ALLOWANCES AND TOLERANCES FOR FITS

Limits and Fits.—Fits between cylindrical parts, i.e., cylindrical fits, govern the proper assembly and performance of many mechanisms. Clearance fits permit relative freedom of motion between a shaft and a hole—axially, radially, or both. Interference fits secure a certain amount of tightness between parts, whether these are meant to remain permanently assembled or to be taken apart from time to time. Or again, two parts may be required to fit together snugly—without apparent tightness or looseness. The designer's problem is to specify these different types of fits in such a way that the shop can produce them. Establishing the specifications requires the adoption of two manufacturing limits for the hole and two for the shaft, and, hence, the adoption of a manufacturing tolerance on each part.

In selecting and specifying limits and fits for various applications, it is essential in the interests of interchangeable manufacturing that 1) standard definitions of terms relating to limits and fits be used; 2) preferred basic sizes be selected wherever possible to reduce material and tooling costs; 3) limits be based upon a series of preferred tolerances and allowances; and 4) a uniform system of applying tolerances (preferably unilateral) be used. These principles have been incorporated in both the American and British standards for limits and fits. Information about these standards is given beginning on page 627.

Basic Dimensions.—The basic size of a screw thread or machine part is the theoretical or nominal standard size from which variations are made. For example, a shaft may have a *basic* diameter of 2 inches, but a maximum variation of minus 0.010 inch may be permitted. The minimum hole should be of basic size wherever the use of standard tools represents the greatest economy. The maximum shaft should be of basic size wherever the use of standard purchased material, without further machining, represents the greatest economy, even though special tools are required to machine the mating part.

Tolerances.—Tolerance is the amount of variation permitted on dimensions or surfaces of machine parts. The tolerance is equal to the difference between the maximum and minimum limits of any specified dimension. For example, if the maximum limit for the diameter of a shaft is 2.000 inches and its minimum limit 1.990 inches, the tolerance for this diameter is 0.010 inch. The extent of these tolerances is established by determining the maximum and minimum clearances required on operating surfaces. As applied to the fitting of machine parts, the word tolerance means the amount that duplicate parts are allowed to vary in size in connection with manufacturing operations, owing to unavoidable imperfections of workmanship. Tolerance may also be defined as the amount that duplicate parts are permitted to vary in size to secure sufficient accuracy without unnecessary refinement. The terms “tolerance” and “allowance” are often used interchangeably, but, according to common usage, *allowance* is a difference in dimensions prescribed to secure various classes of fits between different parts.

Unilateral and Bilateral Tolerances.—The term “unilateral tolerance” means that the total tolerance, as related to a basic dimension, is in *one* direction only. For example, if the basic dimension were 1 inch and the tolerance were expressed as 1.000 − 0.002, or as 1.000 + 0.002, these would be unilateral tolerances because the total tolerance in each is in one direction. On the contrary, if the tolerance were divided, so as to be partly plus and partly minus, it would be classed as “bilateral.”

Thus,

$$1.000^{+0.001}_{-0.001}$$

is an example of bilateral tolerance, because the total tolerance of 0.002 is given in two directions—plus and minus.

When unilateral tolerances are used, one of the three following methods should be used to express them:

1) Specify, limiting dimensions only as

Diameter of hole: 2.250, 2.252

Diameter of shaft: 2.249, 2.247

2) One limiting size may be specified with its tolerances as

Diameter of hole: 2.250 + 0.002, −0.000

Diameter of shaft: 2.249 + 0.000, −0.002

3) The nominal size may be specified for both parts, with a notation showing both allowance and tolerance, as

Diameter of hole: $2\frac{1}{4} + 0.002, -0.000$

Diameter of shaft: $2\frac{1}{4} - 0.001, -0.003$

Bilateral tolerances should be specified as such, usually with plus and minus tolerances of equal amount. An example of the expression of bilateral tolerances is

$$2 \pm 0.001 \quad \text{or} \quad 2\begin{array}{l}+0.001\\-0.001\end{array}$$

Application of Tolerances.—According to common practice, tolerances are applied in such a way as to show the permissible amount of dimensional variation in the direction that is less dangerous. When a variation in either direction is equally dangerous, a bilateral tolerance should be given. When a variation in one direction is more dangerous than a variation in another, a unilateral tolerance should be given in the less dangerous direction.

For nonmating surfaces, or atmospheric fits, the tolerances may be bilateral, or unilateral, depending entirely upon the nature of the variations that develop in manufacture. On mating surfaces, with few exceptions, the tolerances should be unilateral.

Where tolerances are required on the distances between holes, usually they should be bilateral, as variation in either direction is normally equally dangerous. The variation in the distance between shafts carrying gears, however, should always be unilateral and plus; otherwise, the gears might run too tight. A slight increase in the backlash between gears is seldom of much importance.

One exception to the use of unilateral tolerances on mating surfaces occurs when tapers are involved; either bilateral or unilateral tolerances may then prove advisable, depending upon conditions. These tolerances should be determined in the same manner as the tolerances on the distances between holes. When a variation either in or out of the position of the mating taper surfaces is equally dangerous, the tolerances should be bilateral. When a variation in one direction is of less danger than a variation in the opposite direction, the tolerance should be unilateral and in the less dangerous direction.

Locating Tolerance Dimensions.—Only one dimension in the same straight line can be controlled within fixed limits. That dimension is the distance between the cutting surface of the tool and the locating or registering surface of the part being machined. Therefore, it is incorrect to locate any point or surface with tolerances from more than one point in the same straight line.

Every part of a mechanism must be located in each plane. Every operating part must be located with proper operating allowances. After such requirements of location are met, all other surfaces should have liberal clearances. Dimensions should be given between those points or surfaces that it is essential to hold in a specific relation to each other. This restriction applies particularly to those surfaces in each plane that control the location of other component parts. Many dimensions are relatively unimportant in this respect. It is good practice to establish a common locating point in each plane and give, as far as possible, all such dimensions from these common locating points. The locating points on the drawing, the locatingor registering points used for machining the surfaces and the locating points for measuring should all be identical.

The initial dimensions placed on component drawings should be the exact dimensions that would be used if it were possible to work without tolerances. Tolerances should be given in that direction in which variations will cause the least harm or danger. When a variation in either direction is equally dangerous, the tolerances should be of equal amount in both directions, or bilateral. The initial clearance, or allowance, between operating parts should be as small as the operation of the mechanism will permit. The maximum clearance should be as great as the proper functioning of the mechanism will permit.

Direction of Tolerances on Gages.—The extreme sizes for all plain limit gages shall not exceed the extreme limits of the part to be gaged. All variations in the gages, whatever their cause or purpose, shall bring these gages within these extreme limits.

The data for gage tolerances on page 656 cover gages to inspect workpieces held to tolerances in the American National Standard ANSI B4.4M-1981.

Allowance for Forced Fits.—The allowance per inch of diameter usually ranges from 0.001 inch to 0.0025 inch, 0.0015 being a fair average. Ordinarily the allowance per inch decreases as the diameter increases; thus the total allowance for a diameter of 2 inches might be 0.004 inch, whereas for a diameter of 8 inches the total allowance might not be over 0.009 or 0.010 inch. The parts to be assembled by forced fits are usually made cylindrical, although sometimes they are slightly tapered. The advantages of the taper form are that the possibility of abrasion of the fitted surfaces is reduced; that less pressure is required in assembling; and that the parts are more readily separated when renewal is required. On the other hand, the taper fit is less reliable, because if it loosens, the entire fit is free with but little axial movement. Some lubricant, such as white lead and lard oil mixed to the consistency of paint, should be applied to the pin and bore before assembling, to reduce the tendency toward abrasion.

Pressure for Forced Fits.—The pressure required for assembling cylindrical parts depends not only upon the allowance for the fit, but also upon the area of the fitted surfaces, the pressure increasing in proportion to the distance that the inner member is forced in. The approximate ultimate pressure in tons can be determined by the use of the following formula in conjunction with the accompanying table of "Pressure Factors." Assuming that A = area of surface in contact in "fit"; a = total allowance in inches; P = ultimate pressure required, in tons; F = pressure factor based upon assumption that the diameter of the hub is twice the diameter of the bore, that the shaft is of machine steel, and that the hub is of cast iron:

$$P = \frac{A \times a \times F}{2}$$

Pressure Factors

Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor	Diameter, Inches	Pressure Factor
1	500	3½	132	6	75	9	48.7	14	30.5
1¼	395	3¾	123	6¼	72	9½	46.0	14½	29.4
1½	325	4	115	6½	69	10	43.5	15	28.3
1¾	276	4¼	108	6¾	66	10½	41.3	15½	27.4
2	240	4½	101	7	64	11	39.3	16	26.5
2¼	212	4¾	96	7¼	61	11½	37.5	16½	25.6
2½	189	5	91	7½	59	12	35.9	17	24.8
2¾	171	5¼	86	7¾	57	12½	34.4	17½	24.1
3	156	5½	82	8	55	13	33.0	18	23.4
3¼	143	5¾	78	8½	52	13½	31.7	…	…

Allowance for Given Pressure.—By transposing the preceding formula, the approximate allowance for a required ultimate tonnage can be determined. Thus, $a = \frac{2P}{AF}$. The average ultimate pressure in tons commonly used ranges from 7 to 10 times the diameter in inches.

Expansion Fits.—In assembling certain classes of work requiring a very tight fit, the inner member is contracted by sub-zero cooling to permit insertion into the outer member and a tight fit is obtained as the temperature rises and the inner part expands. To obtain the sub-zero temperature, solid carbon dioxide or "dry ice" has been used but its temperature of about 109 degrees F. below zero will not contract some parts sufficiently to permit insertion in holes or recesses. Greater contraction may be obtained by using high purity liquid nitrogen which has a temperature of about 320 degrees F. below zero. During a temperature reduction from 75 degrees F. to –321 degrees F., the shrinkage per inch of diameter varies from about 0.002 to 0.003 inch for steel; 0.0042 inch for aluminum alloys; 0.0046 inch for magnesium alloys; 0.0033 inch for copper alloys; 0.0023 inch for monel metal; and 0.0017 inch for cast iron (not alloyed). The cooling equipment may vary from an insulated bucket to a special automatic unit, depending upon the kind and quantity of work. One type of unit is so arranged that parts are precooled by vapors from the liquid nitrogen before immersion. With another type, cooling is entirely by the vapor method.

Shrinkage Fits.—General practice seems to favor a smaller allowance for shrinkage fits than for forced fits, although in many shops the allowances are practically the same for each, and for some classes of work, shrinkage allowances exceed those for forced fits. The shrinkage allowance also varies to a great extent with the form and construction of the part that has to be shrunk into place. The thickness or amount of metal around the hole is the most important factor. The way in which the metal is distributed also has an influence on the results. Shrinkage allowances for locomotive driving wheel tires adopted by the American Railway Master Mechanics Association are as follows:

Center diameter, inches	38	44	50	56	62	66
Allowances, inches	0.040	0.047	0.053	0.060	0.066	0.070

Whether parts are to be assembled by forced or shrinkage fits depends upon conditions. For example, to press a tire over its wheel center, without heating, would ordinarily be a rather awkward and difficult job. On the other hand, pins, etc., are easily and quickly forced into place with a hydraulic press and there is the additional advantage of knowing the exact pressure required in assembling, whereas there is more or less uncertainty connected with a shrinkage fit, unless the stresses are calculated. Tests to determine the difference in the quality of shrinkage and forced fits showed that the resistance of a shrinkage fit to slippage for an axial pull was 3.66 times greater than that of a forced fit, and in rotation or torsion, 3.2 times greater. In each comparative test, the dimensions and allowances were the same.

Allowances for Shrinkage Fits.—The most important point to consider when calculating shrinkage fits is the stress in the hub at the bore, which depends chiefly upon the shrinkage allowance. If the allowance is excessive, the elastic limit of the material will be exceeded and permanent set will occur, or, in extreme conditions, the ultimate strength of the metal will be exceeded and the hub will burst. The intensity of the grip of the fit and the resistance to slippage depends mainly upon the thickness of the hub; the greater the thickness, the stronger the grip, and *vice versa.* Assuming the modulus of elasticity for steel to be 30,000,000, and for cast iron, 15,000,000, the shrinkage allowance per inch of nominal diameter can be determined by the following formula, in which A = allowance per inch of diameter; T = true tangential tensile stress at inner surface of outer member; C = factor taken from one of the accompanying tables, *Factors for Calculating Shrinkage Fit Allowances*.

For a cast-iron hub and steel shaft:

$$A = \frac{T(2 + C)}{30,000,000} \tag{1}$$

When both hub and shaft are of steel:

$$A = \frac{T(1 + C)}{30,000,000} \tag{2}$$

If the shaft is solid, the factor C is taken from Table 1; if it is hollow and the hub is of steel, factor C is taken from Table 2; if it is hollow and the hub is of cast iron, the factor is taken from Table 3.

Table 1. Factors for Calculating Shrinkage Fit Allowances

Ratio of Diameters $\frac{D_2}{D_1}$	Steel Hub	Cast-iron Hub	Ratio of Diameters $\frac{D_2}{D_1}$	Steel Hub	Cast-iron Hub
1.5	0.227	0.234	2.8	0.410	0.432
1.6	0.255	0.263	3.0	0.421	0.444
1.8	0.299	0.311	3.2	0.430	0.455
2.0	0.333	0.348	3.4	0.438	0.463
2.2	0.359	0.377	3.6	0.444	0.471
2.4	0.380	0.399	3.8	0.450	0.477
2.6	0.397	0.417	4.0	0.455	0.482

Values of factor C for solid steel shafts of nominal diameter D_1, and hubs of steel or cast iron of nominal external and internal diameters D_2 and D_1, respectively.

Example 1: A steel crank web 15 inches outside diameter is to be shrunk on a 10-inch solid steel shaft. Required the allowance per inch of shaft diameter to produce a maximum tensile stress in the crank of 25,000 pounds per square inch, assuming the stresses in the crank to be equivalent to those in a ring of the diameter given.

The ratio of the external to the internal diameters equals $15 \div 10 = 1.5$; $T = 25,000$ pounds; from Table 1, $C = 0.227$. Substituting in Formula (2):

$$A = \frac{25,000 \times (1 + 0.227)}{30,000,000} = 0.001 \text{ inch}$$

Example 2: Find the allowance per inch of diameter for a 10-inch shaft having a 5-inch axial through hole, other conditions being the same as in Example 1.

The ratio of external to internal diameters of the hub equals $15 \div 10 = 1.5$, as before, and the ratio of external to internal diameters of the shaft equals $10 \div 5 = 2$. From Table 2, we find that factor $C = 0.455$; $T = 25,000$ pounds. Substituting these values in Formula (2):

$$A = \frac{25,000(1 + 0.455)}{30,000,000} = 0.0012 \text{ inch}$$

The allowance is increased, as compared with Example 1, because the hollow shaft is more compressible.

Table 2. Factors for Calculating Shrinkage Fit Allowances

$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C
1.5	2.0	0.468	2.4	2.0	0.798	3.4	2.0	0.926
	2.5	0.368		2.5	0.628		2.5	0.728
	3.0	0.322		3.0	0.549		3.0	0.637
	3.5	0.296		3.5	0.506		3.5	0.587
1.6	2.0	0.527	2.6	2.0	0.834	3.6	2.0	0.941
	2.5	0.414		2.5	0.656		2.5	0.740
	3.0	0.362		3.0	0.574		3.0	0.647
	3.5	0.333		3.5	0.528		3.5	0.596
1.8	2.0	0.621	2.8	2.0	0.864	3.8	2.0	0.953
	2.5	0.488		2.5	0.679		2.5	0.749
	3.0	0.427		3.0	0.594		3.0	0.656
	3.5	0.393		3.5	0.547		3.5	0.603
2.0	2.0	0.696	3.0	2.0	0.888	4.0	2.0	0.964
	2.5	0.547		2.5	0.698		2.5	0.758
	3.0	0.479		3.0	0.611		3.0	0.663
	3.5	0.441		3.5	0.562		3.5	0.610
2.2	2.0	0.753	3.2	2.0	0.909	…	…	…
	2.5	0.592		2.5	0.715		…	…
	3.0	0.518		3.0	0.625		…	…
	3.5	0.477		3.5	0.576		…	…

Values of factor C for hollow steel shafts and cast-iron hubs.
Notation as in Table 1.

Table 3. Factors for Calculating Shrinkage Fit Allowances

$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C	$\frac{D_2}{D_1}$	$\frac{D_1}{D_0}$	C
1.5	2.0	0.455	2.4	2.0	0.760	3.4	2.0	0.876
	2.5	0.357		2.5	0.597		2.5	0.689
	3.0	0.313		3.0	0.523		3.0	0.602
	3.5	0.288		3.5	0.481		3.5	0.555
1.6	2.0	0.509	2.6	2.0	0.793	3.6	2.0	0.888
	2.5	0.400		2.5	0.624		2.5	0.698
	3.0	0.350		3.0	0.546		3.0	0.611
	3.5	0.322		3.5	0.502		3.5	0.562
1.8	2.0	0.599	2.8	2.0	0.820	3.8	2.0	0.900
	2.5	0.471		2.5	0.645		2.5	0.707
	3.0	0.412		3.0	0.564		3.0	0.619
	3.5	0.379		3.5	0.519		3.5	0.570
2.0	2.0	0.667	3.0	2.0	0.842	4.0	2.0	0.909
	2.5	0.524		2.5	0.662		2.5	0.715
	3.0	0.459		3.0	0.580		3.0	0.625
	3.5	0.422		3.5	0.533		3.5	0.576
2.2	2.0	0.718	3.2	2.0	0.860	…	…	…
	2.5	0.565		2.5	0.676		…	…
	3.0	0.494		3.0	0.591		…	…
	3.5	0.455		3.5	0.544		…	…

Values of factor C for hollow steel shafts of external and internal diameters D_1 and D_0, respectively, and steel hubs of nominal external diameter D_2.

Example 3: If the crank web in Example 1 is of cast iron and 4000 pounds per square inch is the maximum tensile stress in the hub, what is the allowance per inch of diameter?

$$\frac{D_2}{D_1} = 1.5 \qquad T = 4000$$

In Table 1, we find that $C = 0.234$. Substituting in Formula (1), for cast-iron hubs, $A =$ 0.0003 inch, which, owing to the lower tensile strength of cast iron, is endout one-third the shrinkage allowance in Example 1, although the stress is two-thirds of the elastic limit.

Temperatures for Shrinkage Fits.—The temperature to which the outer member in a shrinkage fit should be heated for clearance in assembling the parts depends on the total expansion required and on the coefficient α of linear expansion of the metal (i.e., the increase in length of any section of the metal in any direction for an increase in temperature of 1 degree F). The total expansion in diameter that is required consists of the total allowance for shrinkage and an added amount for clearance. The value of the coefficient α is, for nickel-steel, 0.000007; for steel in general, 0.0000065; for cast iron, 0.0000062. As an example, take an outer member of steel to be expanded 0.005 inch per inch of internal diameter, 0.001 being the shrinkage allowance and the remainder for clearance. Then

$$\alpha \times t^{\circ} = 0.005$$

$$t = \frac{0.005}{0.0000065} = 769 \text{ degrees F}$$

The value t is the number of degrees F that the temperature of the member must be raised above that of the room temperature.

ANSI Standard Limits and Fits (ANSI B4.1-1967 (R1994)).—This American National Standard for Preferred Limits and Fits for Cylindrical Parts presents definitions of terms applying to fits between plain (non threaded) cylindrical parts and makes recommendations on preferred sizes, allowances, tolerances, and fits for use wherever they are applicable. This standard is in accord with the recommendations of American-British-Canadian (ABC) conferences up to a diameter of 20 inches. Experimental work is being carried on with the objective of reaching agreement in the range above 20 inches. The recommendations in the standard are presented for guidance and for use where they might serve to improve and simplify products, practices, and facilities. They should have application for a wide range of products.

As revised in 1967, and reaffirmed in 1979, the definitions in ANSI B4.1 have been expanded and some of the limits in certain classes have been changed.

Factors Affecting Selection of Fits.—Many factors, such as length of engagement, bearing load, speed, lubrication, temperature, humidity, and materials must be taken into consideration in the selection of fits for a particular application, and modifications in the ANSI recommendations may be required to satisfy extreme conditions. Subsequent adjustments may also be found desirable as a result of experience in a particular application to suit critical functional requirements or to permit optimum manufacturing economy.

Definitions.—The following terms are defined in this standard:

Nominal Size: The nominal size is the designation used for the purpose of general identification.

Dimension: A dimension is a geometrical characteristic such as diameter, length, angle, or center distance.

Size: Size is a designation of magnitude. When a value is assigned to a dimension, it is referred to as the size of that dimension. (It is recognized that the words "dimension" and "size" are both used at times to convey the meaning of magnitude.)

Allowance: An allowance is a prescribed difference between the maximum material limits of mating parts. (See definition of *Fit*). It is a minimum clearance (positive allowance) or maximum interference (negative allowance) between such parts.

Tolerance: A tolerance is the total permissible variation of a size. The tolerance is the difference between the limits of size.

Basic Size: The basic size is that size from which the limits of size are derived by the application of allowances and tolerances.

Design Size: The design size is the basic size with allowance applied, from which the limits of size are derived by the application of tolerances. Where there is no allowance, the design size is the same as the basic size.

Actual Size: An actual size is a measured size.

Limits of Size: The limits of size are the applicable maximum and minimum sizes.

Maximum Material Limit: A maximum material limit is that limit of size that provides the maximum amount of material for the part. Normally it is the maximum limit of size of an external dimension or the minimum limit of size of an internal dimension.*

Minimum Material Limit: A minimum material limit is that limit of size that provides the minimum amount of material for the part. Normally it is the minimum limit of size of an external dimension or the maximum limit of size of an internal dimension.*

Tolerance Limit: A tolerance limit is the variation, positive or negative, by which a size is permitted to depart from the design size.

Unilateral Tolerance: A unilateral tolerance is a tolerance in which variation is permitted in only one direction from the design size.

Bilateral Tolerance: A bilateral tolerance is a tolerance in which variation is permitted in both directions from the design size.

Unilateral Tolerance System: A design plan that uses only unilateral tolerances is known as a Unilateral Tolerance System.

Bilateral Tolerance System: A design plan that uses only bilateral tolerances is known as a Bilateral Tolerance System.

Fit: Fit is the general term used to signify the range of tightness that may result from the application of a specific combination of allowances and tolerances in the design of mating parts.

Actual Fit: The actual fit between two mating parts is the relation existing between them with respect to the amount of clearance or interference that is present when they are assembled. (Fits are of three general types: clearance, transition, and interference.)

Clearance Fit: A clearance fit is one having limits of size so specified that a clearance always results when mating parts are assembled.

Interference Fit: An interference fit is one having limits of size so specified that an interference always results when mating parts are assembled.

Transition Fit: A transition fit is one having limits of size so specified that either a clearance or an interference may result when mating parts are assembled.

Basic Hole System: A basic hole system is a system of fits in which the design size of the hole is the basic size and the allowance, if any, is applied to the shaft.

Basic Shaft System: A basic shaft system is a system of fits in which the design size of the shaft is the basic size and the allowance, if any, is applied to the hole.

* An example of exceptions: an exterior corner radius where the maximum radius is the minimum material limit and the minimum radius is the maximum material limit.

Preferred Basic Sizes.—In specifying fits, the basic size of mating parts may be chosen from the decimal series or the fractional series in the following table.

Table 1. Preferred Basic Sizes

Decimal			Fractional					
0.010	2.00	8.50	$\frac{1}{64}$	0.015625	$2\frac{1}{4}$	2.2500	$9\frac{1}{2}$	9.5000
0.012	2.20	9.00	$\frac{1}{32}$	0.03125	$2\frac{1}{2}$	2.5000	10	10.0000
0.016	2.40	9.50	$\frac{1}{16}$	0.0625	$2\frac{3}{4}$	2.7500	$10\frac{1}{2}$	10.5000
0.020	2.60	10.00	$\frac{3}{32}$	0.09375	3	3.0000	11	11.0000
0.025	2.80	10.50	$\frac{1}{8}$	0.1250	$3\frac{1}{4}$	3.2500	$11\frac{1}{2}$	11.5000
0.032	3.00	11.00	$\frac{5}{32}$	0.15625	$3\frac{1}{2}$	3.5000	12	12.0000
0.040	3.20	11.50	$\frac{3}{16}$	0.1875	$3\frac{3}{4}$	3.7500	$12\frac{1}{2}$	12.5000
0.05	3.40	12.00	$\frac{1}{4}$	0.2500	4	4.0000	13	13.0000
0.06	3.60	12.50	$\frac{5}{16}$	0.3125	$4\frac{1}{4}$	4.2500	$13\frac{1}{2}$	13.5000
0.08	3.80	13.00	$\frac{3}{8}$	0.3750	$4\frac{1}{2}$	4.5000	14	14.0000
0.10	4.00	13.50	$\frac{7}{16}$	0.4375	$4\frac{3}{4}$	4.7500	$14\frac{1}{2}$	14.5000
0.12	4.20	14.00	$\frac{1}{2}$	0.5000	5	5.0000	15	15.0000
0.16	4.40	14.50	$\frac{9}{16}$	0.5625	$5\frac{1}{4}$	5.2500	$15\frac{1}{2}$	15.5000
0.20	4.60	15.00	$\frac{5}{8}$	0.6250	$5\frac{1}{2}$	5.5000	16	16.0000
0.24	4.80	15.50	$\frac{11}{16}$	0.6875	$5\frac{3}{4}$	5.7500	$16\frac{1}{2}$	16.5000
0.30	5.00	16.00	$\frac{3}{4}$	0.7500	6	6.0000	17	17.0000
0.40	5.20	16.50	$\frac{7}{8}$	0.8750	$6\frac{1}{2}$	6.5000	$17\frac{1}{2}$	17.5000
0.50	5.40	17.00	1	1.0000	7	7.0000	18	18.0000
0.60	5.60	17.50	$1\frac{1}{4}$	1.2500	$7\frac{1}{2}$	7.5000	$18\frac{1}{2}$	18.5000
0.80	5.80	18.00	$1\frac{1}{2}$	1.5000	8	8.0000	19	19.0000
1.00	6.00	18.50	$1\frac{3}{4}$	1.7500	$8\frac{1}{2}$	8.5000	$19\frac{1}{2}$	19.5000
1.20	6.50	19.00	2	2.0000	9	9.0000	20	20.0000
1.40	7.00	19.50	…	…	…	…	…	…
1.60	7.50	20.00	…	…	…	…	…	…
1.80	8.00	…	…	…	…	…	…	…

All dimensions are in inches.

Preferred Series of Tolerances and Allowances (In thousandths of an inch)

0.1	1	10	100	0.3	3	30	…
…	1.2	12	125	…	3.5	35	…
0.15	1.4	14	…	0.4	4	40	…
…	1.6	16	160	…	4.5	45	…
…	1.8	18	…	0.5	5	50	…
0.2	2	20	200	0.6	6	60	…
…	2.2	22	…	0.7	7	70	…
0.25	2.5	25	250	0.8	8	80	…
…	2.8	28	…	0.9	9	…	…

Standard Tolerances.—The series of standard tolerances shown in Table 1 are so arranged that for any one grade they represent approximately similar production difficulties throughout the range of sizes. This table provides a suitable range from which appropriate tolerances for holes and shafts can be selected and enables standard gages to be used. The tolerances shown in Table 1 have been used in the succeeding tables for different classes of fits.

Table 1. ANSI Standard Tolerances *ANSI B4.1-1967 (R1987)*

Nominal Size, Inches		Grade									
		4	5	6	7	8	9	10	11	12	13
Over	To	Tolerances in thousandths of an inch[a]									
0	0.12	0.12	0.15	0.25	0.4	0.6	1.0	1.6	2.5	4	6
0.12	0.24	0.15	0.20	0.3	0.5	0.7	1.2	1.8	3.0	5	7
0.24	0.40	0.15	0.25	0.4	0.6	0.9	1.4	2.2	3.5	6	9
0.40	0.71	0.2	0.3	0.4	0.7	1.0	1.6	2.8	4.0	7	10
0.71	1.19	0.25	0.4	0.5	0.8	1.2	2.0	3.5	5.0	8	12
1.19	1.97	0.3	0.4	0.6	1.0	1.6	2.5	4.0	6	10	16
1.97	3.15	0.3	0.5	0.7	1.2	1.8	3.0	4.5	7	12	18
3.15	4.73	0.4	0.6	0.9	1.4	2.2	3.5	5	9	14	22
4.73	7.09	0.5	0.7	1.0	1.6	2.5	4.0	6	10	16	25
7.09	9.85	0.6	0.8	1.2	1.8	2.8	4.5	7	12	18	28
9.85	12.41	0.6	0.9	1.2	2.0	3.0	5.0	8	12	20	30
12.41	15.75	0.7	1.0	1.4	2.2	3.5	6	9	14	22	35
15.75	19.69	0.8	1.0	1.6	2.5	4	6	10	16	25	40
19.69	30.09	0.9	1.2	2.0	3	5	8	12	20	30	50
30.09	41.49	1.0	1.6	2.5	4	6	10	16	25	40	60
41.49	56.19	1.2	2.0	3	5	8	12	20	30	50	80
56.19	76.39	1.6	2.5	4	6	10	16	25	40	60	100
76.39	100.9	2.0	3	5	8	12	20	30	50	80	125
100.9	131.9	2.5	4	6	10	16	25	40	60	100	160
131.9	171.9	3	5	8	12	20	30	50	80	125	200
171.9	200	4	6	10	16	25	40	60	100	160	250

[a] All tolerances above heavy line are in accordance with American-British-Canadian (ABC) agreements.

Table 2. Relation of Machining Processes to Tolerance Grades

This chart may be used as a general guide to determine the machining processes that will under normal conditions, produce work withen the tolerance grades indicated.

(See also *Relation of Surface Roughness to Tolerances* starting on page 702.)