

Principles of dimensioning

Terminology

What is the main image, which we are using in all our projects, designs, drawings?

It is a **line**!

And in Graphics Communications practice there is the whole bunch of different lines, which are used for drawing purposes.

The **alphabet of lines** is a set of standard linetypes established by the American National Standards Institute (ANSI) for technical drawing. The alphabet of lines and the approximate dimensions used to create different linetypes, are referred to as **linetypes** when used with CAD.

The standard linetypes used in technical drawings:

Center lines are used:

- to represent symmetry
- to represent paths of motion
- to mark the centers of circles and the axes of symmetrical parts, such as cylinders and bolts

Break lines are used to show where an object is broken to save drawing space or reveal interior features.

Break lines come in two forms:

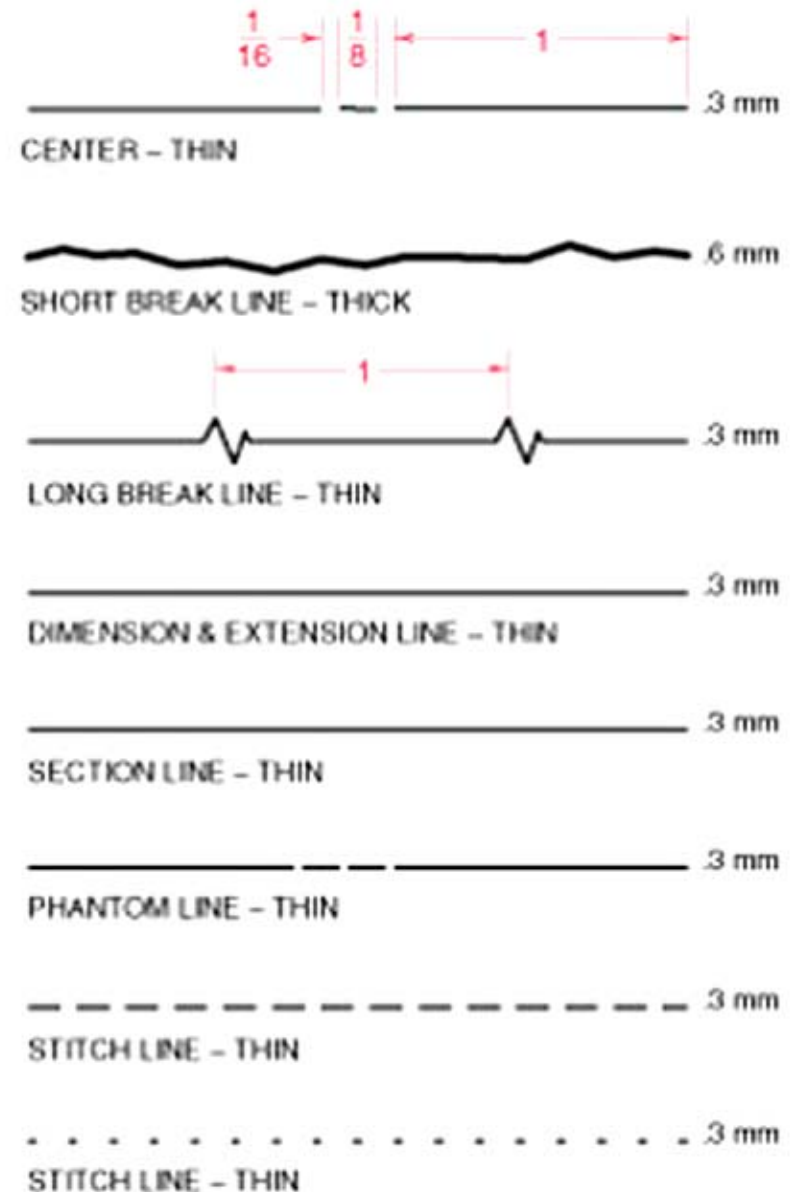
- a freehand thick line and
- a long, ruled thin line with zigzags

Dimension and extension lines are used to indicate the sizes of features on a drawing

Section lines (hatching) are used in section views to represent surfaces of an object cut by a cutting plane.

Phantom lines are used to represent a movable feature in its different positions

Stitch lines - to indicate a sewing or stitching process



Visible lines are used to represent features that can be seen in the current view

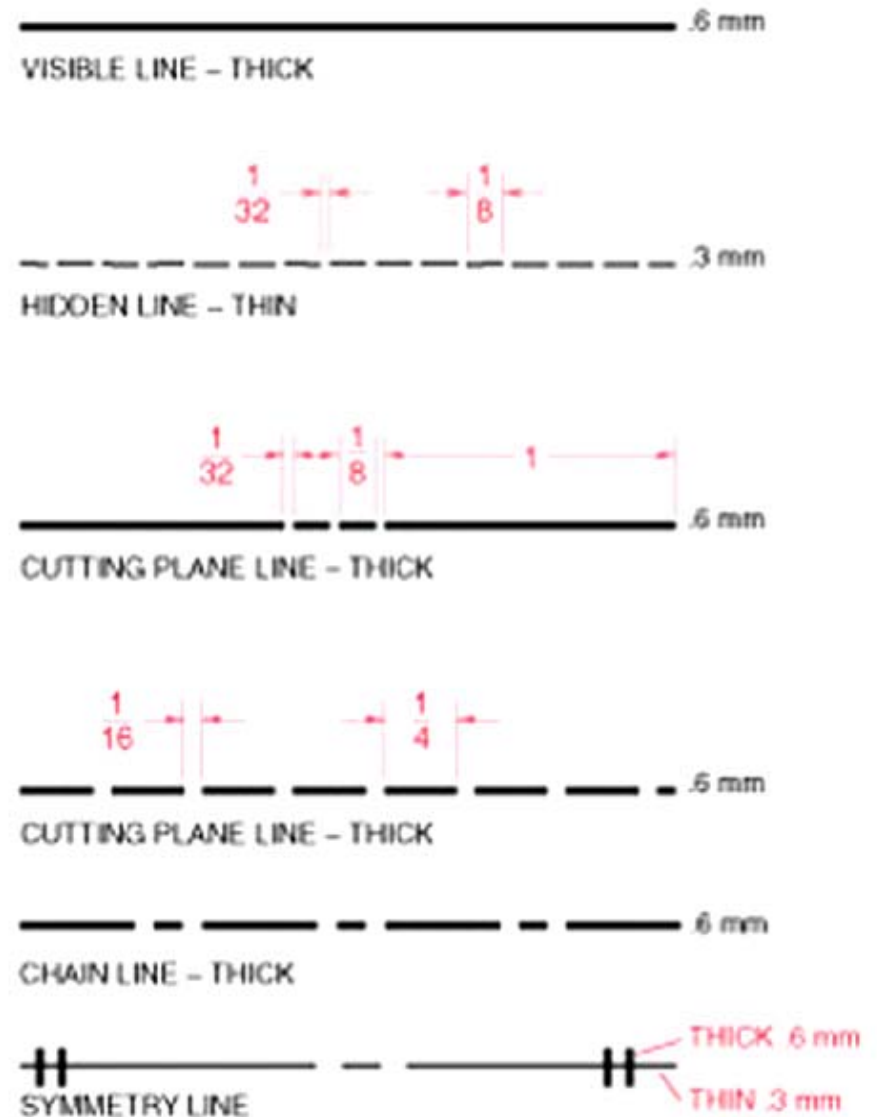
Hidden lines, as you already know, are used to represent features that cannot be seen in the current view

Cutting plane lines are used in section drawings to show the locations of cutting planes

There are also lines used in some particular cases:

Chain lines - to indicate additional treatment for a surface

Symmetry lines - as an axis of symmetry of a particular view



Elements of line technique

According to the Canadian Standard CAN3-B78.1-M83 “Technical Drawings – General Principles”, the thickness of lines should be chosen according to the size and type of the drawing from the following range:

0.25, 0.35, 0.5, 0.7, 1.0, 1.4, and 2.0 mm

However, in many text books (especially, of the US authors) you will come across the thicknesses of lines: **0.3 mm** and **0.6 mm**. In general cases it is good enough since in practical work it is not so easy to see the difference in thickness of 0.1 mm.

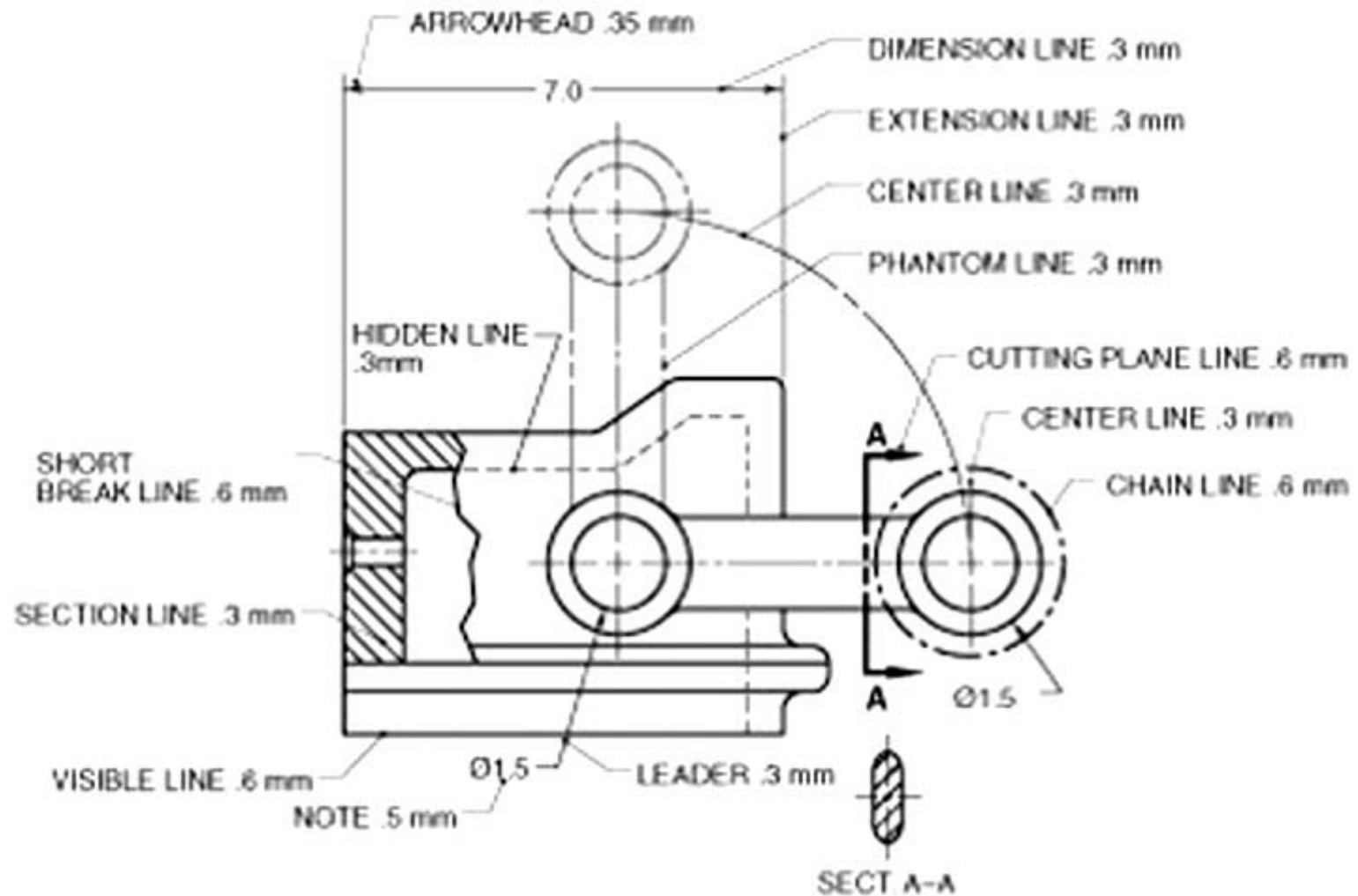
In cases where *other types of lines* are used for special drawings (for example, electrical drawings, pipeline diagrams, or building drawings) the conventions adopted must be clearly indicated by reference to other standards or by notes on the drawings concerned.

However, even for those drawings a simplified representation can be used, where two thicknesses – 0.3 mm and 0.6 mm – are also acceptable.

For all views of one piece or assembly to the same scale, the thickness of the lines should be the same.

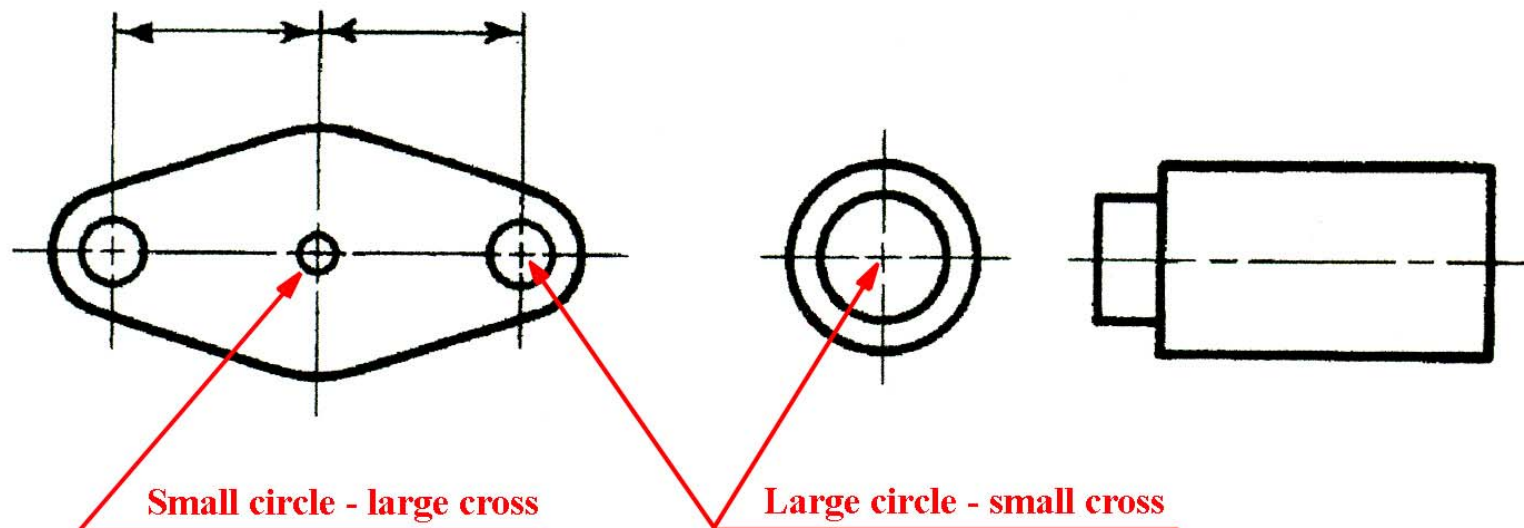
The *minimum space between parallel lines* should never be less than twice the thickness of the heaviest line. It is recommended that these spaces be **0.7 mm** or greater.

The approximate dimensions used to create different linetypes (linetypes when used with CAD):



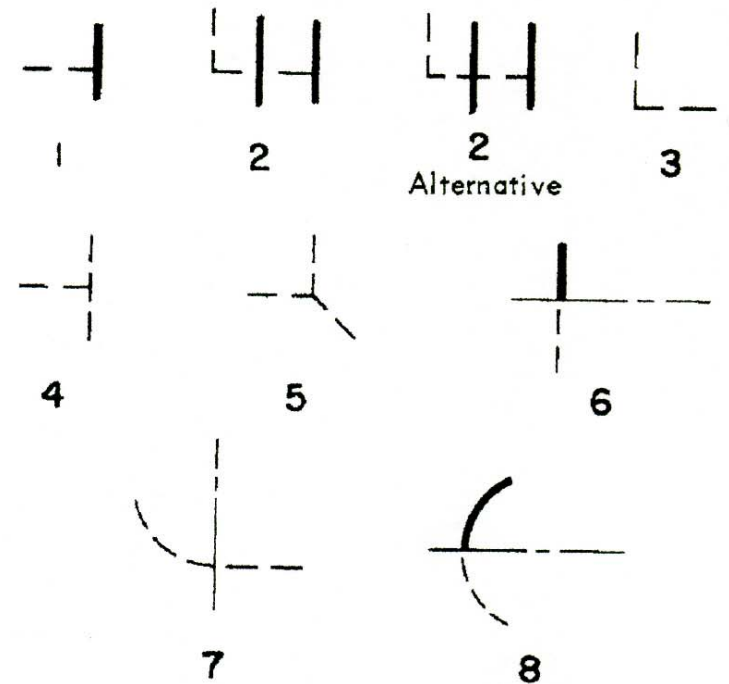
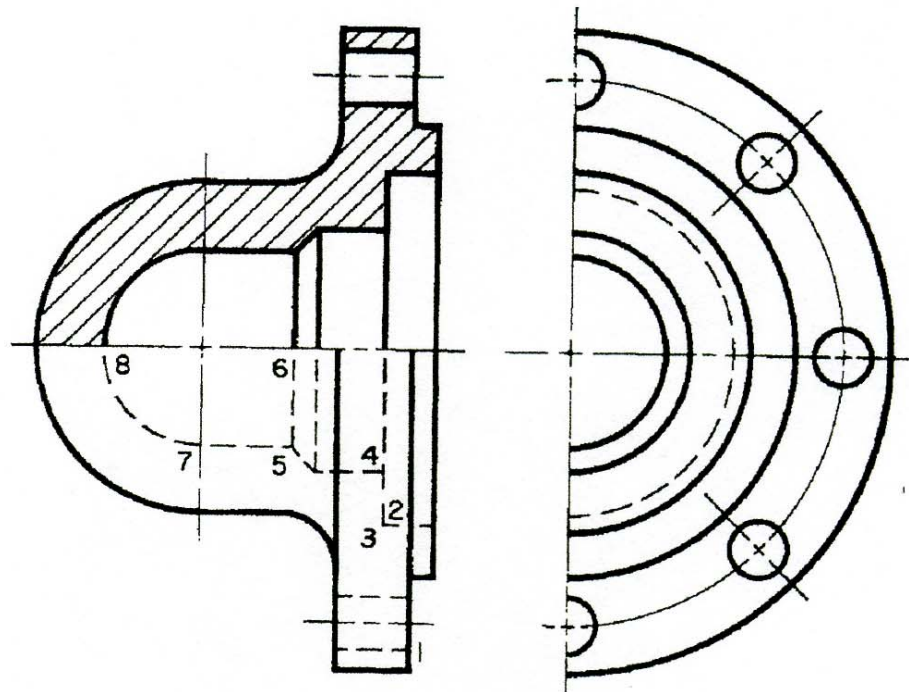
There are two thicknesses of line, thick and thin, and the ratio of the thicknesses must be not less than 2:1. *In practice there are two line weights which are sufficient to follow the standards: a 0.6 mm and a 0.3 mm.* Medium and extra-thick lines should be used only in special cases.

In end views of circular features the point of intersection of two centre lines should be shown by two intersecting short dashes, except for very small circles. These techniques are shown here:



Lines depicting *hidden features* and *phantom details* should always begin and end with a dash in contact with the lines at which they begin and end.,

Exception: when such a dash would form a continuation of a visible detail line. Dashes should join at corners, and arcs should start with dashes at the tangent points.

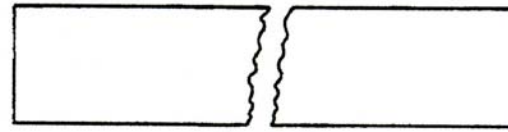


Break Line are used to shorten the view of long uniform or tapered sections, or when only a partial view is necessary, and are used on both detail and assembly drawings.

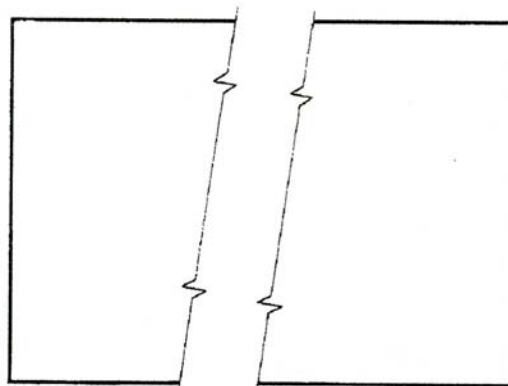
The thin line with freehand zig-zags is recommended for long breaks and may be used for solid details or for assemblies containing open space.

Thick freehand lines are used for short breaks, and the jagged line for wood parts.

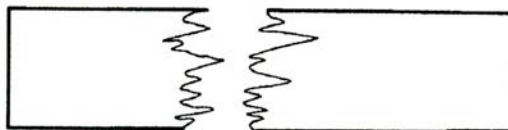
The special thick break lines shown for cylindrical and tubular parts are useful when an end view is not shown, but otherwise the thick freehand break line is adequate.



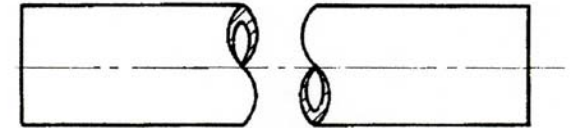
Short Break



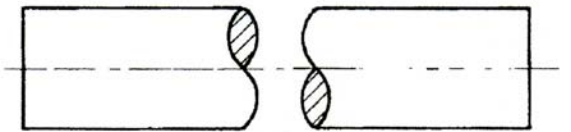
Long Break



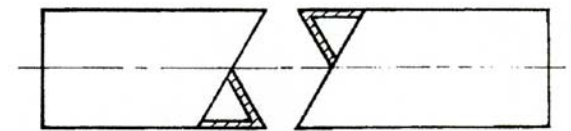
Wood



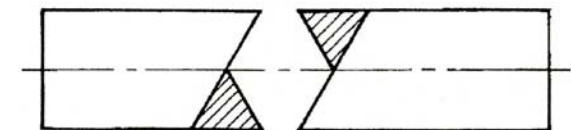
Tubular Round



Solid Round



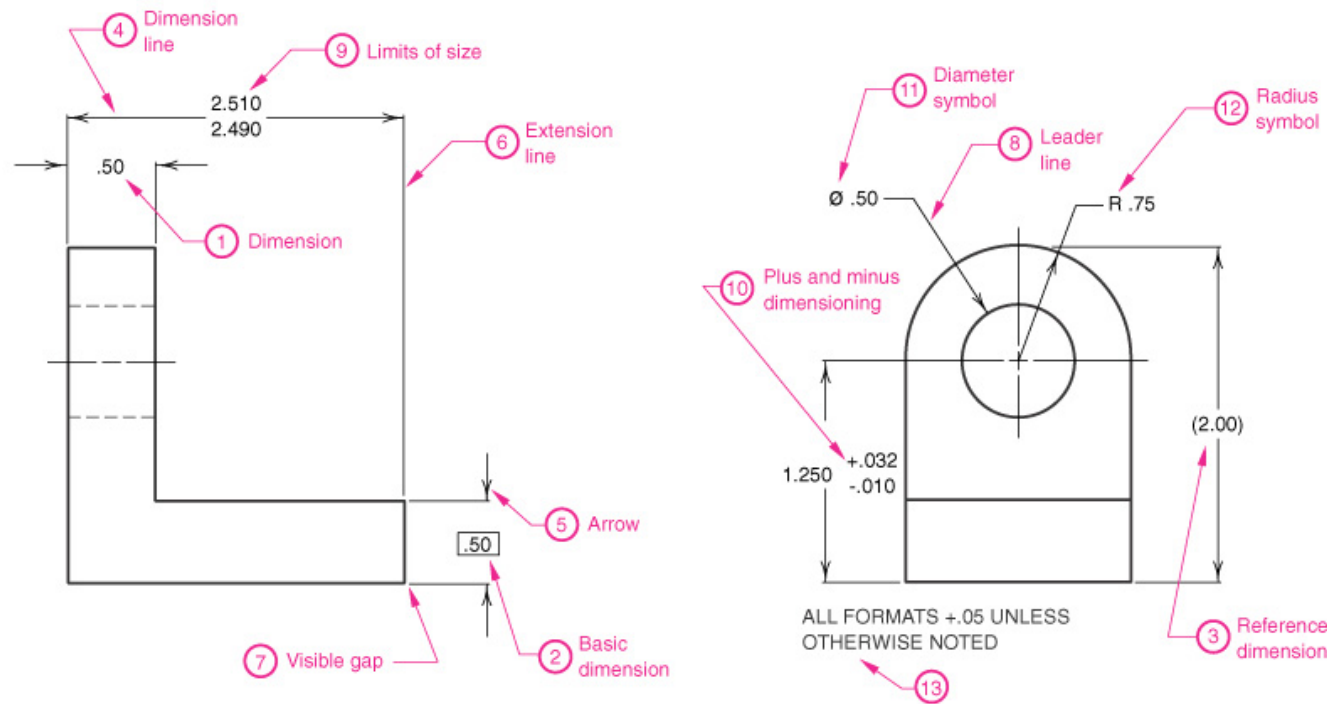
Tubular Rectangular or Square



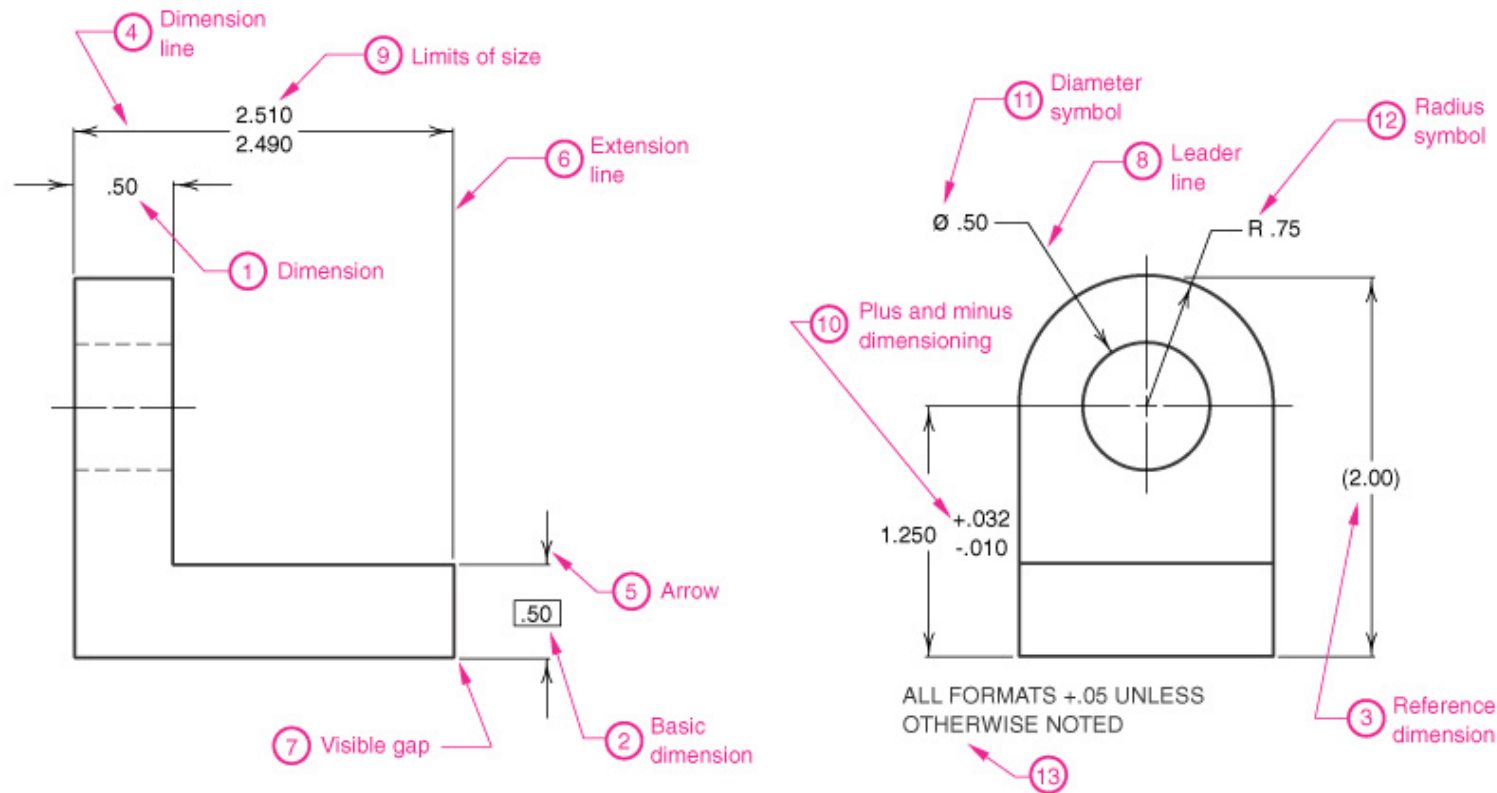
Solid Rectangular or Square

Dimensioning practice

- ❖ Once the shape of a part is defined with an orthographic drawing (i.e. in projections), the size information is added in the form of **dimensions**.
- ❖ Dimensioning a drawing also identifies the **tolerance** (or accuracy) required for each dimension.



1. **Dimension** — the numerical value that defines the size, shape, location, surface texture, or geometric characteristic of a feature.
2. **Basic dimension** — a numerical value defining the theoretically exact size, location, or orientation relative to a coordinate system. Basic dimensions are enclosed in a rectangular box & have no tolerance.
3. **Reference dimension** — a numerical value enclosed in parentheses, provided for information only.



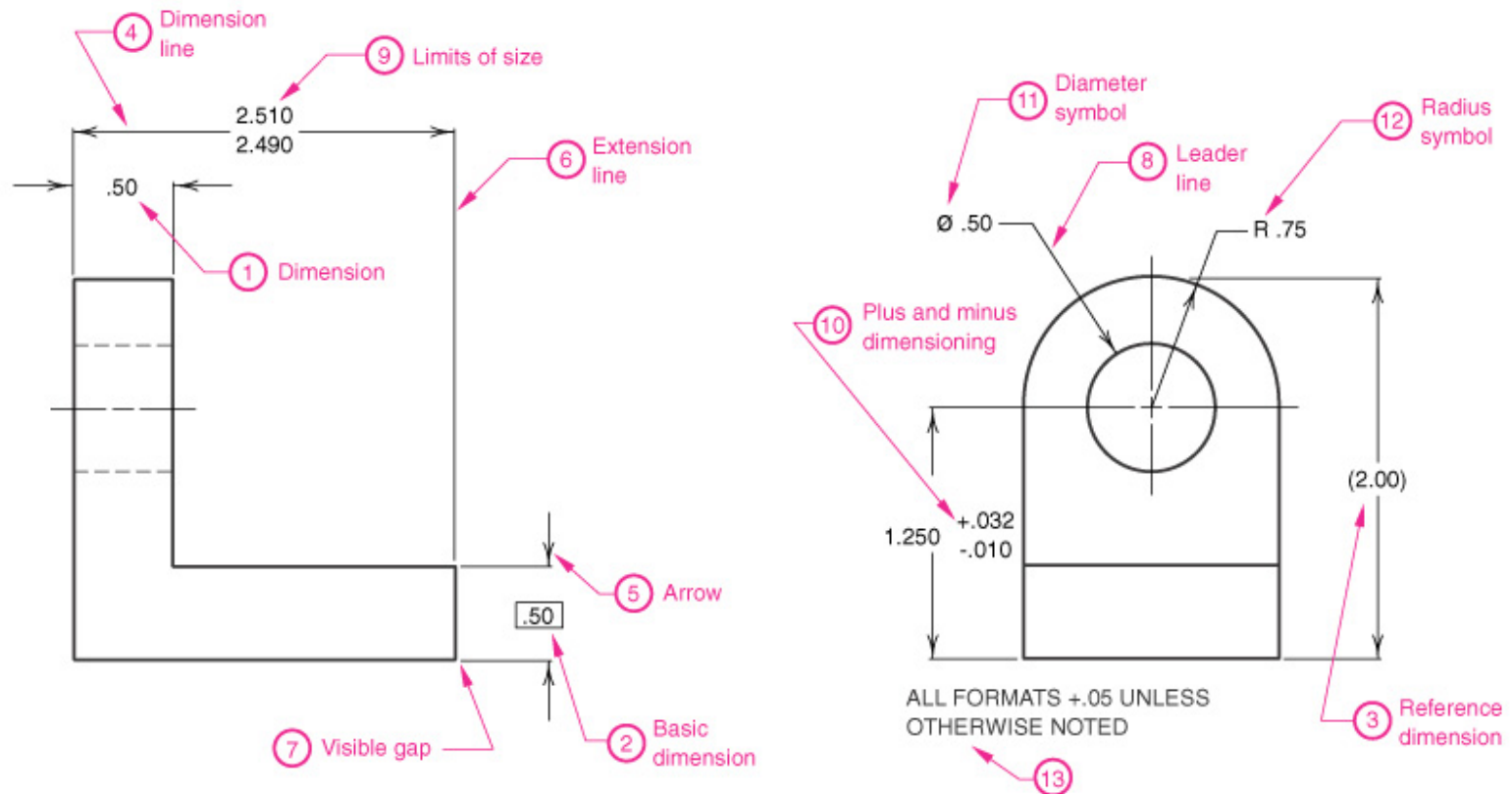
4. Dimension line — a thin, solid line that shows the extent and direction of a dimension.

5. Arrows — symbols at the ends of dimension lines showing the limits of the dimension, leaders, and cutting plane lines.

6. Extension line — a thin, solid line perpendicular to a dimension line, indicating which feature is associated with the dimension.

7. Visible gap — there should be a visible gap of 1 mm between the feature's corners and the end of the extension line.

8. Leader line — a thin, solid line with arrow drawn under an angle & indicating the feature with which a dimension or note is associated.



9. Limits of size — the largest acceptable size and the minimum acceptable size of a feature.

10. Plus and minus dimension — the allowable positive and negative variance from the specified dimension.

11. Diameter symbol — a symbol indicating that the dimension shows the diameter of a circle. The symbol used is the Greek letter phi ϕ .

12. Radius symbol — a symbol indicating that the dimension shows the radius of a circle. The radius symbol used is the capital letter R.

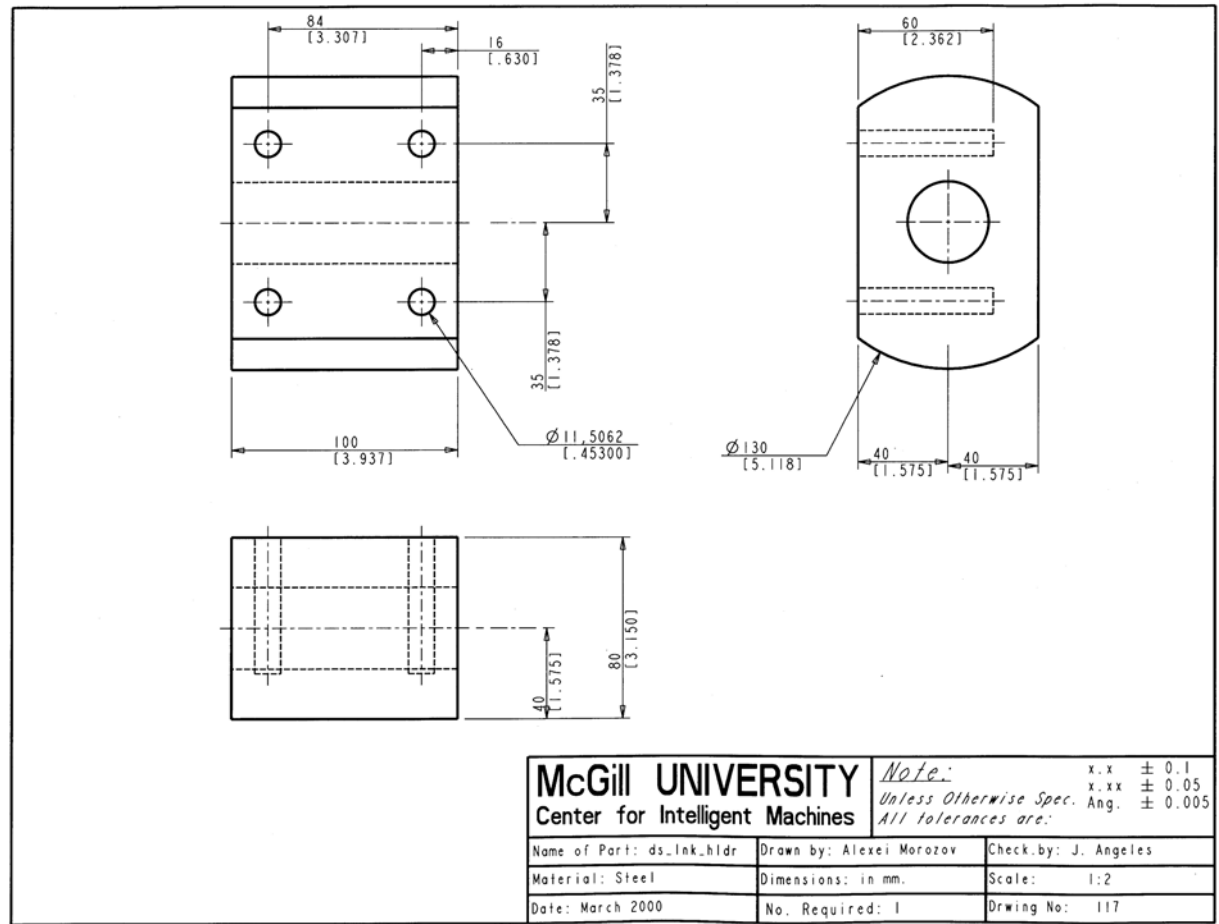
13. Tolerance — the amount that a particular dimension may vary.

Basic concepts and principles

- The **unit of measurement** for dimensioning should be in accordance with the policy of the user. On a drawing for use in American industry for manufacturing, all dimensions are in *inches*, unless otherwise stated.
- Most countries outside of the United States, including Canada, use the metric system of measure, or the international system of units (SI), which is based on the meter.

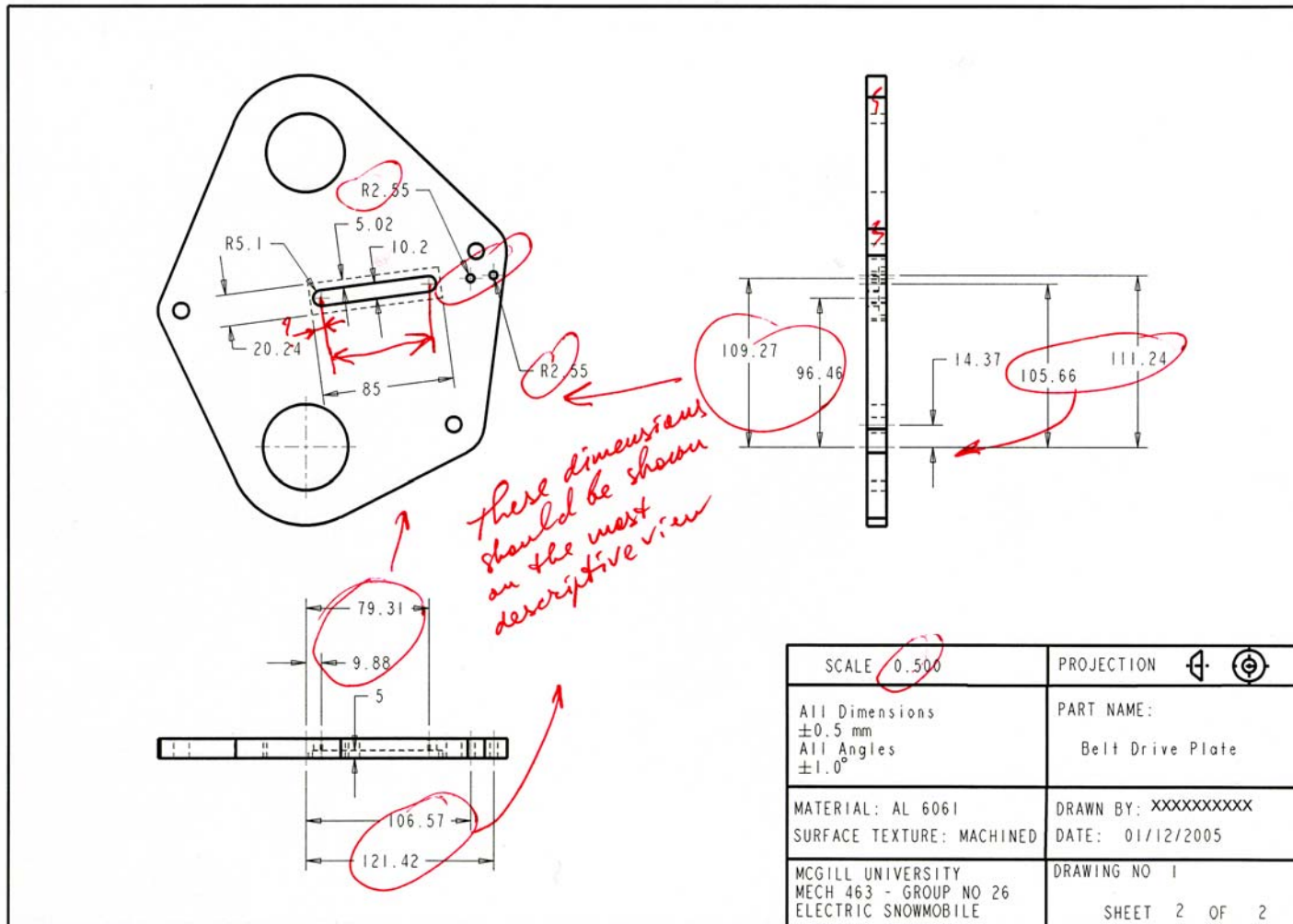
The common metric unit of measure on engineering drawings is the *millimeter*, abbreviated as *mm*.

Sometimes, you will have to place dimensions in two units simultaneously!



Dimension Text Placement

Dimensions should be placed in the most descriptive view of the feature

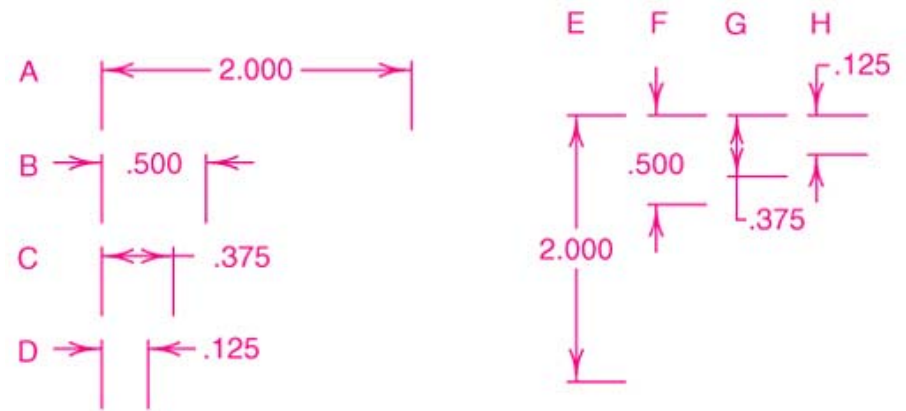


If the view is overcrowded with different dimensions, two and more separate drawings with the same view can be produced.

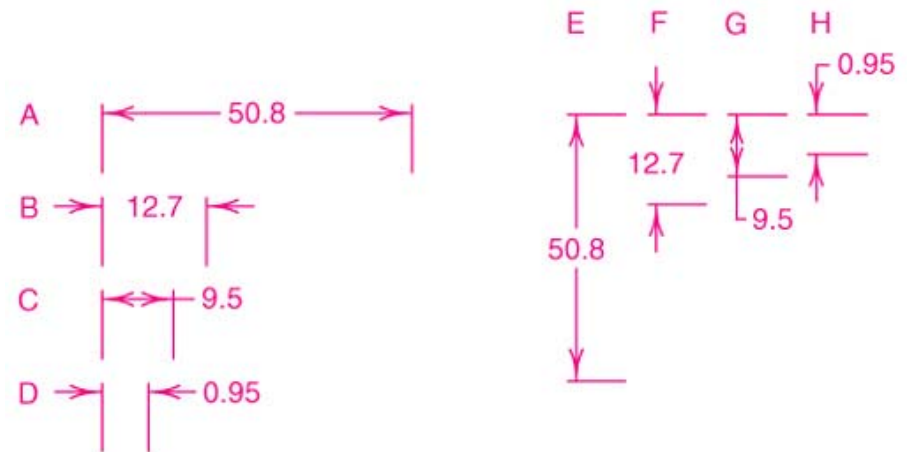
Dimension Text Placement

Text of the dimensions can be placed in different ways:

- dimension *inside* the extension lines, with arrows inside or outside; and
- dimension *outside* the extension lines, with arrows again inside or outside.

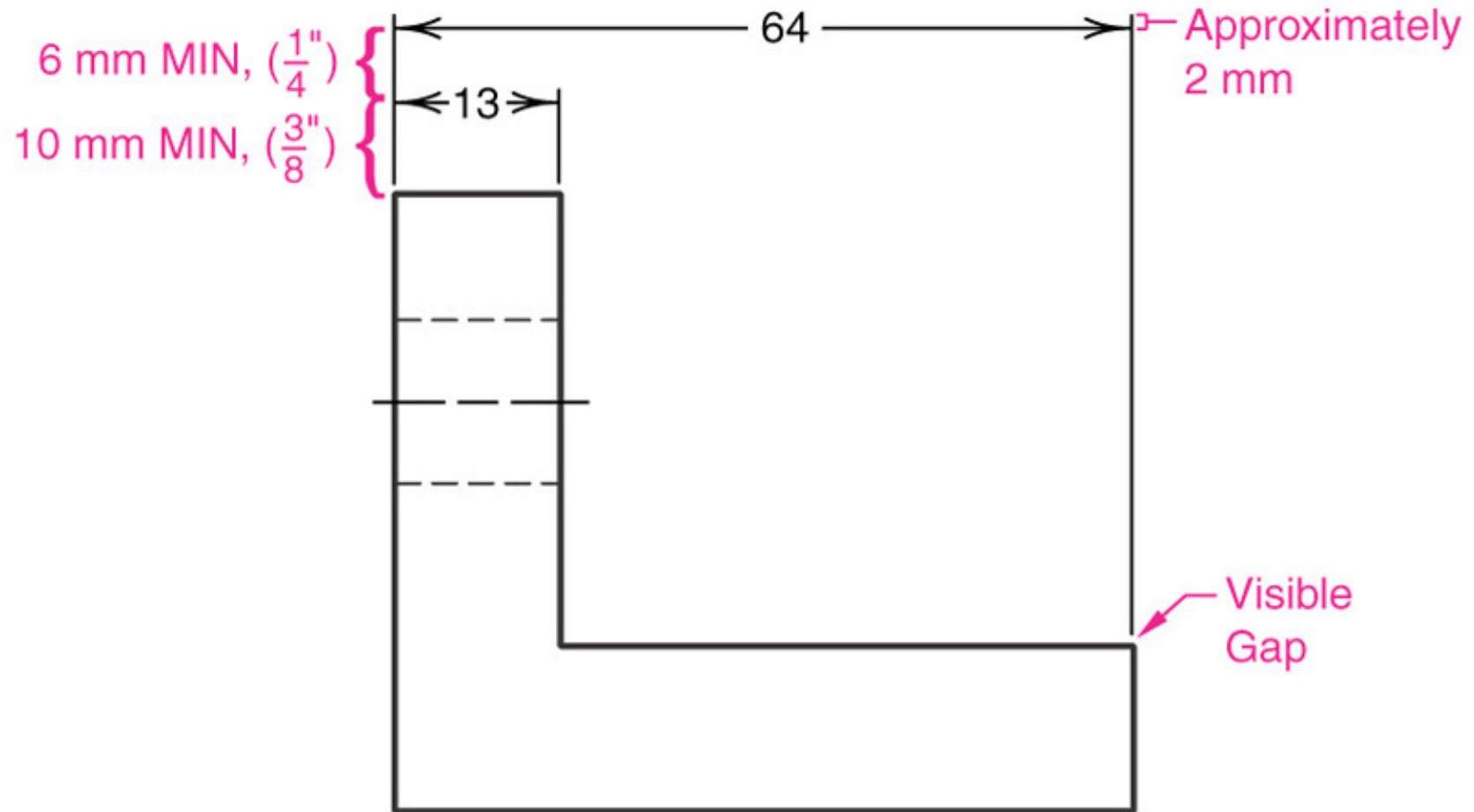


Decimal dimensioning



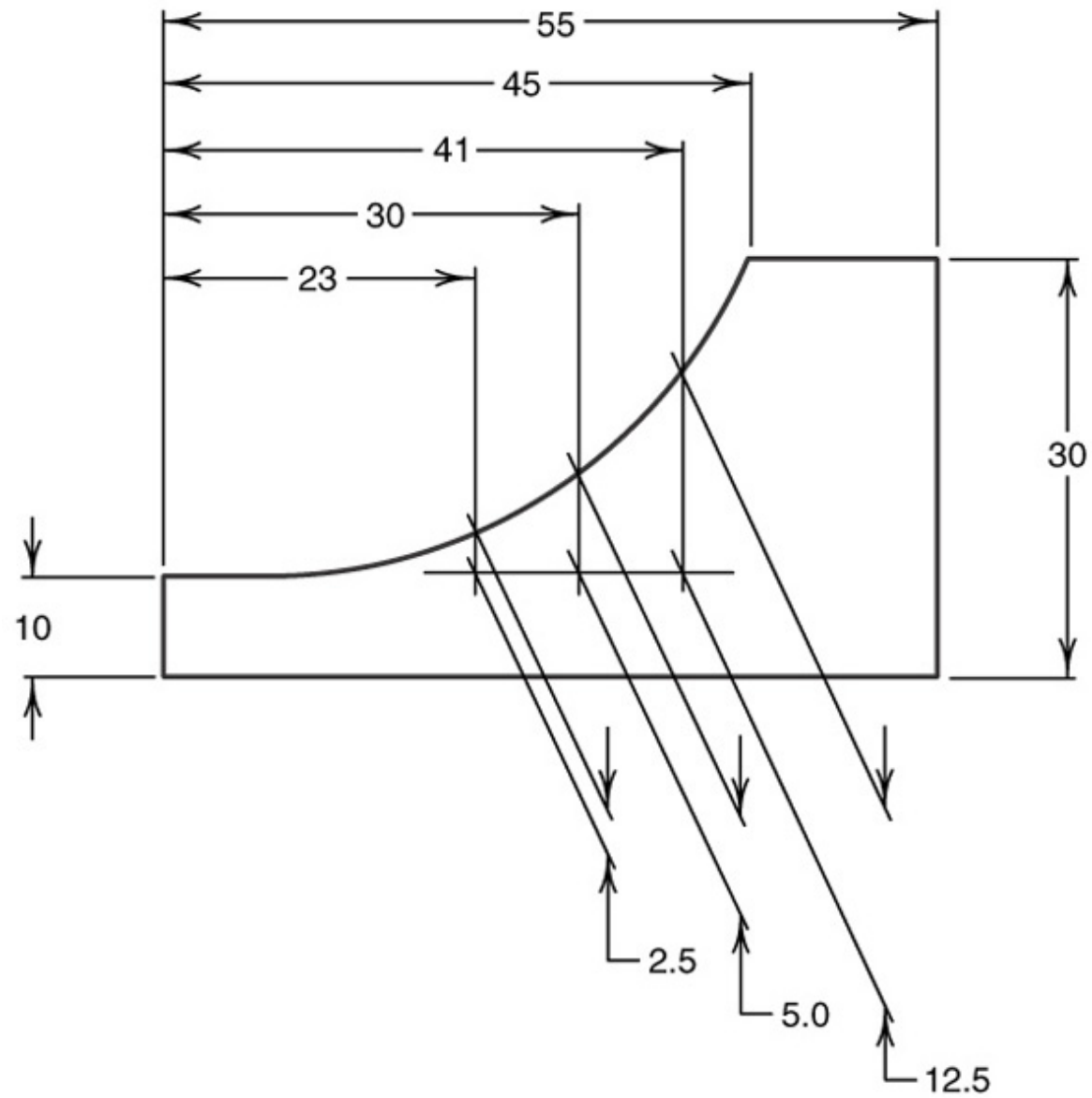
Millimeter dimensioning

The *minimum distance* from the object to the first dimension is 10 mm ($\frac{3}{8}$ "). The minimum spacing between dimensions is 6 mm ($\frac{1}{4}$ "). These values may be increased where appropriate.

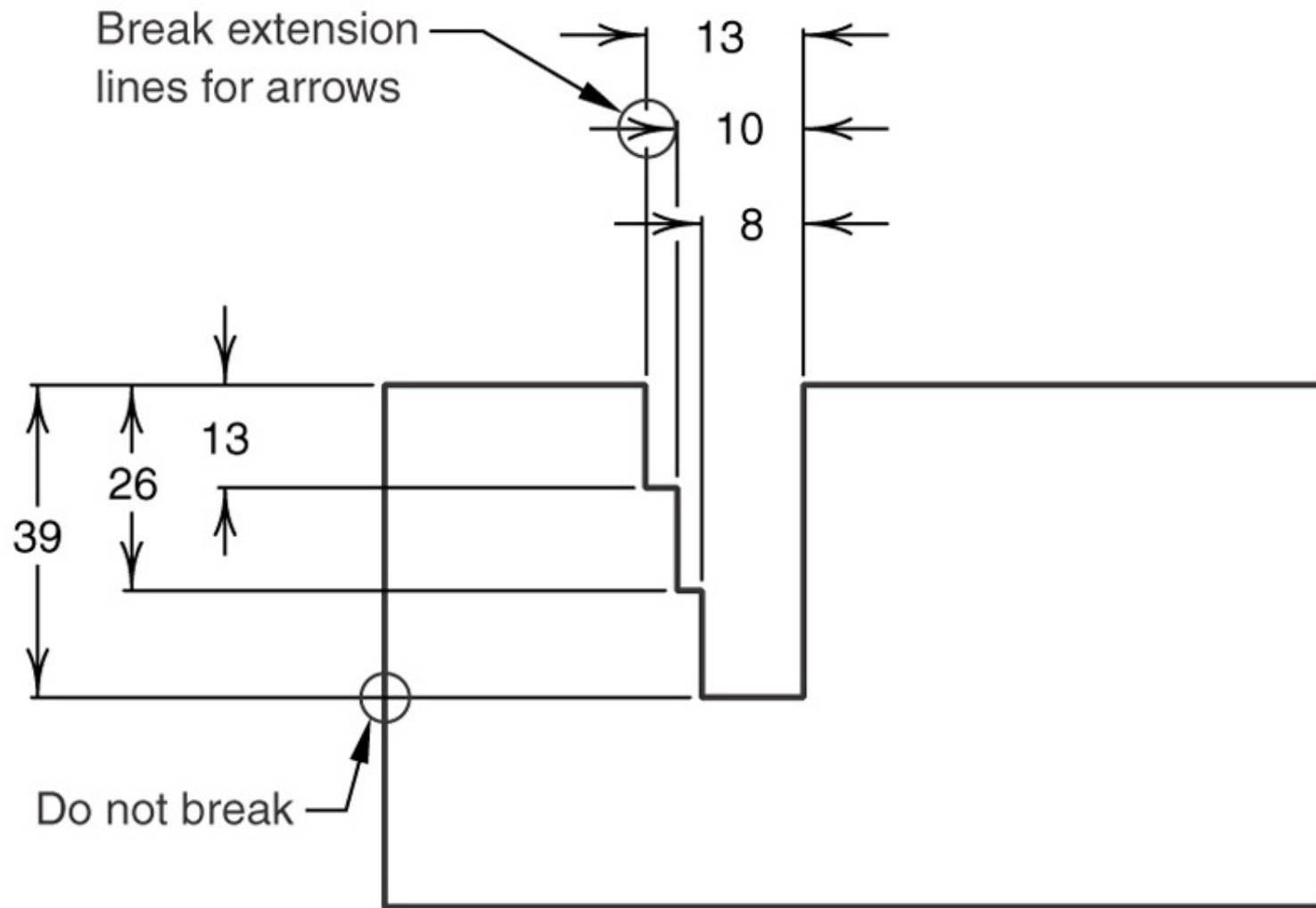


There should be a visible gap between the extension line and the feature to which it refers. Extension line should extend about 2 to 3 mm ($\frac{1}{8}$ ") beyond the last dimension line.

Where space is limited extension lines may be drawn at an angle:

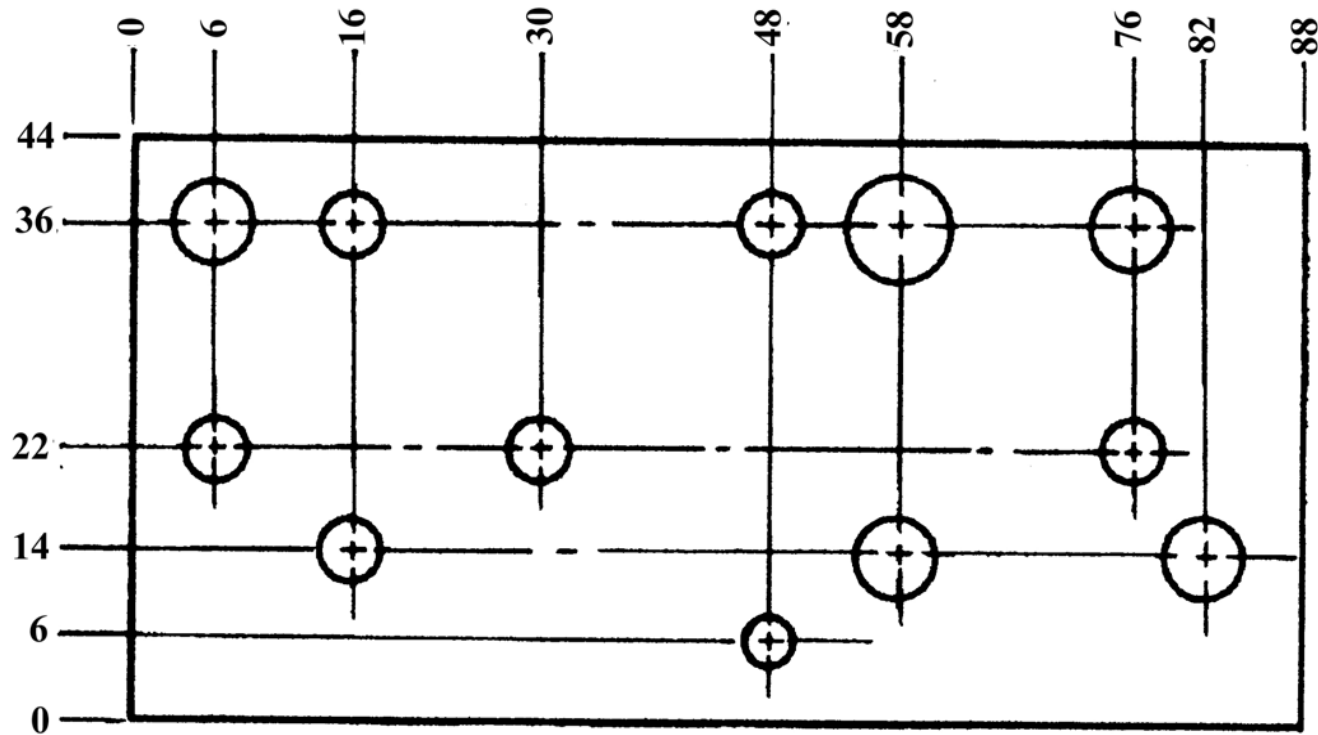


Extension lines should not cross dimension lines, and they should avoid crossing other extension lines whenever possible. When extension lines cross object lines or other extension lines, they should not be broken. When extension lines cross or are close to arrowheads, they should be broken for the arrowhead.



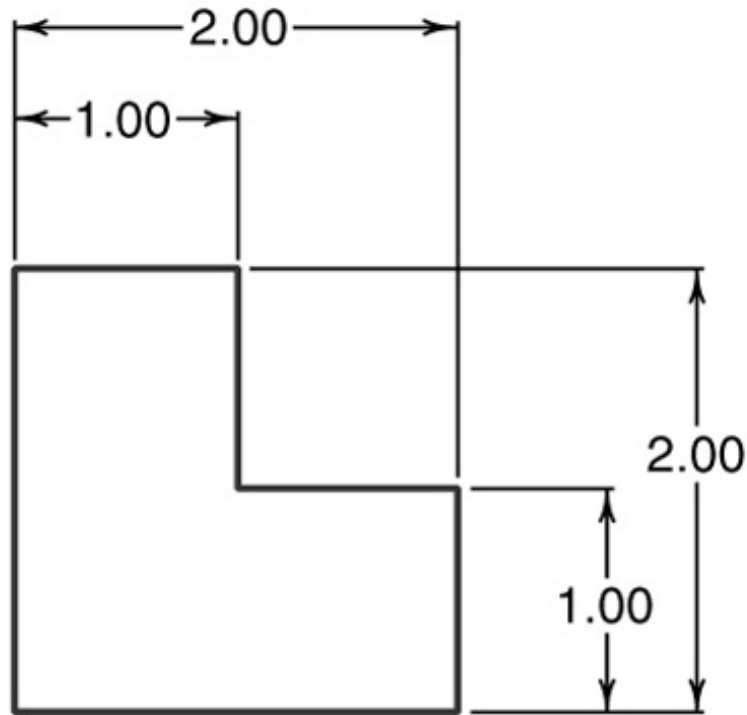
Arrowless dimensioning

To avoid having a large number of dimensions extending away from the part, arrowless dimensioning may be used.

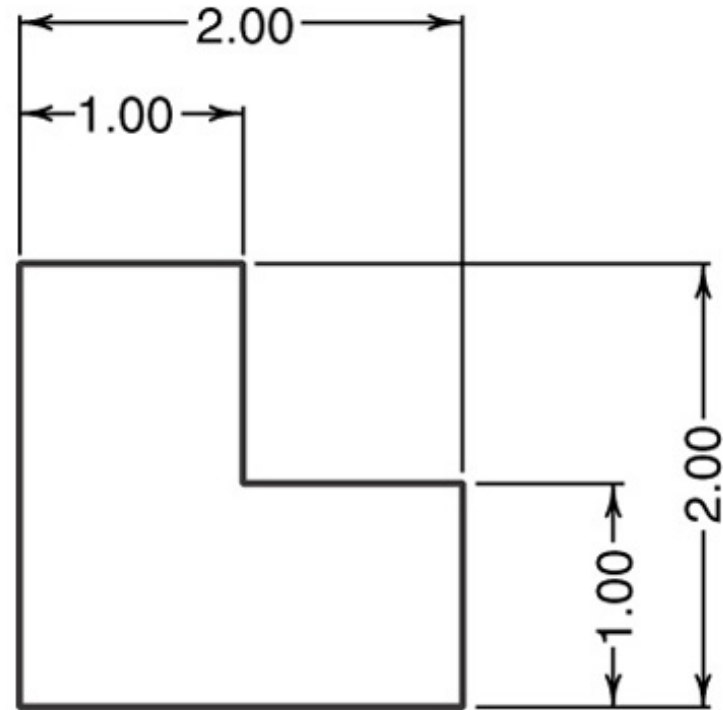


- the “zero” lines represent the vertical and horizontal datum lines,
 - each of the dimensions shown without arrowheads indicates the distance from the zero line.
-
- There shall never be more than one zero line in each direction.
 - Arrowless dimensioning is used for locational dimensions for a series of features such as holes and slots.

All dimensions and note text must be oriented to read from the bottom of the drawing. This is called **unidirectional dimensioning**:



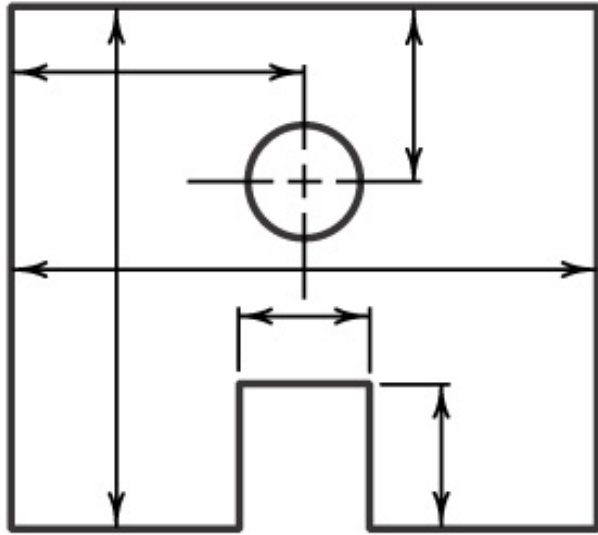
Unidirectional
Current standard



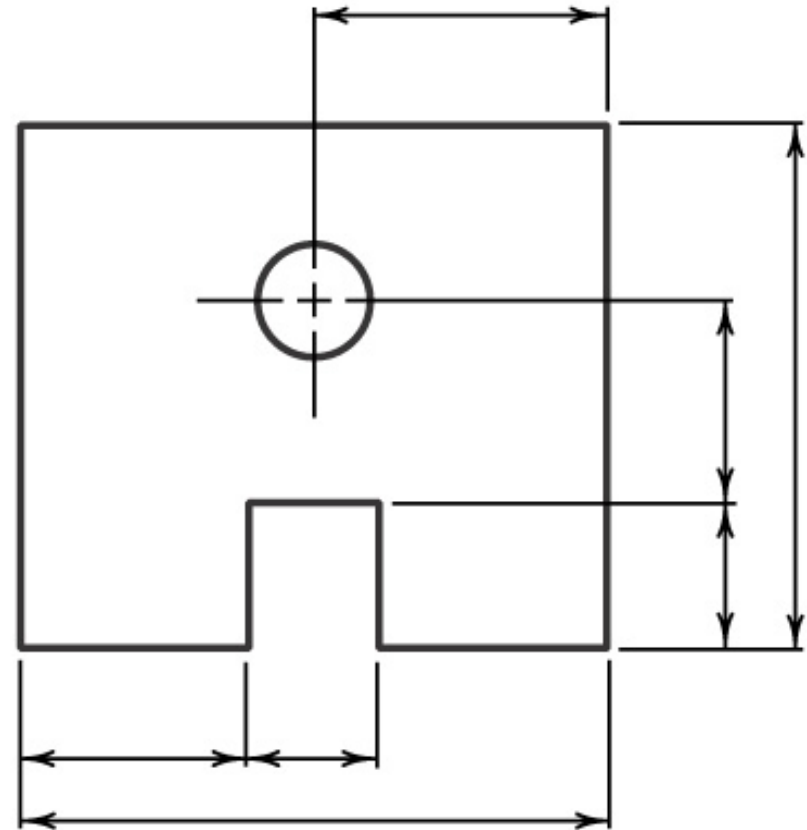
Aligned
Old standard

The **aligned method** (where text is placed parallel to the dimension line) may be seen on older drawings or on architectural drawings but is not approved by the current ANSI standard.

Dimensions must be kept outside the boundaries of views, wherever practical:



(A) No!



(B) Yes

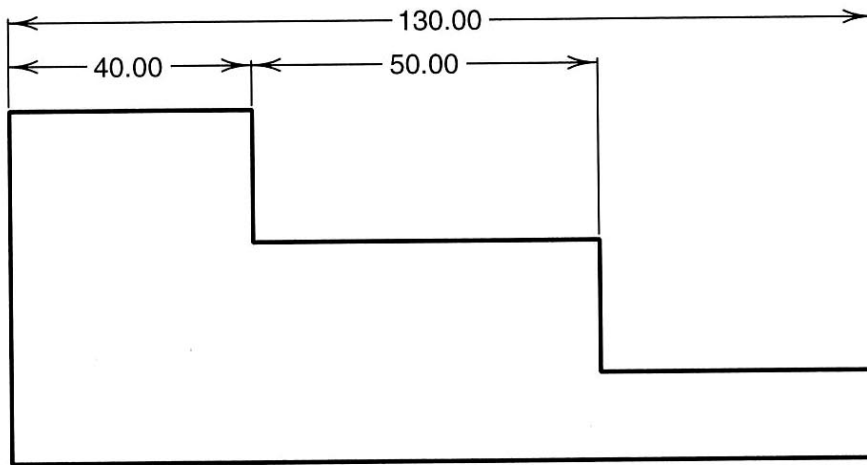
Dimensions may be placed within the boundaries where extension or leader lines would be too long or where clarity would be improved.

Group Dimensions

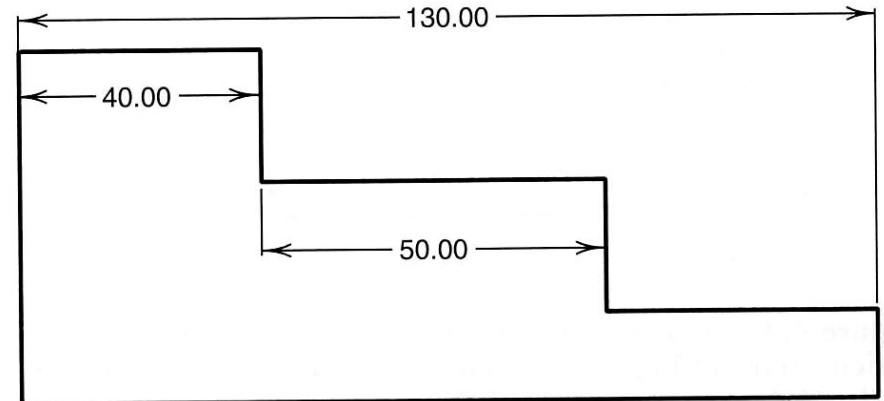
In standard practice, dimensions are grouped on a drawing for uniform appearance.

Avoid using object lines as extension lines for a dimension.

Dimensions can be placed in parallel or in series – one after another.

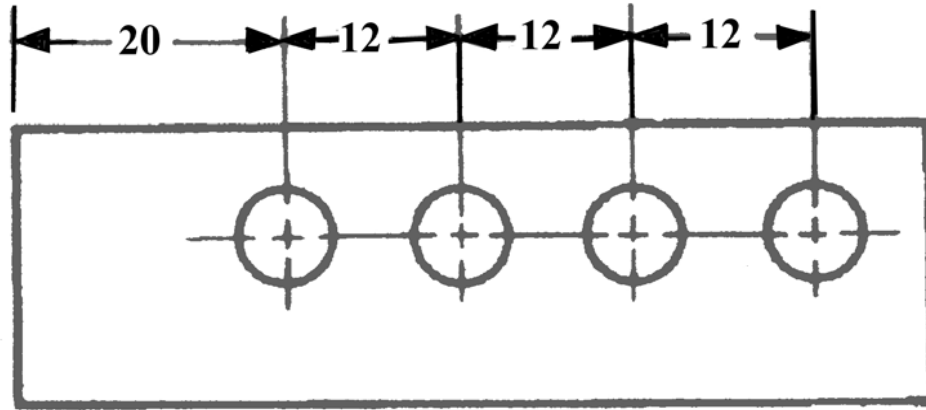


(A) Yes



(B) No!

When a series of dimensions is applied on a point-to-point basis, it is referred to as **chain dimensioning**.

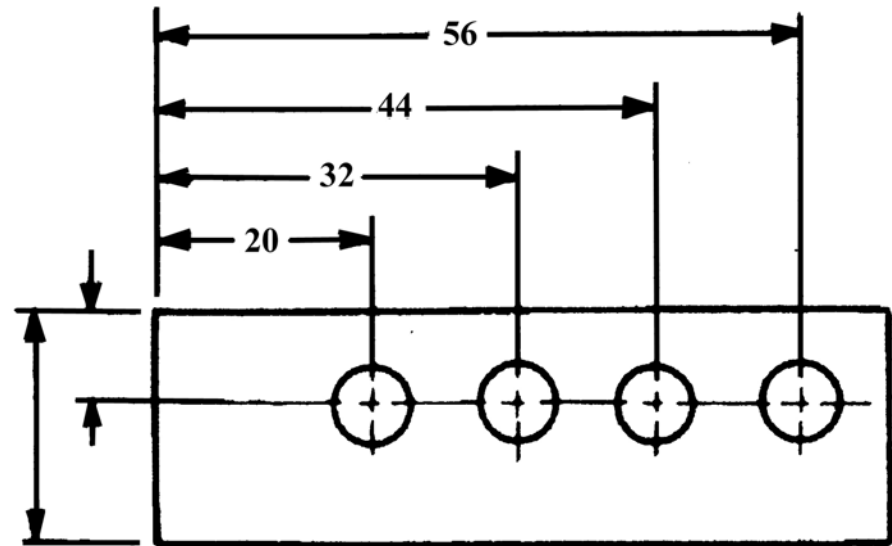


Disadvantage of this system:

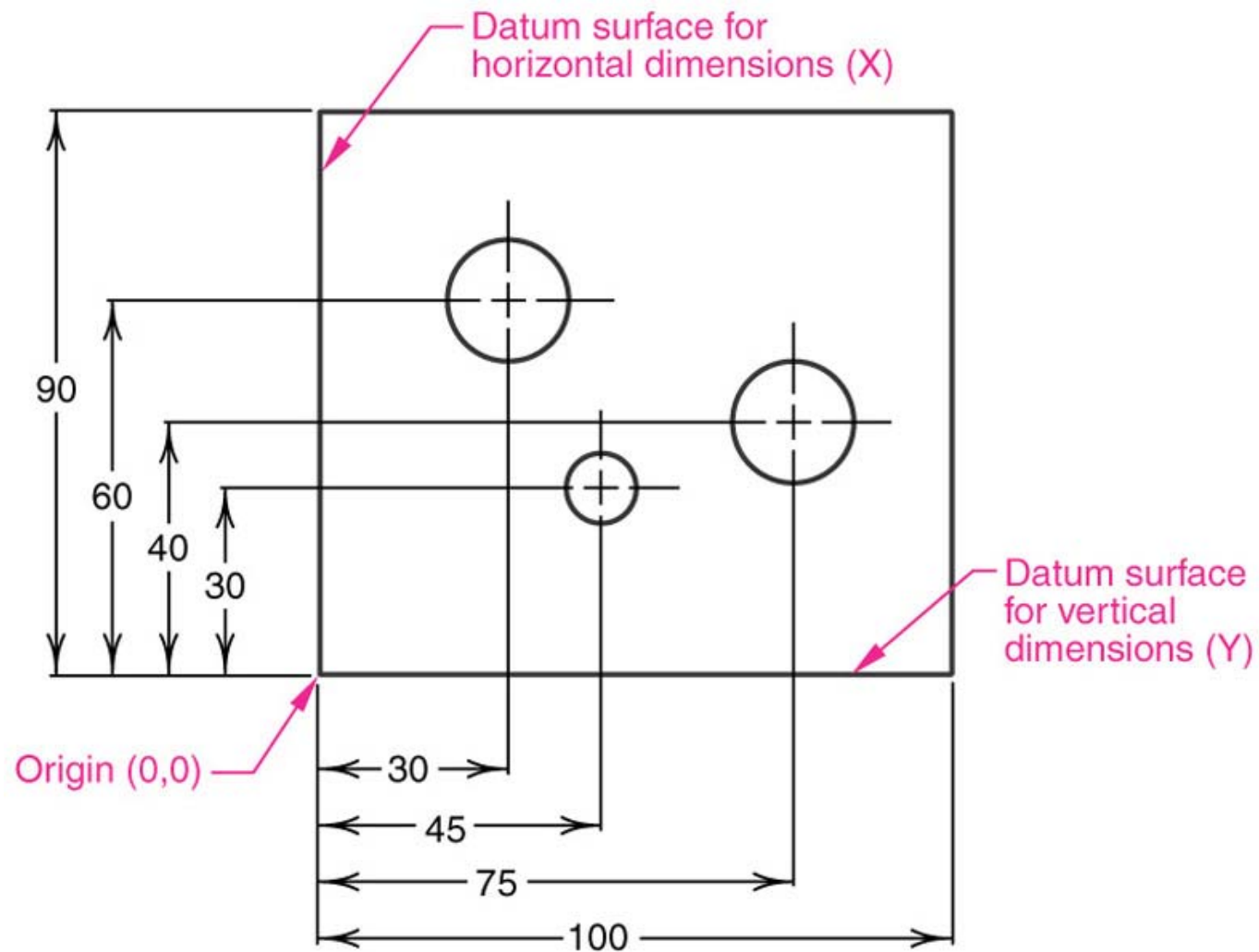
it may result in an undesirable accumulation of tolerances between individual features.

When several dimensions start from a common reference point or line, the method is referred to as **common-point dimensioning**.

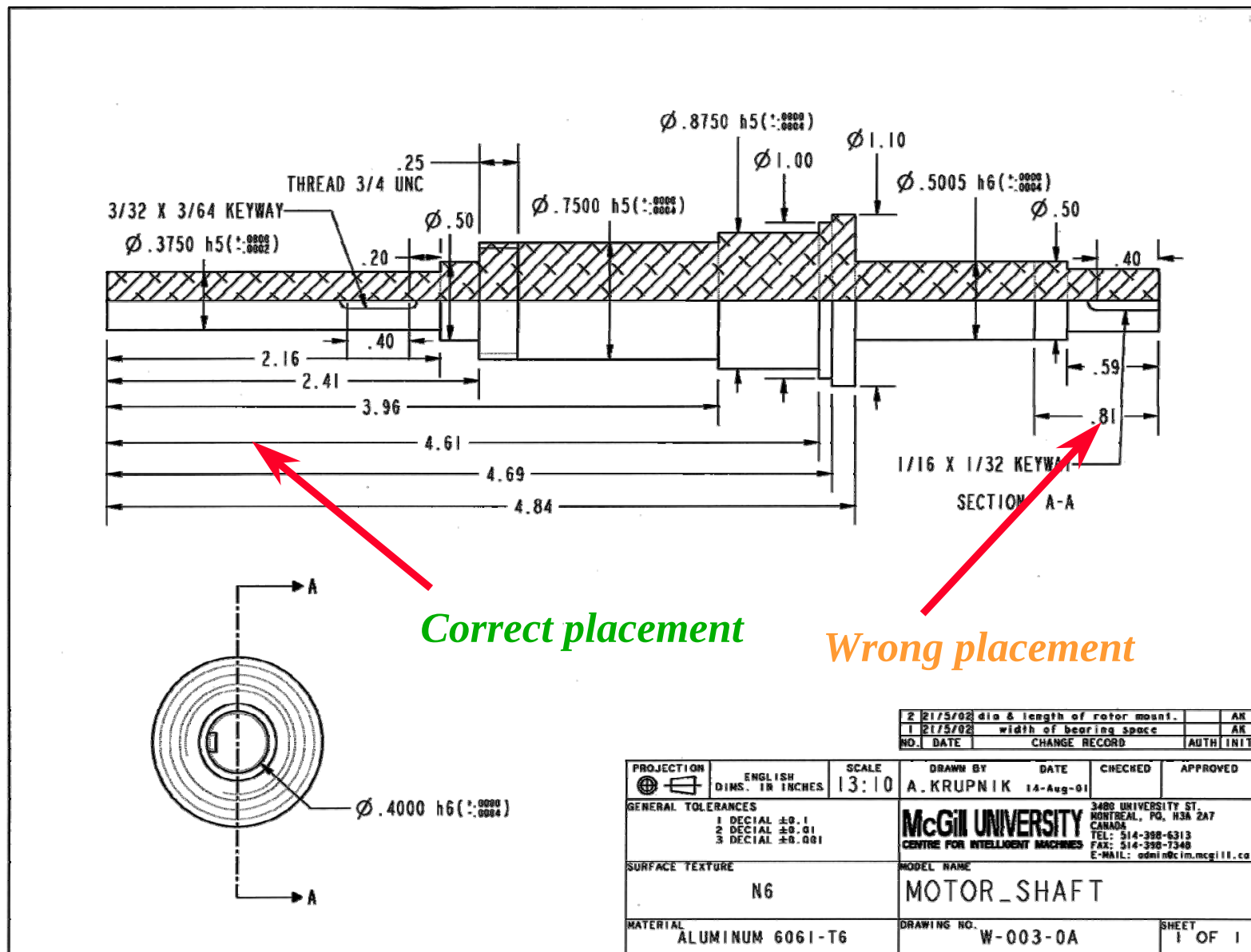
The reference line may be an extension of the outline of the origin, a base line, or a system line.



- The first dimension in a row should be stuck to a base line
- The base surface is the external surface of the part, which has to be machined first

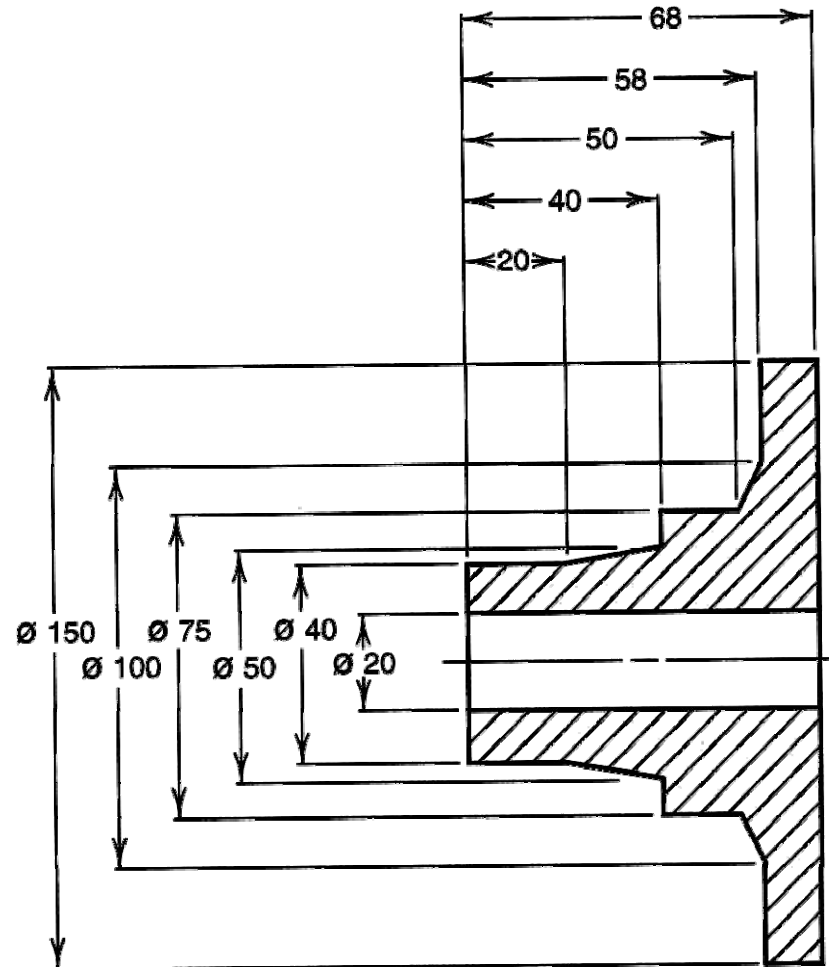


Dimensions can be placed in parallel or in series, but in all cases they should be related to the base (datum) surface.



Stagger Dimensions

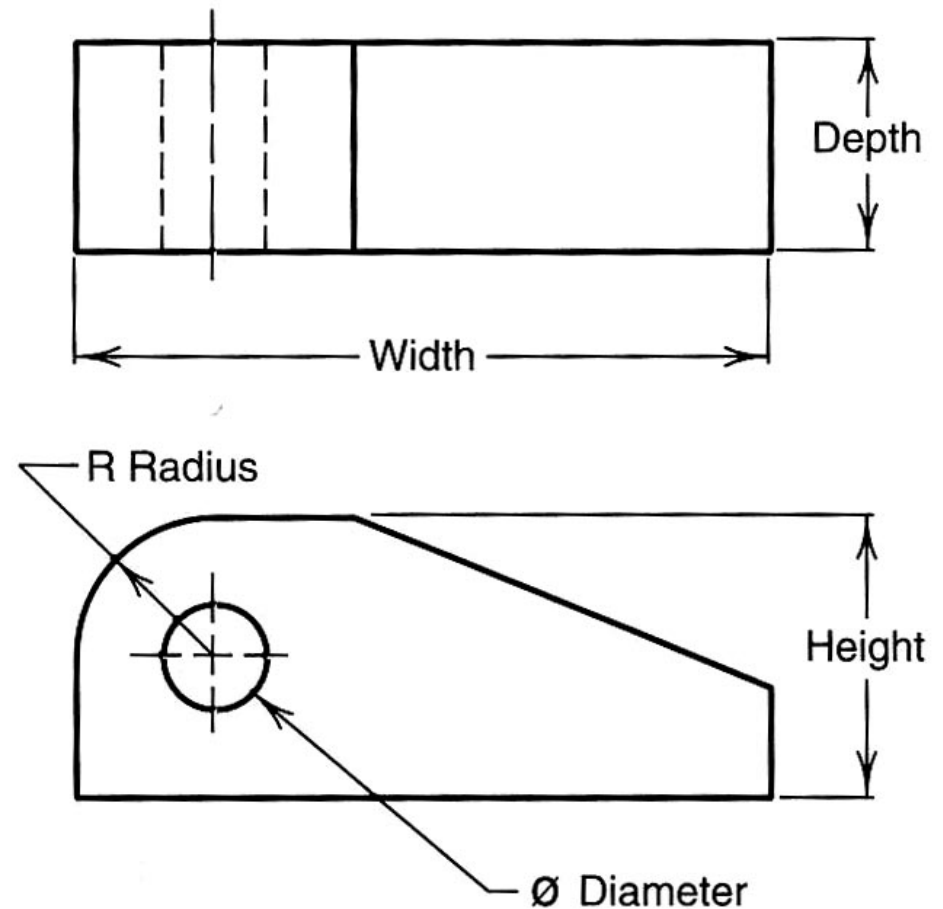
The general practice is to stagger the dimension text on several parallel dimensions



Size Dimension

The dimensions can be classified by types of size:

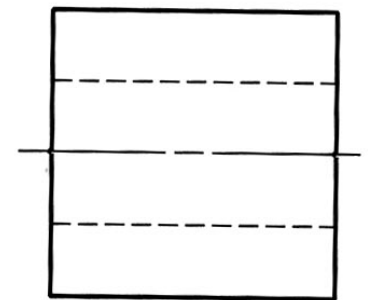
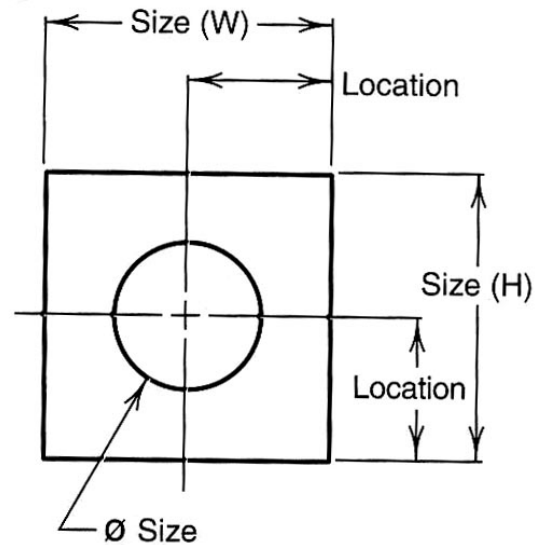
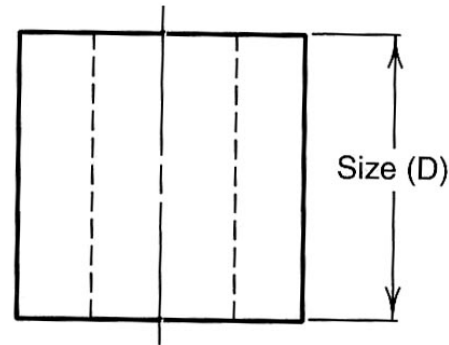
- o *Horizontal* — the left-to-right distance relative to the drawing sheet. Here the *width* is the only horizontal size dimension.
- o *Vertical* — the up and down distance relative to the drawing sheet. Here the *height* and the *depth* are both vertical dimensions, even though they are in two different directions on the part.
- o *Diameter*—the full distance across a circle, measured through the center.
- o *Radius*—the distance from the center of an arc to any point on the arc. Radius is usually used on arcs less than half circles.



Location & Orientation Dimensions

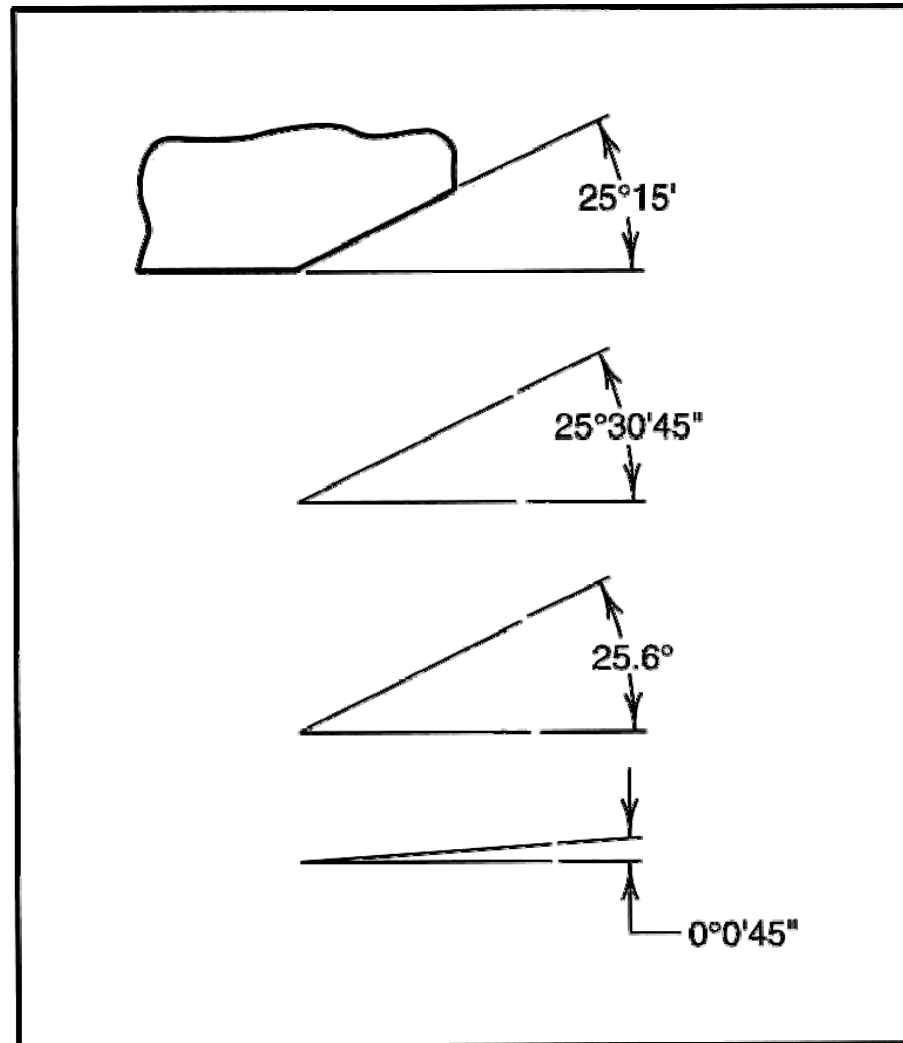
The dimensions can be classified by type of location or orientation:

- o *Horizontal position* – locates the position of the feature in horizontal direction with respect to a base
- o *Vertical position* – locates the position of the feature in vertical direction with respect to a base
- o *Angle* – gives the angle between the horizontal plane and the sloping surface

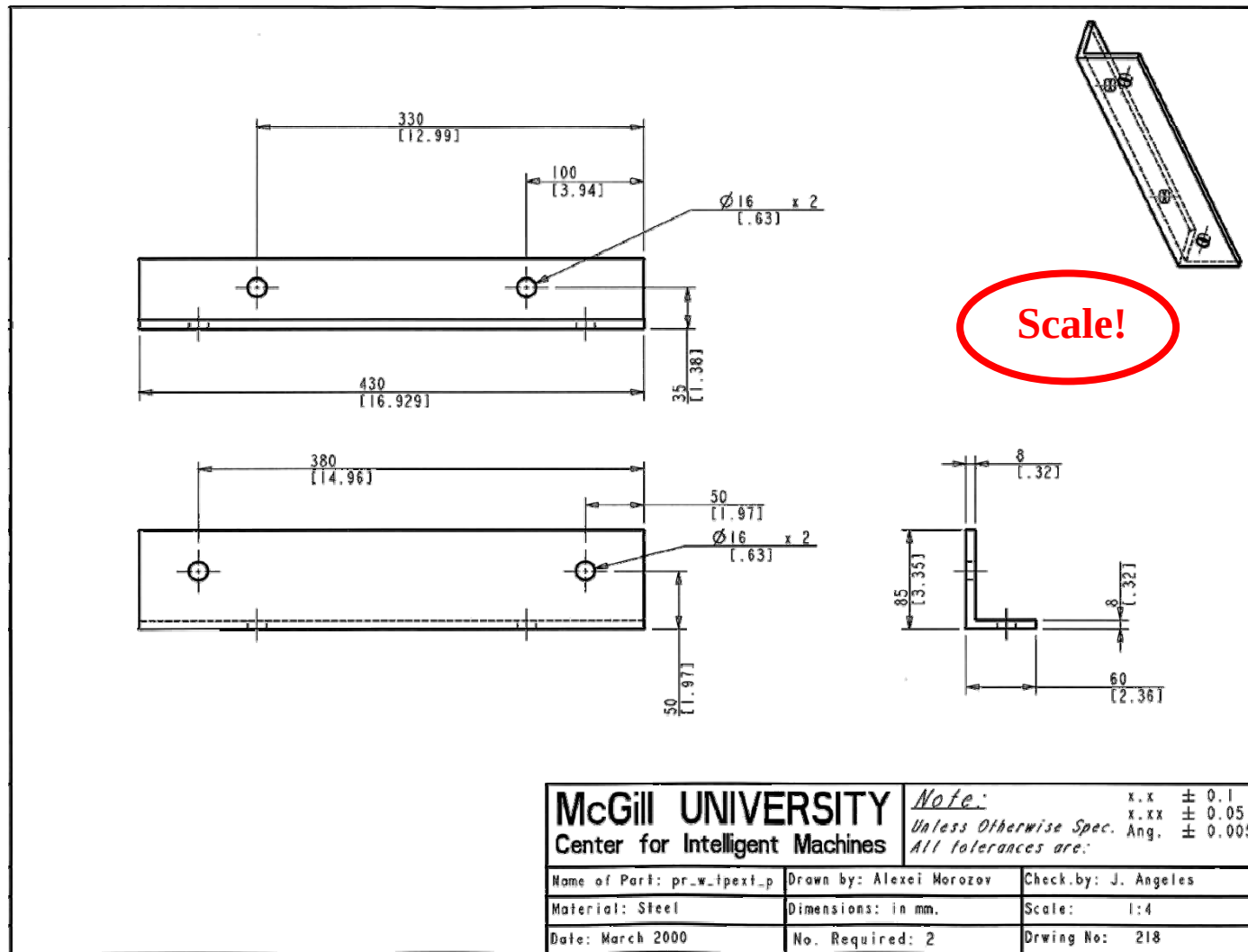


Angular units

- ❑ Angular dimensions are shown either in decimal degrees or in degrees, minutes, and seconds
- ❑ Where only minutes and seconds are specified, the number of minutes or seconds are preceded by the 0°



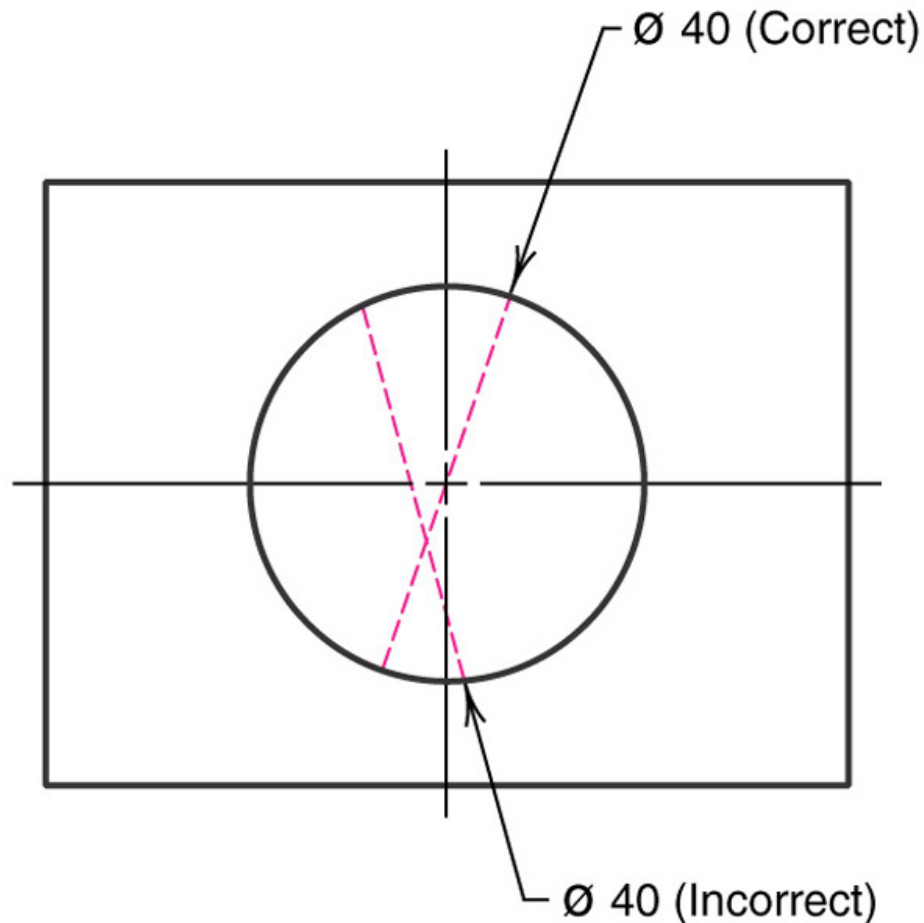
It is also good to have an isometric view of the part on its drawing. Not mandatory but preferable for complex parts.



No dimensions should be shown on isometric view. Scale must be shown if it differs from the general one.

Detail dimensioning (How to dimension various features)

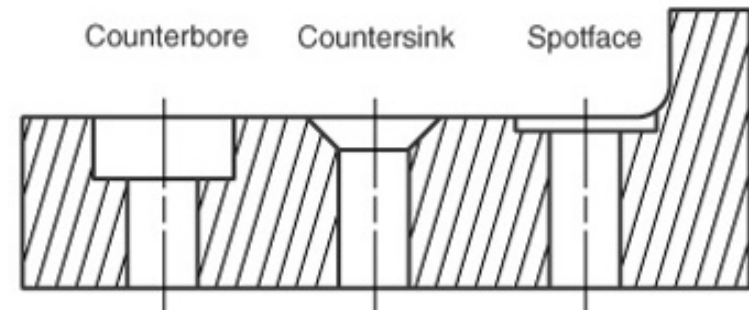
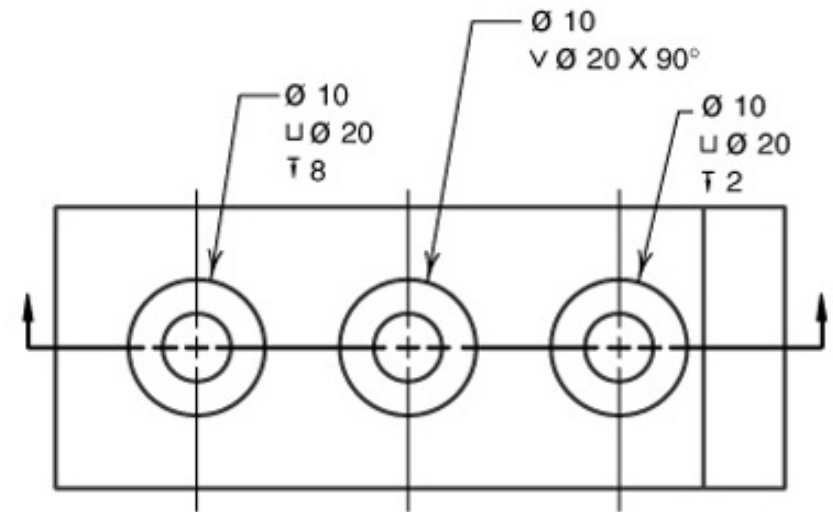
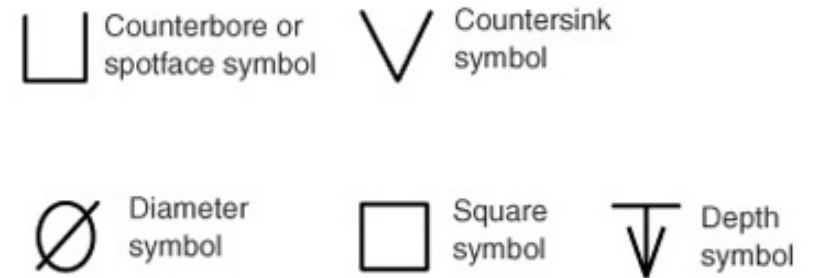
Holes are typically dimensioned in a view which best describes the shape of the hole. Diameters must be dimensioned with the diameter symbol preceding the numerical value. When holes are dimensioned with a leader line, the line must be radial.



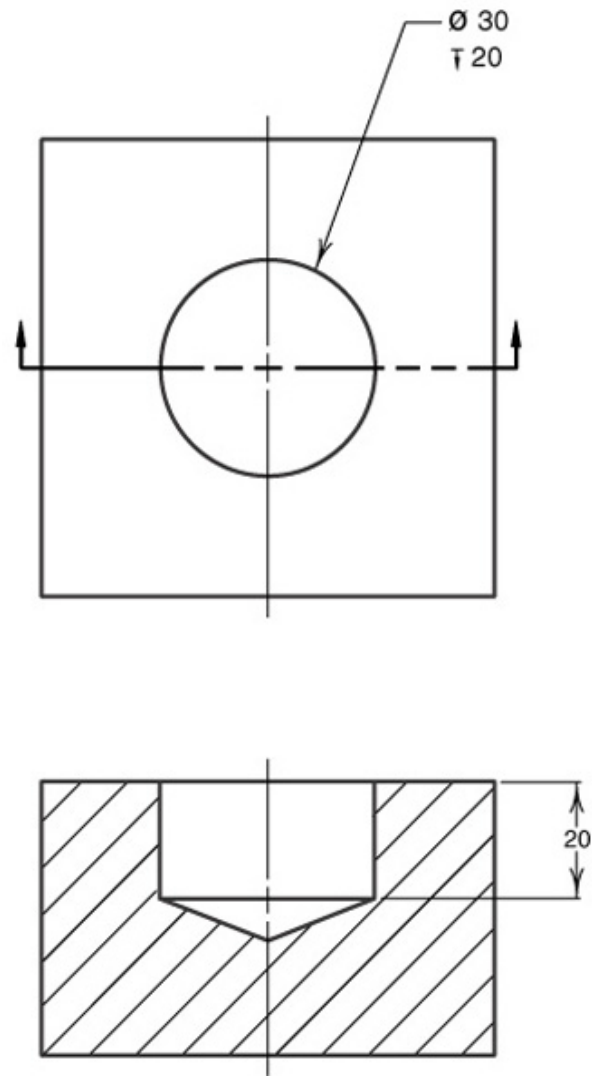
A **radial line** is one that passes through the center of a circle or arc if extended.

Symbols may be used for *spotface*, *counterbore*, and *countersunk* holes. These symbols always precede the diameter symbol.

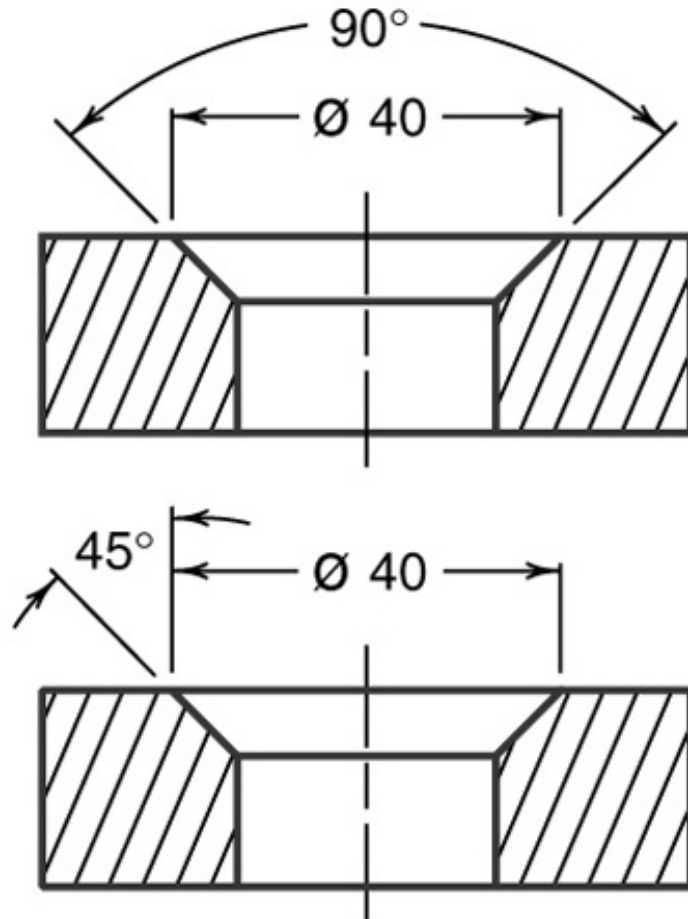
The *depth* symbol may be used to indicate the depth of a hole. The depth symbol is placed preceding the numerical value.



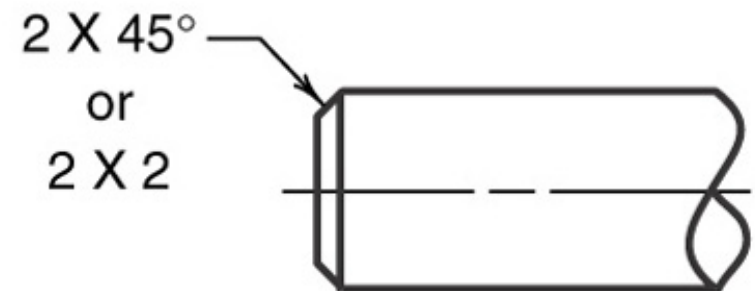
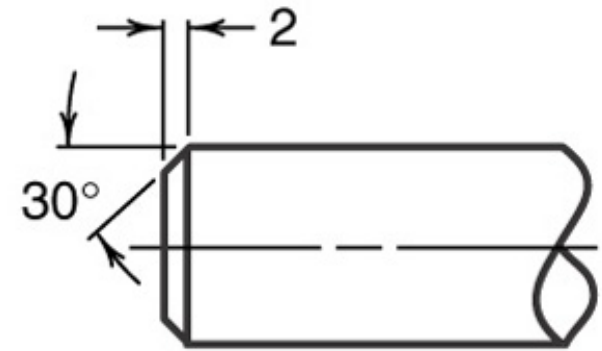
When the depth of a **blind hole** is specified, it refers to the depth of the full diameter of the hole.



Chamfers are dimensioned by providing either an angle and a linear dimension or by providing two linear dimensions.

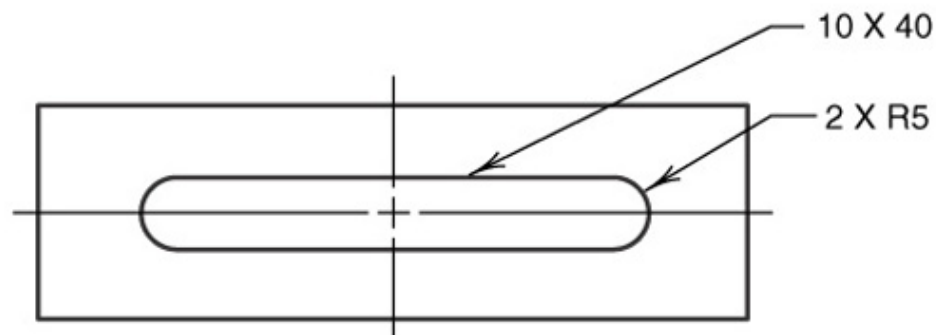
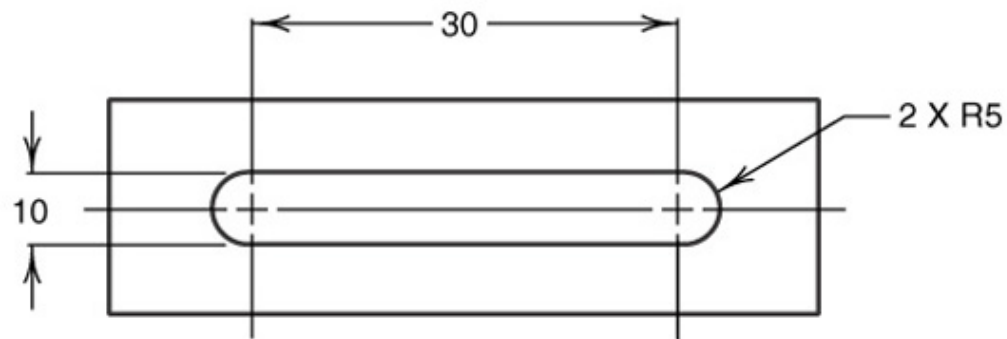
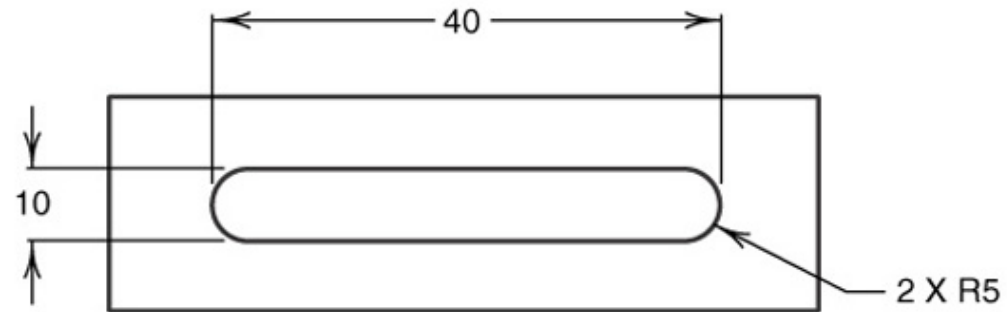


Internal Chamfers



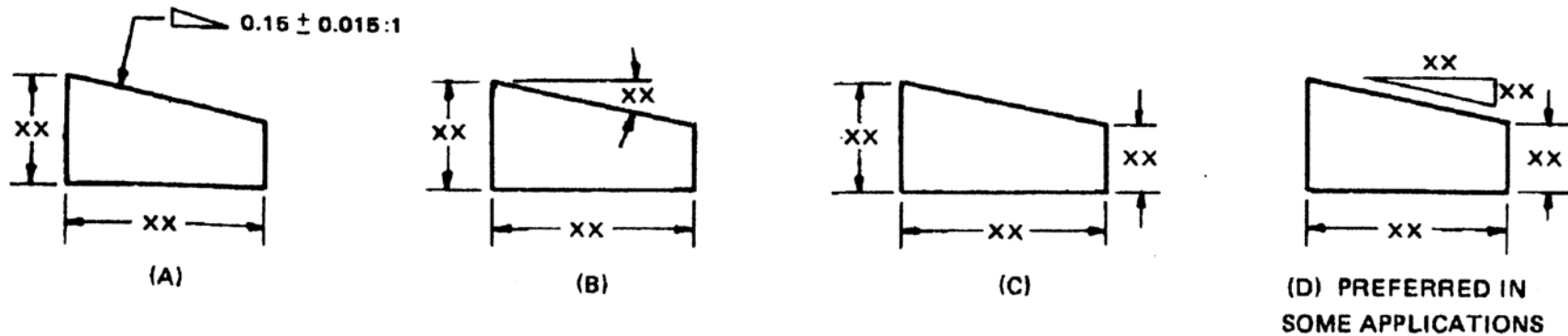
Chamfers of 45° may be specified in a note.

Slotted holes may be dimensioned any of several ways depending on which is most appropriate for the application.



The **slope** of a line or flat surface can be specified:

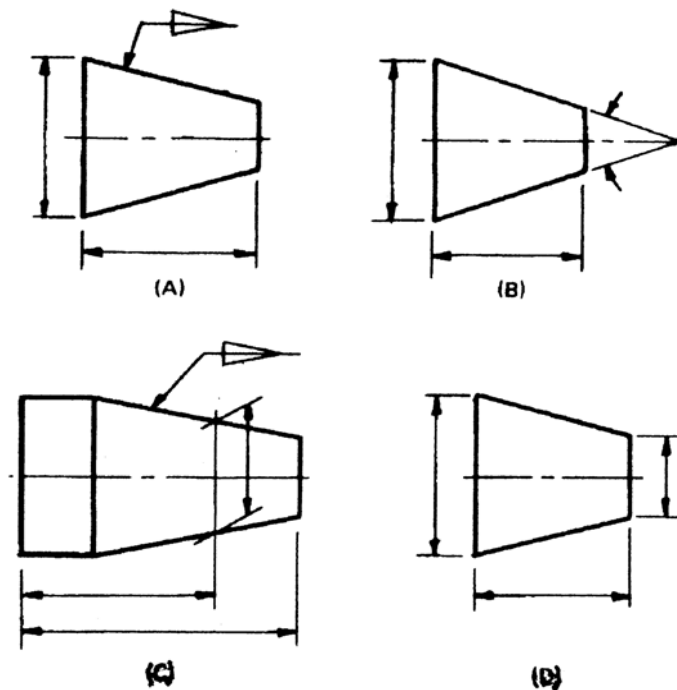
- (a) by the angle; (b) as a ratio combined with the slope symbol; (c) by the dimensions showing the difference in the heights of two points from the base line and the distance between them



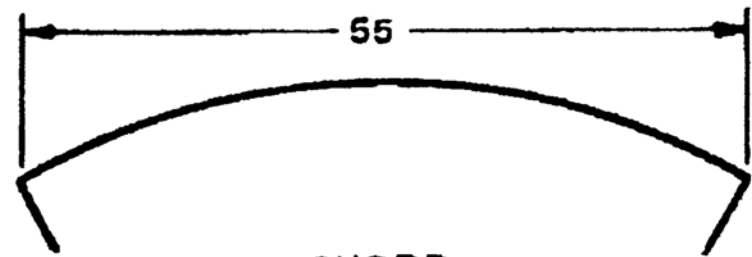
The **tapers** can be specified by:

- The included angle or as a ratio shown with taper symbol;
- the diameter of the larger or smaller ends, or at the selected cross section;
- the dimension locating the cross section;
- the length of the cone.

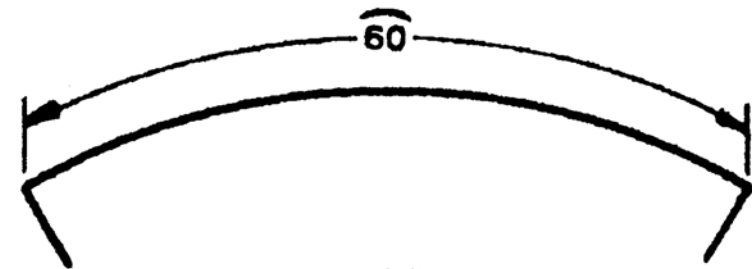
Examples of some typical combinations for dimensioning tapers



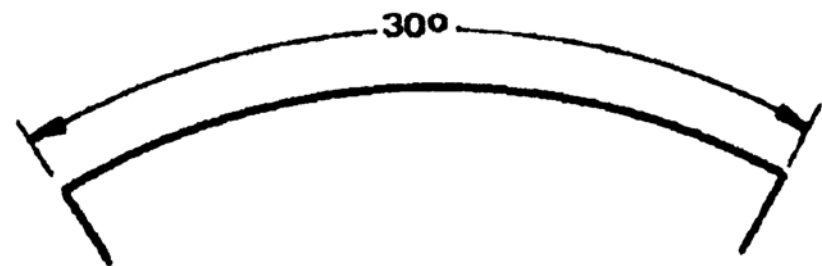
Dimensioning chords, arcs and angles



CHORD



ARC



ANGLE

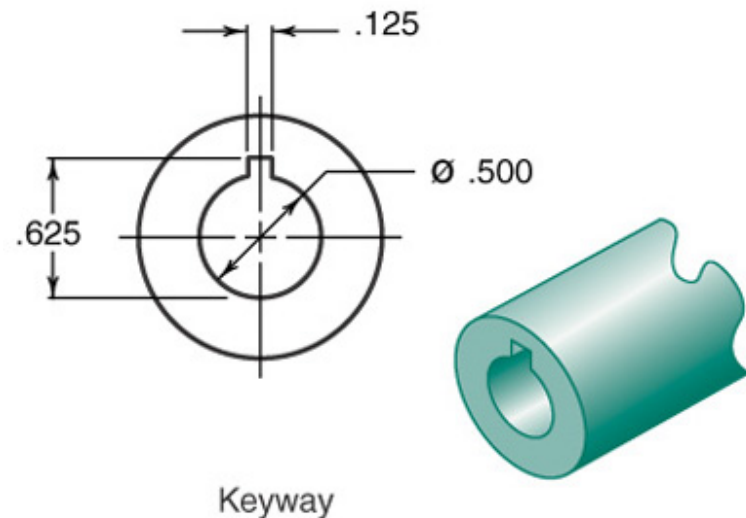
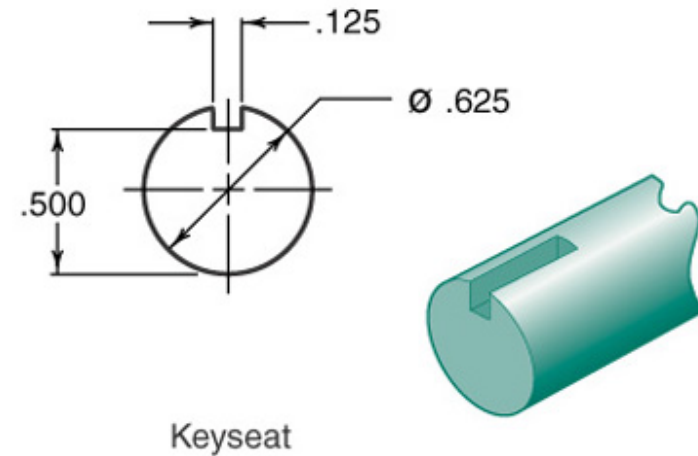
Keyseats and **keyways**, which are fastening devices, are dimensioned in a particular way, because they present some unusual problems.

The height of the keyseat itself is not dimensioned because, once the top of the shaft is cut away, there is no feature left to measure.

Also the dimensions are *unilateral*:

- for the keyseat, the dimension is a minimum;
- for the keyway, the dimension is a maximum.

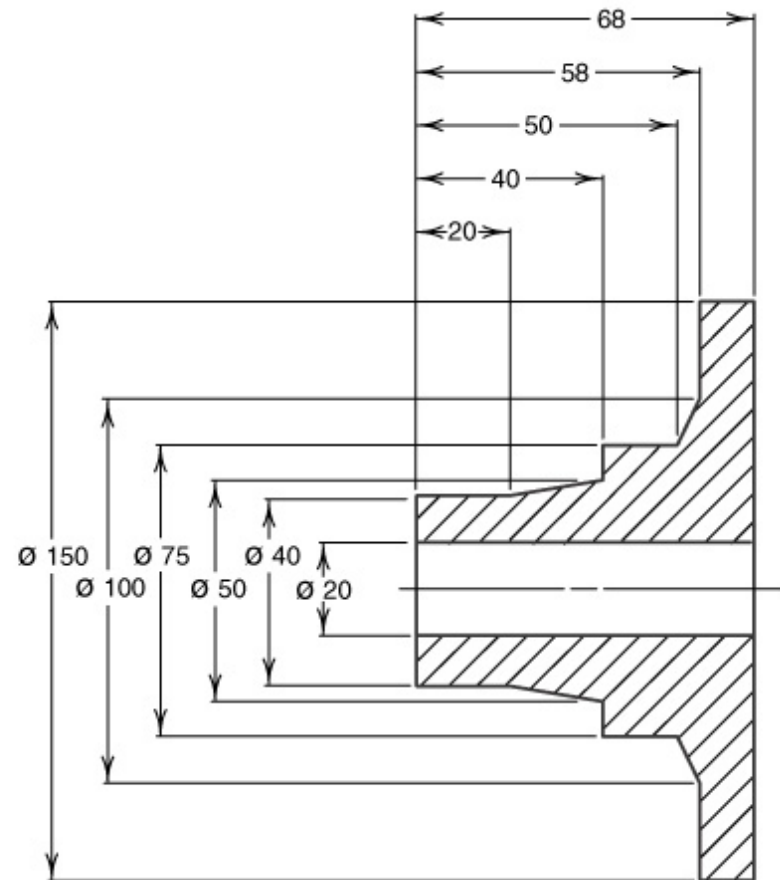
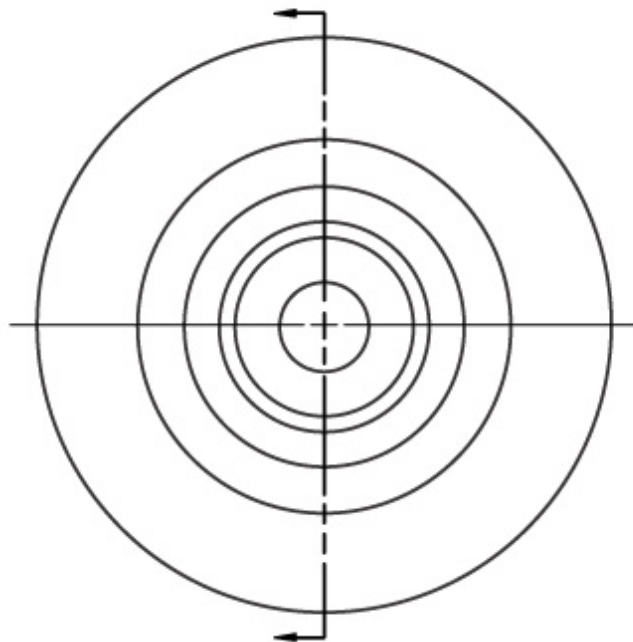
This is to ensure an interference once the key is inserted between the parts.



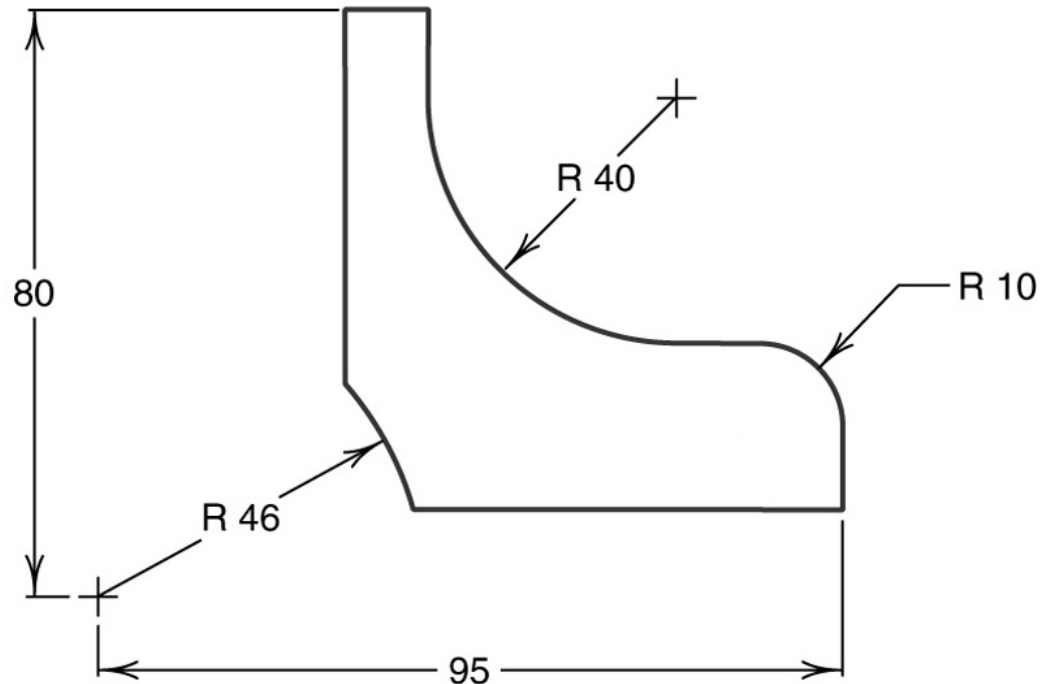
Diameter and radius

If a full circle or an arc of more than half of a circle is being dimensioned, the diameter is specified, preceded by the diameter symbol which is the Greek letter phi ϕ . If the arc is less than half of a circle, then the radius is specified, and the numerical value is preceded by the radius symbol R.

Concentric circles should be dimensioned in the longitudinal view whenever practical.

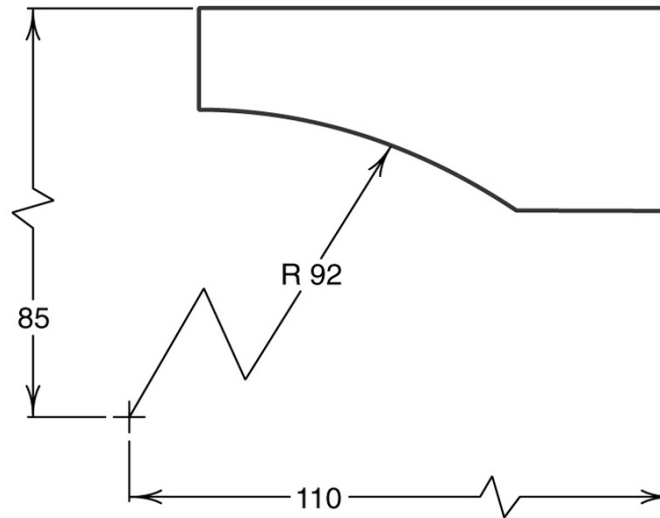


- o The dimension line for radii should have a single arrowhead touching the arc.
- o When there is adequate room, the dimension is placed between the center of the radius and the arrowhead.

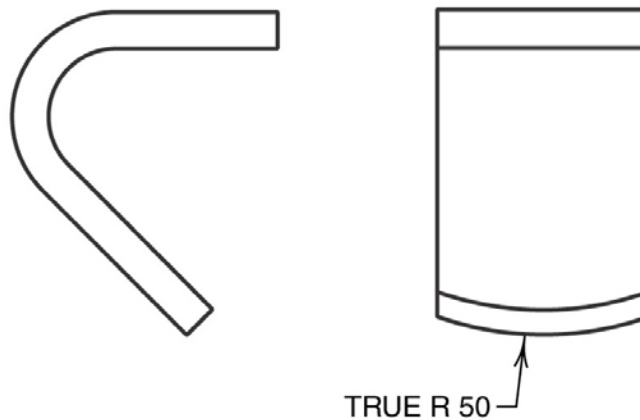


- o When space is limited, a radial leader line is used.
- o When the center of an arc is not clearly defined by being tangent to other dimensioned features on the object, the center of the arc is noted with a small cross.
- o The center of the radius is not noted if the radius is shown tangent to defined surfaces.
- o The center of the arc, which is not tangent to other features, should be dimensioned.

- o If the center of an arc interferes with another view or is outside of the drawing area, foreshortened dimension lines and radius lines may be used.



- o When a radius is dimensioned in a view where it does not appear true shape (it is distorted) the word TRUE appears preceding the radius symbol.



The supreme principle of dimensioning is clarity!

- 1. Each feature of an object is dimensioned once and only once.*
- 2. Dimensions should be placed in the most descriptive view of the feature.*
- 3. Dimensions should specify only the size of a feature. The manufacturing method should only be specified if it is a mandatory design requirement.*
- 4. Angles shown on drawings as right angles are assumed to be 90^0 unless otherwise specified, and they need not be dimensioned.*
- 5. Dimensions should be located outside the boundaries of the object whenever possible.*
- 6. Dimension lines should be aligned and grouped where possible to promote clarity and uniform appearance.*
- 7. Crossed dimension lines should be avoided whenever possible.*
- 8. The space between the first dimension line and the object should be at least 10 mm (0.4 inch). The space between dimension lines should be at least 6 mm (0.24 inch).*
- 9. There should be a visible gap between the object and the origin of an extension line.*
- 10. Extension lines should extend 2-3 mm (0.12 inch) beyond the last dimension line.*
- 11. Extension lines should be broken if they are close to arrowheads.*
- 12. Dimensions should be oriented to be read from the bottom of the drawing.*
- 13. Diameters are dimensioned with a numerical value preceded by the diameter symbol ϕ*
- 14. Radii are dimensioned with a numerical value preceded by the radius symbol **R**.*
- 15. When a dimension is given to the center of an arc or radius, a small cross is shown at the center.*