Tolerancing
11-1 Introduction

Tolerance is the total amount a specific dimension can vary. Tolerance are assigned so that any 2 mating parts will fit together, as shown in (a). In this case, the actual hole may not be less than 1.25 inch and not more than 1.251 inch; these are the limits for dimension, and the difference between is the Tolerance. A metric version is shown in (b).
Allowance

In Fig. (a), the max. shaft is shown solid and the min. shaft is shown phantom. The difference is .001”. This difference .001” is the tolerance for the shaft. Similarly, the tolerance for the hole is .001”. The loosest fit, or maximum clearance, occurs when the smallest shaft is in the largest hole, as shown in (b). The tightest fit is shown in (c).

The difference between the largest allowable shaft size and the smallest allowable hole size is called allowance.
When parts are required to fit properly in assembly but not to be interchangeable, parts are not always toleranced but sometimes just indicated to be made to fit at assemble, as example below.

![Diagram showing fit at assembly dimensions](Image)
Normal size is used for general identification and is usually expressed in decimals or common fractions. In figure (a), the normal size of both hole and shaft would be 1.25”.

Basic size, or basic dimension, is the theoretically exact size from which limits of size are determined by applying allowance and tolerance. It is the size from which limits are determined for the size, shape, or location of a feature. For above example, the basic size is the same as the normal size, 1.25 inch.
**Actual size** is the measured size of the finished part. Allowance is the minimum clearance space (or maximum interference) between mating parts. Allowance represents the tightest possible fit. For clearance fits this difference will be positive, but for interference fits it will be negative.

![Diagram of actual size and allowance](image)

- **Allowance** = \( 1.250 - 1.248 = 0.002 \)
- **Max Clearance** = \( 1.251 - 1.247 = 0.004 \)
Clearance Fit, where an internal member fits into an external member (as a shaft in a hole), always has space or clearance between the parts. See example below.
Interference fit, where the internal number is always larger than the external number, requires that the parts be forced together. In Fig. (a), the smallest shaft is 1.2513” and the largest hole is 1.2506”, so the interference of metal between parts amounts to at least .00070”. An interference fit always has a negative allowance.

Transition fit results in either a clearance or interference condition. In Fig. (b), the smallest shaft, 1.2503”, will fit into the largest hole, 1.2506”. But the largest shaft, 1.2509”, will have to be forced into the smallest hole, 1.2500”.
If allowances and tolerances are specified properly, mating parts are completely interchangeable. But for close fits, it is necessary to specify very small allowances and tolerances, and the cost will be very high. To avoid this expense, either manual or computer-controlled selective assembly is often used.

In *selective assembly*, all parts are inspected and classified into several grades according to actual sizes.
Reamers, broaches, and other standard tools are often used to produce holes, and standard plug gages are used to check the actual size. Shafts are easily machined down to any size desired. Therefore, tolerated dimensions are commonly determined using the basic hole system, in which the minimum holes is taken as the basic size. Then the allowance is determined, and the tolerance are applied.
A **reamer** is a rotary cutting tool used to enlarge the size of a previously formed hole by a small amount but with a high degree of accuracy to leave smooth sides. The following shows some of reamer tools.
Broaching is a machining process that uses a toothed tool, called a broach, to remove material when precision machining is required, especially for odd shapes, like keyholes.
Lapping is a machining process, in which two surfaces are rubbed together with an abrasive between them, by hand movement or by way of a machine.

Honing is an abrasive machining process that produces a precision surface on a metal workpiece by scrubbing an abrasive stone against it.

http://www.engineershandbook.com/MfgMethods/honing.htm
In some industries, they use the basic shaft systems. It is advantageous when several parts having different fits, are required on a single shaft, or when the shaft for some reason cannot be machine to size easily. In this system, the maximum shaft is taken as the basic size, an allowance for each mating part is assigned, and the tolerance are applied.
A tolerance of a decimal dimension must be given in decimal form, as shown in Fig. 11.6. General Tolerance on decimal dimensions in which tolerances are not given may be covered in a print note such as

DECIMAL DIMENSION TO BE HELD TO .001.

11-7 Specification of Tolerances
There are several methods of expressing tolerances in dimensions that are approved by ANSI and they are as follows.

1. **Limit dimensioning.** In this preferred method, the maximum and minimum limits are specified, as shown below.

The maximum value is placed above the minimum value as shown in Fig. (a). In the single-line note form, the low limit precedes the high limit separated by a dash, as shown in Fig. (b).
2. **Plus-or-Minus dimensioning.** In this method the basic size is followed by a plus-or-minus expression for the tolerance. The result can be either unilateral, where the tolerance only applies in one direction so that one value is zero, or bilateral, where either the same or different values are added and subtracted as shown below.

If the plus and minus values are the same, a single value is given, preceded by the plus-or-minus symbol, as shown below.
If the plus and minus values are the same, a single value is given, preceded by the plus-or-minus symbol, as shown below.
3. **Single-limit dimensioning**. It is not necessary to specify both limits. MIN or MAX is often placed after a number to indicate minimum or maximum dimensions desired where other elements of design determine the other unspecified limit.

4. **Angular tolerances** are usually bilateral and in terms of degrees, minutes, and seconds.
In tolerance dimensioning, it is very important to consider the effect of one tolerance on the other. When the location of a surface is affected by more than one tolerance value, the tolerance are cumulative, as shown in Fig. (a) NO GOOD.

As a rule, it is best to dimension each surface so that it is affected by only one dimension. This can be done by referring all dimensions to a single datum surface, such as B, as shown in Fig. (b).
11-9 Tolerances and Machining Processes

Tolerance should be as generous as possible and still permit satisfactory use of the part. Figure below shows a chart, to be used as a general guide, with the tolerances achievable by the indicated machining process.
A system of preferred metric limits and fits by the International Organization for Standardization (ISO) is in the ANSI B4.2 standard. The system is specified for holes, cylinders, and shafts, but it is also adaptable to fits between parallel surfaces such features as keys and slots. The terms for metric fits, illustrated in the figure below, are somewhat similar to those for decimal-inch fits.
Basic size is the size from which limits or deviations are assigned. Basic sizes, usually diameters, should be selected from a table of preferred sizes, as shown below.
International tolerance grade (IT) is a set of tolerances that varies according to the basic size and provide a uniform level of accuracy within the grade. For example, in dimension 50H8 for a close-running fir, the IT grade is indicated by the number 8. (the letter H indicates that the tolerance is on the hole for the 50-mm dimension.)
Tolerance zone refers to the relationship of the tolerance to basic size. It is established by a combination of the fundamental deviation indicated by a letter and the IT grade number. In the dimension 50H8, for the close-running fit, the H8 specifies the tolerance zone, as shown below.

The figure (a) shows the hole-basis system of preferred fits and the figure (b) shows the shaft-basis system of preferred fits (b).
The preferred basic size for computing tolerances are given in Table 11.2 (p.371). Basic diameters should be selected from the first-choice column since these are readily available stock size for round, square, and hexagonal products.
The symbols for either the hole-basis or shaft-basis preferred fits are given in Table 11.2. First should be selected from this Table for mating parts where possible. See 11-13 Preferred Fits for more.
Geometric tolerances state the maximum allowable variations of a form or its position from the perfect geometry implied on the drawing. However, it is impossible to produce perfect forms, it may be necessary to specify the amount of variation permitted.
Geometric tolerances specify either the diameter or the width of a tolerance zone within which a surface or the axis of a cylinder or a hole must be if the part is to meet the required accuracy for proper function and fit.

When tolerances of form are not given on a drawing, it is customary to assume that, regardless of form variations, the part will fit and function satisfactorily.

Tolerances of form and position (or location) control such characteristics as straightness, flatness, parallelism, perpendicularity (squareness), concentricity, roundness, angular displacement, and so on.
11-14 Symbols for Tolerances and Form

ANSI symbols, shown on the Table below, provide an accurate and concise means of specifying geometric characteristics and tolerances in a minimum space. The symbols may be supplemented by notes if the precise geometric requirements cannot be conveyed by the symbols. Note five types of tolerance.

<table>
<thead>
<tr>
<th>Geometric characteristic symbols</th>
<th>Modifying symbols</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Type of Tolerance</strong></td>
<td><strong>Characteristic</strong></td>
</tr>
<tr>
<td>For individual features</td>
<td>Form</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>For individual or related features</td>
<td>Profile</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>For related features</td>
<td>Orientation</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Location</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Runout</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Arrowheads may be filled or not filled*
Combinations of the various symbols and their meanings are given in Figure below.

(d) BASIC DIMENSION SYMBOL

(b) DATUM SYMBOL

Geometric characteristic symbol

Tolerance

Modifier

(b) DATUM SYMBOL

(d) FEATURE CONTROL SYMBOLS

(e) FEATURE CONTROL SYMBOLS WITH DATUM REFERENCES

Modifier applicable to tolerance

Modifier of datum

Datum reference

Reference to two datums
Application of the symbols to drawing are illustrated in the figure below.
1. The *basic dimension symbol* is defined by the enclosing frame symbol, as shown below. The basic dimension is the value used to describe the theoretically exact size, shape, or location of a feature.

![Basic Dimension Symbol](image1)

2. The *datum identifying symbol* consists of a capital letter in a square frame and a leader line extending from the frame to the concerned feature and terminating with a triangle. The triangle may be filled or not filled, as shown below.

![Datum Symbol](image2)
Letters of alphabet (except I, O, and Q) are used as datum-identifying letter. A point, line, plane, cylinder, or other geometric form assumed to be exact for purposes of computation may serve as a datum from which the location or geometric relationship of features of a part may be established, as shown below.
3. Supplementary Symbols includes the symbols for MMC (maximum material condition – or minimum hole diameter, maximum shaft diameter) and LMC (least material condition – or maximum hole diameter, minimum shaft diameter), as shown below.

The symbol for diameter precedes the specified tolerance in a feature control symbol, as shown below. This symbol for diameter should precede the dimension.
4. **Combined symbols** are found when individual symbols, datum reference letters, and needed tolerances are combined in a single frame, as shown below.
Position Tolerances

Figure 11.24 (a) shows a hole located from two surfaces at right angles to each other. In figure 11.24, the center may lie anywhere within a square tolerance zone, the sides of which are equal to the tolerances.

The total variation along either diagonal of the square by the coordination method of dimensioning will be 1.4 times greater than the indicated tolerance. When the location of the hole is off in a diagonal direction, the area of the square tolerance zone is increased by 57% without exceeding the tolerance permitted.
Cumulative Tolerances

If four holes are dimensioned with rectangular coordinates, as in (a), specifying a tolerance describes a square zone in which the center of the hole must be located as shown in Figs. (b) and (c). Because the shape of the square zone, the tolerance for the location of the center of the hole is greater in the diagonal direction than the indicated tolerance.
In Fig. (a), hole A is selected as a **datum**, and the other three located from it. The square tolerance zone for hole A results from the tolerances on the two rectangular coordinate dimensions locating hole A. The tolerance zone for the other three holes result from the tolerances between the holes, while their locations will vary according to the actual location of datum hole A. Two of many possible zone patterns are shown in Figs. (b) and (c).
A true-position dimension specifies the theoretically exact position of a feature. The location of each feature, such as a hole, slot, or stud, is given by untolerance basic Dimensions identified by the enclosing frame or symbol. To prevent misunderstandings, true position should be established with respect to a datum.

In simple arrangements, the choice of a datum may be obvious and not require identification. Positional tolerances are indicated in a feature control frame attached to a feature on the object.
Untoleranced Dimensions

Positional tolerances described a cylindrical zone for the tolerances as shown below. Here, the untoleranced dimensions avoid the accumulation of tolerances even in a chain of dimensions.
This **cylindrical tolerance zone** has a diameter equal to the positional tolerance, and its length is equal to the length of the feature unless otherwise specified. Its axis must be within this cylinder, as shown below.
As shown in the figure below, the positional tolerance specification indicates that all elements on the hole surface must be on or outside a cylinder whose diameter is equal to the minimum diameter or the maximum diameter of the hole minus the positional tolerance, with the centerline of the cylinder located at the true position.
Position Tolerances of Axes

While features, such as holes and bosses, may vary in any direction from the true-position axis, other features, such as slots, may vary on either side of a true-position plane, as shown below.

(a) THIS ON THE DRAWING . . .

(b) . . . MEANS THIS
Maximum Material Condition

Maximum material condition, or MMC, means that a feature of a finished product contains the maximum amount of material permitted by the tolerated dimensions shown for the feature.

Holes, slots, or other internal features are at MMC when at minimum size. Shafts, pads, bosses, and other external features are at MMC when they at their maximum size.

A feature is at MMC for both mating parts when the largest shaft is in the smallest hole and there is the least clearance between the parts.
What must be the size of the gauge pins to fit the position tolerance at MMC?

The gauge pins that fit the MMC condition on the left may also fit this case that is outside the specified positional tolerance.

Bilateral tolerances have traditionally been given on angles, as shown below. Using bilateral tolerances, the wedge-shaped tolerance zone increases as the distance from the vertex of angle increases.
11-15 Form Tolerances For Single Feature
Straightness

*Straightness tolerance*, shown below, specifies a tolerance zone within which an axis or all points of the considered element must lie. Straightness is a condition in which an element of a surface or an axis is a straight line.

Each longitudinal element of the surface must be within the specified tolerance size of the perfect form at MMC and lie between two parallel lines (0.02 apart) where the two lines and the nominal axis share a common plane.

Each circular element of the figure must be within the specified tolerance of size. The centerline of the feature must lie within a cylindrical tolerance zone of 0.04 at MMC. The allowed straightness tolerance increases equal to the amount the feature departs from MMC.
**Flatness**

*Flatness tolerance* specifies a tolerance zone defined by two parallel planes within which the surface must lie, as shown below. Flatness is the condition of a surface having all elements in one plane.

![Diagram of flatness tolerance]

*The surface must be within the specified tolerance of size and must lie between two parallel planes 0.25 apart.*
Roundedness

*Roundness tolerance*, shown in figure below, specifies a tolerance zone bounded by two concentric circles within which each circular element of surface must lie.

Roundness is a condition of a surface of revolution in which, for a cone or cylinder, all points of the surface intersected by any plane perpendicular to a common axis are equidistant from that axis. For a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.
Cylindricity

*Cylindricity tolerance* specifies a tolerance zone bounded by two concentric cylinders within which the surface must lie, as shown below. This tolerance applies to both circular and longitudinal elements of the entire surface. Cylindricity is a condition of a surface of revolution in which all points of the surface are equidistant from a common axis.

![Diagram of Cylindricity](image)
No Tolerance

When no tolerance of form is given, many possible shapes may exist within a tolerance zone, as shown below.

---

THIS ON THE DRAWING . . . . .

. . . . MEANS THIS

Tolerance zone or boundary within which forms may vary when no tolerance of form is given.
Profile

Profile tolerance specifies a uniform boundary or zone along the true profile within which all elements of surface must lie, as shown below.

A profile is the outline of an object in a given plane, or 2-D, figure. Profiles are formed by projecting a 3-D figure onto a plane or by taking cross sections through the figure, with the resulting profile composed of such elements as straight lines, arcs, or other curves.
A tolerance zone is defined by two parallel planes perpendicular to a datum plane, datum axis or axis within which the surface of the feature must lie, as shown below.
The surface must be within the specified tolerance of size and must lie between two parallel planes 0.4 apart which are inclined at 30° to the datum plane A.
If an angular surface is located by a linear and an angular dimension, as shown in (a), the surface must lie within a tolerance zone, as shown in (b). The angular zone will be wider as the distance from the vertex increases. To avoid the accumulation of tolerance further out from the angle’s vertex, the basic angle tolerancing method, as shown in (c), is recommended. The angle is indicated as a basic dimension, and no angular tolerance is specified. The tolerance zone is now defined by two parallel planes, resulting in improved angular control, as shown in (d).

Note that two different tolerance notations result in different tolerance zones.
Parallelism

The surface must be within the specified tolerance of size and must lie between two planes 0.12 apart which are parallel to the datum plane A.
The feature axis must be within the specified tolerance of location and must lie between two planes 0.12 apart which are parallel to the datum plane, regardless of feature size.
The feature axis must be within the specified tolerance of location. Where the feature is at maximum material condition (10.00), the maximum parallelism tolerance is 0.05 diameter. Where the feature departs from its MMC size, an increase in the parallelism tolerance is allowed which is equal to the amount of such departure.
Perpendicularly

**Perpendicularity**

- **Perpendicularity for a Plane Surface**

- 0.12 width tolerance zone

- Datum plane A

- Possible orientation of the surface

- The surface must be within the specified tolerance of location and must lie between two parallel planes 0.12 apart which are perpendicular to the datum plane A.

- **Perpendicularity for a Median Plane**

- 0.12 width tolerance zone

- Datum plane A

- Possible orientation of the feature center plane

- The feature center plane must be within the specified tolerance of location and must lie between two parallel planes 0.12 apart, regardless of feature size, which are perpendicular to the datum plane A.

- **Perpendicularity for an Axis**

- 0.2 width tolerance zone

- Datum axial A

- Possible orientation of the feature axis

- The feature axis must be within the specified tolerance of location and must lie between two planes 0.2 apart, regardless of feature size, which are perpendicular to the datum axis.
(a). A **tolerance zone** is defined by two parallel planes perpendicular to a datum plane, datum axis or axis within which the surface of the feature must lie, as shown below.
(b). A cylindrical tolerance zone perpendicular to a datum plane within which the axis of the feature must lie, as shown below.
Concentricity

The feature axis must be within a cylindrical zone of 0.1 diameter, regardless of feature size, and whose axis coincides with the datum axis.