

Tolerancing

Tolerance is the total amount a dimension may vary and is the difference between the **upper** (maximum) and **lower** (minimum) limits.

Because it is impossible to make everything to an exact size, tolerances are used on production drawings *to control the parts*.

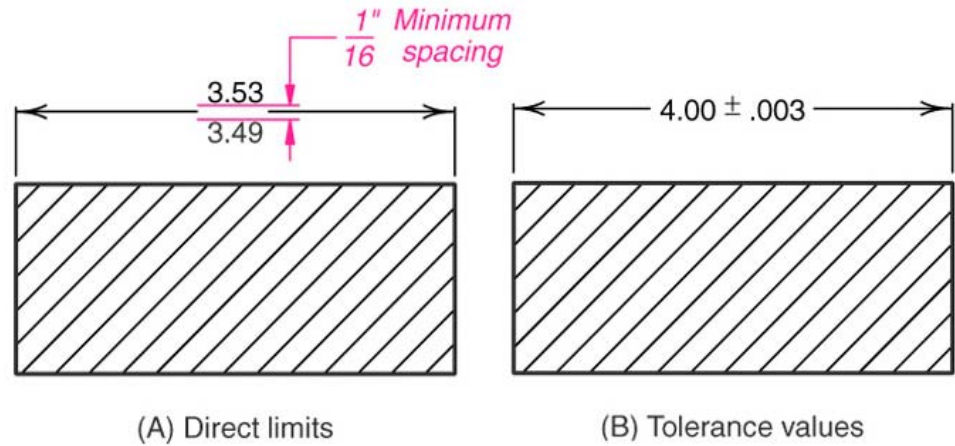
When do we need tolerances?

In particular, tolerances are assigned *to mating parts* in an assembly. For example, in case, when the slot in the part must accommodate another part.

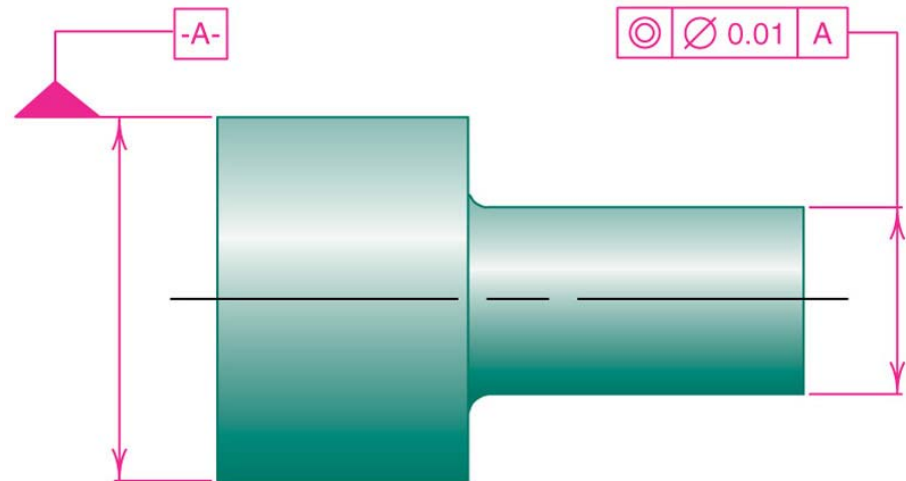
One of the great advantages of using tolerances is that it allows for **interchangeable parts**, thus permitting the replacement of individual parts.

Tolerances can be expressed in several ways:

1. *Direct limits*, or as *tolerance values* applied directly to a dimension.

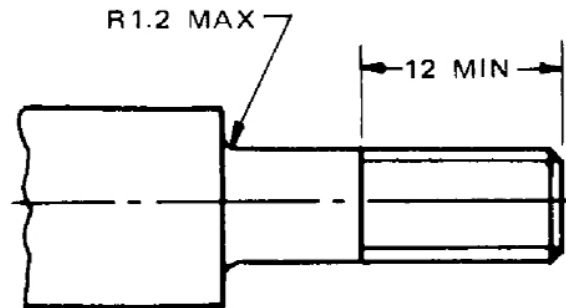


2. *Geometric tolerances*, indicated by special symbols related to part surfaces.



3. Notes referring to *specific conditions*, usually placed next to the corresponding dimensions.

Example: *single limit tolerance* which limits either the maximum or minimum size of a feature or a space, leaving the other limit of size unspecified.



4. A *general tolerance* note in the title block.

2		21/5/02	dia & length of rotor mount.		AK
1		21/5/02	width of bearing space		AK
NO.	DATE	CHANGE RECORD		AUTH	INIT
PROJECTION 	ENGLISH DIMS. IN INCHES	SCALE 13:10	DRAWN BY A. KRUPNIK	DATE 14-Aug-01	CHECKED APPROVED
GENERAL TOLERANCES 1 DECIAL ± 0.1 2 DECIAL ± 0.01 3 DECIAL ± 0.001			McGill UNIVERSITY 3480 UNIVERSITY ST. MONTREAL, PQ, H3A 2A7 CANADA TEL: 514-398-6313 FAX: 514-398-7348 E-MAIL: admin@cim.mcgill.ca		
SURFACE TEXTURE N6			MODEL NAME MOTOR_SHAFT		
MATERIAL ALUMINUM 6061-T6			DRAWING NO. W-003-0A		SHEET 1 OF 1

General Tolerances

- ✓ General tolerances are given in a note or as part of the title block. A general tolerance note would be similar to:

ALL DECIMAL DIMENSIONS TO BE HELD

TO $\pm .002$ "

This means that a dimension such as .500 would be assigned a tolerance of $\pm .002$, resulting in an upper limit of .502 and a lower limit of .498.

- ✓ For metric dimensions, the note would be similar:

ALL METRIC DIMENSIONS TO BE HELD

TO ± 0.05

This means that a dimension such as 65.00 would be assigned a tolerance of ± 0.05 , resulting in an upper limit of 65.05 and a lower limit of 64.95.

- ✓ If fractional dimensions are used, a general note might be:

ALL FRACTIONAL DIMENSIONS $\pm 1/64$ "

UNLESS OTHERWISE SPECIFIED

- ✓ Angular dimensions can be toleranced through a note such as:

ALL ANGULAR TOLERANCES ± 1 DEGREE

- ✓ Another general tolerance method would specify the tolerances on dimensions in terms of the number of decimal places found in the dimensions, as follows:

UNLESS OTHERWISE SPECIFIED,
TOLERANCES ARE AS FOLLOWS:

Decimal inches:

x = $\pm .020$

xx = $\pm .010$

xxx = $\pm .005$

or

UNLESS OTHERWISE SPECIFIED,

1 PLACE DECIMALS +/- .1

2 PLACE DECIMALS +/- .03

3 PLACE DECIMALS +/- .008

Millimeters:

Whole numbers = ± 0.5

x = $\pm .1$

xx = $\pm .025$

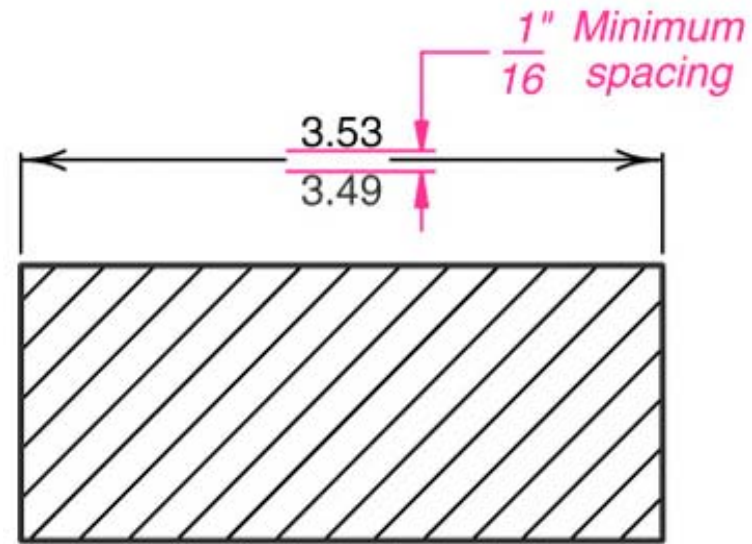
In this method, the dimension applied to each feature automatically identifies the required tolerance. Actual tolerances may vary from one company to another, but the ones given here are common tolerances for machined parts.

General tolerances may contain just one set of figures if all dimensions have the same number of decimal places.

If a dimension has a tolerance added directly to it, that tolerance *supersedes* the general tolerance note. **A tolerance added to a dimension always supersedes the standard tolerance, even if the added tolerance is larger than the standard tolerance.**

Direct Limits

Tolerances can be applied directly to dimensioned features, using limit dimensioning. This is the ASME preferred method; the maximum and minimum sizes are specified as part of the dimension.



(A) Direct limits

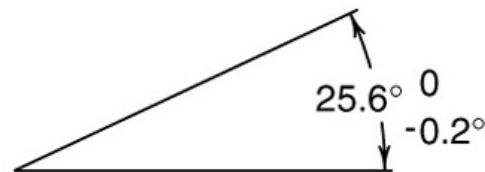
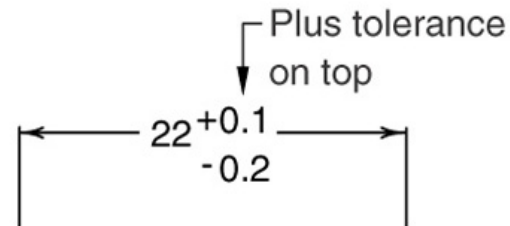
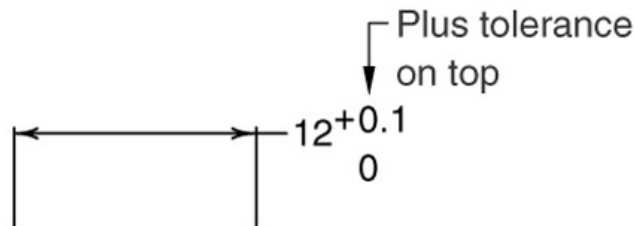
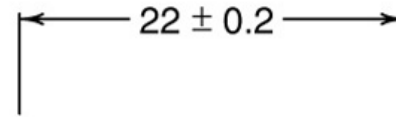
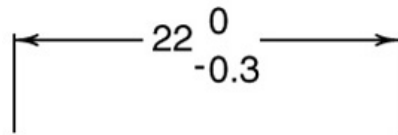
Either the upper limit is placed above the lower limit, or when the dimension is written in a single line, the lower limit precedes the upper limit and they are separated by a dash or by a slash:

3.49 - 3.53

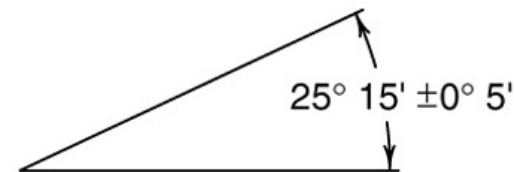
3.49/3.53

Plus and Minus Dimensions

With this approach the basic size is given, followed by a plus/minus sign and the tolerance value.



(A) Unilateral tolerancing

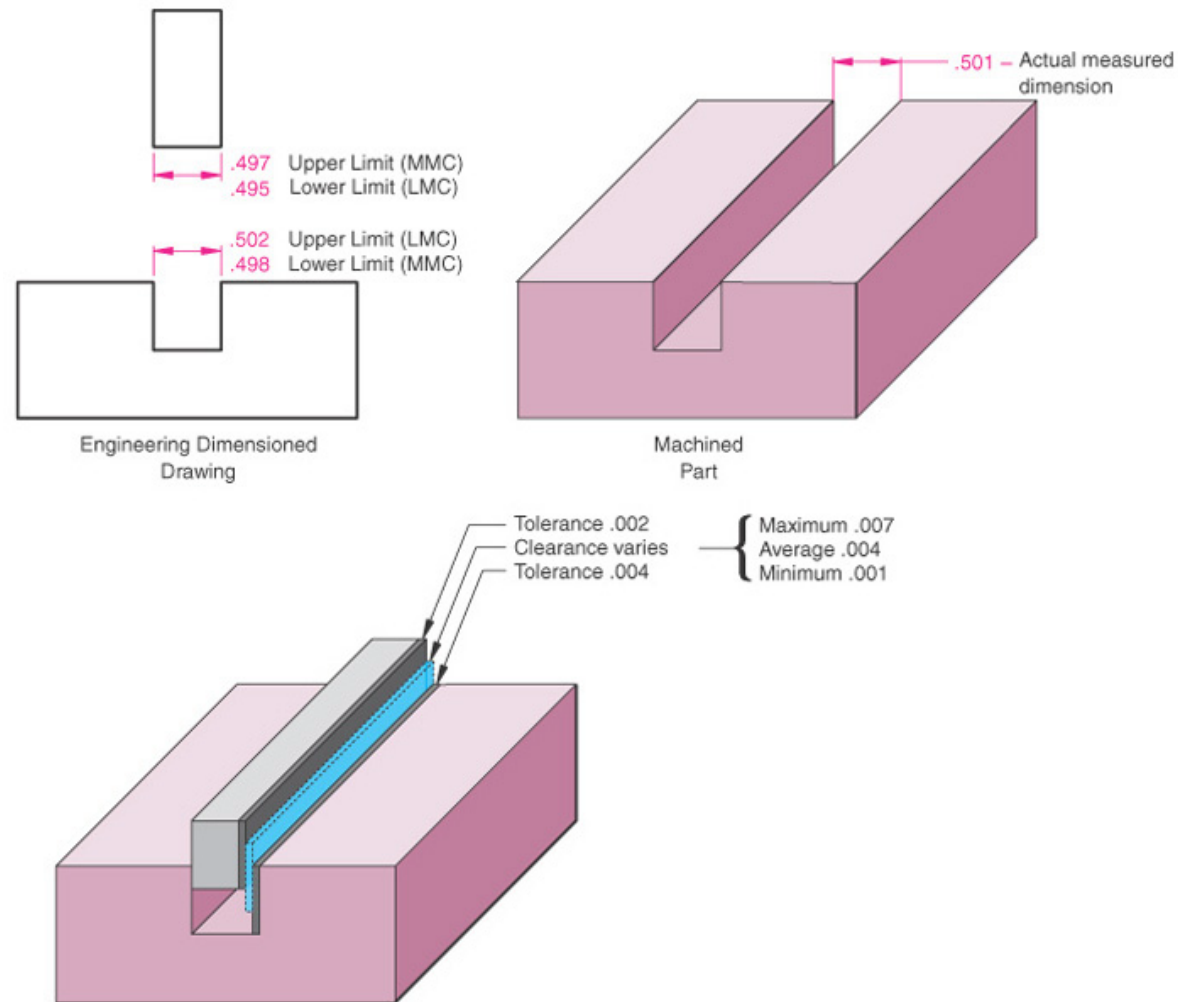


(B) Bilateral tolerancing

Tolerance can be unilateral or bilateral. A **unilateral tolerance** varies in only one direction, while a **bilateral tolerance** varies in both directions from the basic size.

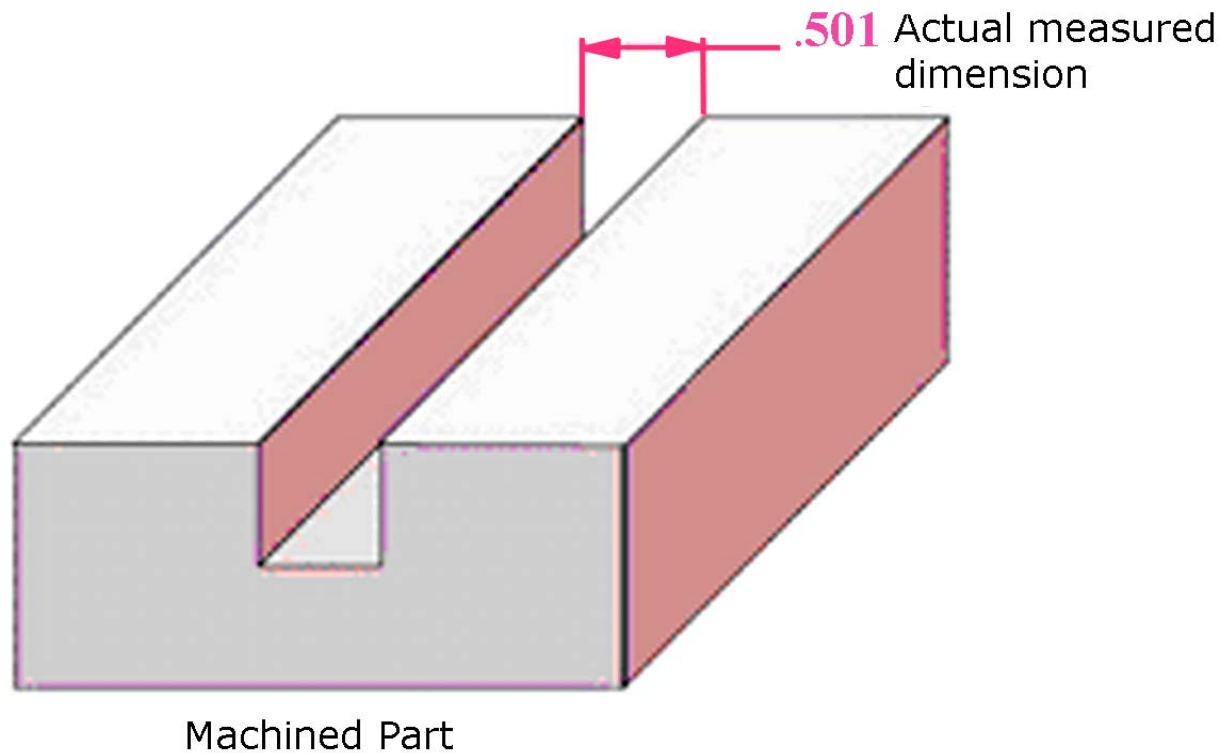
If the variation is equal in both directions, then the variation is preceded by a $\pm$ symbol. *The plus and minus approach can only be used when the two variations are equal.*

Important terms



The figure shows a system of two parts that have toleranced dimensions. These two parts are used as an example in ASME/ANSI standard to define important terms.

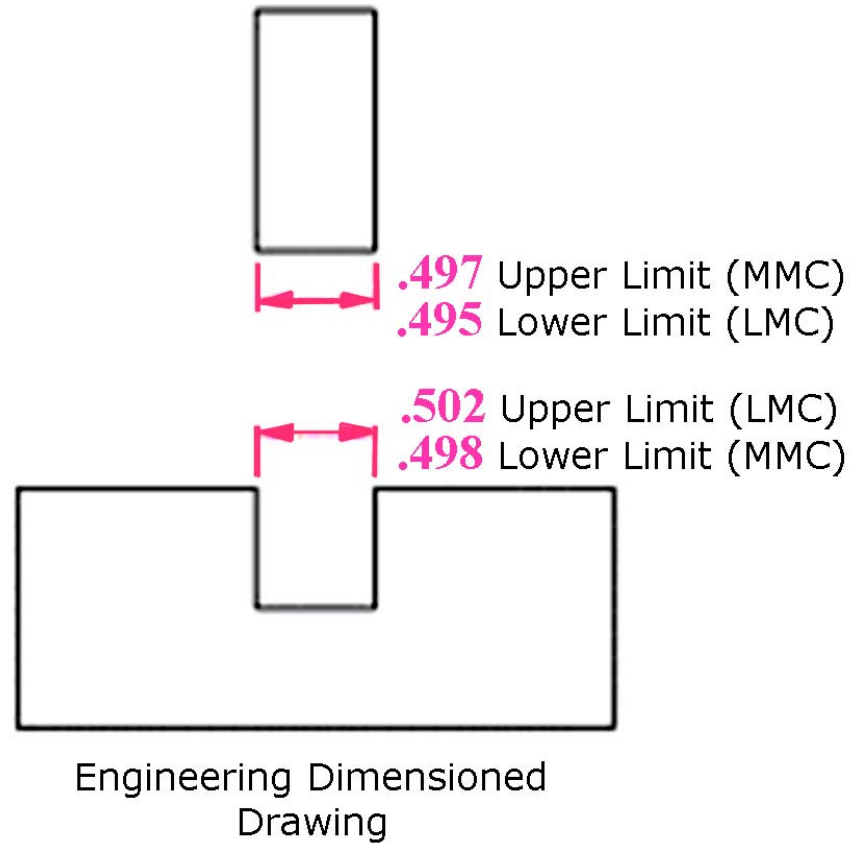
- **Nominal size** – a dimension used to describe the general size, usually expressed in common fractions. The slot in the figure has a nominal size of $1/2$ ".
- **Basic size** – the theoretical size used as a starting point for the application of tolerances. The basic size of the slot is $.500$ ".



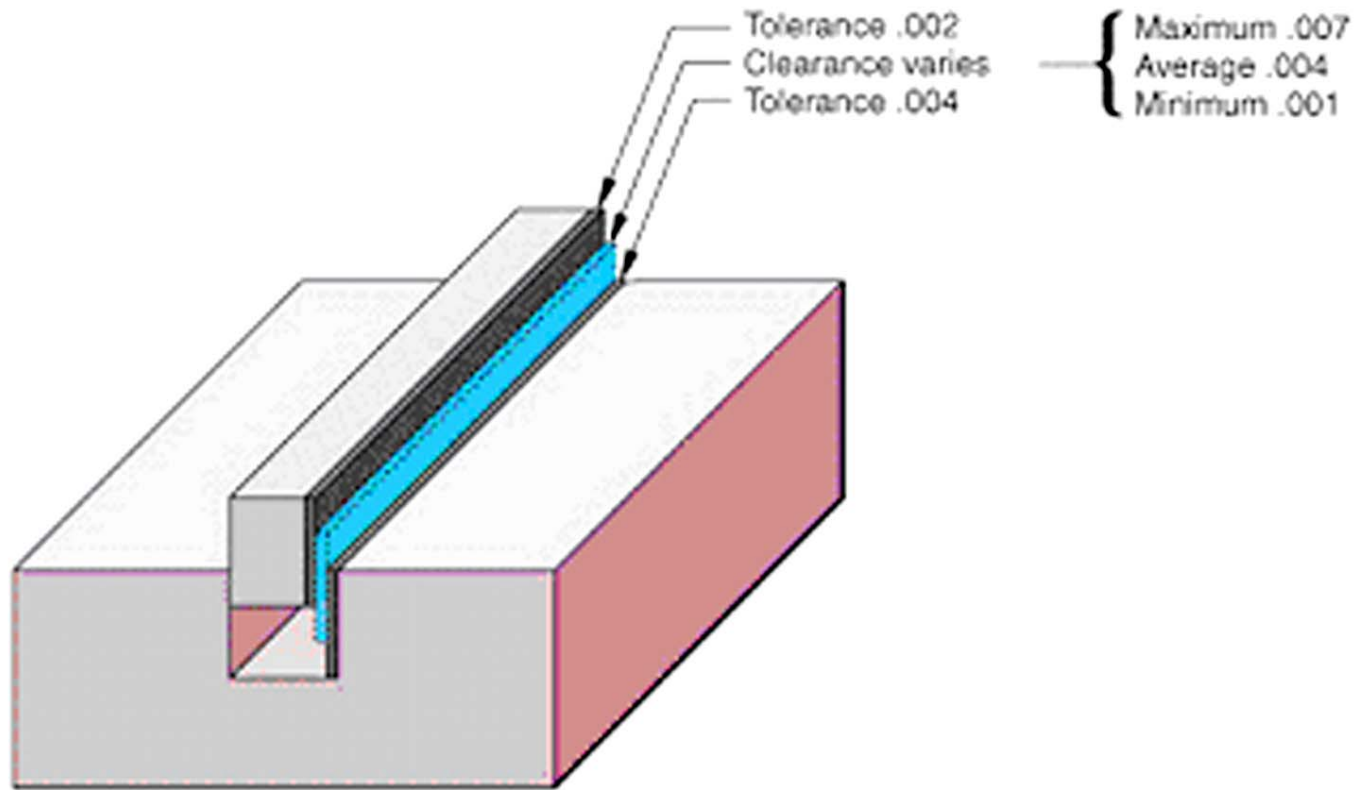
- **Actual size** – the measured size of the finished part after machining. Here it is $.501$ ".

- **Limits** – the maximum and minimum sizes shown by the tolerated dimension. The slot in figure has limits of .502 and .498, and the mating part has limits of .495 and .497.

The larger value for each part is the *upper limit*, and the smaller value is the *lower limit*.

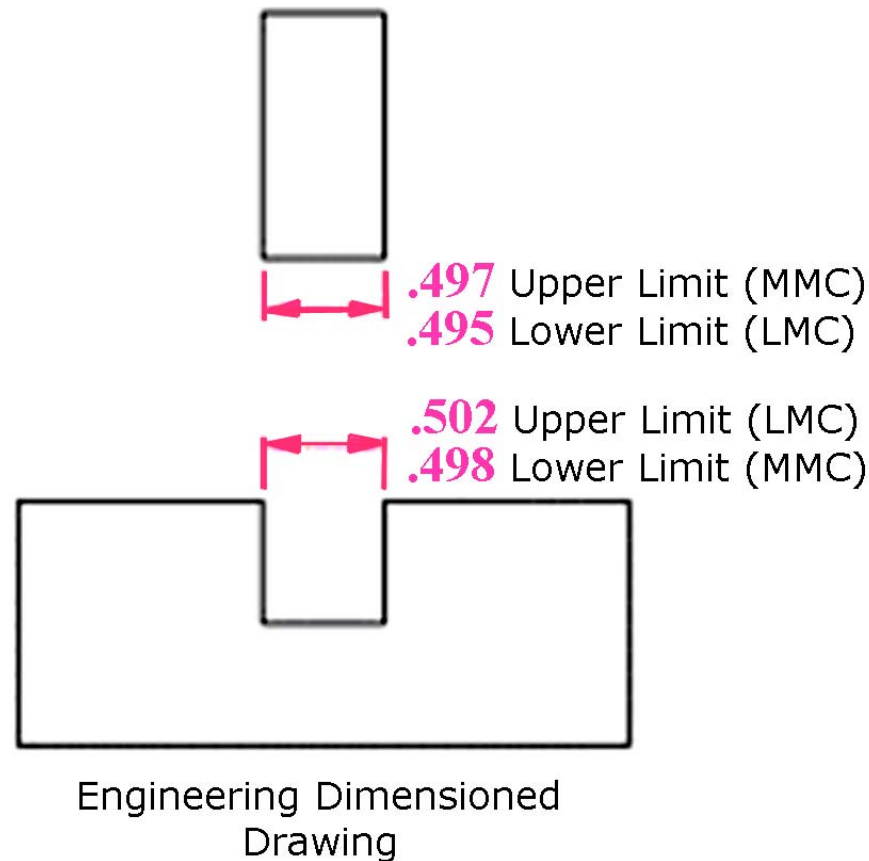


- **Allowance** is the minimum clearance or maximum interference between parts, or the *tightest* fit between two mating parts. In figure the allowance is .001, meaning that the tightest fit occurs when the slot is machined to its smallest allowable size of .498 and the mating part is machined to its largest allowable size of .497. The difference between .498 and .497, or .001, is the allowance.



- **Tolerance** is the total allowable variance in a dimension, which is the difference between the upper and lower limits. The tolerance of the slot in Figure is .004" ($.502 - .498 = .004$) and the tolerance of the mating part is .002" ($.497 - .495 = .002$).

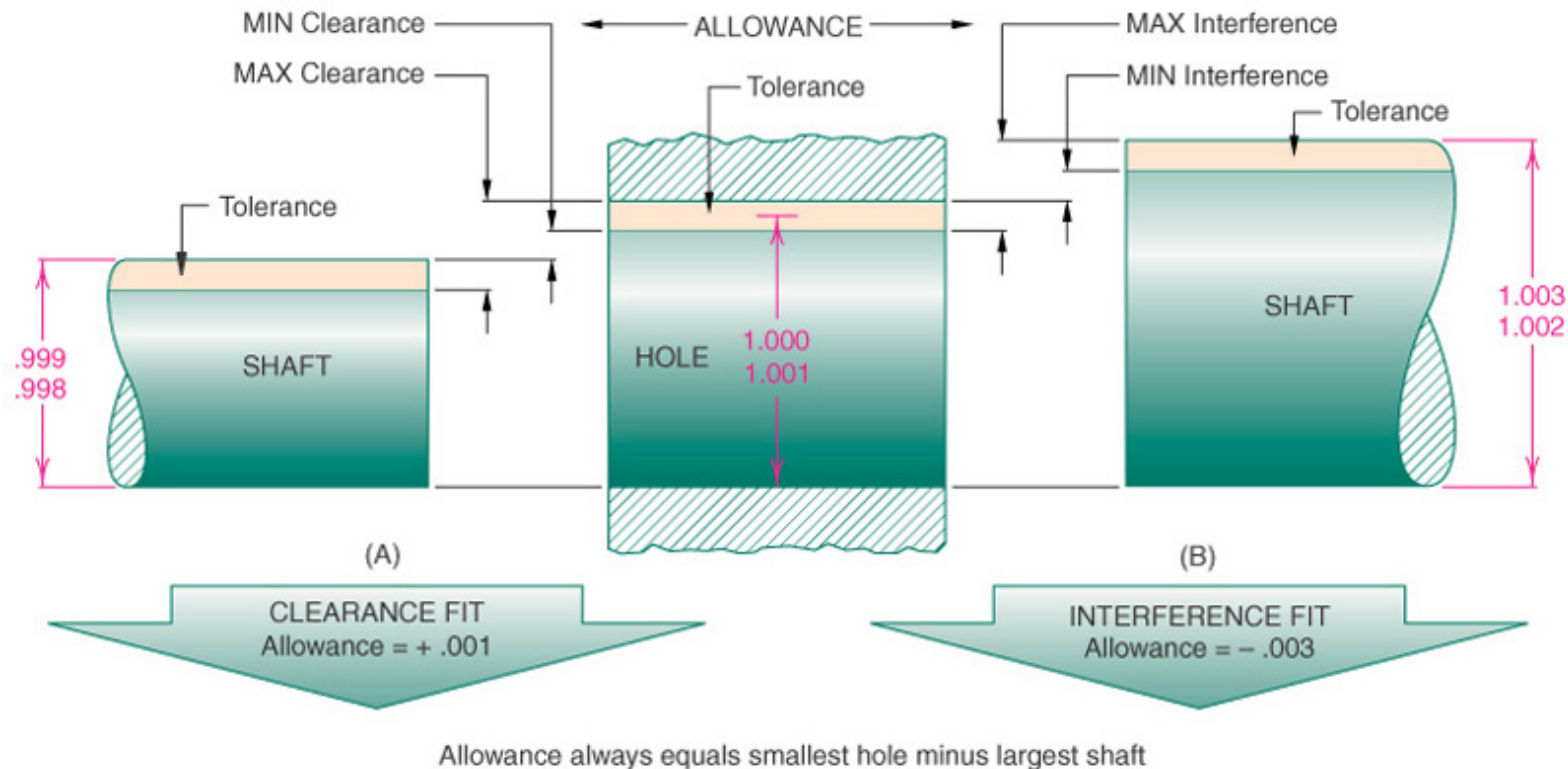
- **Maximum material condition (MMC)** is the condition of a part when it contains the most amount of material. The MMC of an external feature (such as a shaft) is the upper limit. The MMC of an internal feature (such as a hole) is the lower limit.



- **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 is the lower limit of the part. The LMC of an internal feature is the upper limit of the part.

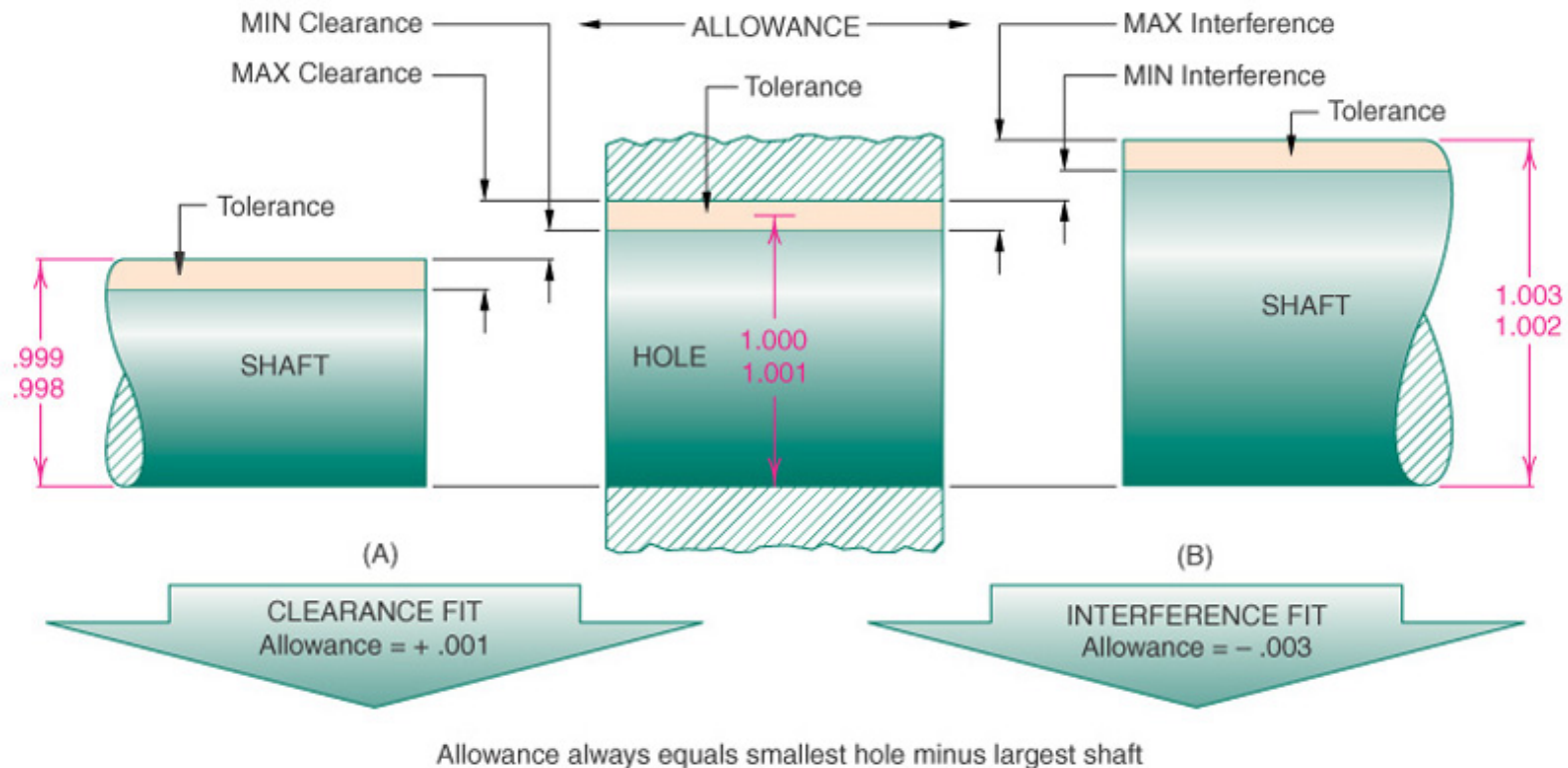
Fit types

The degree of tightness between mating parts is called the **fit**.



Clearance fit occurs when two toleranced mating parts will always leave a space or clearance when assembled. In figure above, the largest that shaft A can be manufactured is .999, and the smallest the hole can be is 1.000. The shaft will always be smaller than the hole, resulting in a *minimum clearance* of +.001, also called an *allowance*. The maximum clearance occurs when the smallest shaft (.998) is mated with the largest hole (1.001), resulting in a difference of +.003.

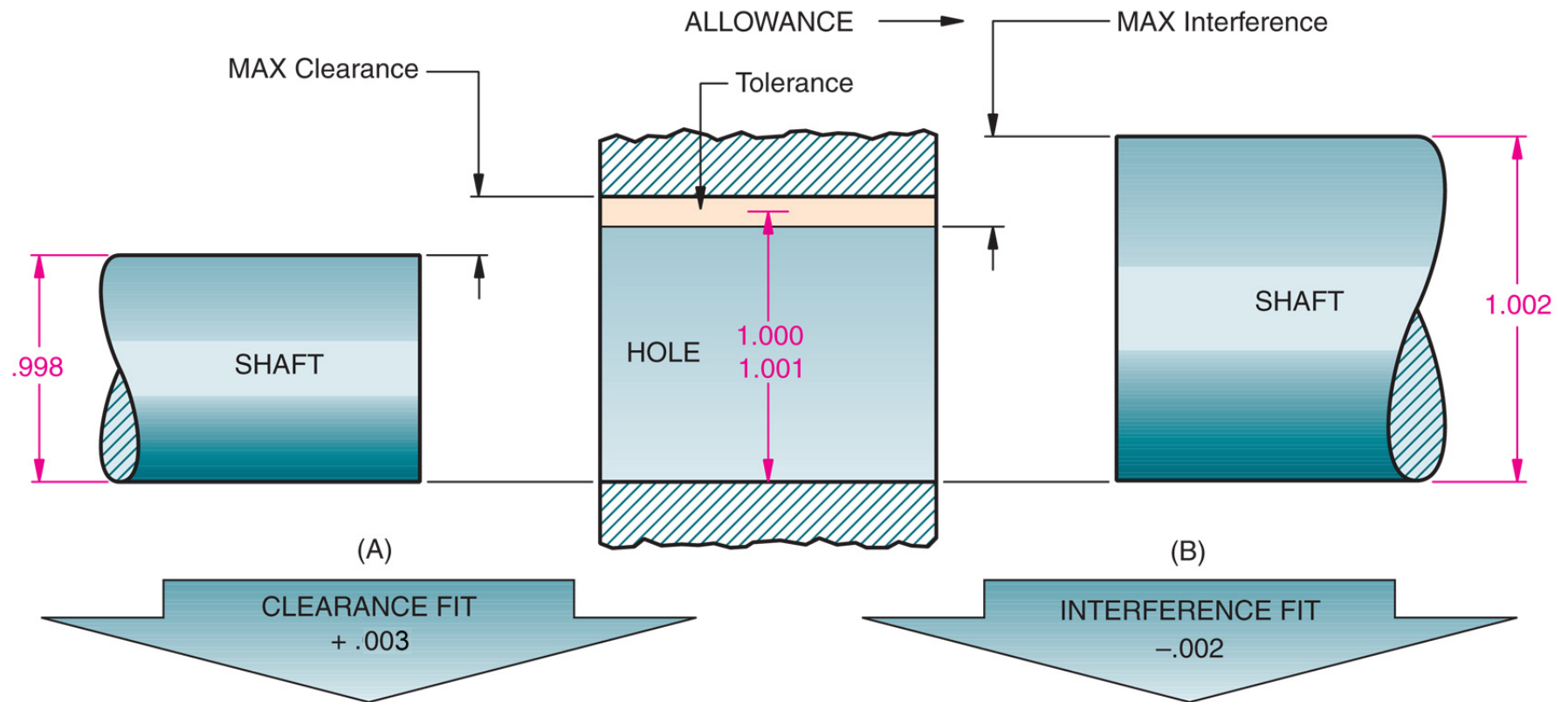
Interference fit occurs when two toleranced mating parts will always interfere when assembled. An interference fit *fixes* or *anchors* one part into the other, as though the two parts were one. In the figure, the smallest that shaft B can be manufactured is 1.002, and the largest the hole can be manufactured is 1.001. This means that the shaft will always be larger than the hole, and the minimum interference is $-.001$.



The maximum interference would occur when the smallest hole (1.000) is mated with the largest shaft (1.003), resulting in an interference of $-.003$.

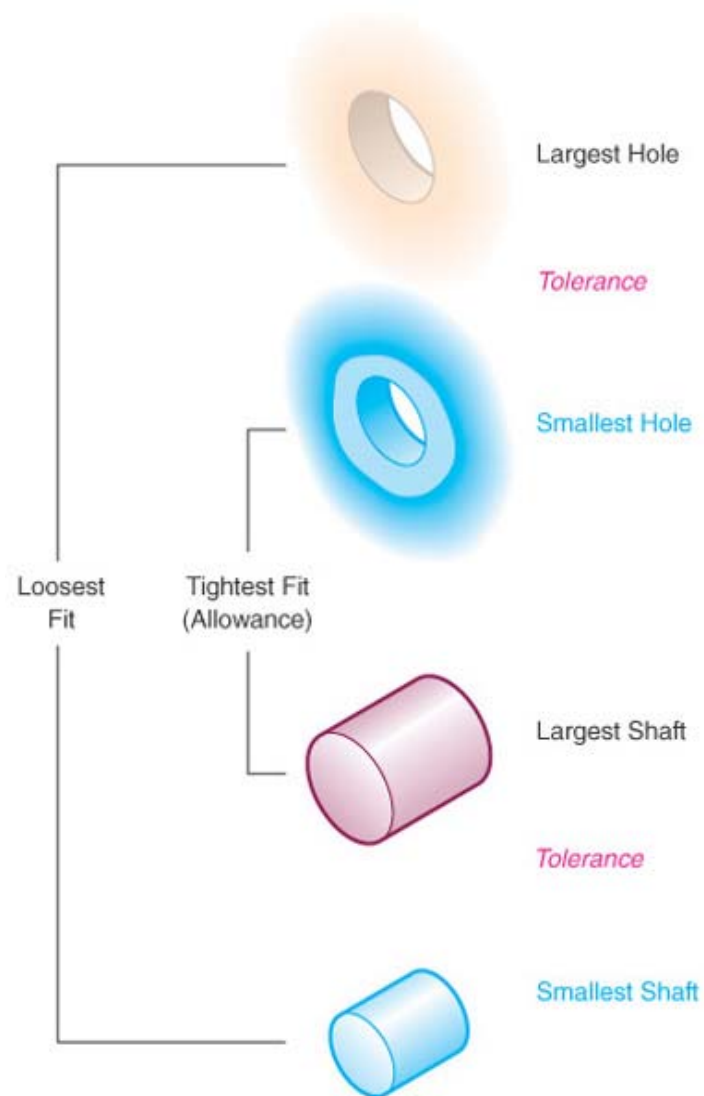
In order to assemble the parts under this condition, it would be necessary to *stretch* the hole or *shrink* the shaft or to use force to press the shaft into the hole. This kind of fit can be used to fasten two parts together without the use of mechanical fasteners or adhesive.

Transition fit occurs when two toleranced mating parts will sometimes be an interference fit and sometimes be a clearance fit when assembled.



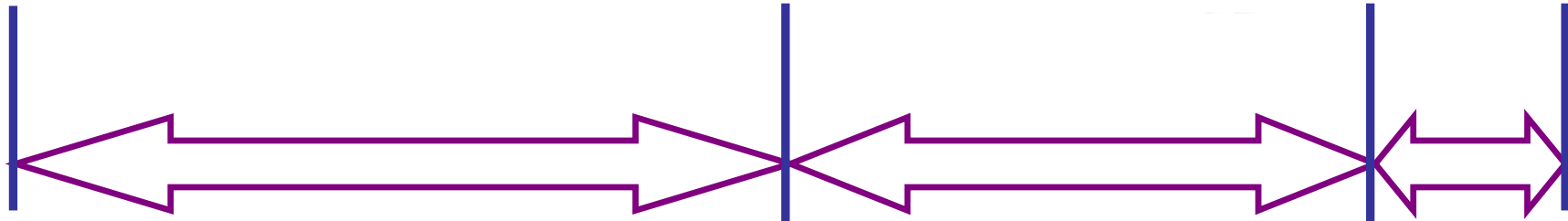
In the figure, the smallest the shaft can be manufactured is $.998$, and the largest the hole can be manufactured is 1.001 , resulting in a max clearance of $+.003$. The largest the shaft can be manufactured is 1.002 , and the smallest the hole can be is 1.000 , resulting in a max interference of $-.002$.

The **loosest fit** is the difference between the smallest feature A (shaft) and the largest feature B (HOLE). The **tightest fit** is the difference between the largest feature A (SHAFT) and the smallest feature B (hole):



Fits for holes and shafts

Class of fit	1	2	3	4	5	6	7	8
Bar graph (basic hole system)								
	Loose fit	Free fit	Medium fit	Snug fit	Wringing fit	Tight fit	Medium force fit	Heavy force and shrink fit



Clearance Fit

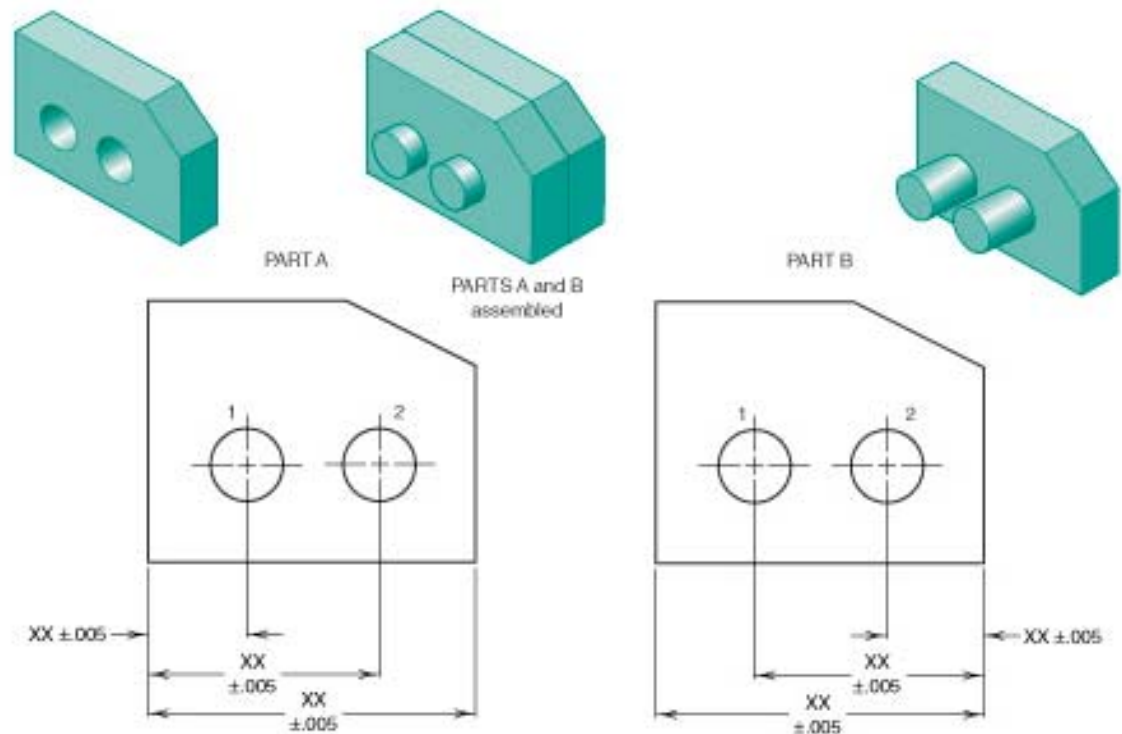
Transition Fit

Interference Fit

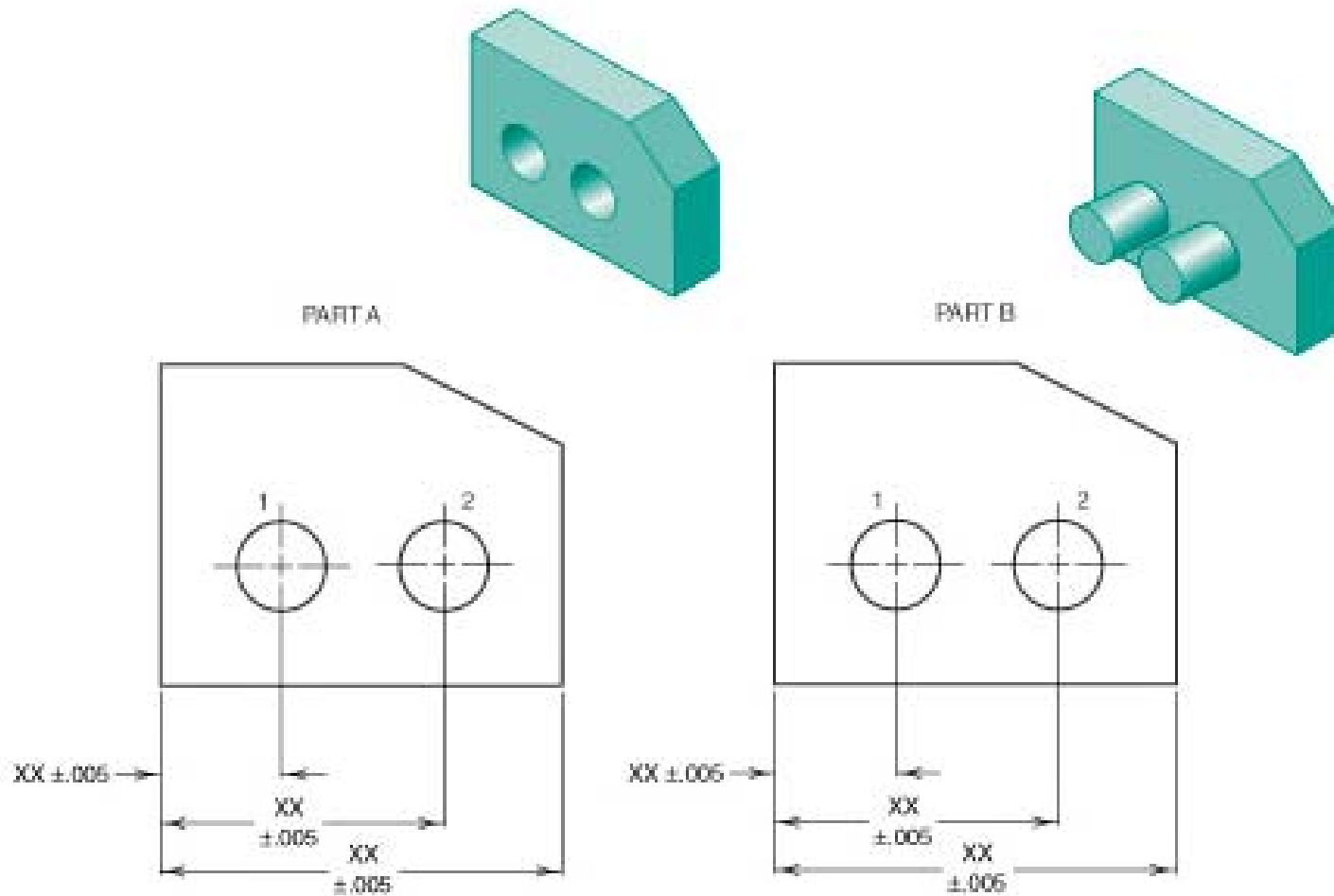
The **additive rule** for tolerances is that tolerances taken in the same direction from one point of reference are additive. The consequence is that tolerances to the same point taken from different directions become additive. The effect is called **tolerance stack-up**.

Example: Part A is manufactured to fit Part B.

When assembled, the cylindrical features on Part B are inserted into Part A so that the inclined surface on each part is aligned. The holes in Part A are dimensioned from the left side of the part, and the holes in Part B are dimensioned from the right side of the part.

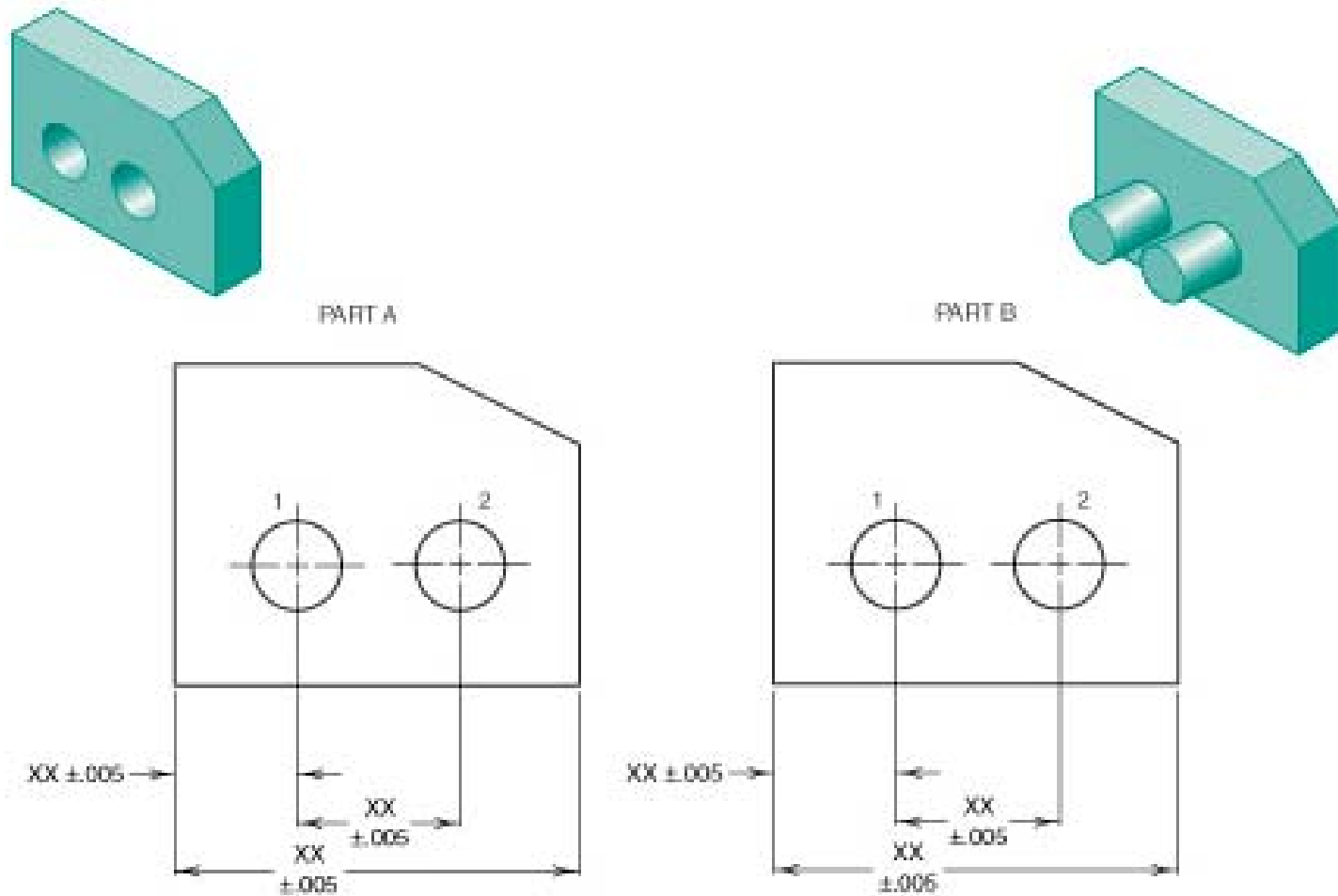


If the distance between the two holes (cylinders) were toleranced on the basis of Part A or Part B alone, the tolerance would be ± 0.010 , which is the sum of the two tolerances taken from the side in each case. However, the two parts are to be fitted together, and the tolerance for the distance between hole and cylinder must be determined on the basis of *both* Parts A and B. Since the distances to the two features were taken from different references in the two parts, the tolerances must again be added. This makes the tolerance for the distance between the hole and the cylinder ± 0.015 .



If the reference for Part B had been the same as that for Part A, that is, the left side of the part, the tolerance for the distance between the holes would have remained at ± 0.010 , which would have been more desirable.

However, the most desirable, and the most recommended, procedure is to relate the two holes directly to each other, not to either side of the part, and not to the overall width of the part. The result will be the best tolerance, ± 0.005 , for the distance between the holes:



Metric limits and fits

The standards used for metric measurements are recommended by the International Standards Organisation (ISO).

The terms used in metric tolerancing are as follows:

Basic Size – the exact theoretical size to which limits of deviation are assigned and are the same for both parts.

Deviation – the difference between the size of the part and the basic size.

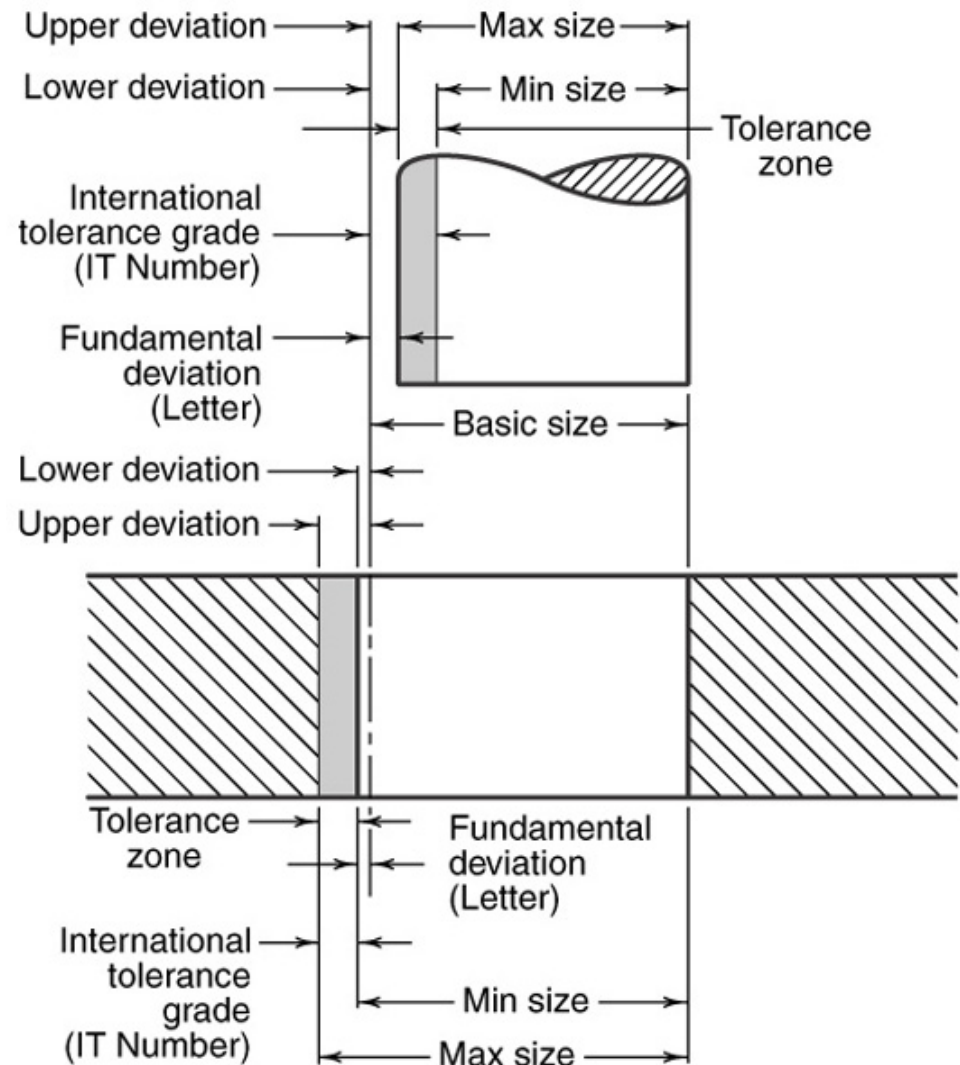
Upper deviation – the difference between the maximum size limit and the basic size.

Lower deviation is the difference between the minimum limit of size and the basic size.

Fundamental deviation is the deviation closest to the basic size (for both parts). It is denoted by a letter.

Tolerance is the difference between the maximum and minimum size limits on a part.

Tolerance zone represents the tolerance and its position in relation to the basic size.



International tolerance grade (IT) – the classification system – representing groups of tolerances which vary depending upon the basic size, but have the same level of accuracy with a given grade. It is denoted by the combinations IT0, IT1, and IT01 to IT16 – altogether 18 IT grades.

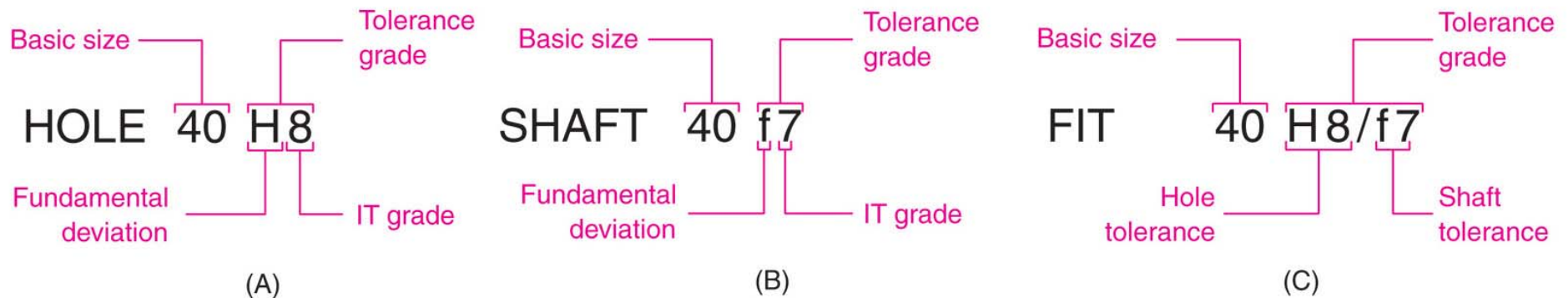
Relation of Machining Processes to Tolerance Grades

MACHINING OPERATION	TOLERANCE GRADES									
	4	5	6	7	8	9	10	11	12	13
Lapping & Honing	■	■								
Cylindrical Grinding	■	■	■	■						
Surface Grinding	■	■	■	■	■					
Diamond Turning	■	■	■	■						
Diamond Boring	■	■	■	■						
Broaching	■	■	■	■	■					
Reaming	■	■	■	■	■	■	■			
Turning	■	■	■	■	■	■	■	■	■	■
Boring	■	■	■	■	■	■	■	■	■	■
Milling	■	■	■	■	■	■	■	■	■	■
Planing & Shaping	■	■	■	■	■	■	■	■	■	■
Drilling	■	■	■	■	■	■	■	■	■	■

Practical Usage of IT Tolerance Grades

	IT Grades																			
	01	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16		
Measuring Tools																				
Fits																				
Material																				
Large Manufacturing Tolerances																				

For example, in the following figure with the notations of hole, shaft and their fit, the numbers 7 and 8 are IT grades:



Hole basis is the system of fits where the minimum hole size is the basic size. The fundamental deviation for a hole basis system is indicated by the uppercase letter “H”.

Shaft basis is the system of fits where the maximum shaft size is the basic size. The fundamental deviation for a shaft basis system is indicated by the lowercase letter “f”.

Metric Tolerance Symbols

Combining the IT grade number and the tolerance position letter establishes the tolerance symbol, which identifies the actual upper and lower limits of a part.

The tolerance size of the part is defined by the basic size followed by a letter and a number, such as 40H8 or 40f7. For example the notation of a metric fit would appear as **40H8**, where:

40	The basic size of 40 millimeters.
H	Fundamental deviation of an internal feature (hole).
8	A close running clearance fit.

Here are the three methods of designating metric tolerances on drawings:

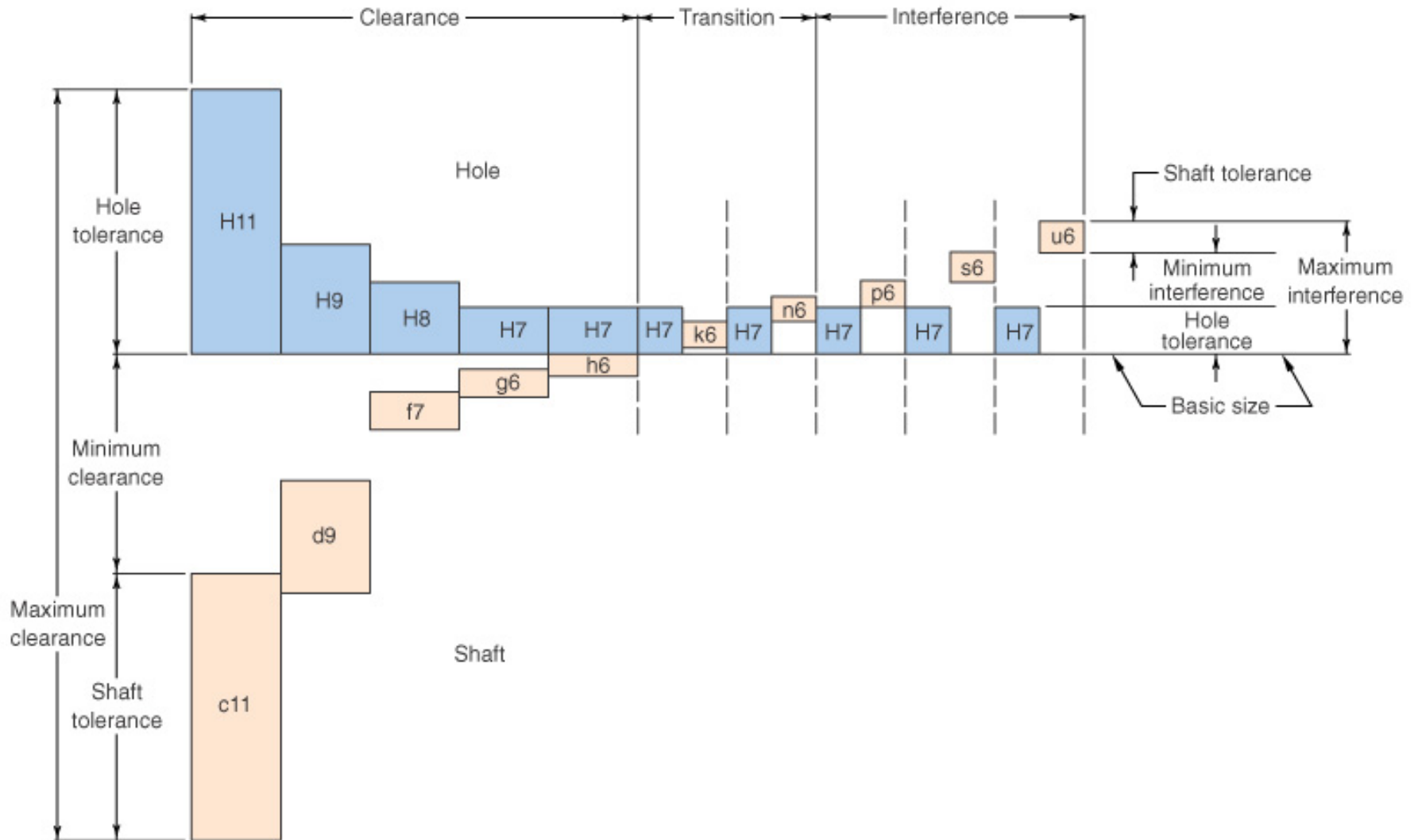
40H8	40H8 $\left(\begin{array}{c} 40.039 \\ 40.000 \end{array} \right)$	$\left(\begin{array}{c} 40.039 \\ 40.000 \end{array} \right)$ 40H8
(A)	(B)	(C)

The types of fits with corresponding letter and number symbols, as well as with upper and lower tolerance limits can be taken from the table:

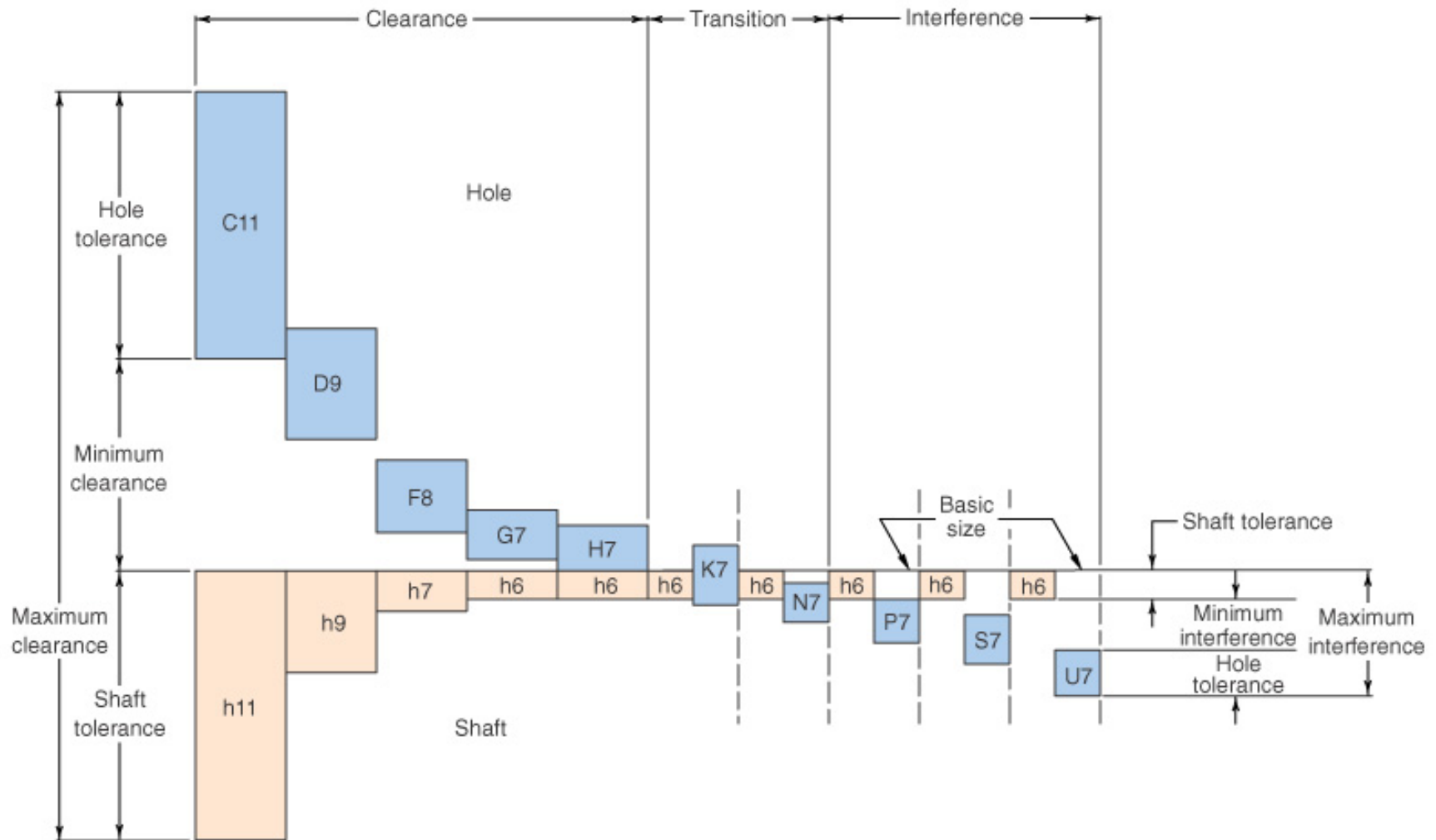
BASIC SIZE		LOOSE RUNNING			FREE RUNNING			CLOSE RUNNING			SLIDING			LOCATIONAL CLEARANCE		
		Hole H11	Shaft c11	Fit	Hole H9	Shaft d9	Fit	Hole H8	Shaft f7	Fit	Hole H7	Shaft g6	Fit	Hole H7	Shaft h6	Fit
40	MAX	40.160	39.880	0.440	40.062	39.920	0.204	40.039	39.975	0.029	40.025	39.991	0.050	40.025	40.000	0.041
	MIN	40.000	39.720	0.120	40.000	39.858	0.060	40.000	39.950	0.025	40.000	39.975	0.009	40.000	39.984	0.000
50	MAX	50.160	49.870	0.450	50.062	49.920	0.204	50.039	49.975	0.089	50.025	49.991	0.050	50.025	50.000	0.041
	MIN	50.000	49.710	0.130	50.000	49.858	0.080	50.000	49.950	0.025	50.000	49.975	0.009	50.000	49.984	0.000
60	MAX	60.190	59.860	0.520	60.074	59.900	0.248	60.046	59.970	0.106	60.030	59.990	0.059	60.030	60.000	0.049
	MIN	60.000	59.670	0.140	60.000	59.826	0.100	60.000	59.940	0.030	60.000	59.971	0.010	60.000	59.981	0.000
80	MAX	80.190	79.550	0.530	80.074	79.900	0.248	80.046	79.970	0.106	80.030	79.990	0.059	80.030	80.000	0.049
	MIN	80.000	79.660	0.150	80.000	79.826	0.100	80.000	79.940	0.030	80.000	79.971	0.010	80.000	79.981	0.000
100	MAX	100.220	99.830	0.610	100.087	99.880	0.294	100.054	99.964	0.125	100.035	99.988	0.069	100.035	100.000	0.057
	MIN	100.000	99.610	0.170	100.000	99.793	0.120	100.000	99.929	0.036	100.000	99.966	0.012	100.000	99.978	0.000
120	MAX	120.220	119.820	0.620	120.087	119.880	0.294	120.054	119.964	0.125	120.035	119.988	0.069	120.035	120.000	0.057
	MIN	120.000	119.600	0.180	120.000	119.793	0.120	120.000	119.929	0.036	120.000	119.966	0.012	120.000	119.978	0.000
160	MAX	160.250	159.790	0.710	160.100	159.855	0.345	160.063	159.957	0.146	160.040	159.986	0.078	160.040	160.000	0.065
	MIN	160.000	159.540	0.210	160.000	159.755	0.145	160.000	159.917	0.043	160.000	159.961	0.014	160.000	159.975	0.000
200	MAX	200.290	199.760	0.820	200.115	199.830	0.400	200.072	199.950	0.168	200.046	199.985	0.040	200.046	200.000	0.075
	MIN	200.000	199.470	0.240	200.000	199.715	0.170	200.000	199.904	0.050	200.000	199.956	0.015	200.000	199.971	0.000
250	MAX	250.290	249.720	0.860	250.115	249.830	0.400	250.072	249.950	0.168	250.046	249.985	0.090	250.046	250.000	0.075
	MIN	250.000	249.430	0.280	250.000	249.715	0.170	250.000	249.904	0.050	250.000	249.956	0.015	250.000	249.971	0.000
300	MAX	300.320	299.670	0.970	300.130	299.810	0.450	300.081	299.944	0.189	300.052	299.983	0.101	300.052	300.000	0.084
	MIN	300.000	299.350	0.330	300.000	299.680	0.190	300.000	299.892	0.056	300.000	299.951	0.017	300.000	299.968	0.000
400	MAX	400.360	399.600	1.120	400.140	399.790	0.490	400.089	399.938	0.208	400.057	399.982	0.111	400.057	400.000	0.093
	MIN	400.000	399.240	0.400	400.000	399.650	0.210	400.000	399.881	0.062	400.000	399.946	0.018	400.000	399.964	0.000
500	MAX	500.400	499.520	1.280	500.155	499.770	0.540	500.097	499.932	0.228	500.063	499.980	0.123	500.063	500.000	0.103
	MIN	500.000	499.120	0.480	500.000	499.615	0.230	500.000	499.869	0.068	500.000	499.940	0.020	500.000	499.960	0.000

This is a **hole basis table**. The *hole basis system* for clearance, interference, and transition fits means that the fundamental deviation of the hole (i.e difference between the minimum size limit and the basis size) is zero.

The hole based system of fits is shown here:



If the *shaft basis system* for clearance, interference, and transition fits is used, that means that the fundamental deviation for shaft is zero. The metric preferred shaft basis system of fits in this case is as follows:

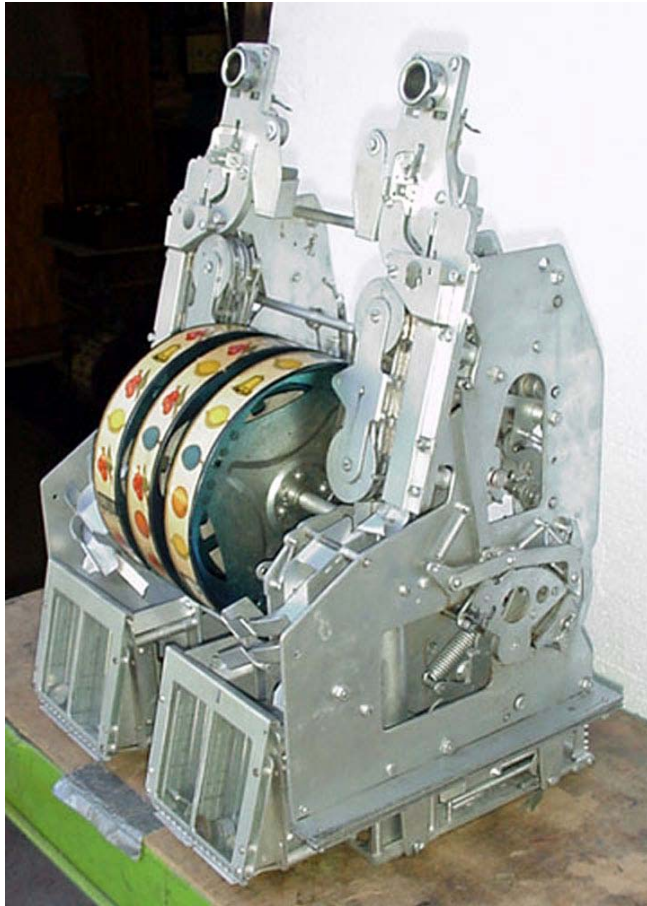


The description and application of the hole basis system and shaft basis system are given in the following table:

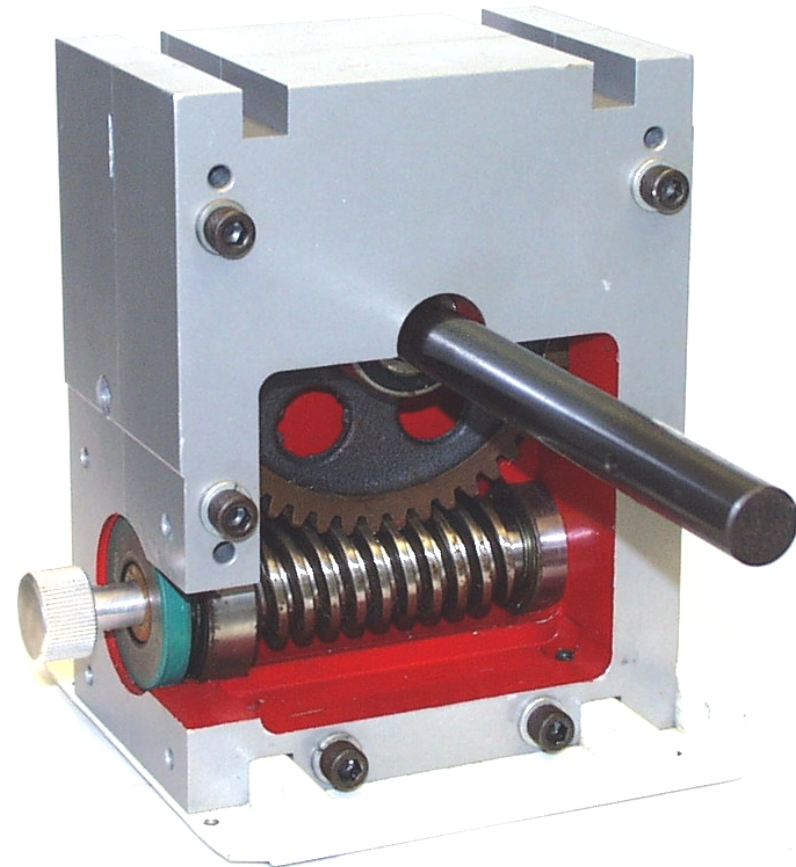
ISO Symbol		Description	
Hole Basis	Shaft Basis		
Clearance fits	H11/c11	C11/h11	Loose running fit for wide commercial tolerances or allowances on external members
	H9/d9	D9/h9	Free running fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or 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
	H7/g6	G7/h6	Sliding fit not intended to run freely but to move and turn freely and locate accurately
Transition fits	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
	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

*Transition fit for basic sizes in range from 0 through 3 mm

Example – *Fitting bearings*



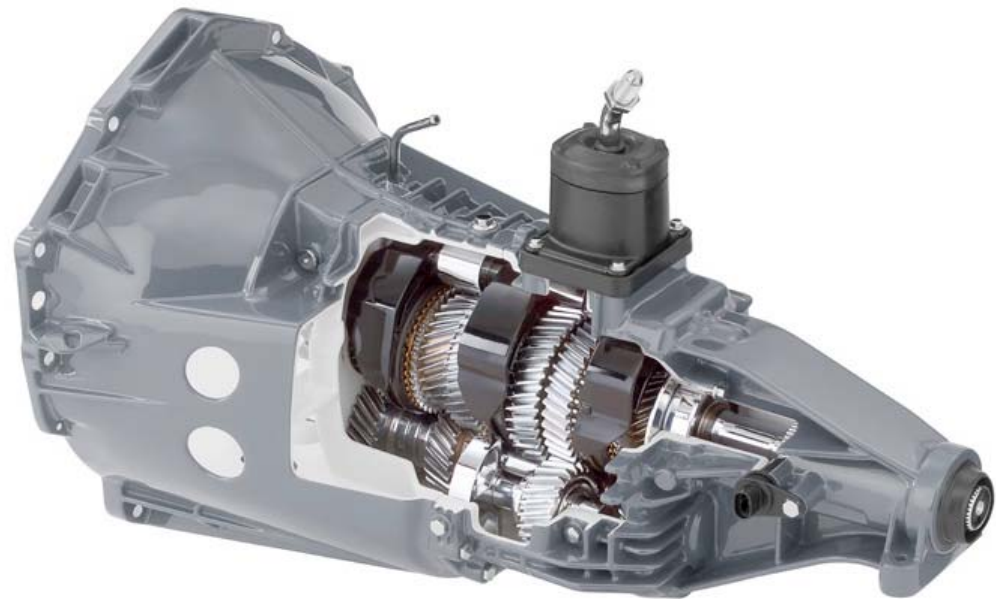
Slot machine



Worm gear box



Tractor Transmission



Five-speed Manual Transmission

Most commercial bearings are produced to *metric dimensions* → the fits are specified according to the tolerance system of the ISO.

Recommended tolerance grades for the *bearing seats on shafts* and *housing bore fits with the outer race* (for bearings carrying moderate to heavy loads):

Bearing Bore Diameter Range	Tolerance Grade
10-18 mm	j5
20-100 mm	k5
105-140 mm	m5
150-200 mm	m6
Housing bore (any)	H8

Question 1: What we can see from these notations?

- From the letters **j**, **k**, **m**, and **H** we can see that **the hole basis system is used for bearings**.
 - The bearing bore and bearing outer diameter are those expected from the bearing manufacturer.
 - You must control the shaft diameter and the housing bore to the specified minimum and maximum dimensions.
- The Bearing Bore / Shaft fit is mainly a *transition fit for accurate location* (compromise between clearance and interference, or more interference) – **j**, **k**, **m**.
- The Housing / External Race fit is a clearance fit (close running fit for accurate location at moderate speeds and moderate load) – **H**.

Question 2: How can bearings be fixed/located on the shaft or in the hole?

Simplest way – *press fitting*.

Instead of press fitting (especially in case of a running fit) you may and some times should use means of axial location: *retaining rings, collars, shoulders, spacers or locknuts*.

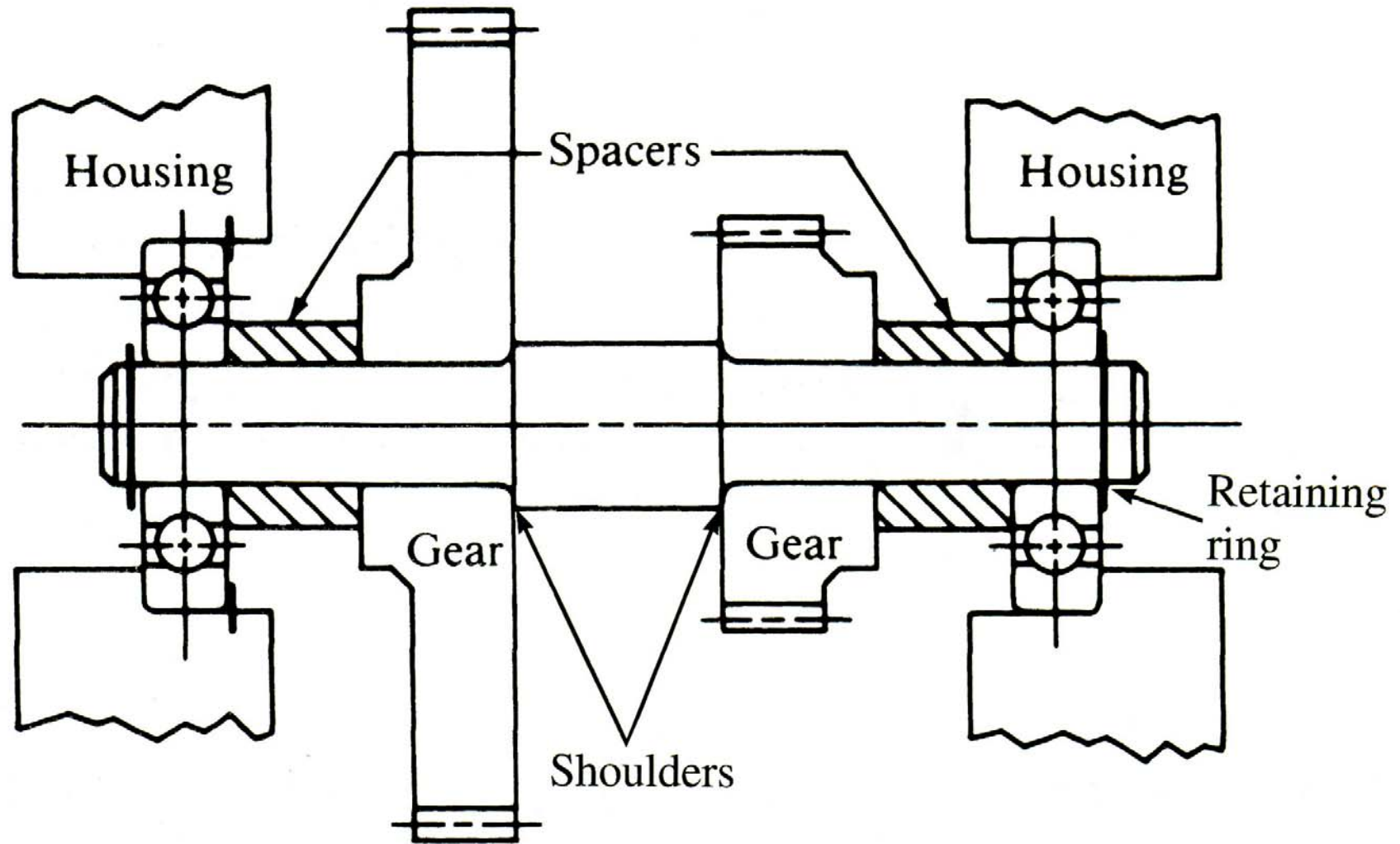
Retaining rings – placed on a shaft, in grooves in the shaft, or in a housing to prevent the axial movement of a machine element.



Types of retaining rings

 INTERNAL	BASIC N5000 For housings and bores Size Range .250—10.0 in. 6.4—254.0 mm.	 EXTERNAL	BOWED 5101* For shafts and pins Size Range .188—1.750 in. 4.8—44.4 mm.	 EXTERNAL	REINFORCED 5115 For shafts and pins Size Range .094—1.0 in. •	 EXTERNAL	HEAVY-DUTY 5160 For shafts and pins Size Range .394—2.0 in. 10.0—50.8 mm.
 INTERNAL	BOWED N5001* For housings and bores Size Range .250—1.750 in. 6.4—44.4 mm.	 EXTERNAL	BEVELED 5102 For shafts and pins Size Range 1.0—10.0 in. 25.4—254.0 mm.	 EXTERNAL	BOWED E-RING 5131 For shafts and pins Size Range .110—1.375 in. 2.8—34.9 mm.	 EXTERNAL	KLIPRING® 5304 T-5304 For shafts and pins Size Range .156—1.000 in. 4.0—25.4 mm.
 INTERNAL	BEVELED N5002 / N5003 For housings and bores Size Range 1.0—10.0 in. 25.4—254.0 mm 1.56—2.81 in. 39.7—71.4 mm	 EXTERNAL	CRESCENT® 5103 For shafts and pins Size Range .125—2.0 in. 3.2—50.8 mm.	 EXTERNAL	E-RING 5133 For shafts and pins Size Range .040—1.375 in. 1.0—34.9 mm.	 EXTERNAL	GRIPRING® 5555 For shafts and pins Size Range .079—.750 in. 2.0—19.0 mm.
 INTERNAL	CIRCULAR 5005 For housings and bores Size Range .312—2.0 in. •	 EXTERNAL	CIRCULAR 5105 For shafts and pins Size Range .094—1.0 in. •	 EXTERNAL	RADIAL GRIPRING® 5135 for shafts and pins Size Range .094—.375 in. 2.4—9.5 mm.	 EXTERNAL	HIGH-STRENGTH 5560* For shafts and pins Size Range .101—.328 in. •
 INTERNAL	INVERTED 5008 For housings and bores Size Range .750—4.0 in. 19.0—101.6 mm.	 EXTERNAL	INTERLOCKING 5107* For shafts and pins Size Range .469—3.375 in. 11.9—85.7 mm.	 EXTERNAL	PRONG-LOCK® 5139* For shafts and pins Size Range .092—.438 in. •	 EXTERNAL	PERMANENT SHOULDER 5590* For shafts and pins Size Range .250 .750 6.4 19.0 mm.
 EXTERNAL	BASIC 5100 For shafts and pins Size Range .125—10.0 in. 3.2—254.0 mm.	 EXTERNAL	INVERTED 5108 For shafts and pins Size Range .500—4.0 in. 12.7—101.6 mm.	 EXTERNAL	REINFORCED E-RING 5144 For shafts and pins Size Range .094—.562 in. 2.4—14.3 mm.	*Non-Stocking Ring Type: Available on special order only	

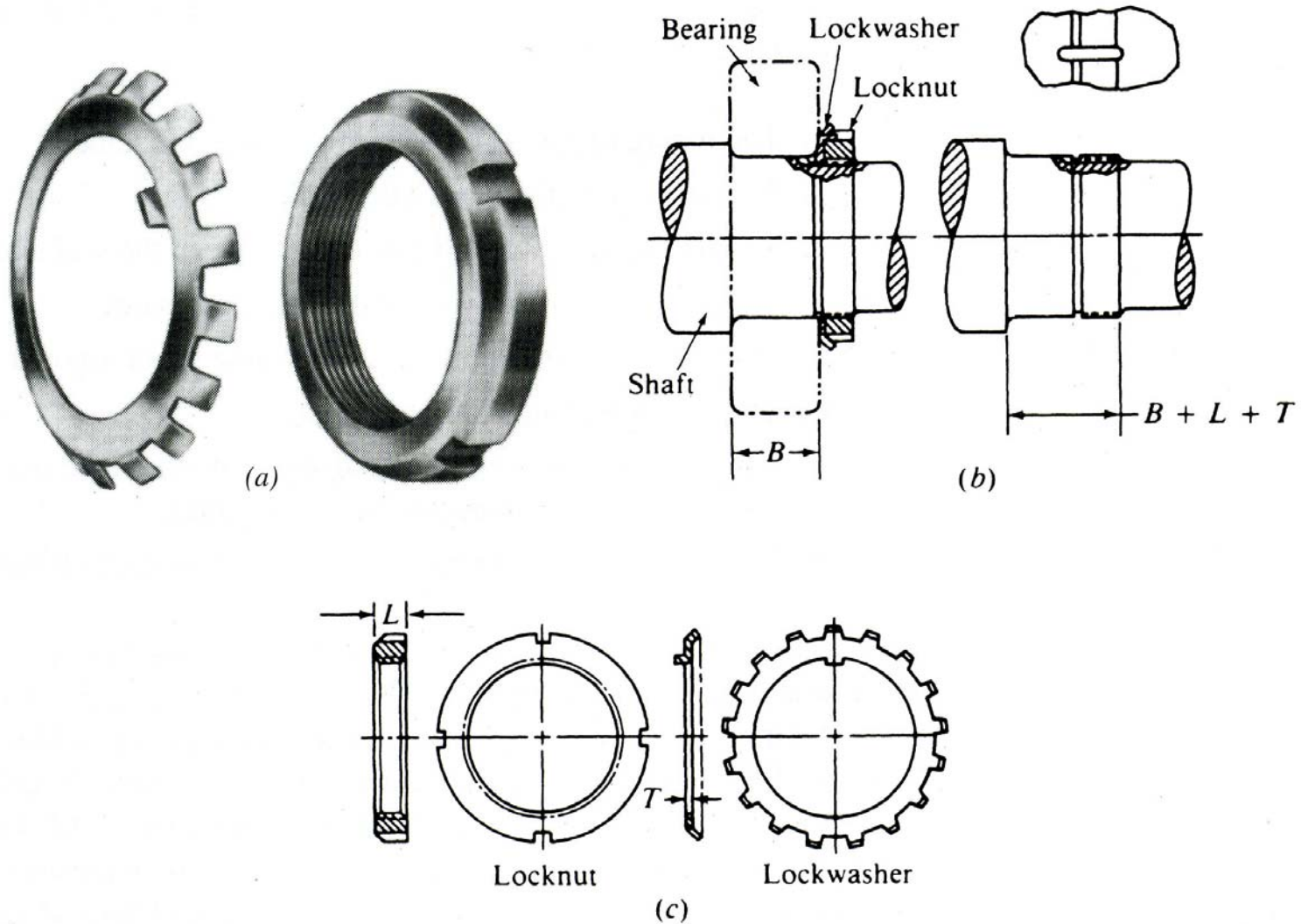
Shoulder – a vertical surface produced when a diameter change occurs on a shaft.



Spacer – a ring slid over the shaft against the machine element that is to be located, i.e. it is positioned between two elements and thus controls only the relative position between them.

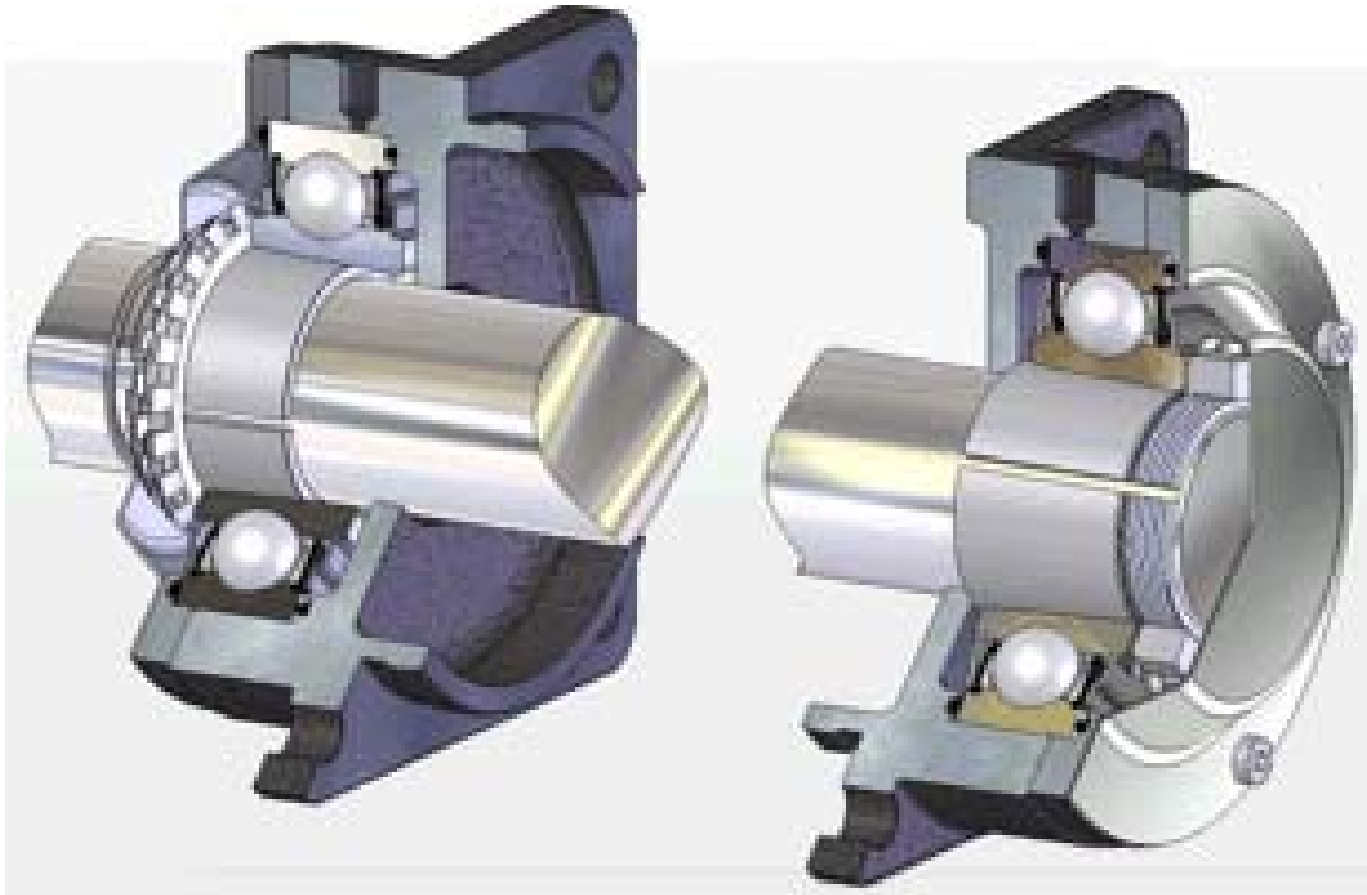
Locknut – can be used when an element is located at the end of the shaft. Needs the thread on the shaft.

Lockwasher can be added to fix the locknut.



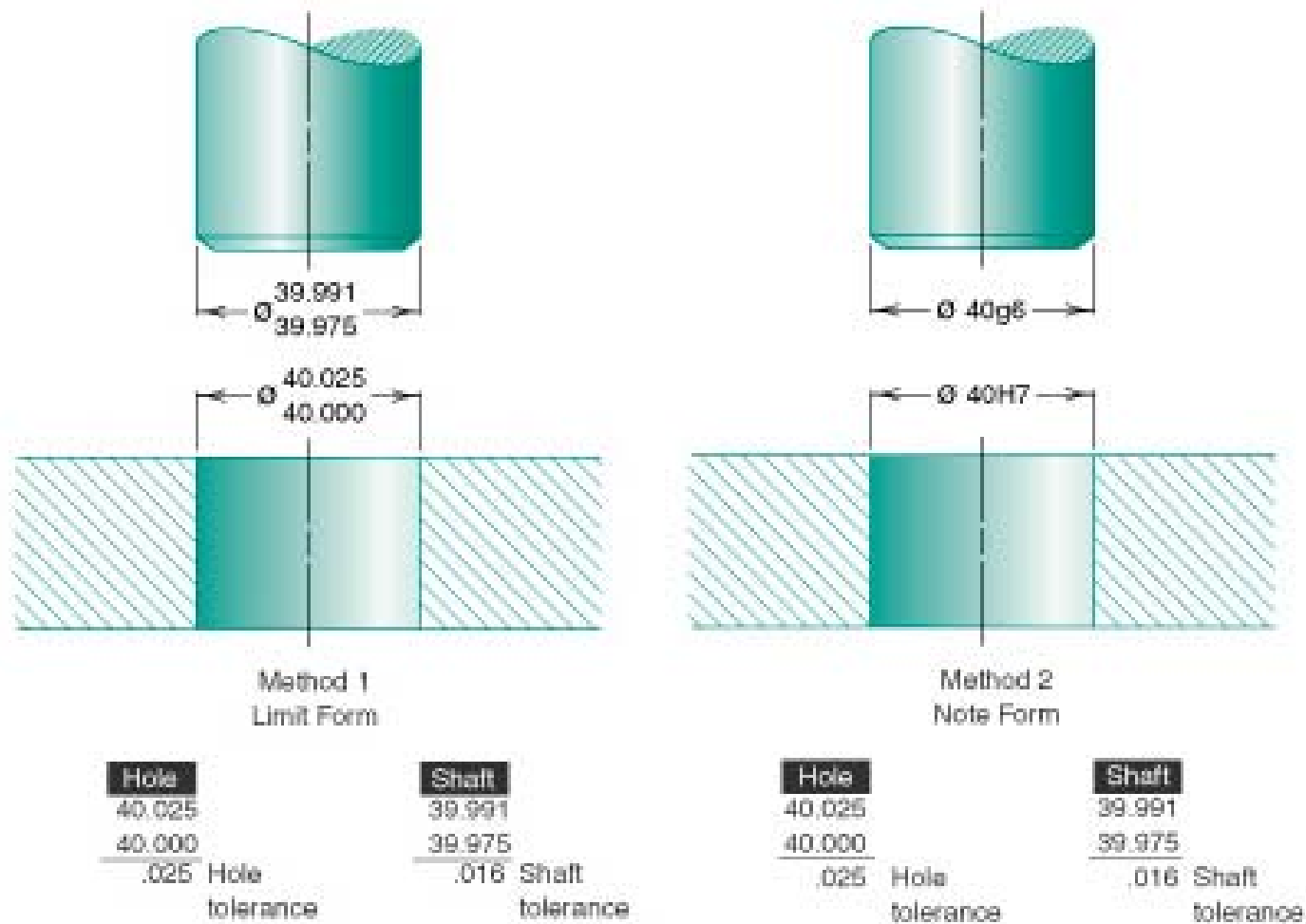
Collar – a ring similar to a spacer slid over the shaft, but positioned adjacent to a machine element for the purpose of axial location. It is held in position, typically, by set screws, and axial location can be set virtually anywhere along the shaft.

Adapter Sleeve – similar to collar, but with tapered outer surface & slit, and is placed between the bearing and the shaft. Locknuts are used to clamp the collar.



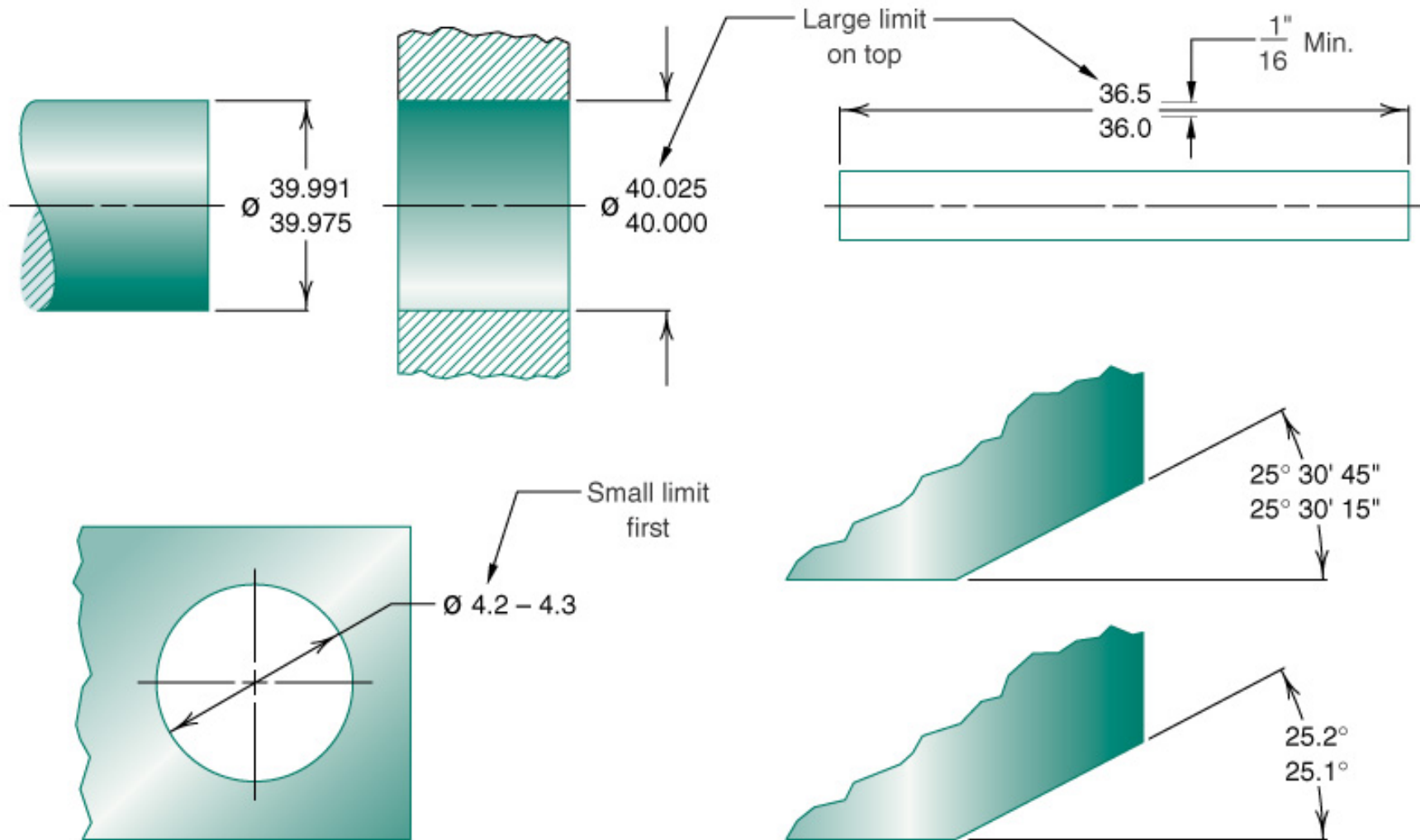
To summarize:

Two methods are acceptable for *representing metric tolerances* on a drawing, the **limit form** and the **note form**:



The limit form gives the actual tolerance values, and the note form uses the basic size and letters that refer to standard tables to determine the size. From the table we can find that this tolerance notation relates to the sliding fit in the hole basis system.

Metric tolerances also have their standards of applying to a technical drawing:

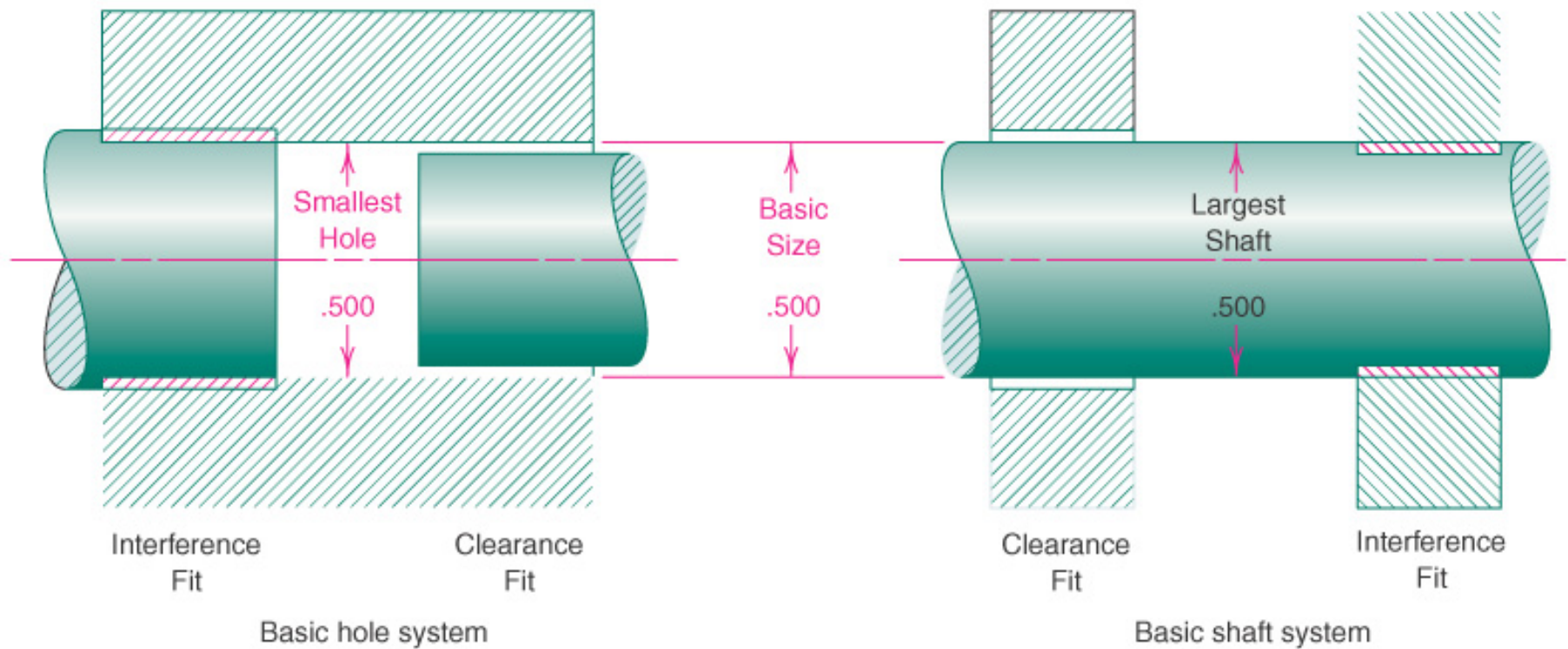


English unit fits

Similar to the metric system, a special group of English unit tolerance relationships, called **preferred precision fits**, have been developed. ANSI Standard B4.1 specifies a series of standard fits between cylindrical parts, based on the basic hole system. The different fit classes are as follows:

- *Running and sliding fit (RC)* – the loosest of the fit classes, when a shaft must move freely inside a bearing or hole, and the positioning of the shaft is not critical.
- *Clearance location fit (LC)* – tighter than the RC class fits, but the shaft and hole *may* be the same size, which is called a **line-to-line fit**. Shaft is located more accurately, but it may still be loose.
- *Transition location fits (LT)* – a *transition* between LC and LN fits.
- *Interference location fits (LN)* – here the shaft can be *line to line* with the hole, but it is almost always larger than the hole. Is used where a part must be positively located relative to another part.
- *Force and shrink fit (FN)* – pure *interference* fit, where the shaft is *always* considered larger than the hole. Is used to transmit torque; to secure a bearing or pulley to a shaft, even if there is a twisting force; or to anchor parts that may slide along a shaft.

Also, similar to the metric system, there are basic hole and basic shaft systems for applying English unit tolerances to parts. It depends on whether the basic size refers to the size of the largest shaft or the smallest hole:



When a shaft and a hole are the same size, it is referred to as a **line-to-line fit**.