16.4 DIMENSIONING

Dimensions describe the details of a part so it can be constructed to the proper size. They show the sizes and locations necessary to manufacture and inspect the part. Dimensions are placed between points that have a specific relationship to each other to ensure the function of the part. More than one view may illustrate a feature. The dimension of the feature should be placed in the view that best describes the feature. Dimensions are not repeated in different views.

The international system of units (SI) is now commonly used in the United States on engineering drawings to conform to global trade and multinational company affiliations. The SI linear unit on engineering drawings is the millimeter, abbreviated as mm. However, the U.S. linear unit of decimal inches is still being used as we make the transition to all metric. All drawings should clearly state by a note that, unless otherwise specified, all dimensions are in millimeters or inches. ANSI/ASME Y14.5M-1994 is the current standard for dimensioning.
All dimensions are subjected to a tolerance (amount of permissible variability unless noted as basic dimensions or reference dimensions). Basic dimensions are identified by an enclosing frame symbol as illustrated in Figure 16-8. The basic dimension is the theoretically exact size or location of a feature. It is the basis from which permissible variations are established by tolerances. Reference dimensions are supplied for information only. They represent intended sizes, but they do not govern the manufacture or inspection of the part. Figure 16-9 shows an example of a part with three holes dimensioned in a series. The dimension that is not to be used in manufacturing the part is enclosed in parentheses. Tolerances are applied to the 20 and 60 mm dimensions. These two dimensions are used to control the manufacturing process applied to produce the part.

Dimensions are related by three primary methods. Chain dimensions are used when the tolerance between adjacent features is more important than the overall tolerance accumulation. Baseline dimensions are used when the location of features must be controlled from a common reference plane. Direct dimensioning is applied to control specific feature locations. These three types of dimensioning methods are shown in Figure 16-10. A general tolerance is applied to all dimensions.

![Figure 16-8. Basic dimension symbol.](image)

![Figure 16-9. Application of a reference dimension.](image)

![Figure 16-10. Different types of dimensioning.](image)

### 16.5 TOLERANCING

Modern mass production calls for parts made at remote locations to be brought together for assembly and to fit properly without modification. Manufacturers depend on the capability of a part to be assembled with...
its intended mating part. Assembly would be no problem if all parts could be made exactly to size. Some parts, such as gage blocks, can be made very close to a target dimension, but such accuracy is very expensive.

Exact sizes of parts are impossible to produce. For practical reasons, parts are made to varying degrees of accuracy depending on their functional requirements. A tolerance refers to the degree of accuracy that is required in a dimension. In general, the cost of manufacturing a component increases with smaller tolerances on its dimensions.

The major terms used in tolerancing are defined as follows:

- **Nominal size** is the stated designation used for the purpose of general identification. A 9/32 drill is an example of a nominal size.

- **Basic size** is that size from which limits of size are derived by the application of allowances and tolerances. The basic size for a 9/32 drill is 0.28125 in.

- **Limits** are the extreme allowable sizes for a feature. In Figure 16-11, the limits for the shaft are 1.247 and 1.248 in.

- **Tolerance** is the permissible variation in a dimension. It is the difference between the largest and smallest acceptable sizes for a feature. The difference in diameters, 0.001 in., is the tolerance on the shaft diameter in Figure 16-11.

- **Allowance** is the minimum clearance between mating parts. In the case of a shaft and mating hole, it is the difference in the diameters of the largest shaft and smallest hole as shown in Figure 16-12.

- **Maximum material condition** (MMC) is the condition of a part when it contains the most amount of material. The MMC of an external feature of size, such as a shaft, is the upper limit. The MMC of an internal feature of size, such as a hole, is the lower limit.

- **Least material condition** (LMC) is the condition of a part when it contains the least amount of material possible. The LMC of an external feature of size is the lower limit. The LMC of an internal feature of size is the upper limit.

Figure 16-13 shows three methods that can be used to express tolerances for dimensions: limit dimensioning, unilateral tolerances, and bilateral tolerances.

- **Limit dimensioning**—the maximum and minimum sizes of a feature are specified as shown in Figure 16-13(a).
16.3 AUXILIARY AND SECTION VIEWS

Auxiliary views are used to show the true size and shape of features not parallel to any principal views. Primary auxiliary views are projected from a principal view. Secondary auxiliary views also can be used by making a projection from a primary auxiliary view. Figure 16-5 shows an example of a slot seen in its true size and shape (TSS) in a primary auxiliary view.

Sectional views are drawn to show interior details. They are often clearer than exterior views, which contain numerous hidden lines. The location and position of a sectional view is indicated by a cutting plane line and arrows indicating the line of sight in the section. Section lines are used to show the solid material in a sectional view. General sectioning is indicated by thin lines at a 45° angle. Special sectioning symbols may be used for specific materials. An example of a full section is shown in Figure 16-6.
randomly selected mating parts and (2) the tolerance should be as large as possible. The cost of producing any manufactured component increases with smaller tolerances.

16.6 FITS

A fit signifies the type of clearance that exists between mating parts. The three most common types of fits are clearance, interference, and transition. Clearance fits provide some gap between mating parts. Interference fits have no clearance and force is required for the assembly to occur. Transition fits are tolerated to result in either a clearance or an interference fit. Standard systems of fits are applied to holes and shafts that govern the tolerances according to the basic size of the components. Fits can either be based on a standard hole system or a standard shaft system. In a standard hole system, the smallest allowable hole is taken as the basic size from which the limits of tolerance are applied. In a standard shaft system, the largest allowable shaft is taken as the basic size.

There are standardized American National Standard and metric sizes for holes and shafts to achieve different types of fits. The American National Standard system is a set of classes of fits based on the basic hole system. The basic hole system is widely used because drills and reamers are used to produce standard-sized holes. The types of fit covered by this standard are:

- RC—running and sliding fits;
- LC—clearance locational fits;
- LT—transition locational fits;
- LN—interference locational fits, and
- FN—force and shrink fits.

Tables of standard sizes and tolerances are needed to use this system. Since the system is organized on a hole basis, the basic shaft size and the type of fit are needed to determine the dimension and tolerance for the mating parts. For example, an RC 4 fit refers to a close running fit, whereas RC 9 refers to a
loose running fit. Tables will supply the tolerances (in number of thousandths) to add to the basic size to determine the upper limit on the hole and to subtract from the basic size to determine the upper and lower limits on the shaft. The standard tables are too extensive to be reprinted here, but should be available in any text on engineering drawing or the Machinery's Handbook.

**Example 16.6.1.** Find the dimensions and tolerances for a 2.500 in. diameter hole and shaft with an RC 9 fit.

**Solution.** Table 16-2 (based on American National Standard fits—ANSI B 4.1-1967, 1979) provides the following information (Oberg et al. 2000).

<table>
<thead>
<tr>
<th>Limits of Clearance</th>
<th>Hole</th>
<th>Shaft</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.0</td>
<td>7.0</td>
<td>-9.0</td>
</tr>
<tr>
<td>20.5</td>
<td>0</td>
<td>-13.5</td>
</tr>
</tbody>
</table>

(Units are thousandths of an inch) (Oberg et al. 2000)

The lower limit of the hole has no tolerance since it represents the basic size. The limit dimensions on the hole are found as:

Upper limit: 2.5000 + 0.0070 = 2.5070 in.
Lower limit: 2.5000 + 0.0000 = 2.5000 in.

The limit dimensions on the shaft are found as:

Upper limit: 2.5000 - 0.0090 = 2.4910 in.
Lower limit: 2.5000 - 0.0135 = 2.4865 in.

The limits of clearance represent the largest and smallest clearances that can result in assembly. They can be readily verified by calculating the extreme shaft and hole combinations:

LMC hole - LMC shaft = largest clearance (loosest fit)

\[ 2.5070 - 2.4865 = 0.0205 \text{ in.} \]

MMC hole - MMC shaft = smallest clearance (tightest fit or allowance)

\[ 2.5000 - 2.4910 = 0.0090 \text{ in.} \]

The metric system of fits is organized in a fashion similar to the American National Standard system. The metric system can operate on a hole basis or shaft basis. An International Tolerance (IT) Grade is associated with a particular size and level of accuracy. For example, the designation 40H8 refers to a 40 mm basic-size hole with an IT grade of 8. The designation 40T7 refers to a 40 mm basic-size shaft with an IT grade of 7. The combination of 40H8/T7 refers to a particular fit, in this case, a close running fit. Tables of standard metric values are needed to determine the appropriate tolerances.

### 16.7 TOLERANCES FOR 100% INTERCHANGEABILITY

The most common requirement of a tolerance on a dimension calls for parts to be 100% interchangeable. Any random combination of mating parts will be guaranteed to assemble. The extreme or most difficult conditions for assembly are used to find the unknown tolerance. In the extreme condition, internal dimensions are taken at the minimum value and external dimensions are taken at the maximum value. A path equation is used to add signed dimensions to find the value of an unknown tolerance.

**Example 16.7.1.** Figure 16-14 shows a car radio tuner knob (k) being assembled with a bearing (b) and a spacer (a) in a cavity (c). For the knob to turn freely, a 0.003 in. clearance (g) must exist between the knob flange (a) and the top of cavity. Find the tolerance x of the depth of the cavity.

**Solution.** A sign convention is used. Dimensions are positive going from the bottom to the top. Starting at the bottom of the cavity in Figure 16-14, the signed dimensions for
the extreme conditions are added to form a closed path. The extreme conditions are found when the maximum-size knob, bearing, and spacer are combined with a minimum clearance (g) and minimum cavity (c):

\[ s_{\text{max}} + b_{\text{max}} + k_{\text{max}} + g_{\text{min}} - c_{\text{min}} = 0 \]

or:

\[ 0.379 + 0.254 + 0.128 + 0.003 - (0.766 - x) = 0 \]

\[ x = 0.002 \]

16.8 SURFACE FINISH SYMBOLS

Surface texture is the variation of height, width, and orientation of the irregularities on a surface. Surface texture can strongly affect the performance of a part in service. Surface texture specifications are critical to assuring the proper function of parts such as bearings or dies. Figure 16-15 illustrates standard surface texture criteria. The important terms used in surface texture specification are defined as follows:

- **Roughness** refers to the finest irregularities in a surface. Roughness is strongly dependent on the type of manufacturing process used to generate a surface.

- **Average roughness** is the arithmetic average of the absolute values of height deviations from the mean plane or centerline of a surface. It is typically measured in microinches or micrometers.

- **Roughness spacing** is the average spacing between successive peaks within the roughness sampling length.

- **Cutoff** is the sampling length used for calculation of the average roughness. When it is not specified, a value of 0.030 in. (0.8 mm) is assumed.

- **Waviness** is the widely spaced, repeated variation on a surface.

- **Waviness height** is the peak-to-valley distance between waves.

- **Waviness spacing** is the average spacing between successive peaks within the waviness sampling length.

- **Lay** is the direction of the surface pattern. This is dependent on the method used to generate the surface.

Figure 16-16 illustrates standard lay designations. Figure 16-17 illustrates applications of surface texture symbols.
Figure 16-16. Standard lay designations.

Figure 16-17. Application of surface texture symbols: average roughness is given in microinches. All other values are given in inches.