimension locates one small hole from the center; the other gives the distances between the small holes.

Figure (c) shows another example of coordinate dimensioning. The three small holes are on a bolt circle whose diameter is given for reference purposes only. From the main center, the small holes are located in 2 mutually perpendicular directions.

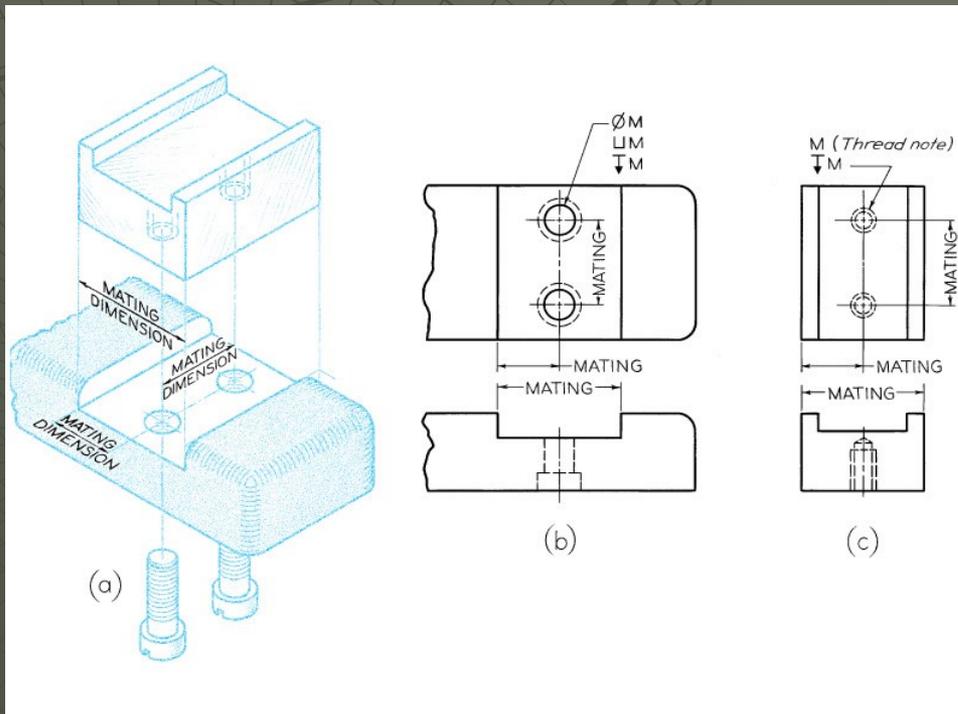
Another example of locating holes by means of linear measurements is shown in (d). In this case, one measurement is made at an angle to the coordinate dimensions due to the direct functional relationship of the 2 holes. In (e), the holes are located from two baselines, or datums.

Figure (f) shows a method of giving all dimensions from a common datum. Each dimension except the first has a single arrowhead and is accumulative in value.



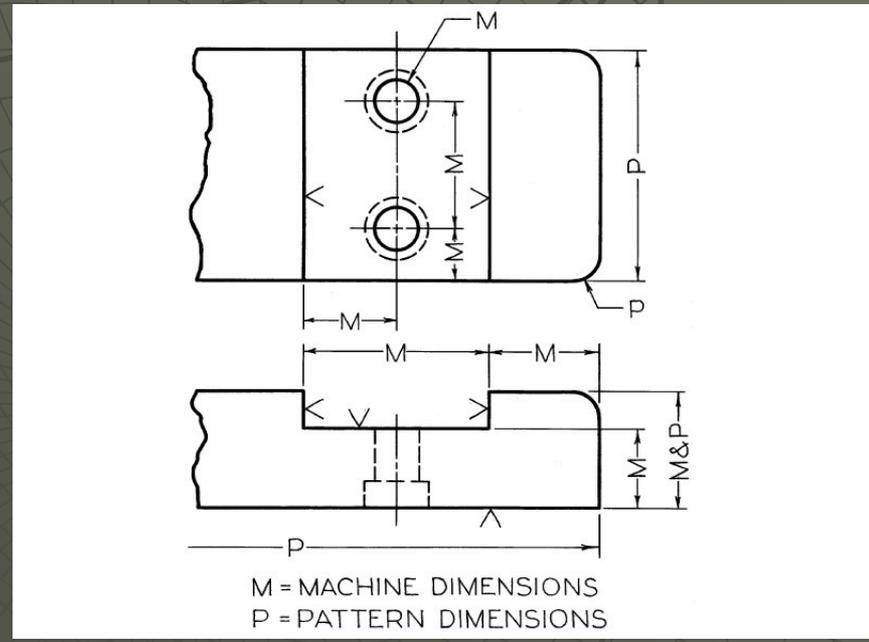
9-24 Mating Dimensions

In dimensioning a single part, its relation to mating parts must be taken into consideration. For instance, in (a) a guide block fits into a slot in a base. Those dimensions common to both parts are **mating dimensions**. These mating dimensions should be given on the **multiview** drawings in the corresponding locations as shown in (b) and (c).



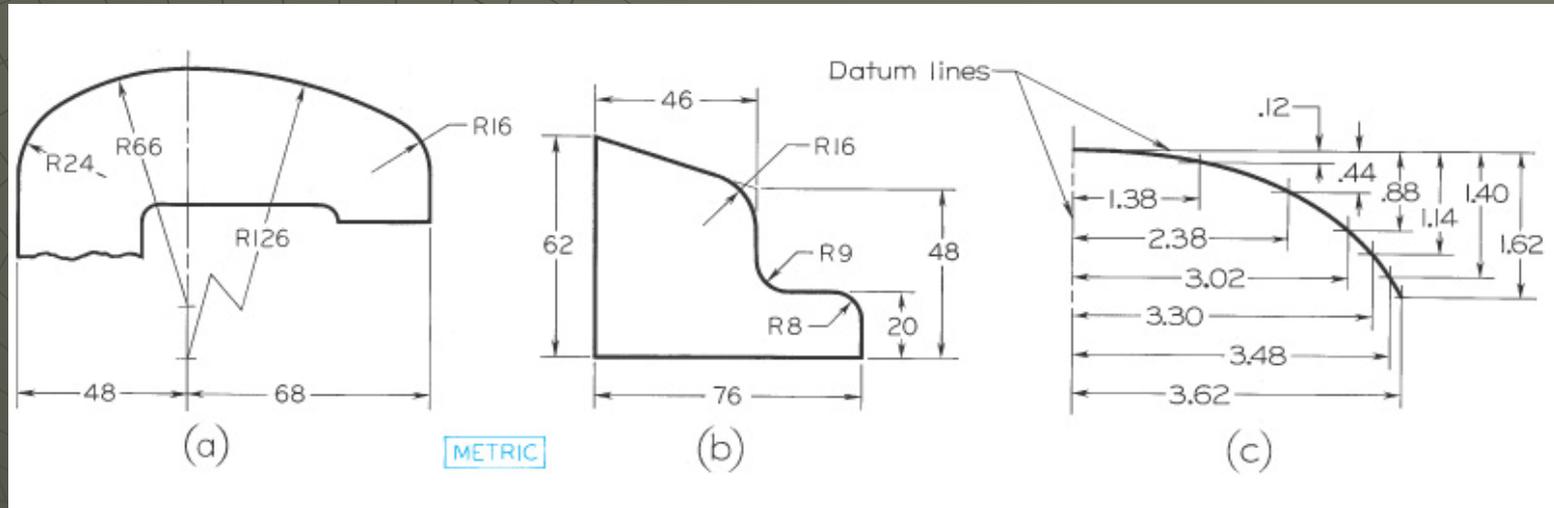
9-25 Machine, Pattern, and Forging Dimensions

Figure below shows the same part as previous slide, with the machine dimensions and pattern dimensions identified by the letters M and P. The pattern maker is interested only in the dimensions required to make the pattern. The machinist, in general, concerns only with the dimensions needed to machine the part.



9-26 Dimensioning of Curves

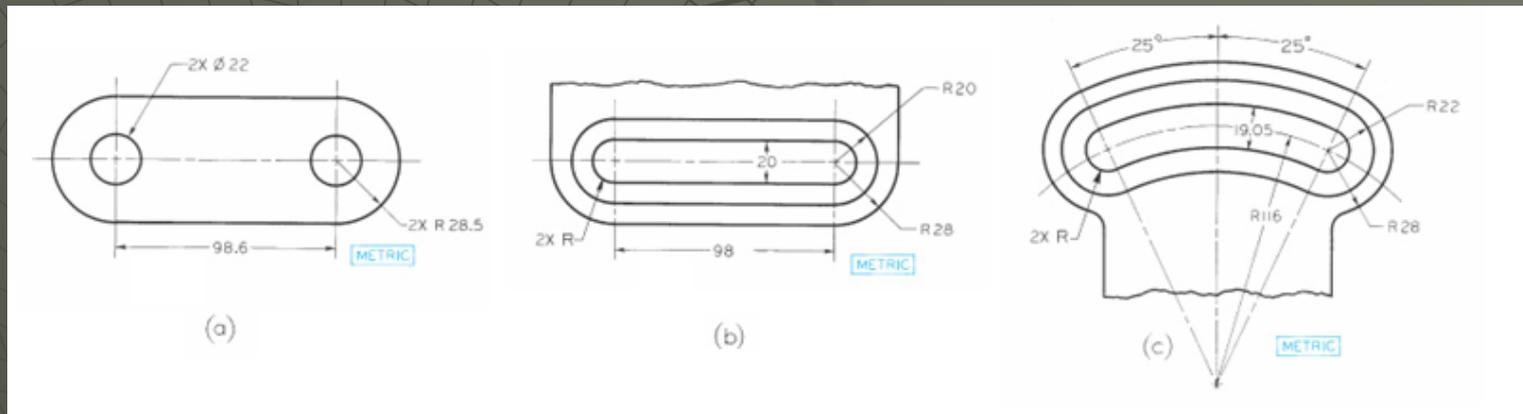
Curved shapes may be dimensioned by giving a group of radii as shown in (a).



Another method is to dimension the outline envelope of a curved shape so that the various radii are self-locating from “floating center” as shown in (b). Either a circular or a noncircular curve may be dimensioned by means of coordinate dimensions, or datums, as shown in (c).

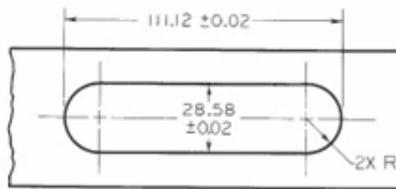
9-27 Dimensioning of Rounded-end shapes

The method used for dimensioning rounded-end shapes depends on the degree of accuracy required. When precision is not necessary, the methods used are those that are convenient for manufacturing, as in (a) to (c).



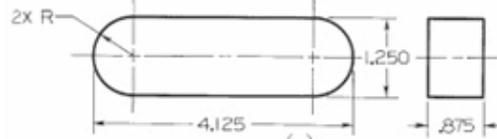
When accuracy is required, the methods shown in (d) to (g) are recommended. Overall lengths of rounded-end shapes are given in each case, and radii are indicated, but without specific values.

The center-to-center distance may be required for accurate locations of some holes. In (g), the hole location is more critical than the location of the radius, so the two are located independently, as shown.

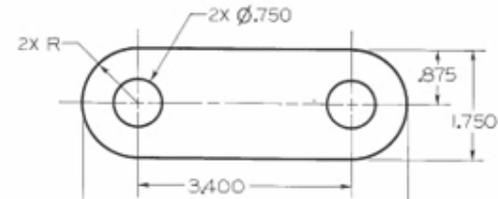


(d)

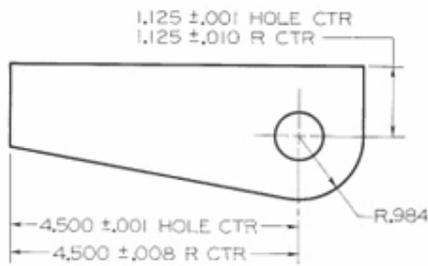
METRIC



(e)



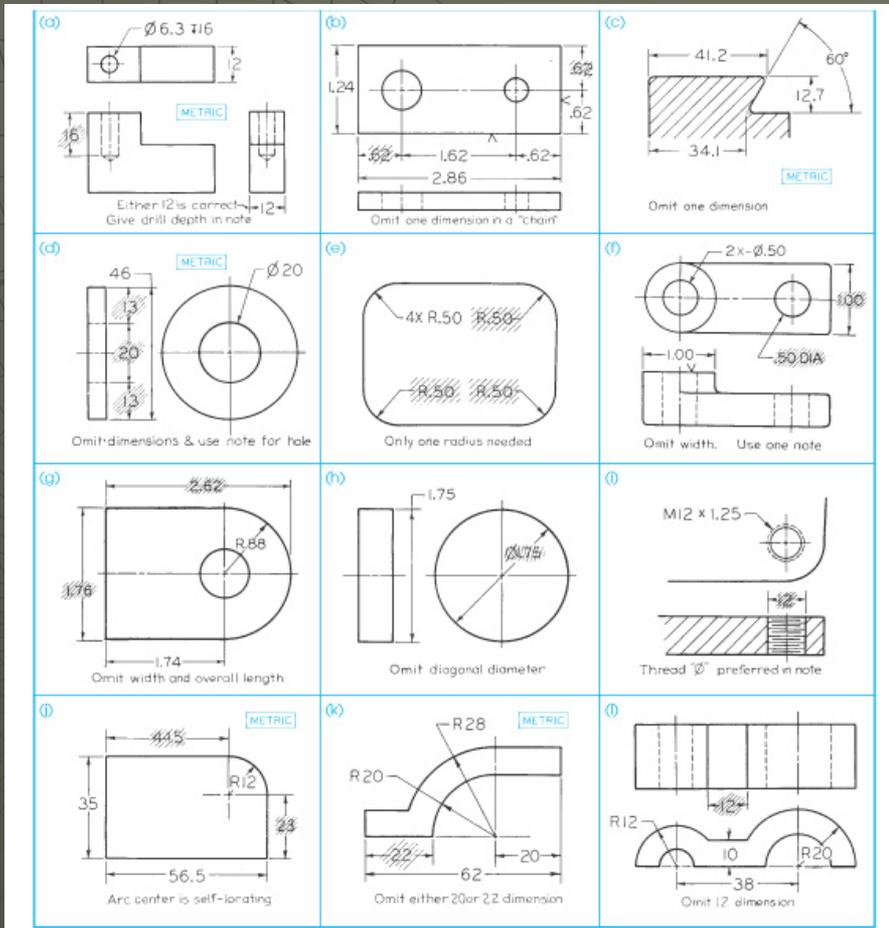
(f)



(g)

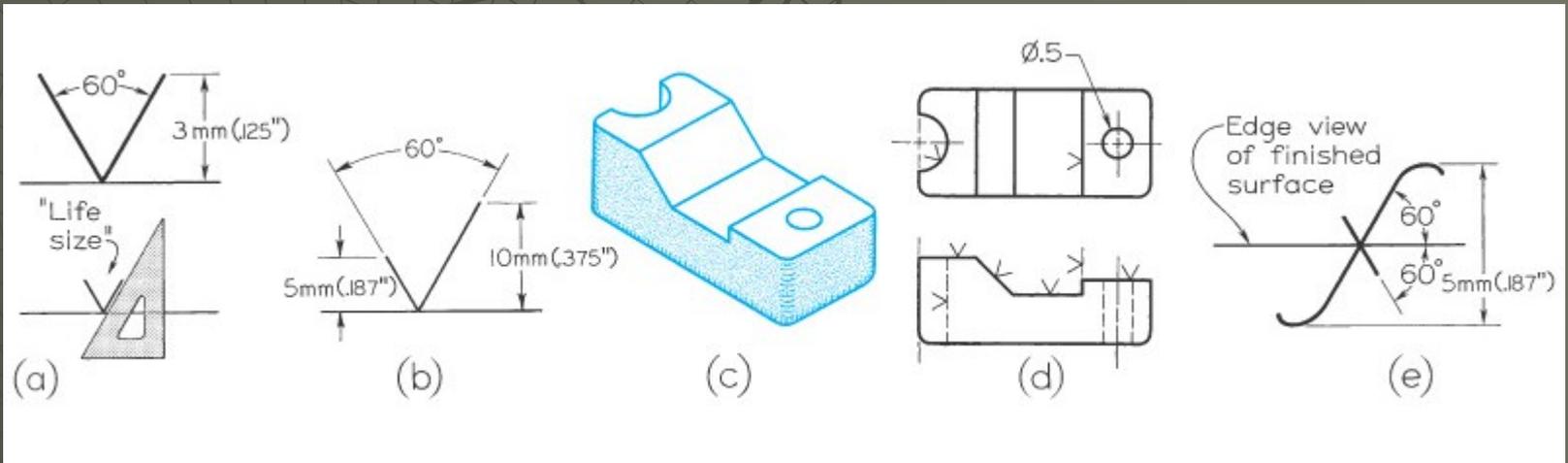
9-28 Superfluous Dimensions

All necessary dimensions must be shown, but avoid giving unnecessary or superfluous dimensions, as shown below.

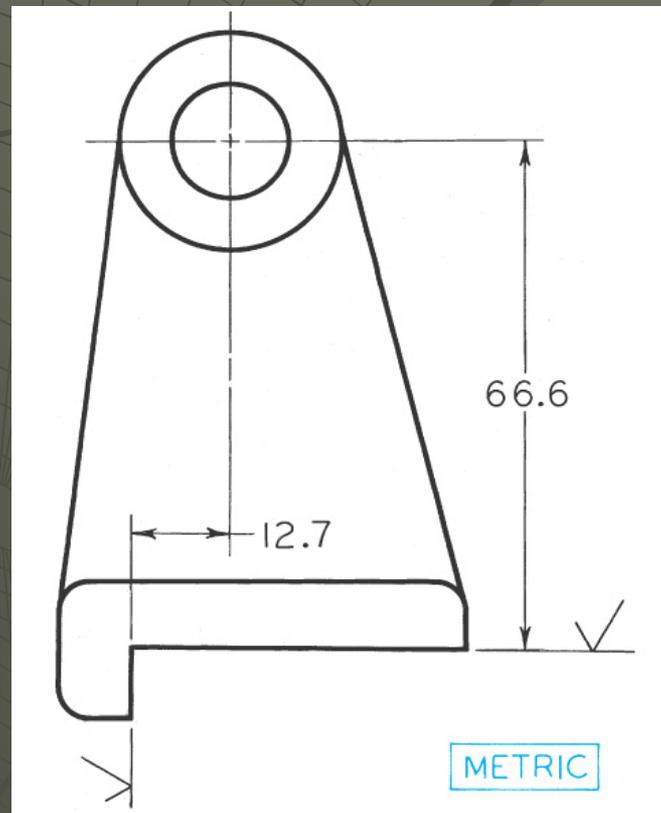


9-29 Finish Marks

A **finish mark** is used to indicate that a surface is to be machined, or finished, as on a rough casting or forging. As shown below, three styles of finish mark, the general symbol, the new basic symbol (preferred by ANSI), and the old symbol, are used to indicate an ordinary smooth machined surface.

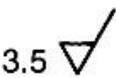
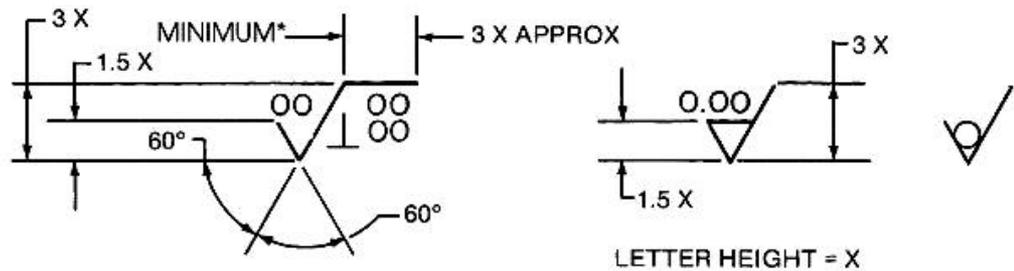


The point of the finish symbol should be directed inward toward the body of metal in a manner similar to that of a tool bit. The symbol must **not be shown upside down** as seen below.

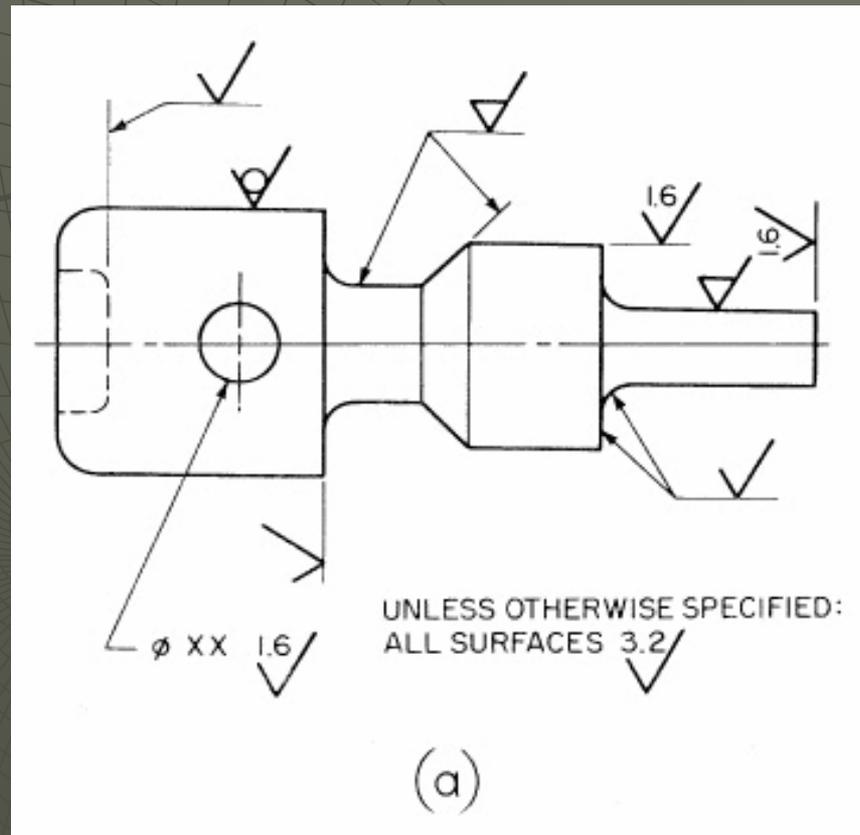


9-30 Surface Roughness, Waviness, and Lay

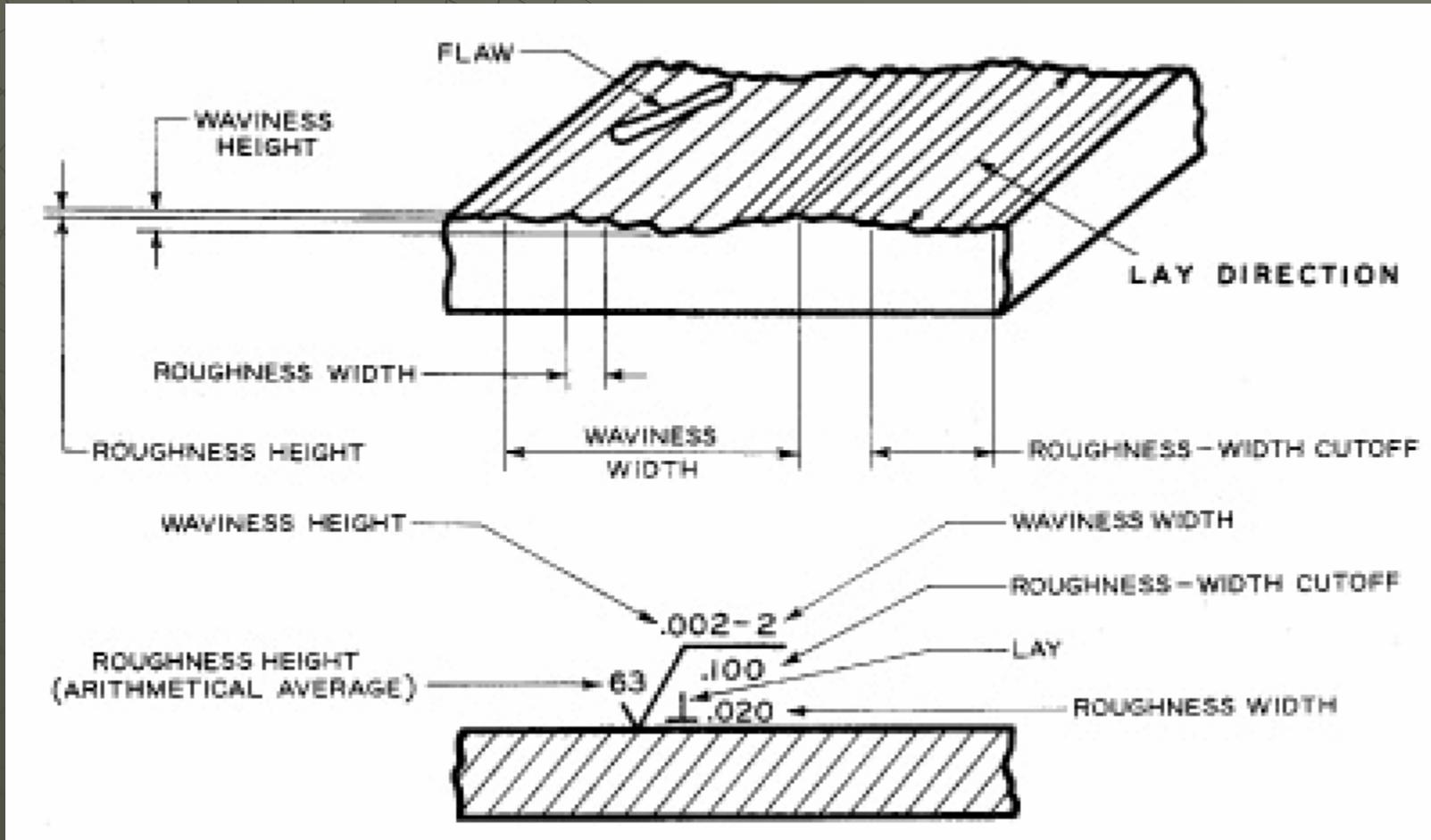
The system of **surface texture symbols** recommended by ANSI/ASME for use on drawing, regardless of the system of measurement used, is now broadly accepted by American industry. The symbols are used to define surface texture, roughness and lay.

Symbol	Meaning
(a) 	Basic Surface Texture Symbol. Surface may be produced by any method except when the bar or circle, (b) or (d), is specified.
(b) 	Material Removal By Machining Is Required. The horizontal bar indicates that material removal by machining is required to produce the surface and that material must be provided for that purpose.
(c) 3.5 	Material Removal Allowance. The number indicates the amount of stock to be removed by machining in millimeters (or inches). Tolerances may be added to the basic value shown or in a general note.
(d) 	Material Removal Prohibited. The circle in the vee indicates that the surface must be produced by processes such as casting, forging, hot finishing, cold finishing, die casting, powder metallurgy or injection molding without subsequent removal of material.
(e) 	Surface Texture Symbol. To be used when any surface characteristics are specified above the horizontal line or to the right of the symbol. Surface may be produced by any method except when the bar or circle, (b) or (d), is specified.
(f) 	
* This dimension is adjusted by +1 for each line of values beyond the two lines shown below the horizontal line.	

Applications of the surface texture symbols are given in (a) below. Note that the symbols read from the bottom and/or the right side of the drawing and that they are not drawn at any angle or upside down.

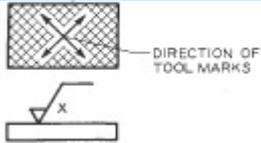
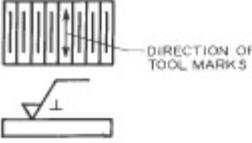
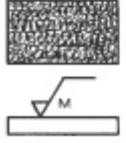
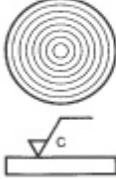
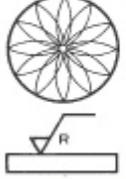


Measurements for **roughness** and **waviness** , unless otherwise specified, apply in the direction that gives the maximum reading, usually across the lay, as shown in (b). The recommended roughness height values are given in Table 9.1.

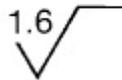


When it is necessary to indicate the roughness-width cutoff values, the standard values used are listed in Table 9.2. if no values is specified, the 0.80 value is assumed. When maximum waviness height values are required, the recommended values to be used are as given in Table 9.3. When it is desired to indicate lay, the lay symbols in figure below are added to the surface texture symbols as per the examples given.

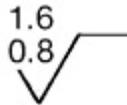
LAY SYMBOLS

SYM	DESIGNATION	EXAMPLE	SYM	DESIGNATION	EXAMPLE
— —	Lay parallel to the line representing the surface to which the symbol is applied.		X	Lay angular in both directions to line representing the surface to which symbol is applied.	
⊥	Lay perpendicular to the line representing the surface to which the symbol is applied.		M	Lay multidirectional	
C	Lay approximately circular relative to the center of the surface to which the symbol is applied.		R	Lay approximately radial relative to the center of the surface to which the symbol is applied.	

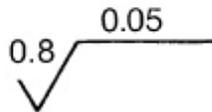
Selected applications of the surface texture values to the symbols are given and explained below.



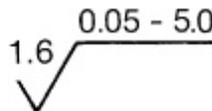
Roughness average rating is placed at the left to the long leg. The specification of only one rating shall indicate the maximum value and any lesser value shall be acceptable. Specify in micrometers (microinch).



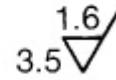
The specification of maximum and minimum roughness average values indicates permissible range of roughness. Specify in micrometers (microinch).



Maximum waviness height rating is the first rating placed above the horizontal extension. Any lesser rating shall be acceptable. Specify in millimeters (inch).



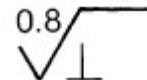
Maximum waviness spacing rating is the second rating placed above the horizontal extension and to the right of the waviness height rating. Any lesser rating shall be acceptable. Specify in millimeters (inch).



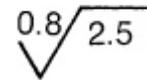
Material removal by machining is required to produce the surface. The basic amount of stock provided for material removal is specified at the left of the short leg of the symbol. Specify in millimeters (inch).



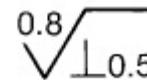
Removal of material is prohibited.



Lay designation is indicated by the lay symbol placed at the right of the long leg.

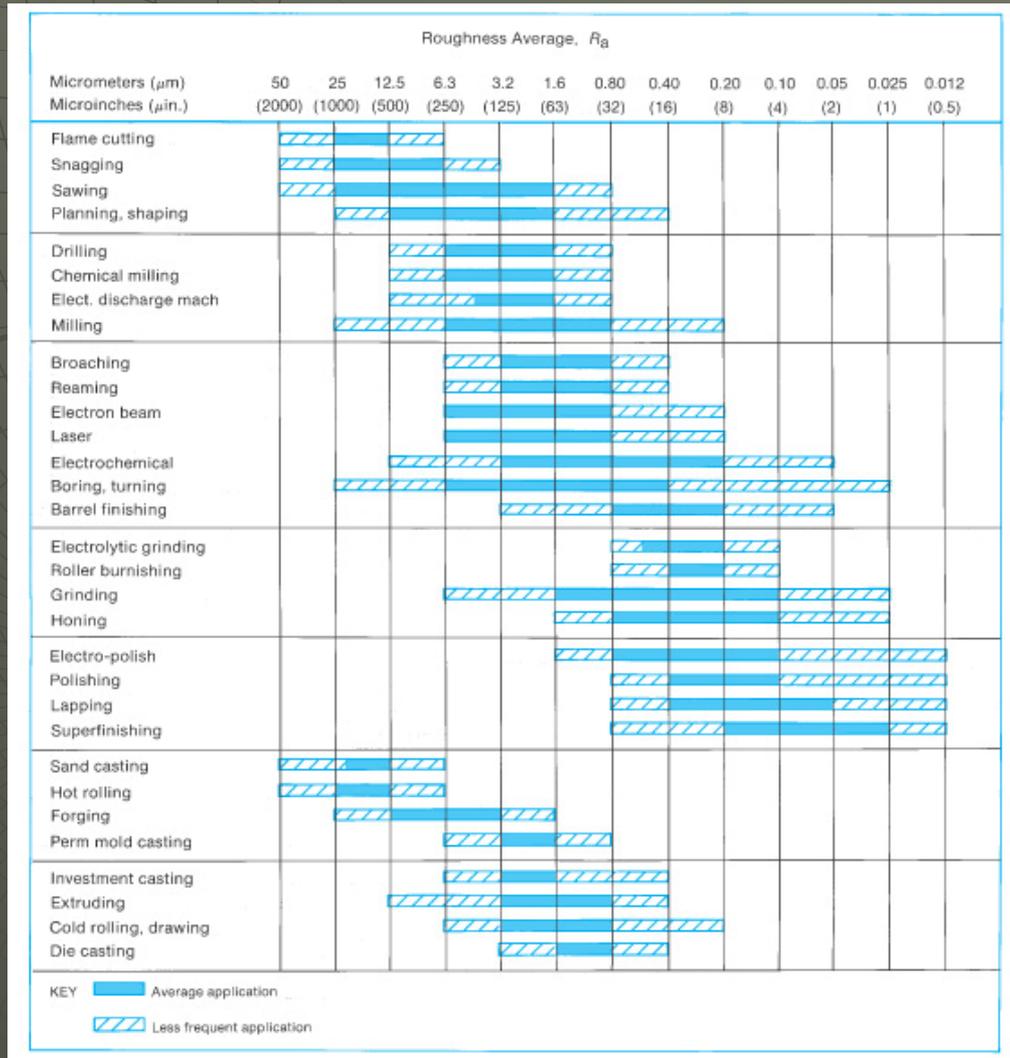


Roughness sampling length or cutoff rating is placed below the horizontal extension. When no value is shown, 0.80 mm (0.030 inch) applies. Specify in millimeters (inch.)



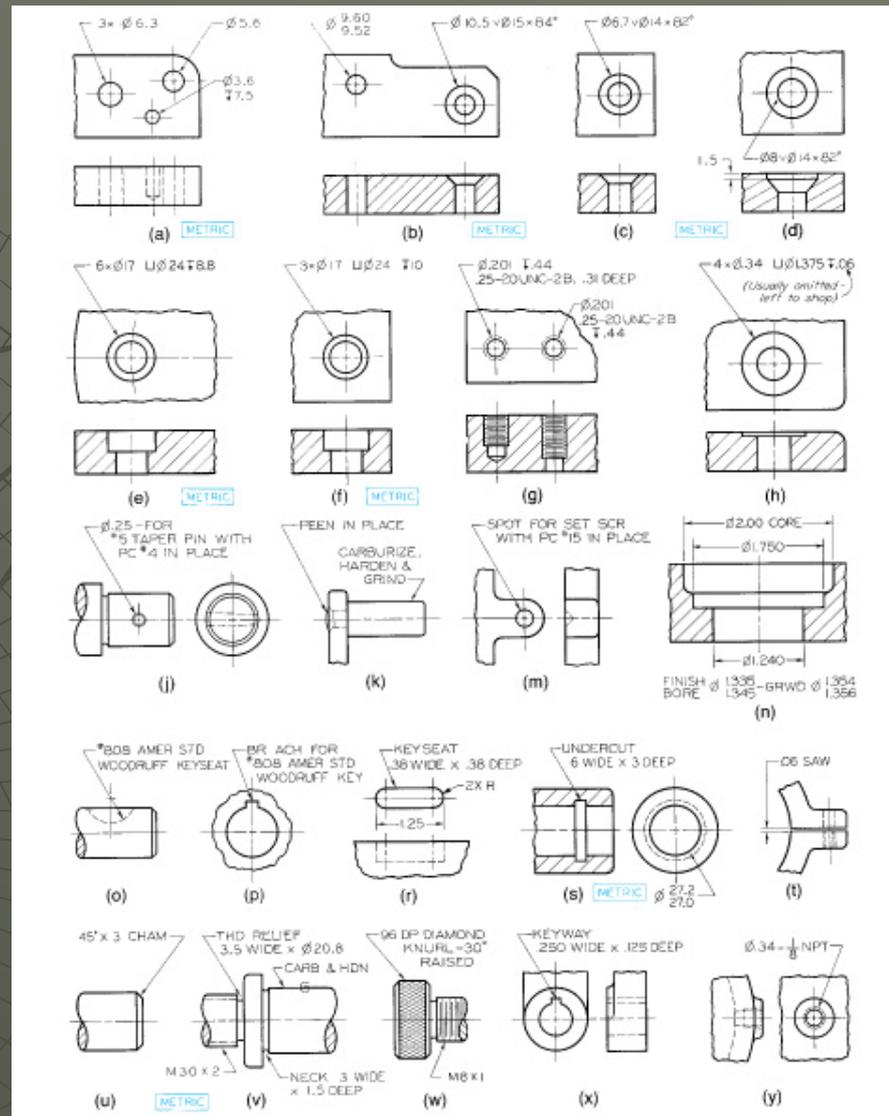
Where required maximum roughness spacing shall be placed at the right of the lay symbol. Any lesser rating shall be acceptable. Specify in millimeters (inch).

A typical range of surface roughness values that may be obtained from various production method is shown in the Table below. Preferred roughness-height values are shown at the top of chart.



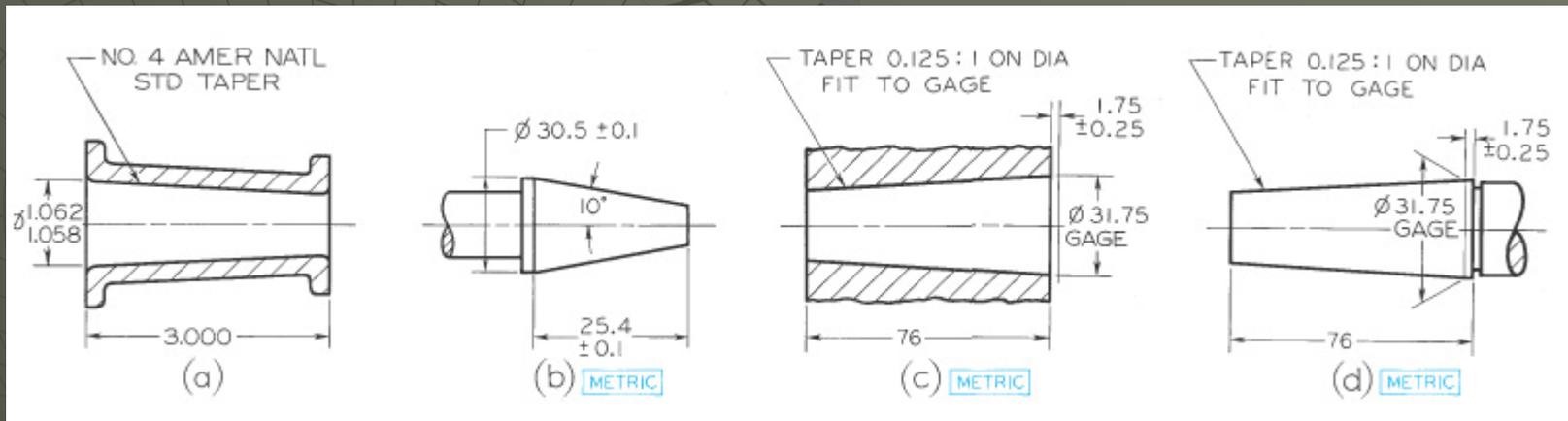
9-31 Notes

It is usually to supplement the direct dimensions with notes. Notes are classified as general notes when they apply to an entire drawing and as local notes when they apply to specific items. Some examples of local notes are at right.



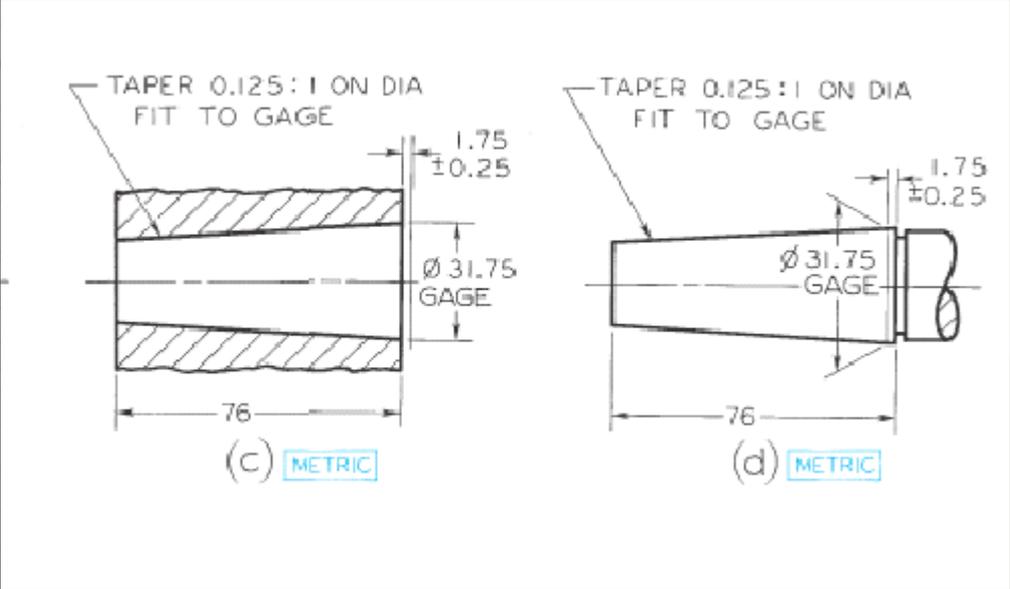
9-33 Dimensioning of Tapers

Standard machine tapers are used on machine spindles, shanks of tools, or pins and are described in “Machine Tapers” in ANSI/ASME B5.10—1994. such standard tapers are dimensioned on a drawing by giving the diameter, the length, and a note, as shown in Fig. (a) below.



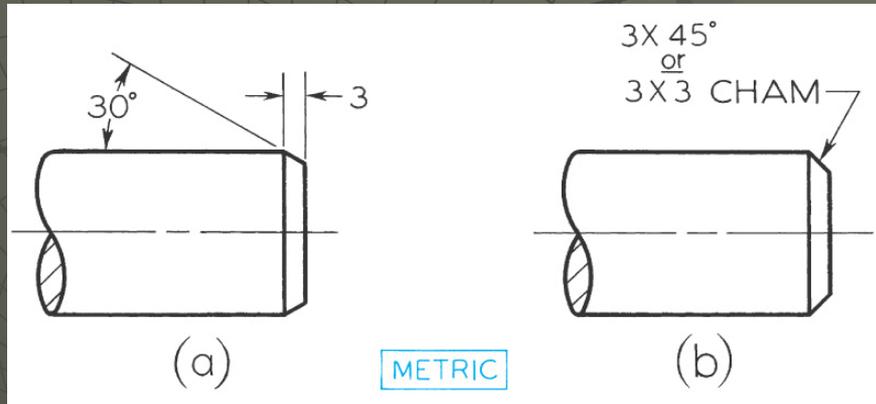
For not-too-critical requirements, a taper may be dimensioned by giving the diameter at the large end, the length, and the included angle, all with proper tolerances, as shown in Fig. (b).

For close-fitting tapers, the amount of *taper per unit on diameter* is indicated as shown in Figs. (c) and (d). A gage line is selected and located by a comparatively generous tolerance, while other dimensions are given appropriate tolerances as required.



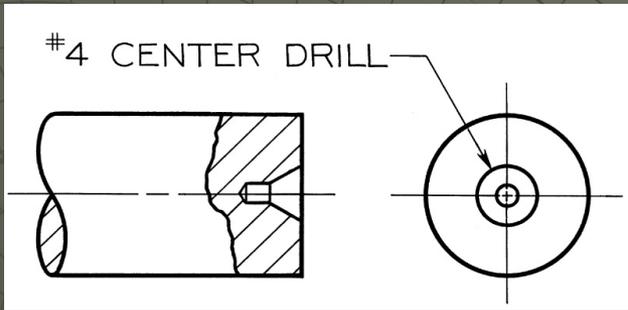
9-34 Dimensioning of Chamfers

A chamfer is a beveled or sloping edge. It is dimensioned by giving the length of the offset and the angle, as shown in Fig. (a). A 45-degree chamfer also may be dimensioned in a manner similar to that shown in Fig. (a), but usually it is dimensioned by note, as in Fig. (b).



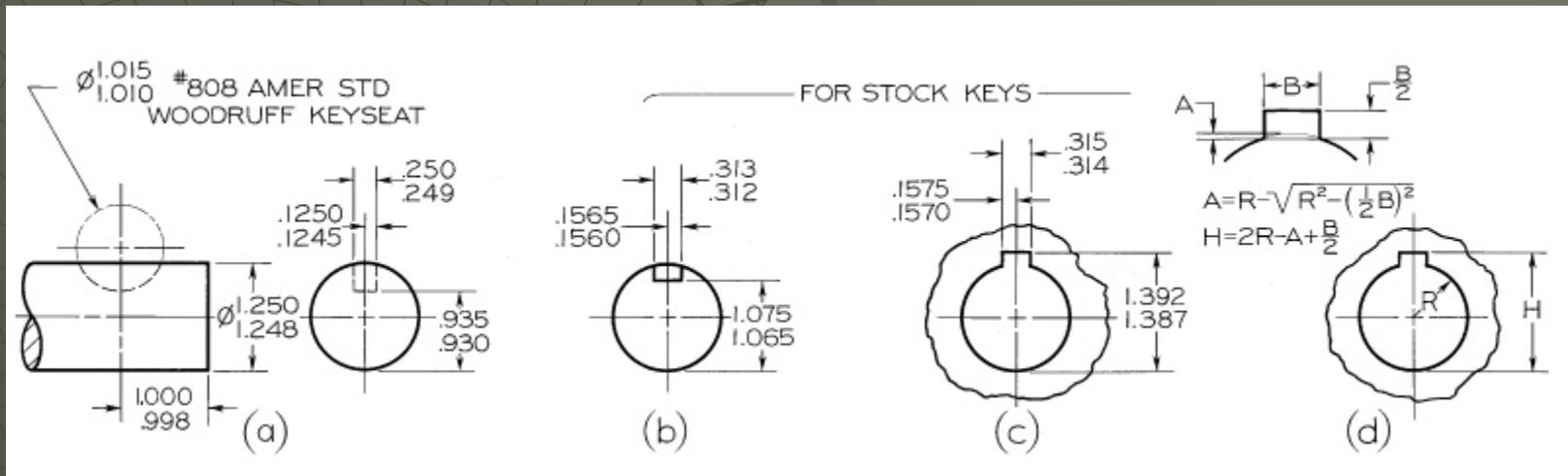
9-35 Shaft Centers

Shaft centers are required on shafts, spindles, and other conical or cylindrical parts for turning, grinding, and other operation. Such a center may be dimensioned as shown below. Normally the centers are produced by a combined drill and countersink.



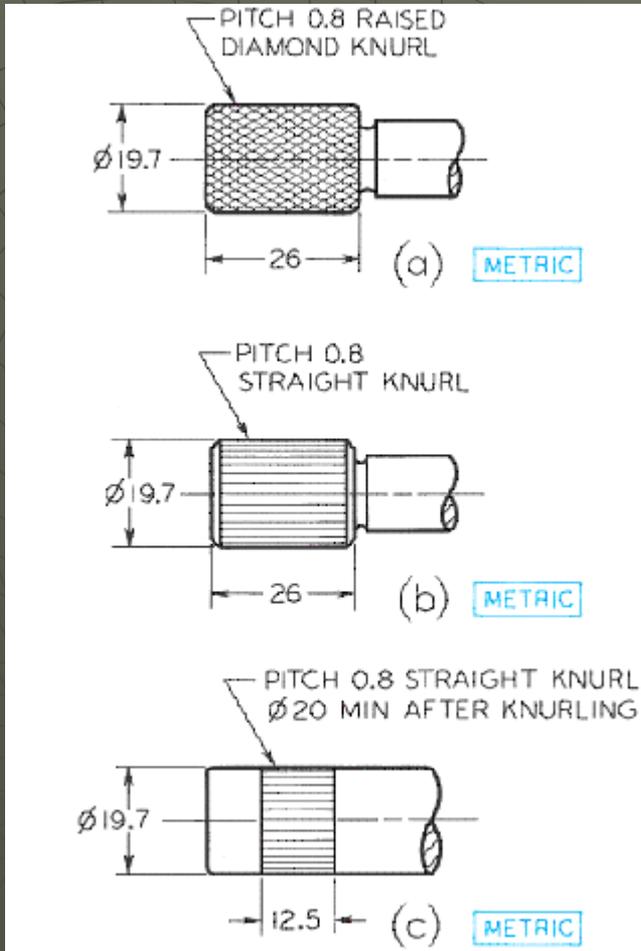
9-36 Dimensioning Keyways

Methods of dimensioning keyways for Woodruff keys and stock keys are shown below. The preferred method of dimensioning the depth of keyway is to give the dimension from the bottom of the keyway to the opposite side of the shaft or hole, as shown.



The method of computing such a dimension is shown in Fig. (d). Values for A may be found in machinists; handbooks.

9-37 Dimensioning of Knurls

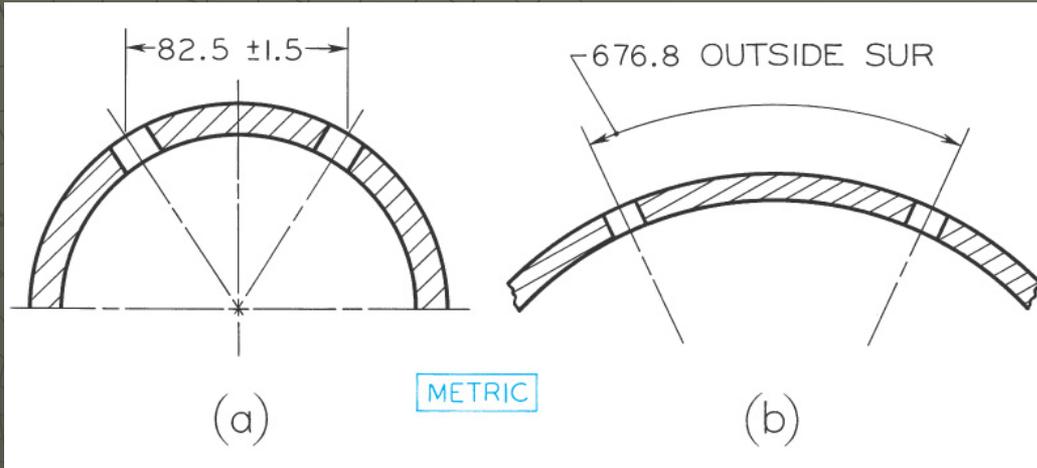


A knurl is a roughened surface to provide a better handgrip or to be used for a press fit between 2 parts. For handgrip purposes, it is necessary only to give the pitch of the knurl, the type of knurling, and the length of knurled area, as shown in Figs. (a) and (b).

To dimension a knurl for a press fit, the toleranced diameter before knurling should be given, as shown in Fig. (c). A note should be added that gives the pitch and type of knurl and the minimum diameter after knurling.

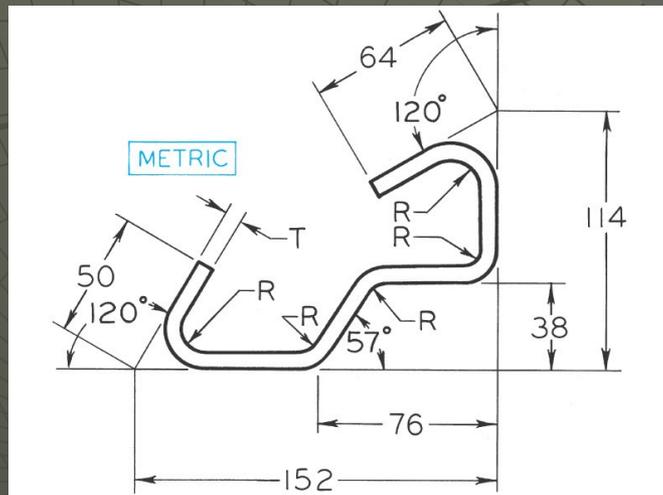
9-38 Dimensioning Along Curved Surfaces

When angular measurements are unsatisfactory, chordal dimensions, as shown in (a), or linear dimensions on the curved surfaces, as shown in (b), may be given.

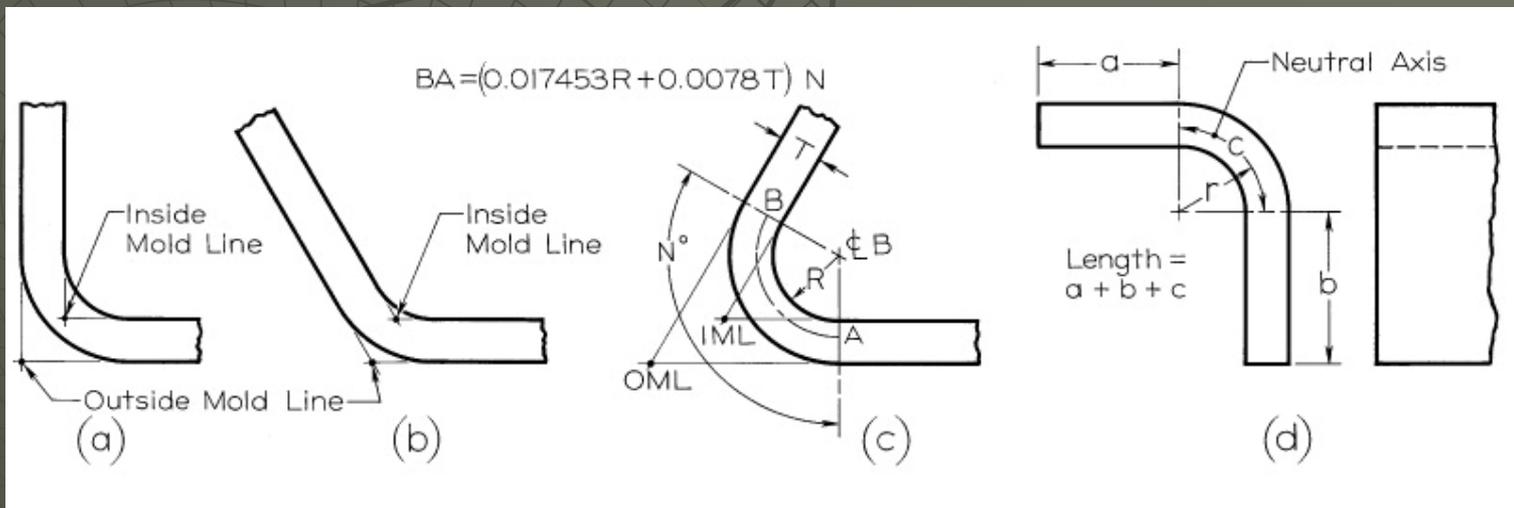


9-39 Sheet-Metal Bends

In sheet metal dimensioning, allowance must be made for bends. The intersection of the plane surfaces adjacent to a bend is called the **mold line**, and this line, rather than the center of arc, is used to determine dimensions, as shown below.

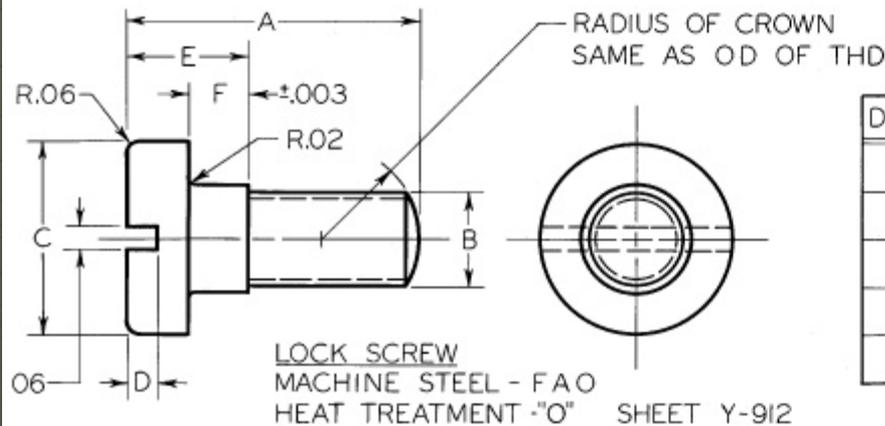


The following procedure for calculating bends is typical. If the two inner plane surfaces of an angle are extended, their line of intersection is called the IML or *inside mold line*, as shown in Figs. (a) –(c). Similarly, if the two outer plane surfaces are extended, they produce the OML or *outside mold line*. The *centerline of bend* refers primarily to the machine on which the bend is made and is at the center of the bend radius.



9-40 Tabular Dimensions

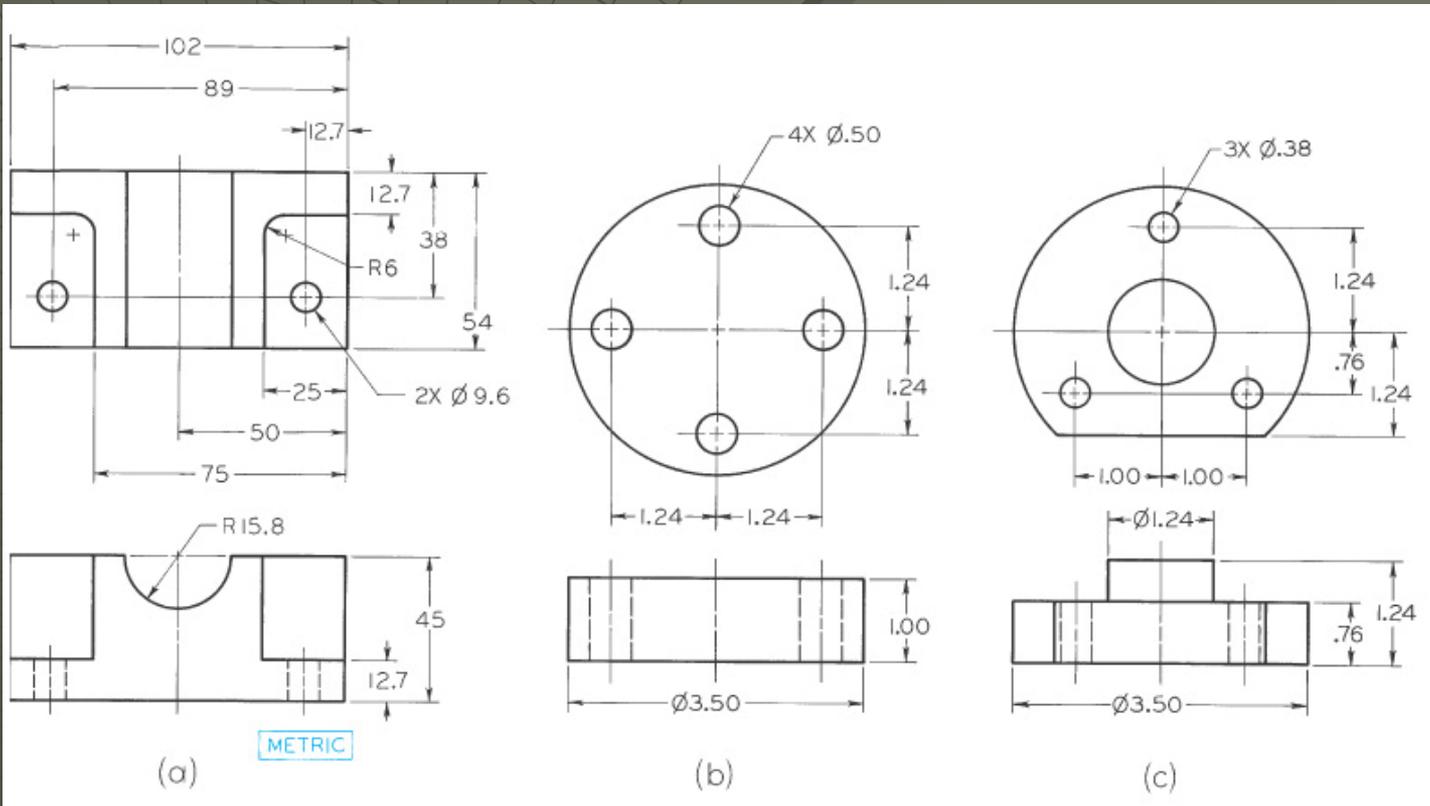
A series of objects having like features but varying in dimensions may be represented by one drawing, as shown in figure below. Letters are substituted for dimension figures on the drawing, and the varying dimensions are given in tabular form. The dimensions of many standard parts are given in this manner in catalogs and handbooks.



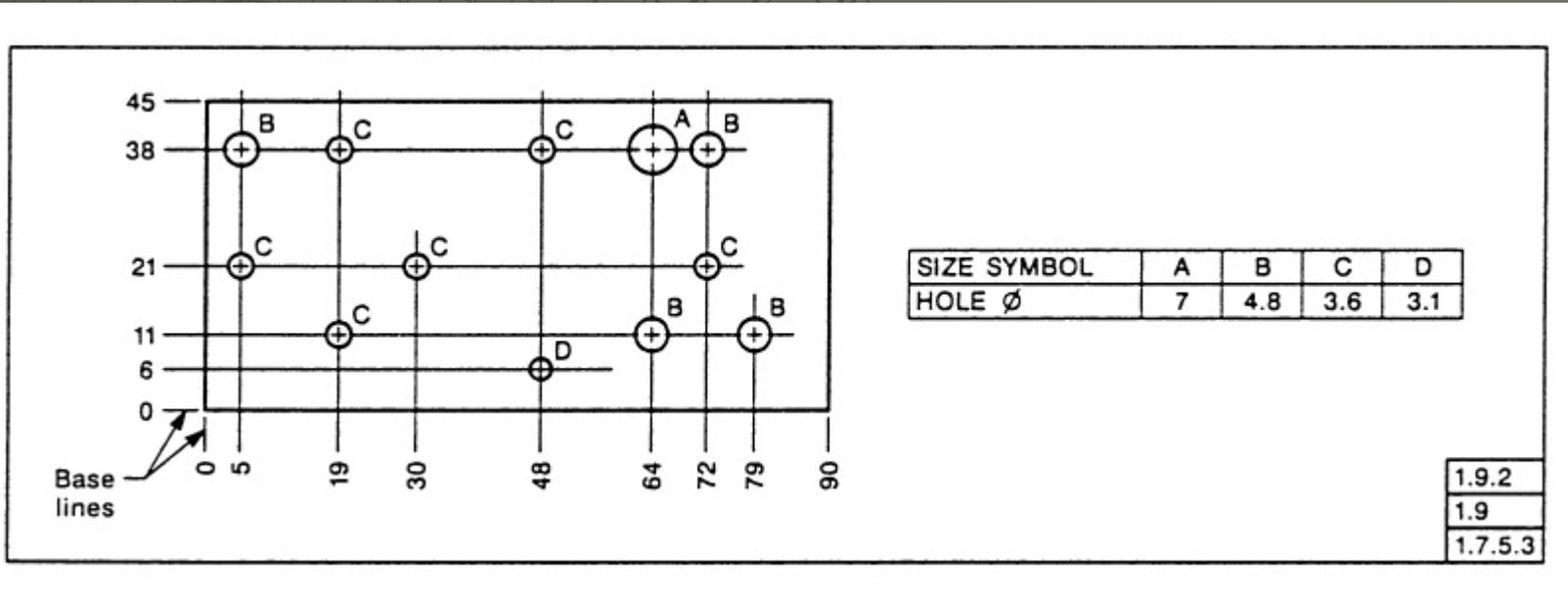
DETAIL	A	B	C	D	E	F	UNC THD	STOCK	LBS
1	.62	.38	.62	.06	.25	.135	.312-18	Ø.75	.09
2	.88	.38	.62	.09	.38	.197	.312-18	Ø.75	.12
3	1.00	.44	.75	.12	.38	.197	.375-16	Ø.875	.19
4	1.25	.50	.88	.12	.50	.260	.437-14	Ø.1	.30
5	1.50	.56	1.00	.16	.62	.323	.5-13	Ø1.25	.46

9-42 Coordinate Dimensioning

A set of 3 mutual perpendicular datum or reference planes is usually required for coordinate dimensioning. These planes either must be obvious or must be clearly identified, as shown below.



The designer selects as origins for dimensions those surfaces or features most important to the functioning of the part. Enough of these features are selected to position the part in relation to the set of mutually perpendicular planes. All related dimensions are then made from these planes. Rectangular coordinate dimensioning without dimension lines is shown below.



SIZE SYMBOL	A	B	C	D
HOLE \varnothing	7	4.8	3.6	3.1

1.9.2
1.9
1.7.5.3