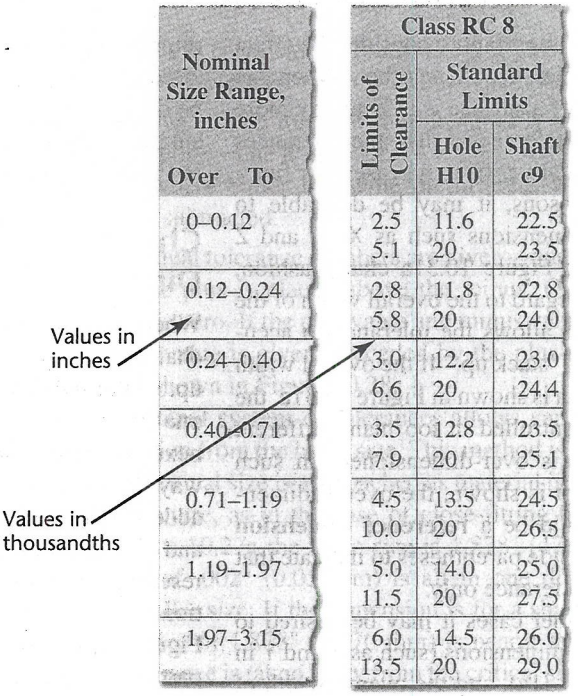
10.6 USING AMERICAN NATIONAL STANDARD LIMITS AND FIT TABLES

The American National Standards Institute has issued ANSI B4.1-1967 (R1994), "Preferred Limits and Fits for Cylindrical Parts," defining terms and recommending preferred standard sizes, allowances, tolerances, and fits in terms of the decimal inch. This standard gives a series of standard classes of fits on a unilateral-hole basis so that the fit produced by mating parts of a class of fit will produce approximately similar performance throughout the range of sizes. These tables give standard allowances for any given size or type of fit; they also prescribe the standard limits for the mating parts that will produce the fit.

The tables are designed for the basic hole system (see Appendixes 5–9). For coverage of the metric system of tolerances and fits, see Appendixes 13–16.

Table 10.1 gives the three general types of fits, the five subtypes, their letter symbols, and descriptions.

In the fit tables for each class of fit, the range of nominal sizes of shafts or holes is given in inches. To simplify the tables and reduce the space required to present them, the other values are given in thousandths of an inch as in the example shown in Figure 10.32. Minimum and maximum limits of clearance are given; the top number is the least clearance, or the allowance, and the lower number the maximum clearance, or loosest fit. Then, under the heading "Standard Limits," are the limits for the hole and for the shaft that are to be applied to the basic size to obtain the limits of size for the parts, using the basic hole system.

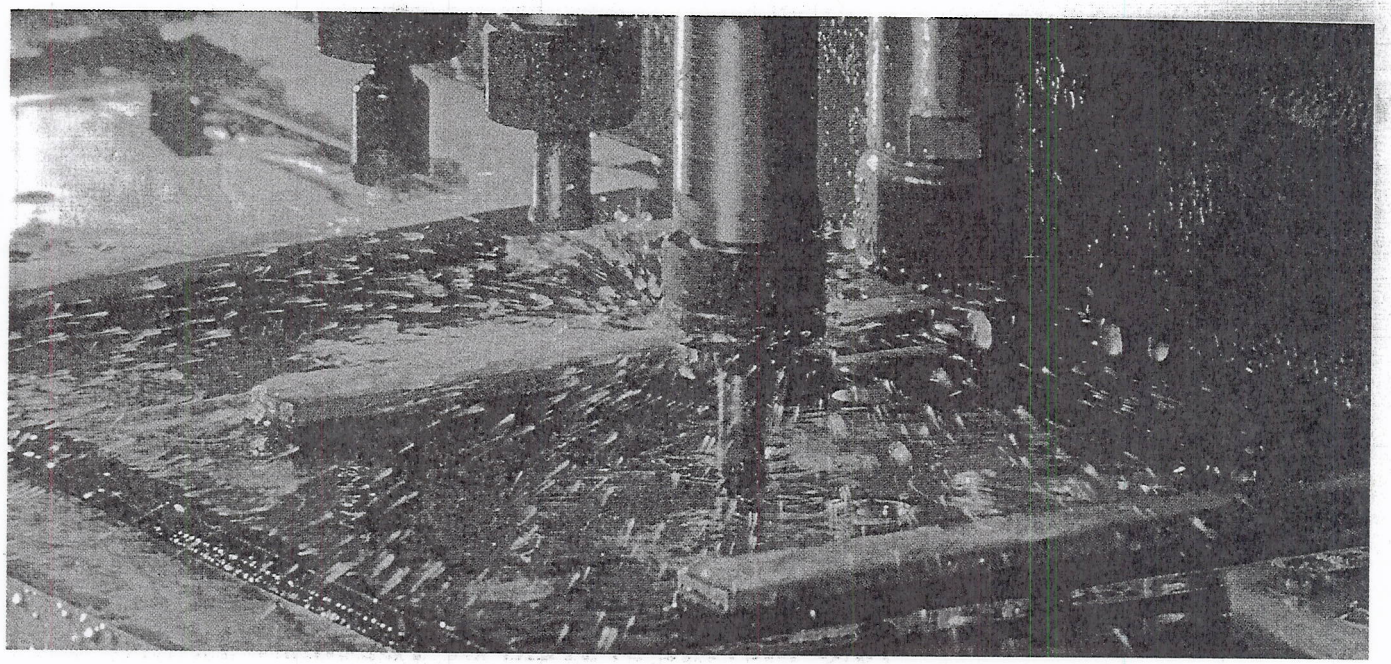


Nominal Size Range, inches Over	To	Class RC 8 Limits of Clearance	Standard Limits Hole H10	Standard Limits Shaft c9
0	0.12	2.5 5.1	11.6 20	22.5 23.5
0.12	0.24	2.8 5.8	11.8 20	22.8 24.0
0.24	0.40	3.0 6.6	12.2 20	23.0 24.4
0.40	0.71	3.5 7.9	12.8 20	23.5 25.1
0.71	1.19	4.5 10.0	13.5 20	24.5 26.5
1.19	1.97	5.0 11.5	14.0 20	25.0 27.5
1.97	3.15	6.0 13.5	14.5 20	26.0 29.0

10.32 Portion of RC8 Fit Table. The International Standards Organization (ISO) publishes a similar series of fit tables for metric values.

Table 10.1 General Fit Types and Subtypes.

Fit Type	Symbol	Subtype	Description
Clearance	RC	Running or sliding fits	Running and sliding fits (Appendix 7) are intended to provide a similar running performance, with suitable lubrication allowance, throughout the range of sizes. The clearances for the first two classes, used chiefly as slide fits, increase more slowly with diameter than the other classes, so that accurate location is maintained even at the expense of free relative motion.
Locational	LC	Clearance fits	Locational fits (Appendixes 6–8) are fits intended to determine only the location of the mating parts; they may provide rigid or accurate location, as with interference fits, or provide some freedom of location, as with clearance fits. Accordingly, they are divided into three groups: clearance fits, transition fits, and interference fits.
	LT	Transition clearance or interference fits	
	LN	Locational interference fits	
Interference	FN	Force or shrink fits	Force or shrink fits (Appendix 11) constitute a special type of interference fit, normally characterized by the maintenance of constant bore pressures throughout the range of sizes. The interference therefore varies almost directly with diameter and the difference between its minimum and maximum value is small in order to maintain the resulting pressures within reasonable limits.



Milling machines can produce parts within tolerances of .5 mm, or a few thousandths of an inch. *Courtesy of Ron Sherman/Creative Eye/MIRA.com*

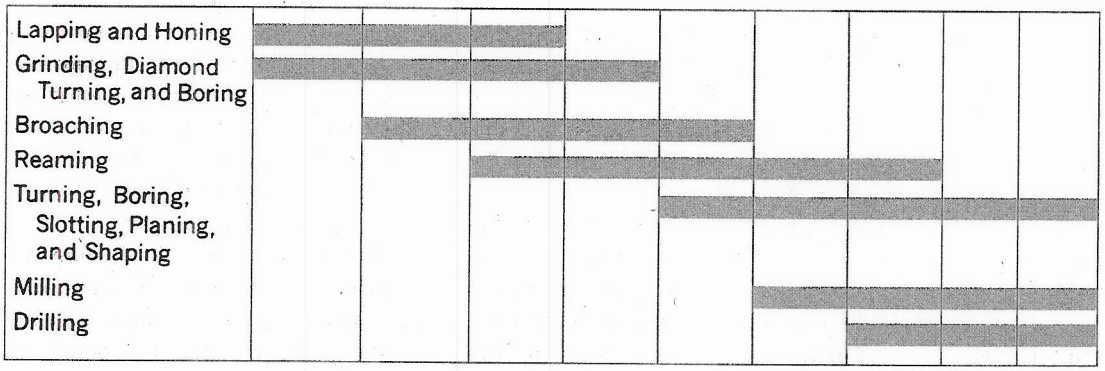
10.7 TOLERANCES AND MACHINING PROCESSES

Tolerances should be as generous as possible and still permit satisfactory use of the part. The tighter the tolerance, the more expensive it is to manufacture the part. Great savings can be gained from the use of less expensive tools, from lower labor and inspection costs, and from reduced scrapping of material.

Table 10.2 is a chart to be used as a general guide, with the tolerances achievable by the indicated machining processes. You can convert these to metric values by multiplying by 25.4 and rounding to one less decimal place.

Table 10.2 Tolerances Related to Machining Processes.

Range of Sizes From	Range of Sizes To and Including	Tolerances								
.000	.599	.00015	.0002	.0003	.0005	.0008	.0012	.002	.003	.005
.600	.999	.00015	.00025	.0004	.0006	.001	.0015	.0025	.004	.006
1.000	1.499	.0002	.0003	.0005	.0008	.0012	.002	.003	.005	.008
1.500	2.799	.00025	.0004	.0006	.001	.0015	.0025	.004	.006	.010
2.800	4.499	.0003	.0005	.0008	.0012	.002	.003	.005	.008	.012
4.500	7.799	.0004	.0006	.001	.0015	.0025	.004	.006	.010	.015
7.800	13.599	.0005	.0008	.0012	.002	.003	.005	.008	.012	.020
13.600	20.999	.0006	.001	.0015	.0025	.004	.006	.010	.015	.025



10.11 GEOMETRIC DIMENSIONING AND TOLERANCING

Geometric tolerances state the maximum allowable variations of a form or its position from the perfect geometry implied on the drawing. The term "geometric" refers to various forms, such as a plane, a cylinder, a cone, a square, or a hexagon. Theoretically, these are perfect forms, but because 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.

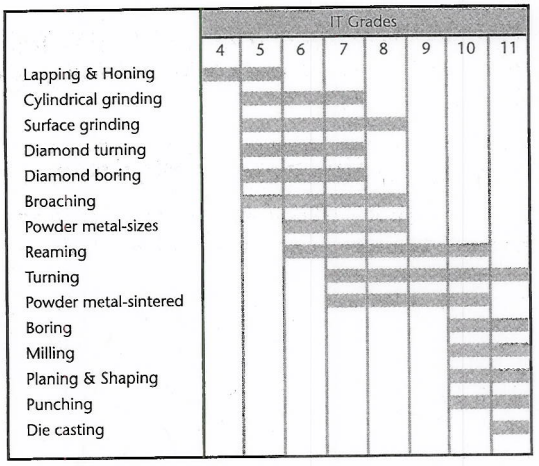
Tolerances of form and position (or location) control such characteristics as straightness, flatness, parallelism, perpendicularity (squareness), concentricity, roundness, angular displacement, and so on.

Methods of indicating geometric tolerances by means of geometric characteristic symbols, rather than by traditional notes, are recommended. See the latest Dimensioning and Tolerancing Standard, ANSI/ASME Y14.5M-1994, for more complete coverage.

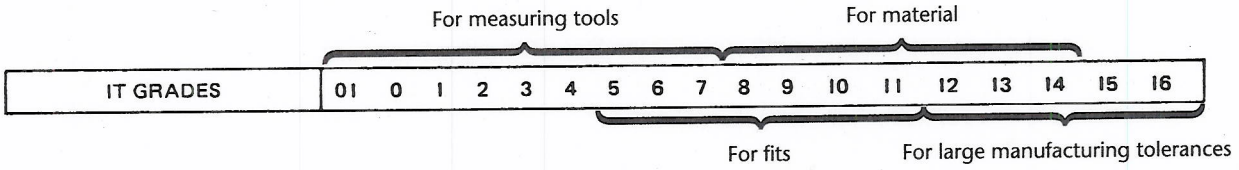
Table 10.4 Preferred Fits. *Reprinted from B4.2-1978, by permission of The American Society of Mechanical Engineers. All rights reserved.*

ISO Symbol				
	Hole Basis	Shaft Basis*	Description	
Clearance Fits	H11/c11	C11/h11	**Loose-running** fit for wide commercial tolerances or allowances on external members.	More Clearance →
	H9/d9	D9/h9	**Free-running** fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures.	
	H8/f7	F8/h7	**Close-running** fit for running on accurate machines and for accurate location at moderate speeds and journal pressures.	
Transition Fits	H7/g6	G7/h6	**Sliding** fit not intended to run freely, but to move and turn freely and locate accurately.	
	H7/h6	H7/h6	**Locational clearance** fit provides snug fit for locating stationary parts; but can be freely assembled and disassembled.	
	H7/k6	K7/h6	**Locational transition** fit for accurate location, a compromise between clearance and interference.	
	H7/n6	N7/h6	**Locational transition** fit for more accurate location where greater interference is permissible.	
Interference Fits	H7/p6	P7/h6	**Locational interference** fit for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements.	← More interference
	H7/s6	S7/h6	**Medium drive** fit for ordinary steel parts or shrink fits on light sections, the tightest fit usable with cast iron.	
	H7/u6	U7/h6	**Force** fit suitable for parts that can be highly stressed or for shrink fits where the heavy pressing forces required are impractical.	

**The transition and interference shaft-basis fits shown do not convert to exactly the same hole-basis fit conditions for basic sizes in the range from Q through 3 mm. Interference fit P7/h6 converts to a transition fit H7/p6 in the above size range.*



10.36 International Tolerance Grades Related to Machining Processes. *Reprinted from B4.2-1978, by permission of The American Society of Mechanical Engineers. All rights reserved.*



10.37 Practical Use of the International Tolerance Grades

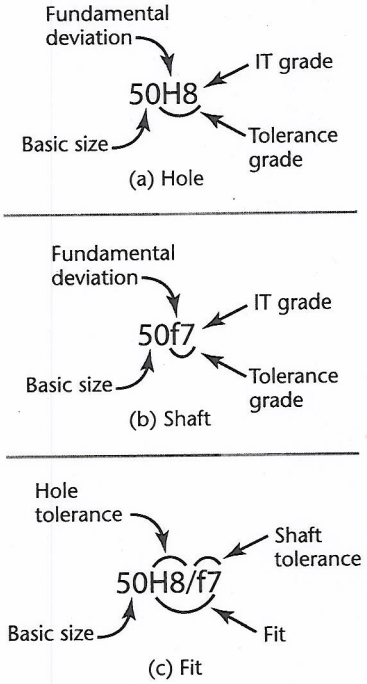
fundamental deviation is specified by the uppercase letter H.

Basic shaft system The basic shaft system of preferred fits is a system in which the basic diameter is the maximum size of the shaft. The fundamental deviation is given by the lowercase letter f, as shown in Figure 10.38b.

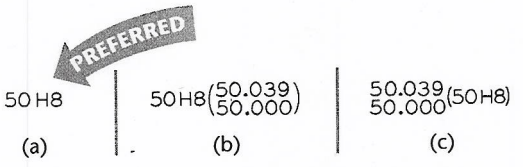
Interference fit An interference fit results in an interference between two mating parts under all tolerance conditions.

Transition fit A transition fit results in either a clearance or an interference condition between two assembled parts.

Tolerance symbols **Tolerance symbols** are used to specify the tolerances and fits for mating parts, as shown in Figure 10.38c. For the hole-basis system, the 50 indicates the diameter in millimeters, the capital letter H indicates the fundamental deviation for the hole, and the lowercase letter f indicates the deviation for the shaft. The numbers following the letters indicate the IT grade. Note that the symbols for the hole and shaft are separated by a slash. Tolerance symbols for a 50-mm-diameter hole may be given in several acceptable forms, as shown in Figure 10.39. The values in parentheses are for reference only and may be omitted. For upper and lower limit values, see Appendix 11.



10.38 Applications of Definitions and Symbols to Holes and Shafts. *Reprinted from B4.2-1978, by permission of The American Society of Mechanical Engineers. All rights reserved.*



10.39 Acceptable Methods of Giving Tolerance Symbols. *Reprinted from Y14.5M-1994, by permission of The American Society of Mechanical Engineers. All rights reserved.*

10.12 SYMBOLS FOR TOLERANCES OF POSITION AND FORM

Since traditional notes for specifying tolerances of position (location) and form (shape) may be confusing or unclear, may require too much space, and may not be understood internationally, most multinational companies have adopted symbols for such specifications (ANSI/ASME Y14.5M-1994). These ANSI symbols, shown in Table 10.5, provide an accurate and concise means of specifying **geometric characteristics** and tolerances in a minimum of space. A **feature control frame** specifies the tolerance for the geometric characteristic to be controlled and any modifying conditions that are required. The symbols may be supplemented by notes if the precise geometric requirements cannot be conveyed by the symbols. For construction details of the geometric tolerancing symbols, see Appendix 39.

Table 10.5 Geometric Characteristic and Modifying Symbols. *Reprinted from Y14.5M-1994, by permission of The American Society of Mechanical Engineers. All rights reserved.*

Geometric Characteristic Symbols				Modifying Symbols	
	Type of Tolerance	Characteristic	Symbol	Term	Symbol
For individual features	Form	Straightness	—	At maximum material condition	Ⓜ
		Flatness	▱	At least material condition	Ⓛ
		Circularity (roundness)	○	Projected tolerance zone	Ⓟ
		Cylindricity	⌭	Free state	Ⓕ
For individual or related features	Profile	Profile of a line	⌒	Tangent plane	Ⓣ
		Profile of a surface	⌓	Diameter	⌀
For related features	Orientation	Angularity	∠	Spherical diameter	S⌀
		Perpendicularity	⊥	Radius	R
		Parallelism	//	Spherical radius	SR
	Location	Position	⌖	Controlled radius	CR
		Concentricity	◎	Reference	( )
	Runout	Symmetry	⌯	Arc length	⌒
		Circular runout *	↗	Statistical tolerance	ⓈⓉ
		Total runout *	⌰	Between *	←→

**Arrowheads may be filled or not filled.*