UNIT **5b**

BASIC DIMENSIONING

8.1 Introduction

Dimensioning refers to the addition of size values to drawing entities. Dimensions are required for points, lines, arcs, circles, etc. that are related functionally or control relationship of other features. Basic dimensioning is the addition of functional or design and nominal sizes to features on drawing views. This is probably good only for sketches and preliminary design drawings. Working drawings need tolerances in addition to functional size values. Tolerances are allowable variation on functional. Most CAD software can automatically add basic dimensions to a drawing. But some fine tuning would normally be necessary to achieve acceptable results. ASME (ANSI) Y14.5M is the standard for dimensioning practice in the USA. Students should familiarize themselves with this standard; even though the main guidelines are incorporated in the discussions that follow from a practical perspective.

8.2 Engineering Drawing and Units

An engineering drawing is a precise technical graphic model that communicates design intent. It is used by manufacturers to make a product and inspectors use it to determine if the product should be accepted. An engineering drawing should convey the following information:

1. Shape or geometric characteristics of component (drawing views).
2. Overall size of component and its features.
3. Tolerances on sizes.
4. Material for the component.
5. Specifications or notes for requirements such as heat treatment, surface finish, etc.

Dimensions in engineering drawings are shown in units of length and angle. The standard unit of length in SI Units is the meter. In drawing practice, the preferred SI unit of length is the millimeter. One meter (1 m) is equal to one thousand millimeters (1000 mm). Fractions are not allowed in SI units, only decimal values are allowed. Architectural drawings may be dimensioned in millimeter (mm) and meters (m). Meters and kilometers (km) are used for civil dimensioning.

Angle refers to the relative orientation of lines on a plane or the relative orientation of planes in space. Angle is conventionally measured in degrees (^{o}). There are 360 degrees in a circle; 60 minutes in a degree; and 60 seconds in a minute. The degree is the common unit of angular measure in Metric and English drawings.

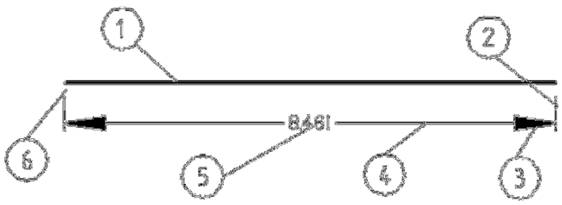
8.3 Size Descriptions

- A dimension is a number in a standard unit of measure shown on a drawing to indicate size, location, or orientation of graphic features.
- A *design siz*e is the functional size of an object and it is equal to the full-size value of the object. Only design sizes are shown as dimensions in engineering drawings.
- Tolerances are small variations permitted on functional sizes for ease of manufacture. They are in the one-hundredths and one-thousandths of functional sizes
- A *plot or print size* is the actual size of a graphic entity on a physical drawing sheet. Plot sizes are not shown in engineering drawings but a scale factor is usually indicated on the drawing.
- The scale factor is the ratio between the design size and the plot size.

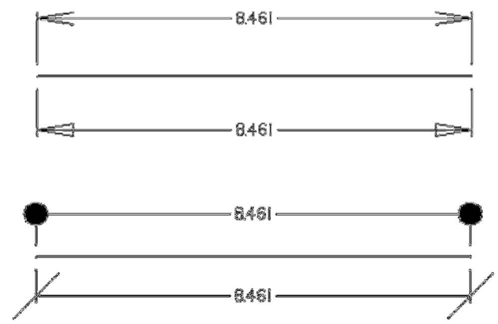
8.4 Dimension Elements and Symbols

Several elements define a dimension in engineering drawings. Fig. 8. 1a shows these elements. The sizes of dimension elements are related to the text height of the dimension value in CAD. The recommended text height for the dimension value is 3 mm (.125”).

1. Graphic feature (line in Fig. 8.1a)
2. Extension line
3. Dimension line terminator
4. Dimension line
5. Dimension value (number)
6. Visible gap



a) Elements of a dimension



b) Dimension line terminators

Fig. 8.1: Dimensional elements and terminators

The graphic feature represents a dimensional entity in a view or image of a drawing. It may be a line, arc, circle, fillet, etc. The extension line connects the object feature with the dimension line. Sometimes, leaders are used in place of extension and dimension lines, especially when dimensioning arcs and circles. The dimension line terminators indicate the limits of a dimension. They occur in pairs, one at the end of the dimension line. It is a filled arrow in this figure but it could be an unfilled arrow, an open arrow, a slash (/), or a filled small circle (●) as shown in Fig. 8.1b. The dimension line is always parallel to a line feature in an object but perpendicular to the extension line. The dimension value is a number representing the size of the dimension. It may be placed above or under the dimension line. Sometimes, it is placed in a gap on the dimension line which is broken to allow this type of placement. It could be aligned with the dimension line or placed horizontally. The visible gap is a space that demarcates the object feature from dimensional elements. This is very important in dimension placement.

Table 1 show some dimensioning symbols commonly associated with basic dimensioning. These symbols have been standardized so as to eliminate language translation. This makes it possible for drawings prepared in different countries to be read and interpreted correctly. Fig. 8. 2 shows a dimensioned component, how many dimensioning symbols can you identify in it?

Table 8.1: Common symbols

Diameter	∅	Counterbore	⌴
Spherical Diameter	S∅	Countersink	⌵
Radius	R	Depth	↧
Spherical Radius	SR	Reference	( )
Conical Taper	⌲	Not To Scale	—
Square	□	Slope	⌳
Feature Repetition	2x	Arc Length	⌒

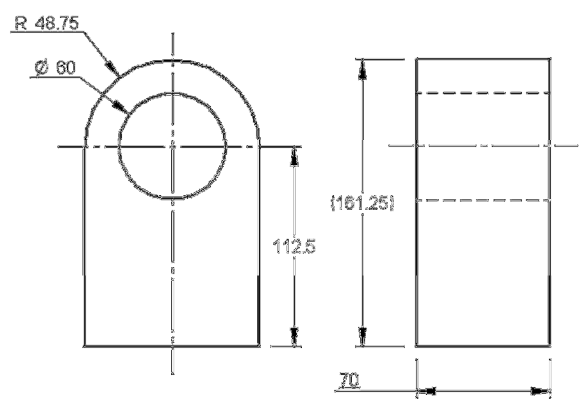


Fig. 8.2: Dimensioned component

8.5 Dimension Types and Line Spacing

A dimension may describe size, location, or orientation (angle) of a feature. Fig. 8.3 shows the basic types of dimension: S-size, L-location, A-angle. The size dimension gives the design size of a feature. A location dimension gives the distance(s) of a key point on a feature from a reference point, line or plane. For example, the center point of a circle is a key point commonly used in dimensioning the location of a circle. An orientation dimension gives the angular position of one feature relative to another. Beveled and sloping features are common in many components. The orientation of the faces on which such features appear need to be dimensioned with the size of the angles associated with the orientations.

Fig. 8.4 shows the recommended minimum gaps for dimension placement. The first dimension line should be at least 10 mm (0.375”) from a visible outline, others 6 mm (0.25”) from the next dimension line. Larger dimensions should be place over smaller ones as indicated in Fig. 8.4.

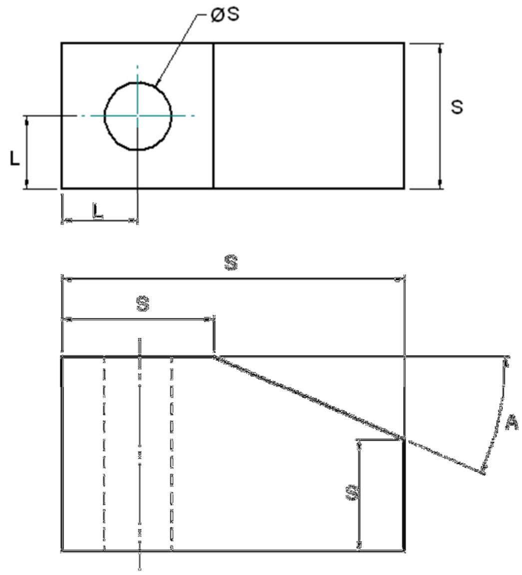


Fig. 8.3: Types of dimensions

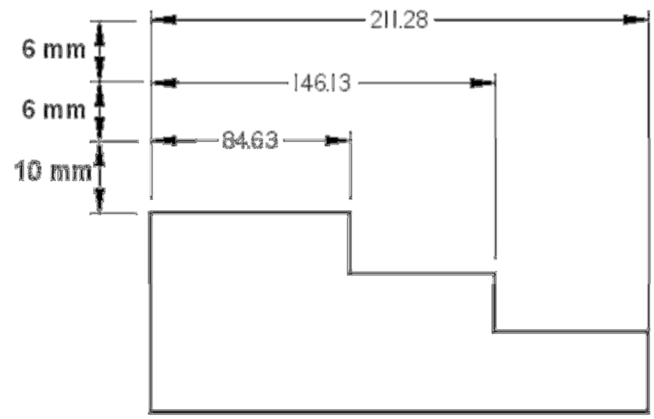


Fig. 8.4: Spacing of dimensions

8.6 Placing Dimensions on Object Features

Placing dimensions on the features of an object on a view must be done systematically and with thoughtfulness. The overriding concern is to present dimensions with clarity. A thought about how the dimension may be verified by measurement or inspection should be considered when placing dimensions. The following guidelines can be helpful when dimensioning:

1. There are two types of sizes, namely linear and angular.
2. Some features (e.g. holes, arcs) have two types of dimensions, namely size and location.
3. Use visible lines only for dimensioning features.
4. Do not use hidden lines for dimensioning features.
5. Spacing between visible outline and first dimension line should be at least 10 mm (3/8”).
6. Spacing between adjacent dimension lines should be at least 6 mm (1/4”).
7. Provide a visible gap between extension line and the feature referenced.
8. Place dimension outside views except it helps clarity placing them inside.
9. Dimensions common to two views should be placed between the views, except when clarity is impaired.
10. A dimension should be shown only on one view or once in a drawing. There should be no duplication of the dimension of the same feature. However, for different features of the same size, the size of each feature must be shown individually once.
11. Use of reference dimensions should be minimized or avoided completely.
12. Circles and arcs should be dimensioned in the view revealing their true shape.
13. Dimensions should be grouped together as much as possible.
14. Dimension values should not overlap themselves, dimension, extension, or visible lines.
15. Dimension text should be horizontal or aligned with (parallel to) dimension line.
16. Smaller dimensions should be placed inside larger dimensions.
17. Minimize or avoid leader lines crossing dimension or extension lines.
18. Minimize extension lines crossing themselves or visible lines.
19. Leader lines should be inclined at 15^o to 75^o; but 30^o to 60^o is preferred.
20. Use datum dimensioning. Avoid chain dimensioning, especially for mechanical objects.

8.6.1 Dimensioning Arcs and Circles

Fig. 8.5 shows the dimensioning of arcs. Arcs should be dimensioned on the view revealing the arc contour. The symbol “R” for radius must precede the value of the dimension of an arc. If the center point of an arc is not obvious, then it must be shown by dimensions. Fig. 8.6 shows the dimensioning of circles. The symbol Ø for diameter must precede the value of the dimension of a circle. The center points of a circle must be dimensioned for location reasons as shown in Fig. 8.6. Fig. 8.7 shows dimension placements for the diameters of some objects. Notice that the information in two views in Fig. 8.7a is presented in one view in Fig. 8.7b because the section view allows direct dimensioning of the bore. Fig. 8.7c could be sectioned also.

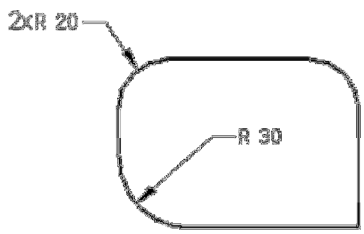


Fig. 8.5: Arc dimensions

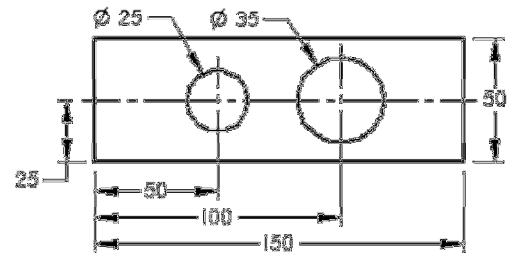
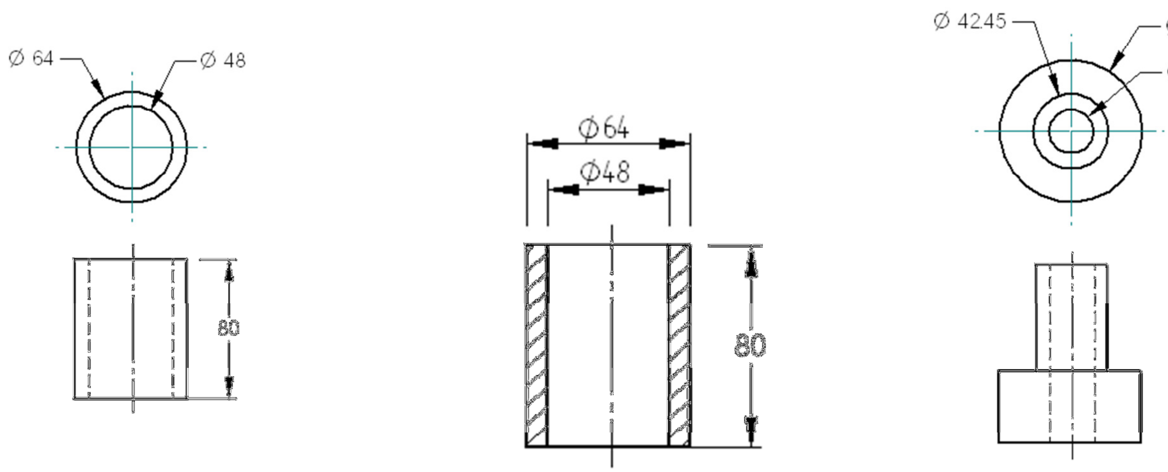


Fig. 8.6: Circle dimensions



a) Diameter on profile view b) Section view showing diameter c) Multiple diameters on profile view

Fig. 8.7: Dimensioning diameters

9.6.2 Dimensioning Angles

Fig. 9-8 shows the dimensioning of angles. Angular dimensions should be expressed in degrees, minutes and seconds or the decimal equivalent. In mechanical drawings, angles are specified in decimal units.

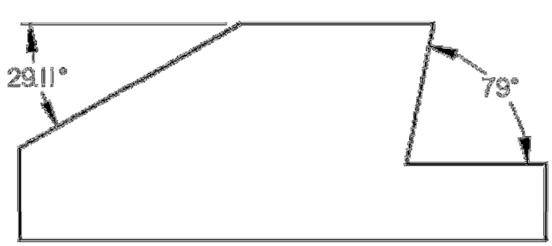


Fig. 9.8: Angular dimensions

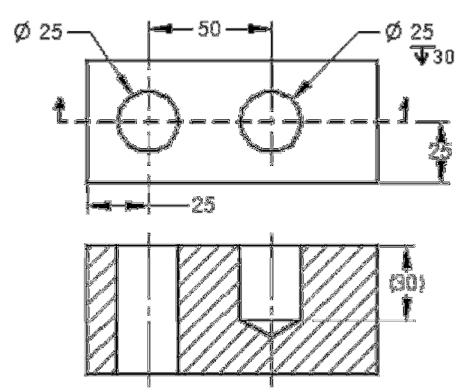


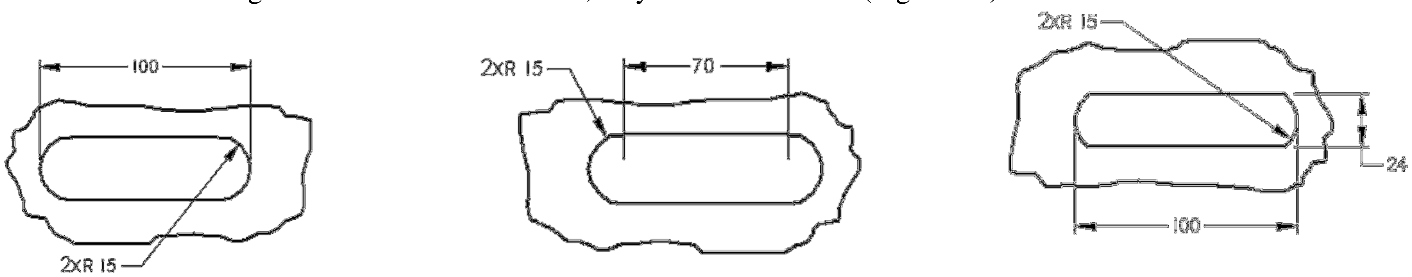
Fig. 9.9: Hole dimensions

9.6.3 Dimensioning Holes

Fig. 9.9 shows one through hole and one blind hole. Holes should be dimensioned on the view showing the circle outline. The depth of through holes is not specified on a drawing however, the depth of a blind hole must be specified either by the depth symbol or directly by the size. The depth of the blind hole in Fig. 9.9 is specified as a reference dimension for interpretation of the depth symbol on the top view only; it should be omitted in practice because the depth symbol is used on the top view.

8.6.4 Dimensioning Slots

Slots are common features on shafts and other components. Proper dimensioning of slots depends on their function and form. Length shown may be between centers (Fig. 8.10a) or full depending on which is critical (Fig. 8.10b). If the end radii are larger than the width of the slot, they should be shown (Fig. 8.10c).

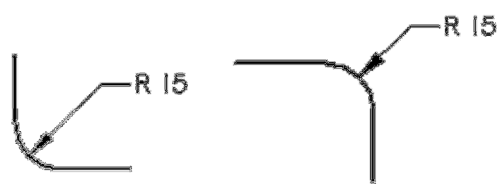


(a) Full length (b) Length between centers (c) Slot width

Fig. 8.10: Dimensioning slots

8.6.5 Dimensioning Fillets and Rounds

Fig. 8.11 shows a fillet and a round. Fillets and rounds are arcs that provide for the smooth transition of faces on an object. The help in removing rough edges as well alleviate stress intensification associated with geometric discontinuities in mechanical components. Fillets are used for interior faces and are concave arcs. Rounds are used for exterior faces and are convex arcs. Fillets and rounds should be dimensioned on the view revealing the arc as shown in Fig. 8.12. The symbol "R" for radius must precede the value of the dimension of a fillet or round.



(a) Fillet (b) Round

Fig. 8.11: Fillets and rounds

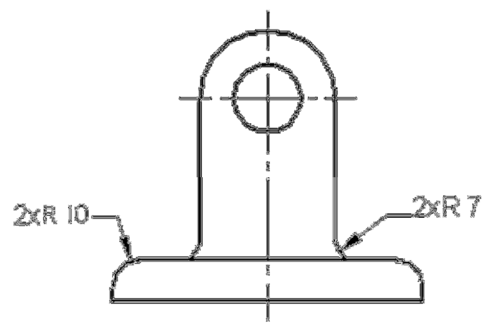
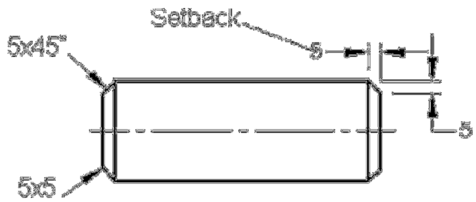


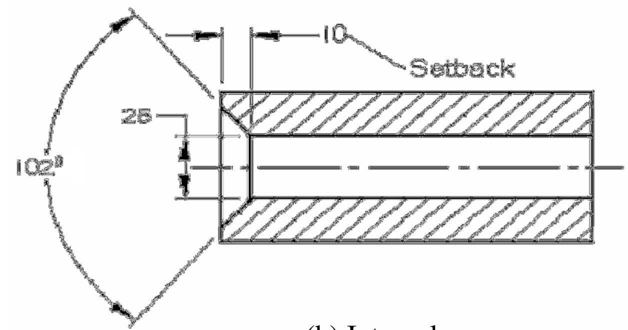
Fig. 8.12: Fillets and rounds on a component

8.6.6 Chamfer Dimensions

Fig. 8.13 shows external and internal chamfers. Chamfers are beveled edges on objects and they remove freehand edges from components and make assembly easier. Chamfers may be specified by notes or dimensions as shown in Fig. 8.13a for external chamfer. The setback lengths on the horizontal and vertical directions are used to specify a chamfer by dimensions. The horizontal length is given first (right end of Fig. 8.13a). Alternatively, the horizontal setback length and angle may be used for specification (left end of Fig. 8.13a). Fig. 8.13b shows the dimensioning of an internal chamfer. Notice that three dimensions are needed in this format by dimensions. Half of the included angle could have been used instead. If the specification or note format is used, the setback and half included angle are sufficient for dimensioning.



(a) External



(b) Internal

Fig. 8.13: Chamfers

8.6.7 Dimensioning Counterbores, Countersinks, and Spotfaces

Fig. 8.14 shows features of a counterbore, countersink, and spotface. Please take time to study the symbols associated with each feature in this figure. A counterbore is a cylindrical recess on a face of an object. It is made by enlarging smaller holes with a boring tool. For the counterbore feature, Φ30 refers to the size of the through hole, Φ40 refers to the size of the step hole, and 20 refer to the depth of the step hole. A countersink is a conical recess on a face of an object. It is made with a special tool and may be used as seats for screws and centers for cylindrical components like shafts and spindles. For the countersink feature, Φ30 refers to the size of the through hole, Φ37 refers to the size of the tapered hole at the surface of the part, and 82^0 refer to the included angle of the tapered hole. A spotface is like a counterbore except that the depth is much smaller. They act as seats for washers and screws. For the spotface feature, Φ30 refers to the size of the through hole and Φ60 refers to the size of the step hole. Notice that the depth of the spotface is not specified. This is because the spotface tool is manufactured for specific depth, often not more than 2 mm.

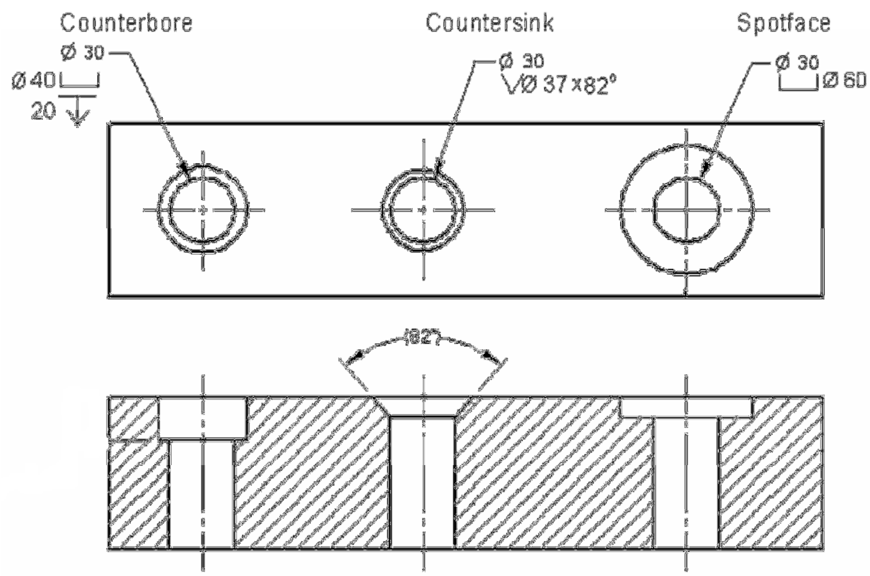


Fig. 8.14: Dimensioning counterbore, countersink and spotface.

8.6.8 Dimensioning Keyseats and Keyways

Keyseats are external slots on shafts, axles, etc. that accept keys. Keyways are internal slots on hubs of cranks, levers, gears, pulleys, sprockets, etc. Fig. 8.15 shows a keyseat and a keyway. Dimensions should be placed such that measurement or inspection of keyseats or keyways can easily be carried out. The length of the keyseat should be shown on the longitudinal view. A broken section is commonly employed for this as shown in Fig. 8.16 where three types of keyseats are indicated. If the keyseat does not start or end at the edge of the shaft, the location dimension must be included as shown in Fig. 8.16a and Fig. 8.16b.

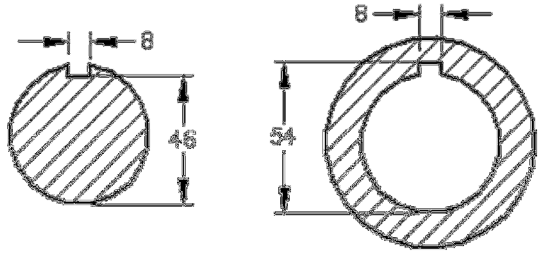


Fig. 8.15: Keyseat and keyway

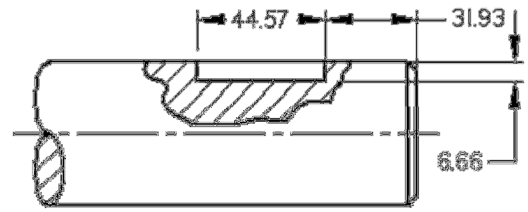


Fig. 8.16: a) Regular keyseat

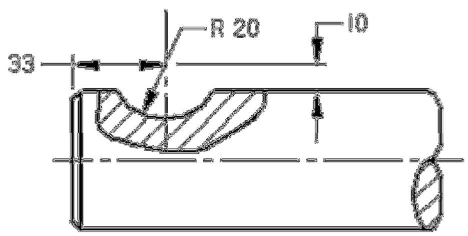


Fig. 8. 16 b) Woodruff keyseat

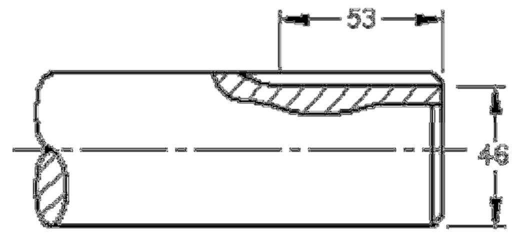
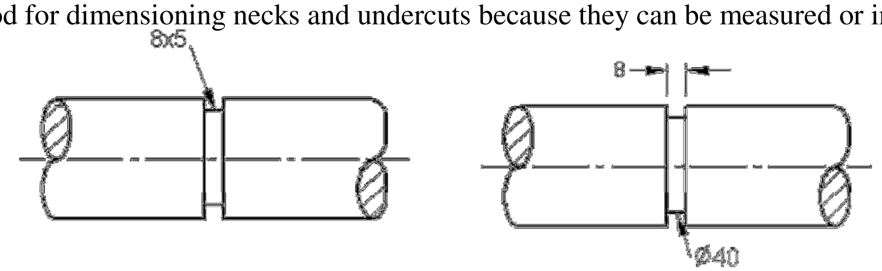


Fig. 8. 16 c) Sledge runner keyseat

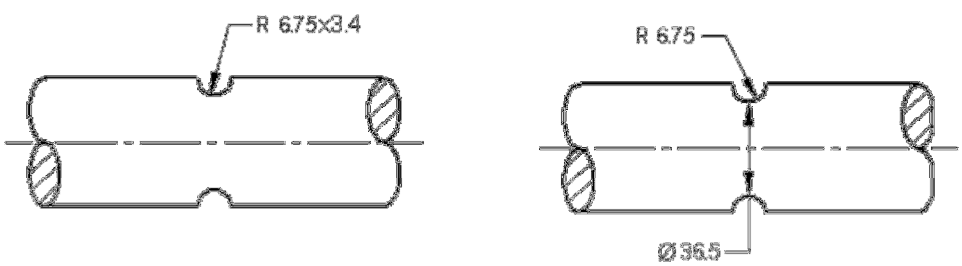
8.6.9 Dimensioning Necks and Undercuts

Necks and undercuts are used to alleviate the influence of stress concentration and relieve the ends of threads. Necks are common on cylindrical sections while undercuts are used on faces. There are rectangular, circular and truncated conical necks or undercuts and are shown in Figs. 8.17, 8.18, and 8.18. The sizes of these features are specified by the width and depth, the width value preceding the depth value as indicated in Figs. 8.17a, 8.18a, and 8.19a. Alternatively, the diameter value of the neck section is given as shown in Figs. 8.17b, 8.18b, and 8.19b. This is the preferred method for dimensioning necks and undercuts because they can be measured or inspected easily this way.



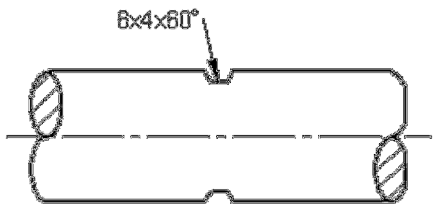
(a) Depth specified (b) Diameter specified

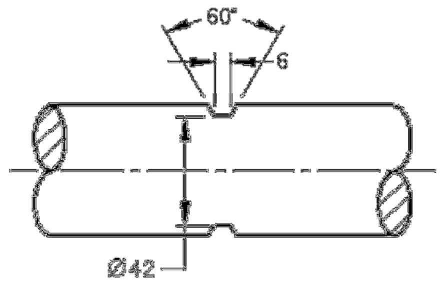
Fig. 8.17: Rectangular neck



(a) Depth specified (b) Diameter specified

Fig. 8.18: Circular neck



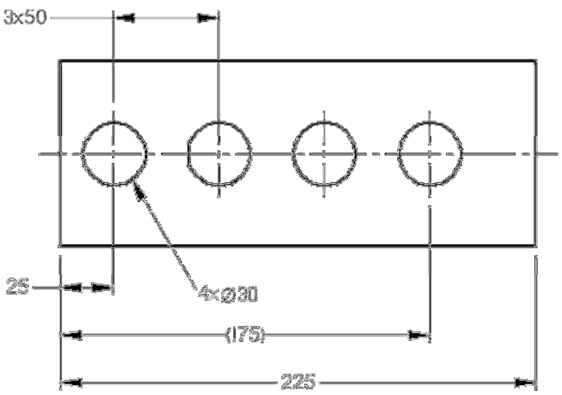


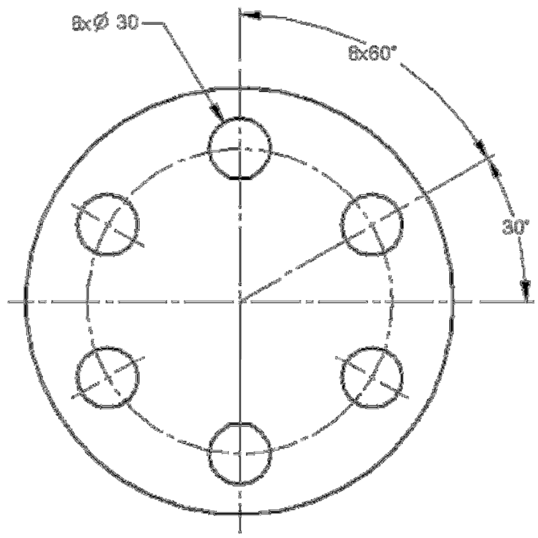
(a) Depth specified (b) Diameter specified

Fig. 8.19: Truncated conical neck

8.6.10 Dimensioning Repeated Features

Some features like holes are repeated on components. Each feature is not dimensioned separately. Instead, the location and or size for one of the features are given and then the total number is included. Fig. 8. 20a has four holes spaced equally as a linear array. Fig. 8. 20b has six holes spaced equally on a circle diameter as a radial array.





(a) Linear array (b) Polar array

Fig. 8.20: Repeated Features

8.7 Dimensioning Methods

Three methods of dimensioning are in common practice. These are datum, chain and tabular. The datum method is depicted in Fig. 8.22a and is preferred. A datum may be a point, line, or surface on a component that is assumed to be exact. It is used as a reference for locating other features on the component. A datum point is often chosen at the far left and bottom point on a part in view. The chain method is illustrated in Fig. 8.23b and is popular in architectural drawings. This method is not recommended for mechanical parts. The tabular method is shown in and Fig. 8.23c. This is used in industry to save space and provide information clearly and concisely.

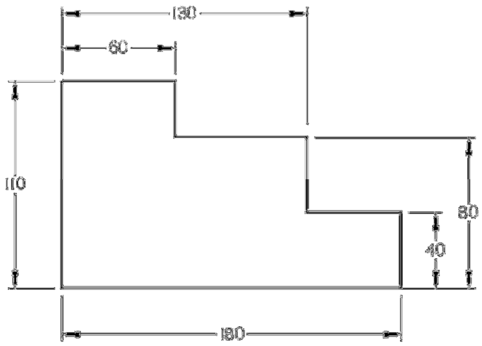


Fig. 8.23a: Datum dimensioning.

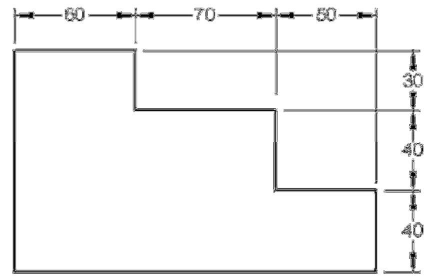


Fig. 8.23b: Chain method.

Table 2: Values of dimensions

Size	2	4	6
A	6.5	10	12.5
B	.875	1.25	1.4375
C	2.75	3.375	3.875

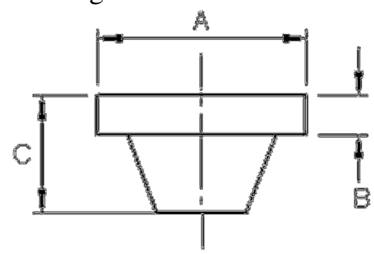


Fig. 8.23c: Tabular method

8.8 Dimensioning Examples

In discussing dimensioning, distinction is made in this section about plain and dimensioned drawing views. Plain drawing views are orthographic views that may be a combination of standard orthoviews, auxiliary views, and section views with hidden lines and center lines. Dimensioned drawing views are obtained by adding dimensions and tolerances to plain drawing views. Chapter 6, focused on the creation of plain standard views, Chapter 7 focused on auxiliary views, and Chapter 8 treated section views. Plain drawing views are normally created manually before dimensions are added in traditional drafting. In CAD environments, they can be constructed from sketches, drawings, etc. or generated from 3D models. If generated from 3D models, hidden lines will be shown by most CAD packages; but some may not show center lines. The user then would have to add center lines and hidden lines if not already added to the generated views in order to create plain multiviews. Fortunately, routines or commands are normally available in these CAD packages for adding center lines or hidden lines automatically in a separate step.

Modern CAD packages are becoming highly automated in drafting skills especially in the drawing aspect. Automatic dimensioning is progressively been improved. As the capabilities of solid modeling software increase, the importance of traditional drafting is decreasing. It seems reasonable to expect that traditional drafting may not be around for very long. However, drafting embodies a knowledge base, drawing, and annotation skills. In most CAD packages with solid modeling capabilities, drawing is virtually automated, since they can be generated in a few steps of menu selection or text commands. So if drawing is fully automated, drafting knowledge and annotations skills I believe will still remain as desirable and required skills in technology disciplines. What then becomes of traditional drafting occupation? Apparently, designers, architects and engineers will be required to do drafting tasks. Print checking and reading will become dominant skills for technology personnel because they will be required in interpreting and ensuring quality assurance of computer generated drawings. Therefore, even when traditional drafting as a career may be at risk, drafting skills will survive, especially annotation skills.

To maximize CAD productivity, the design drafter must master the use of some features in CAD packages such as textstyle and dimension style. When dimension in CAD environment, textstyle and dimension styles are a great advantage. A text style defines a set of character attributes for specific applications. A dimension style defines a set of attributes for dimension display in specific applications such as mechanical, civil, or architectural. Refer to Appendix A for more on this topic.

8.8.1 Manual Dimension Placement

In this section, the basic dimensioning of a component is demonstrated. Manual dimension placement involves adding dimensional values one at a time by a CAD user. The procedure for dimension placement is virtually the same as would be done in traditional drafting, except that the CAD user has the computer to his or her advantage. In either case, plain drawing views must be ready before dimensions can be placed. The drawing views should be reviewed and checked fore correctness before placing dimensions on them. Correcting drawing view errors after placing dimensions is tedious and time wasting even in a CAD environment. In this example, dimensions will be placed manually using CAD software: Solid Edge. The steps for the task of dimension placement are outlined below.

Step 1: Create the Drawing Views
Fig. 8.24 is the plain multiview drawing of a component generated from a solid model. The isometric insert is included for completeness. The isometric view is called an insert because it is not required for a detail drawing. However, it does help visualization when added. If a solid model is available as it is in this example, it is just a few clicks on buttons to insert the view with the other standard views. There is need to provide a good gap between views to make room to the dimensions.

Step 2: Add horizontal dimensions (Top and Front views or Top and Right views)
Fig. 8.25 shows the added horizontal dimensions to Fig. 8.24.

Step 3: Add vertical dimensions (Front and Right view, Top view)
Fig. 8.26 shows the added vertical dimensions to Fig. 8.25.

Step 4: Add angular dimensions (not applicable in this example)
In Fig. 8.26, dimensions common to features on adjacent views have been placed between the views. This is the recommended practice. The dimension 50 in the top view may be omitted since it is obvious by visual inspection. However, it is always preferred to explicitly specify dimensions in engineering drawings.

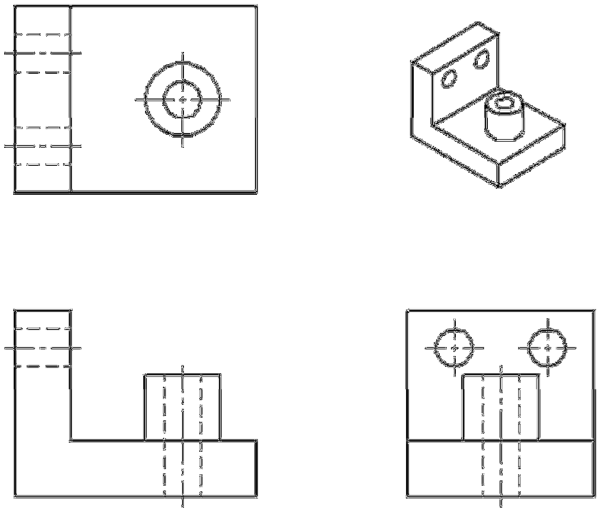


Fig. 8.24: Plain multiviews of a component

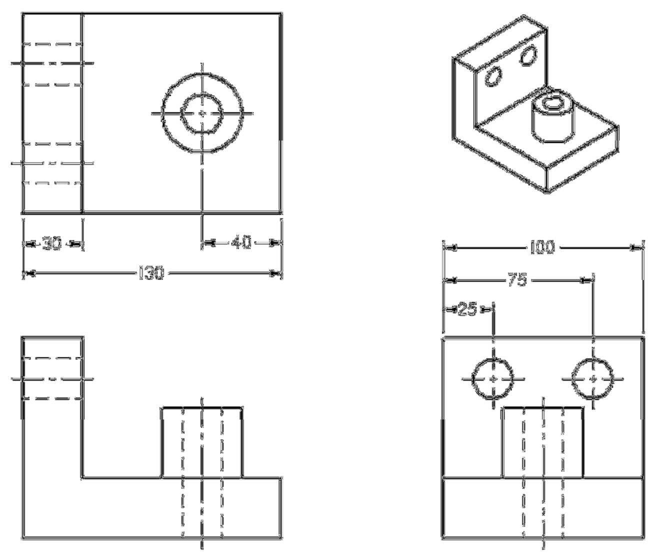


Fig. 8.25: Adding horizontal dimensions to plain multiviews

Step 5: Add arc and circle dimensions (only circles are in the views here)
Fig. 8.27 shows the circle dimensions to Fig. 8.26. Note that only one of the two circles in the right view of Fig. 8.27 is dimensioned with the 2x multiplier added. The 2x is indicative of a repeated feature, twice in this case. Though this example does not present all the features in drafting practice, the principles of dimensioning are the same. When these principles are consistently applied, good annotated drawings will result.

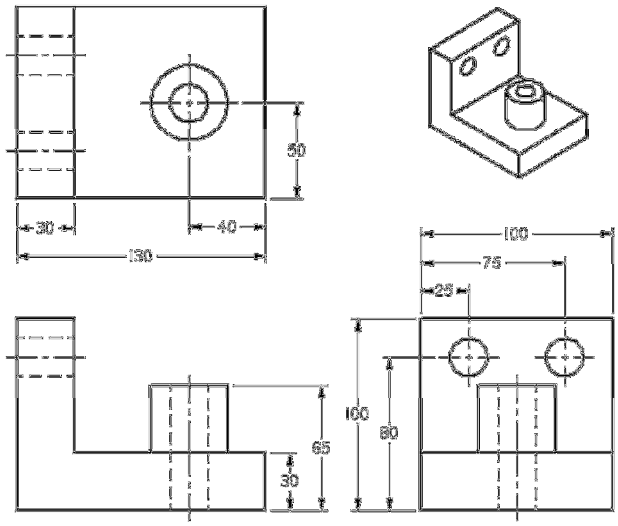


Fig. 8.26: Adding vertical dimensions to multiviews

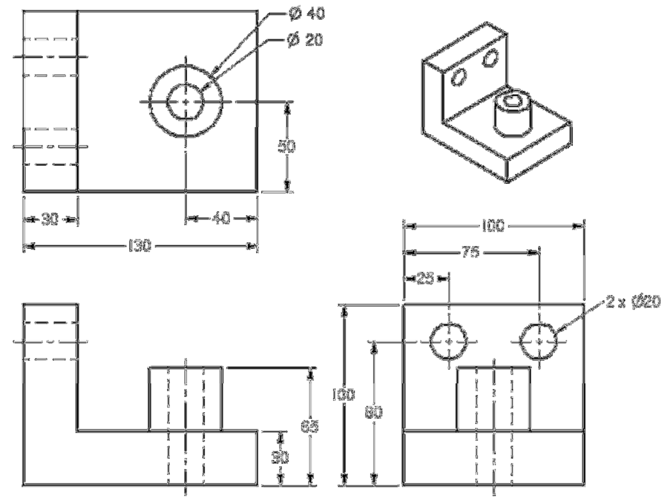


Fig. 8.27: Adding circle dimensions to multiviews

Step 6: Check Drawing Dimensions

- Check all features for location dimensions (three holes and a boss)
- Check all dimensions (verify size and location dimensions)

Step 7: Add notes (not included).

Step 8: Generate check print for review
Print the drawing for review. It may surprise you what you discover in a check print, especially as a new person in the field of drafting. Never turn in a dimensioned drawing without a thorough check on the layout of views and placed dimensions. Errors in dimensional values are hardly tolerated because of associated production cost, rework and company image.

8.8.2 CAD Automatic Dimension Placement

Some solid modeling CAD packages have routines for adding basic dimensions automatically to the plain drawing views that are generated from the solid models. Dimensions from the solid model that can be automatically retrieved are those explicitly defined during the construction of the solid model. Any relevant dimension not defined in the solid construction will have to be manually added later. In some CAD software, angular dimensions are not retrieved from the solid model, so they will have to be manually added. The positions of some of the retrieved dimensions may not be satisfactory. Hence some form of manual tuning will normally be necessary after automatic center line and dimension placement routines have been used. In the previous section, the emphasis was on manual skills in basic dimensioning. In this section, advantage will be taken of automatic annotations routines of CAD software. Taking note of the points highlighted above, the steps for the dimensioning task are outlined. Again, Solid Edge is used in this example.

Step 1: Generate Drawing Views
Let's use the same component of the previous section. Using the view placement routine, generate the drawing views for the component from the solid model as shown in Fig. 8.28. Ensure that enough space is provided between views to make room to the dimensions when placing the views.

Step 2: Add Center lines
As can be observed in Fig. 8.28, hidden lines were added to the views when the appropriate routine was used. Center lines needs to be added. The CAD software used here has a routine for automatic center line addition to the views. After applying this routine, plain drawing views are obtained as shown in Fig. 8.28.

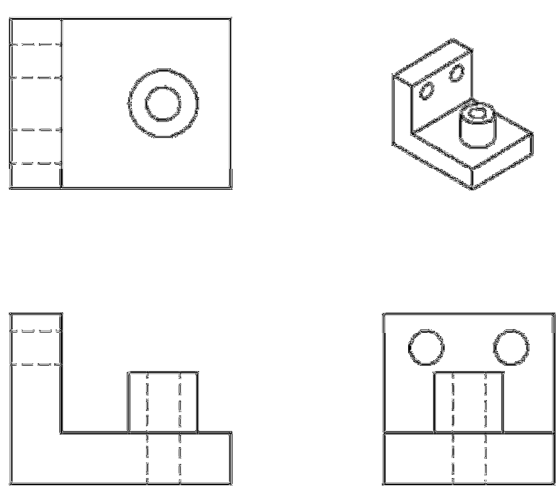


Fig. 8.28: Generated views of a component

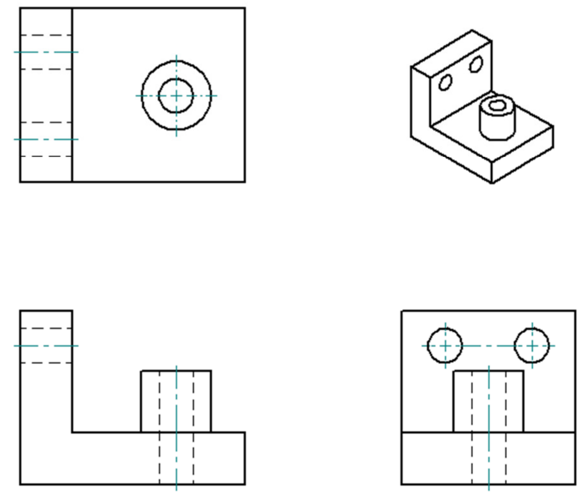


Fig. 8.29: Plain multiview drawing

Step 3: Add dimensions
Using the routine of the CAD software for automatic dimension addition to the views, the dimensions were added to the views of Fig. 8.29 with the result of Fig. 8.30.

Step 4: Add missing dimensions and fine tune dimension positions and placements
A look at Fig. 8.30 will show that:
a) The positioning of the dimension of the small circle in the top view needs adjustment for clarity.
b) The positioning of dimension 100 in the front view needs adjustment.
c) The dimensions 35 and 30 in the front view are chained. This is not recommended formechanical components, therefore re-dimensioning is necessary.
d) The two circles in the right view have no size dimensions. They must be added.
e) The two circles in the right view have no location dimensions. They must be added.
f) Fig. 8.31 is the fine-tuned dimensioning of Fig. 8.30.

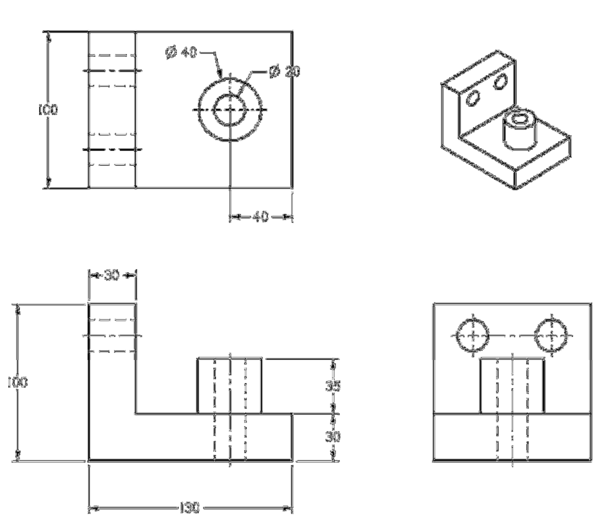


Fig. 8.30: Adding dimensions to multiview drawing

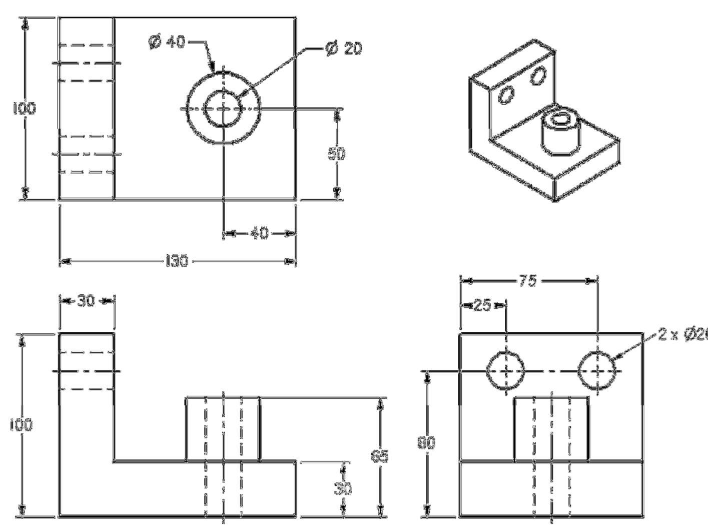


Fig. 8.31: Dimensioned multiview drawing

A drawing such as that of Fig. 8.27 or Fig. 8.31 is partially annotated because it lacks tolerances and possibly some vital specifications on material, finishes, heat treatment, etc. That is, they are not working drawings yet. A fully annotated orthographic view drawing of a component may be referred to as *ortho-detail* drawing. This is a multiview drawing of a component with all specifications and dimensional information necessary for the manufacture and inspection of the component. Similarly, a fully documented orthographic view drawing of an assembly or subassembly may be referred to as *ortho-assembly* drawing. These connotations help to distinguish design documentation drawings into orthographic type drawings and pictorial type drawings. Isometric pictorial drawings are the most common, so we have *iso-detail* and *iso-assembly* drawings.

8.9 SUMMARY

A dimension is a number in a standard unit of measure shown on a drawing to indicate size, location, or orientation of graphic features. The dimensions required on engineering drawings are the design dimensions. A *design siz*e is the functional size of an object and it is equal to the full-size value of the object. The actual size of a feature on a hard copy of a drawing is called the plot size. A *plot size* is the actual size of a graphic entity on a drawing sheet. Plot size is also known as print size. Plot sizes are not shown in engineering drawings but a scale factor is usually indicated on the drawing. The scale factor is the ratio between the design size and the plot size.

Dimensions are part of annotations on engineering drawings. Annotations are textual information and symbols added to views for explicit documentation and communication. There are standard and conventional annotation symbols for dimension features such as diameter, radius, counterbore, countersink, etc. Also, there are national and international guidelines for dimension text height, dimension spacing, etc. In the U.S., these are specified in ASME (ANSI) Y14.5M document, the standard for dimensioning practice in the country.

The two types of dimensions are linear and angular. Linear dimension may describe the size or location of a feature. In CAD practice, dimension elements are best defined in a dimension style where values can be set for them. ANSI recommended text height for the dimension value is 3 mm (.125”). The sizes of dimension elements can be related to the text height of the dimension value in CAD. Only visible lines should be used only for dimensioning features. Spacing between visible outline and first dimension line should be at least 10 mm (3/8”) and spacing between adjacent dimension lines should be at least 6 mm (1/4”). A clear and visible gap must exist between extension line and the feature referenced. As much as it is possible, place dimension outside views but when it helps clarity, place them inside a view. Dimensions common to two views should be placed between the views, except when clarity is impaired. There should be no duplication of the dimension of the same feature. However, for different features of the same size, the size of each feature must be shown individually once. Circles and arcs should be dimensioned in the view revealing their true shape. Dimension values should not overlap themselves, dimension, extension, or visible lines. Smaller dimensions should be placed inside larger dimensions. Minimize or avoid leader lines crossing dimension or extension lines and minimize extension lines crossing themselves or visible lines.