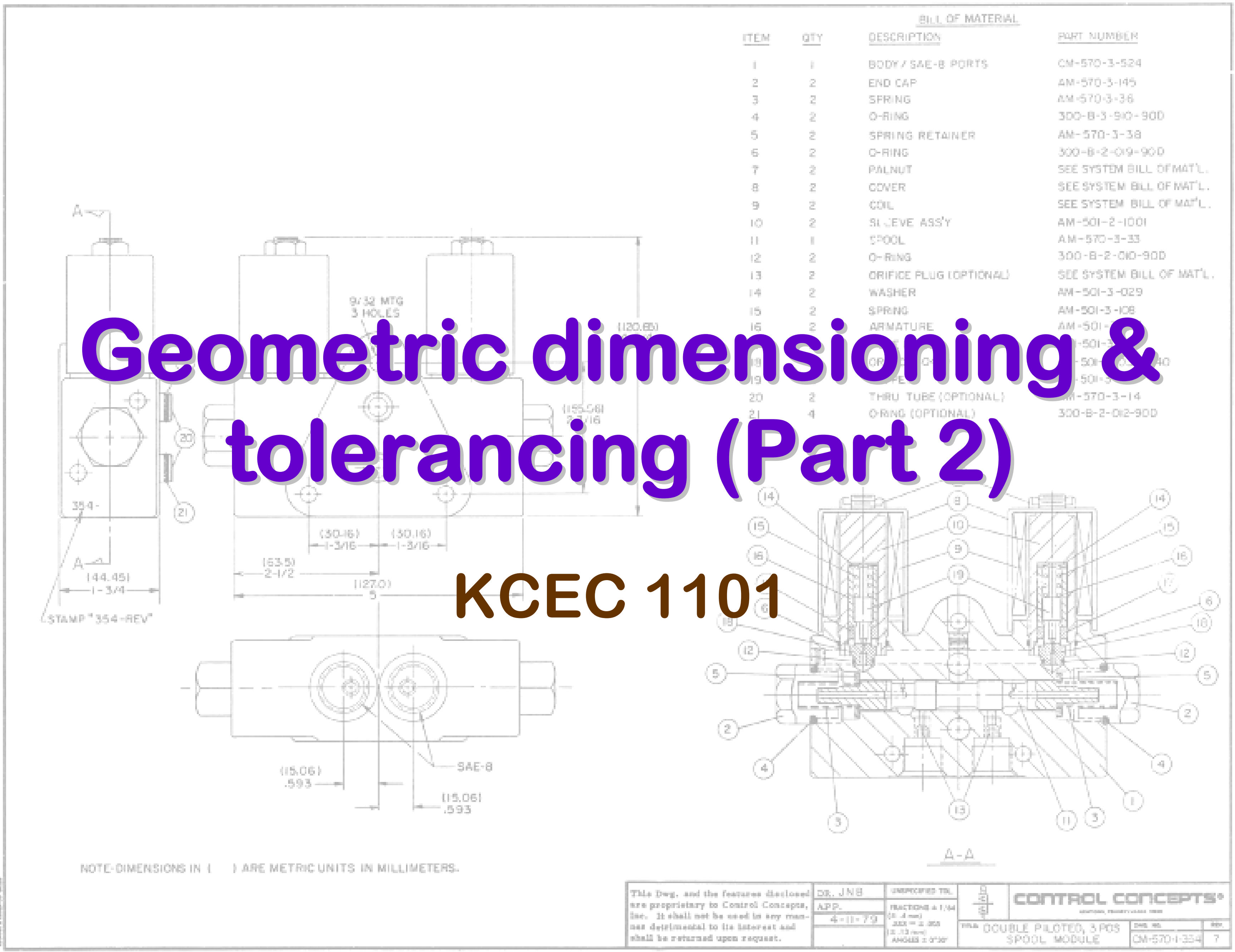
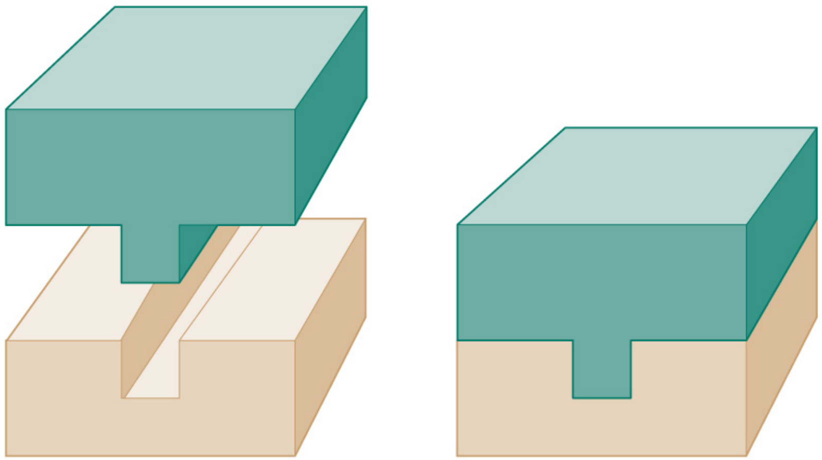
Geometric dimensioning & tolerancing (Part 2)



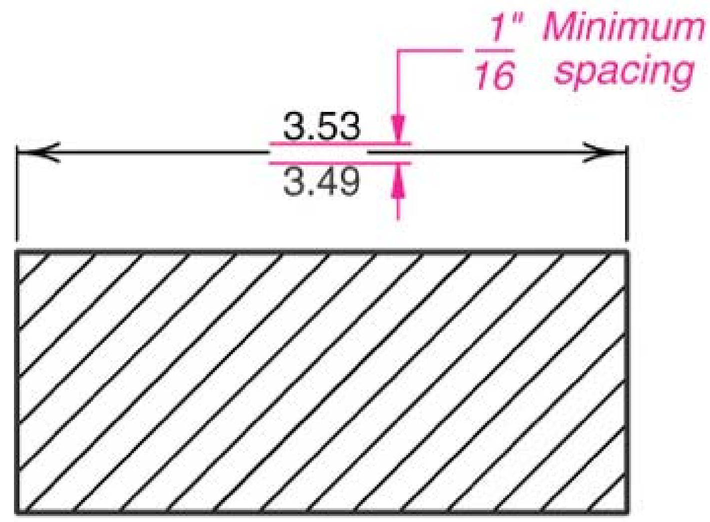
KCEC 1101

Tolerances

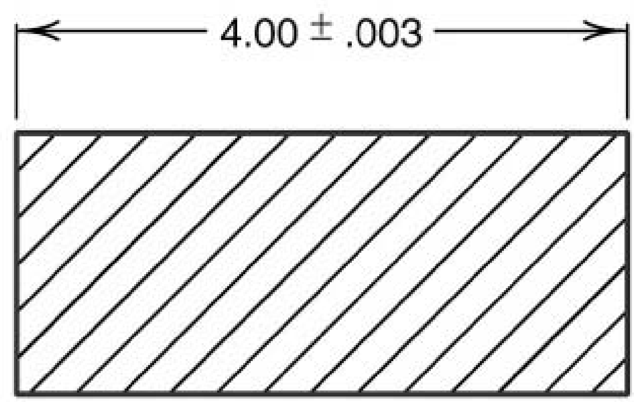
- **Geometric tolerances** are controls that effect the shape and/or orientation of features.
- **Tolerance** is the total amount a dimension may vary and is the difference between the maximum and minimum limits.
- Tolerances are assigned to mating parts. For example, the slot in the part must accommodate another part. A system is two or more mating parts. One of the great advantages of using tolerances is that it allows for **interchangeable parts**, thus permitting the replacement of individual parts.



- **For example, A tolerance of 4.650 ± 0.003 means that upper limit (largest value) for part is 4.653, the lower limit (smallest value) is 4.647, and the tolerance is 0.006**



(A) Direct limits

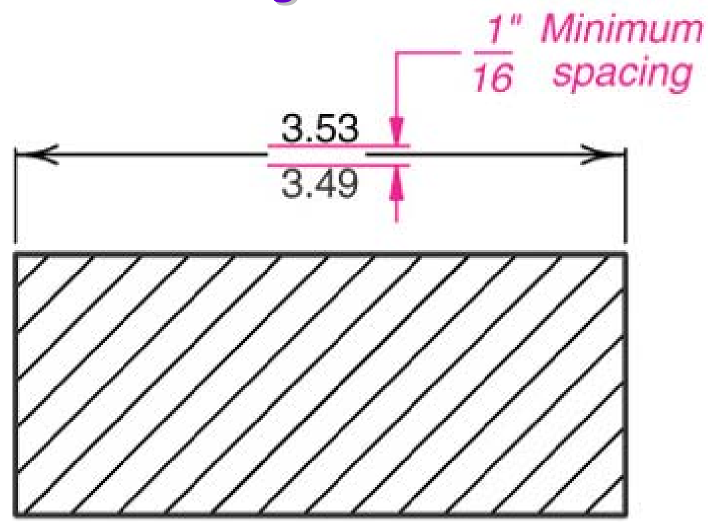


(B) Tolerance values

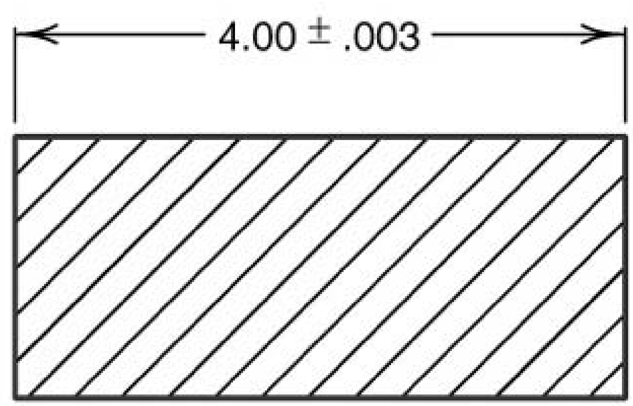
Tolerances are expressed as:

1. Direct limits, or as tolerance values applied directly to a dimension
2. Geometric tolerance
3. Notes referring to specific conditions
4. A general tolerance note in the title block

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

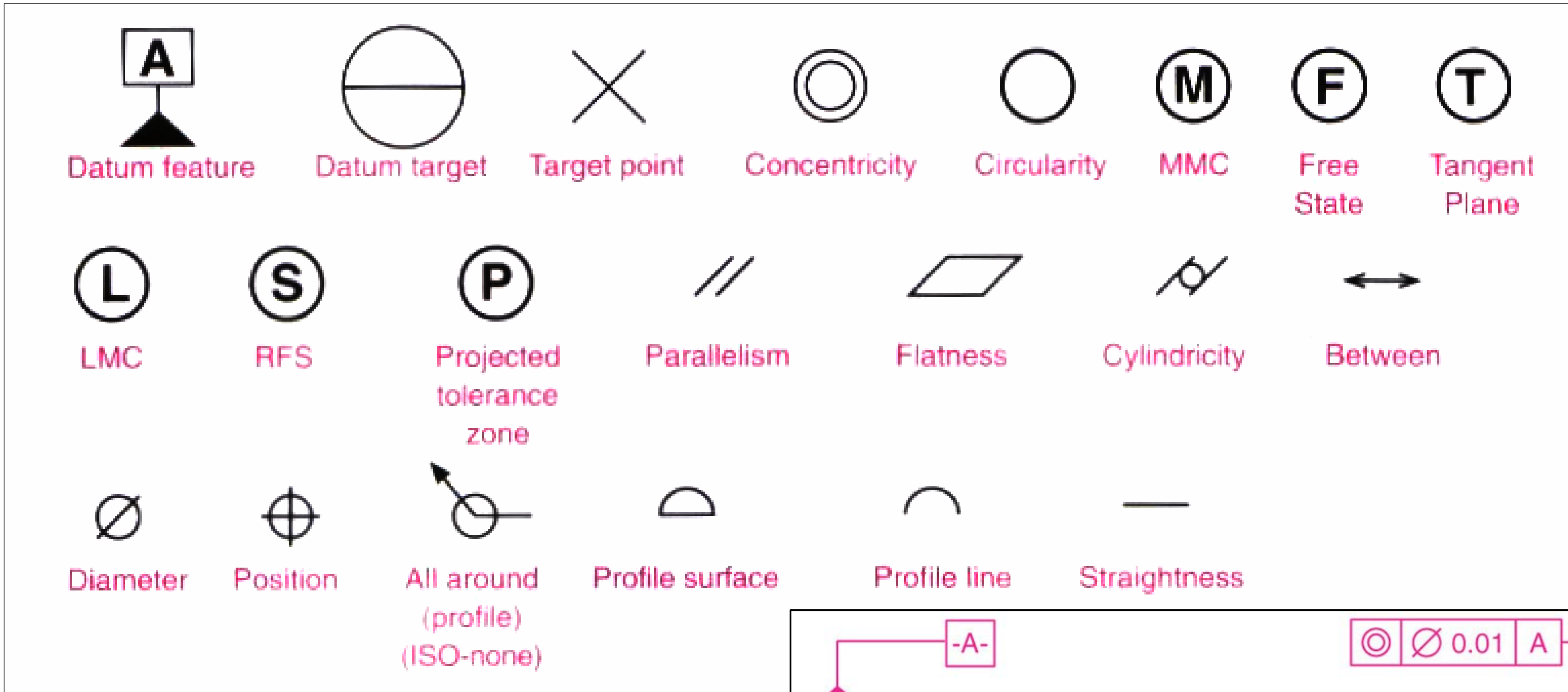


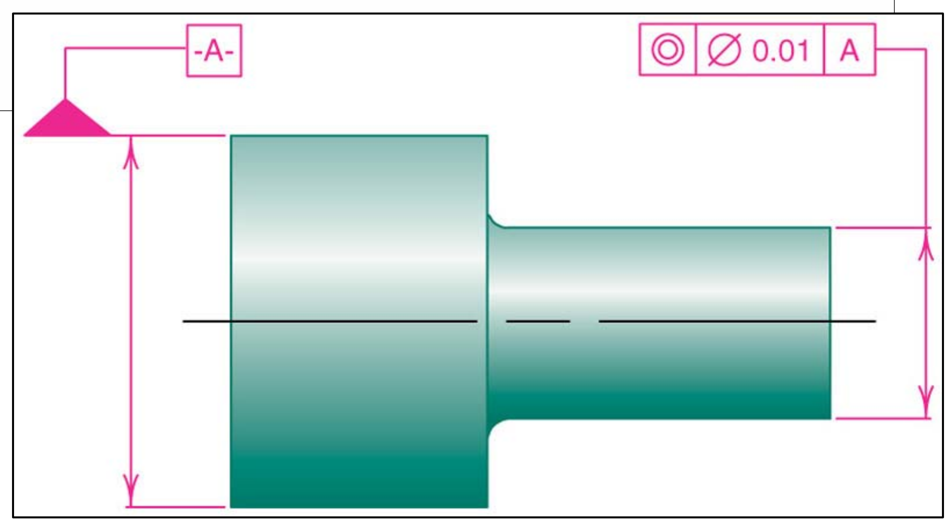
(A) Direct limits



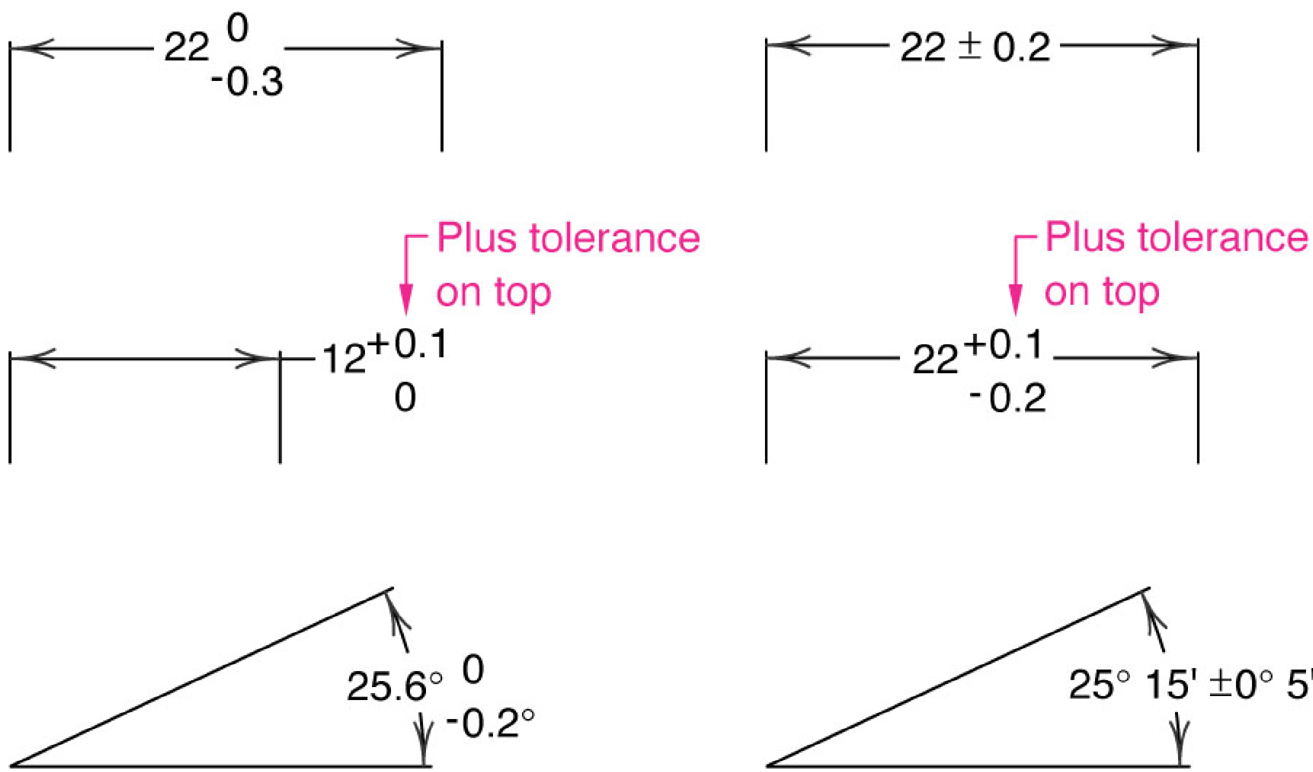
(B) Tolerance values

2. Geometric tolerances & dimensioning.





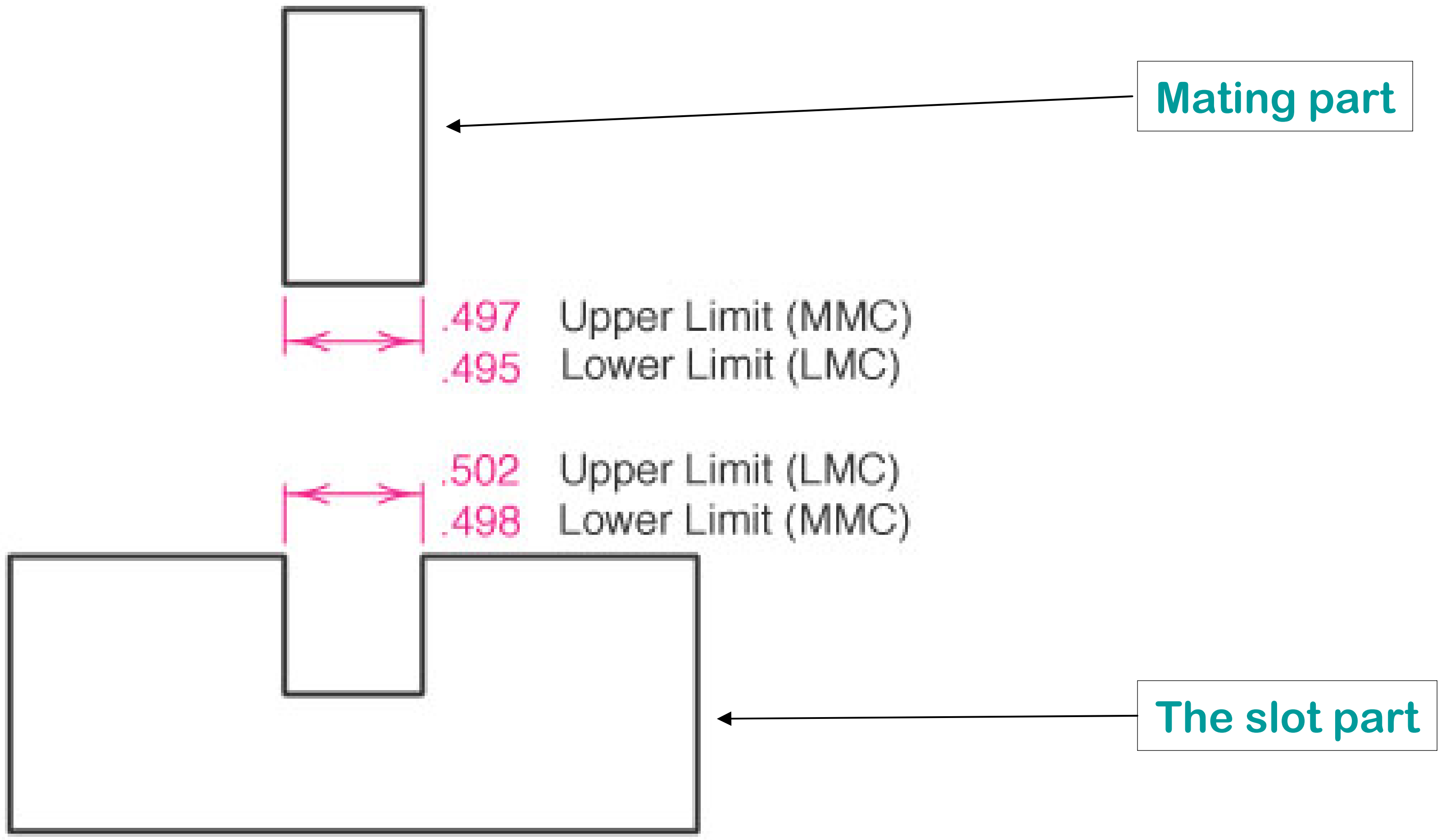
- Tolerance can be **unilateral** or **bilateral.**
- A unilateral tolerance varies in only one direction, while a bilateral varies in both directions.
- If the variation is equal in both directions, then the variation is preceded by a $\pm$ symbol. The plus and minus approach can only be used when the two variations are equal.



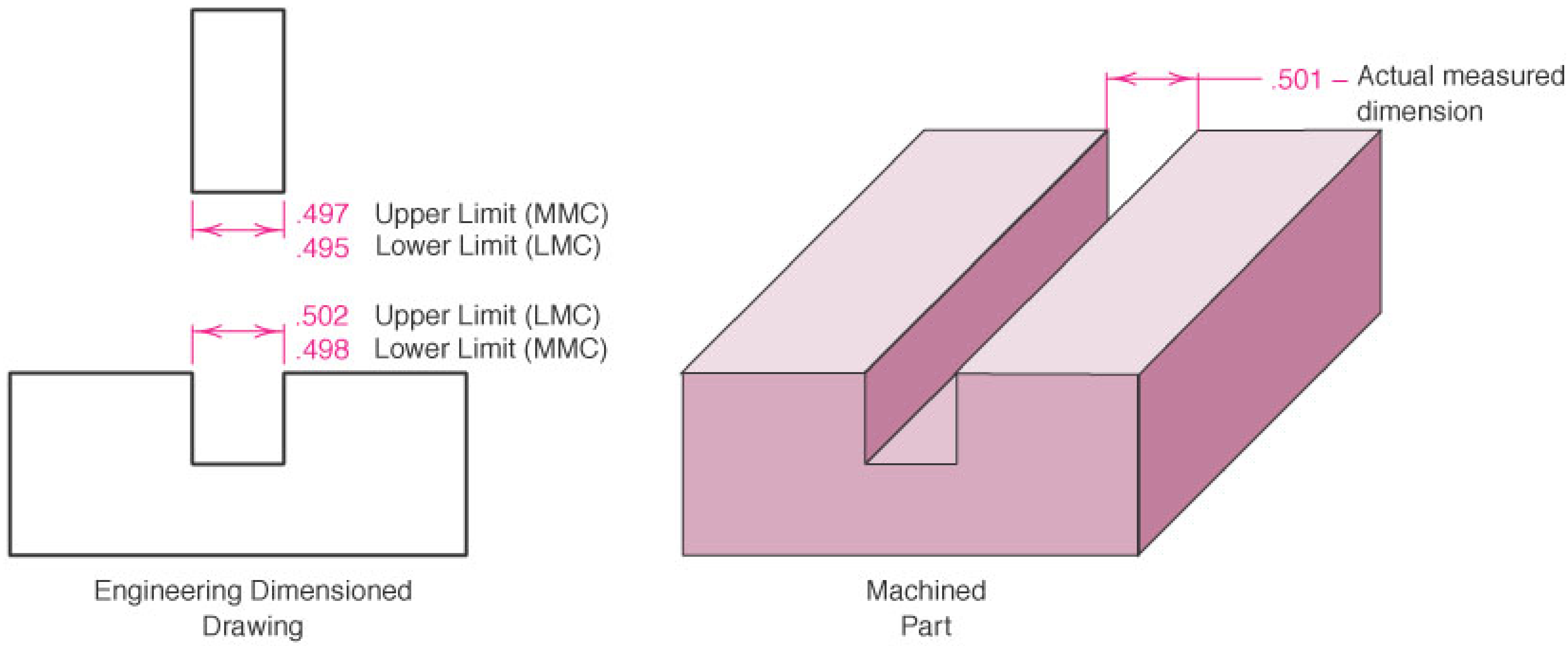
(A) Unilateral tolerancing

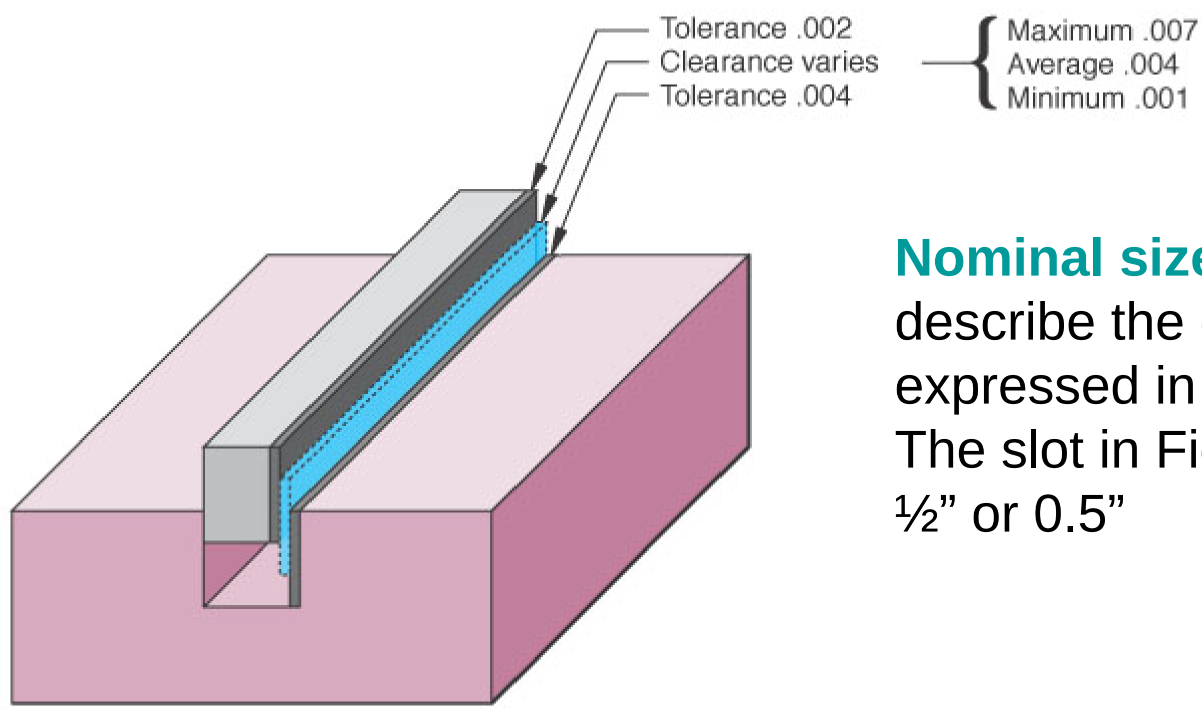
(B) Bilateral tolerancing

Important terms in ASME Y14.5M-1994
(A system of two part with tolerance dimension)

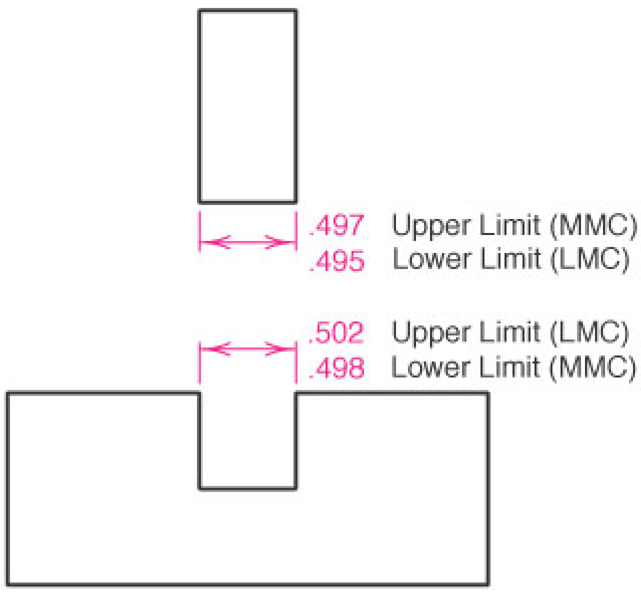


Engineering Dimensioned Drawing

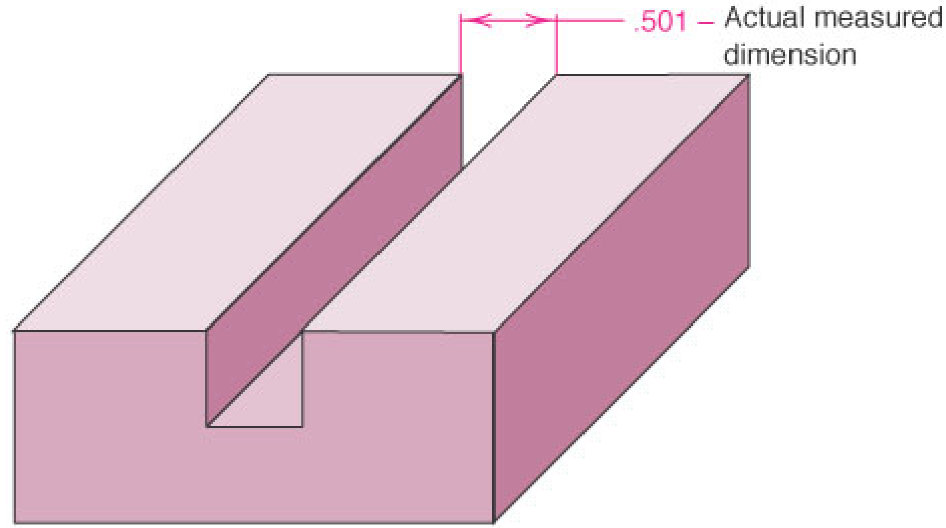




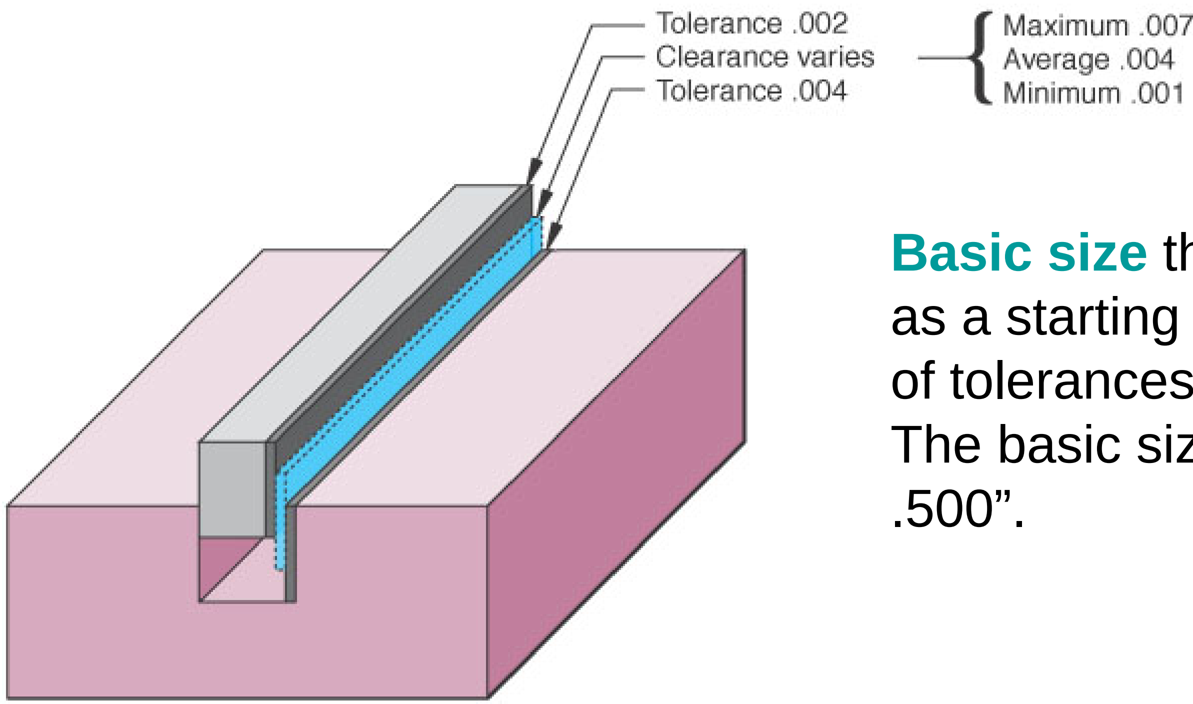
Nominal size - a dimension used to describe the general size usually expressed in common fractions. The slot in Fig has a nominal size ½” or 0.5”



Engineering Dimensioned Drawing

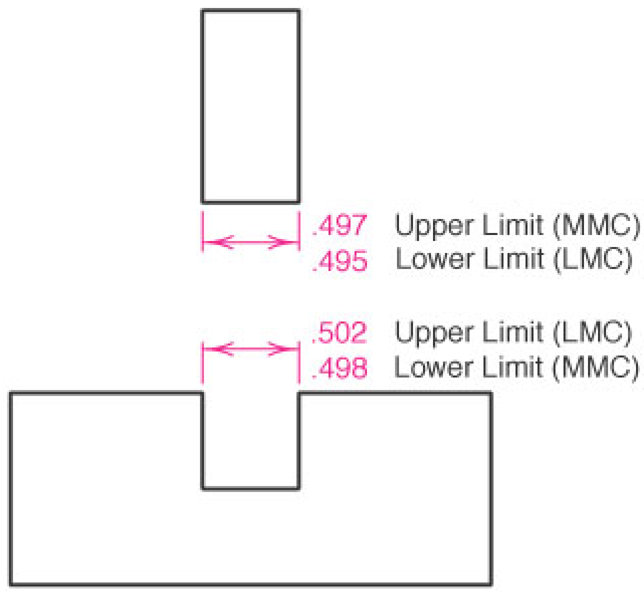


Machined Part

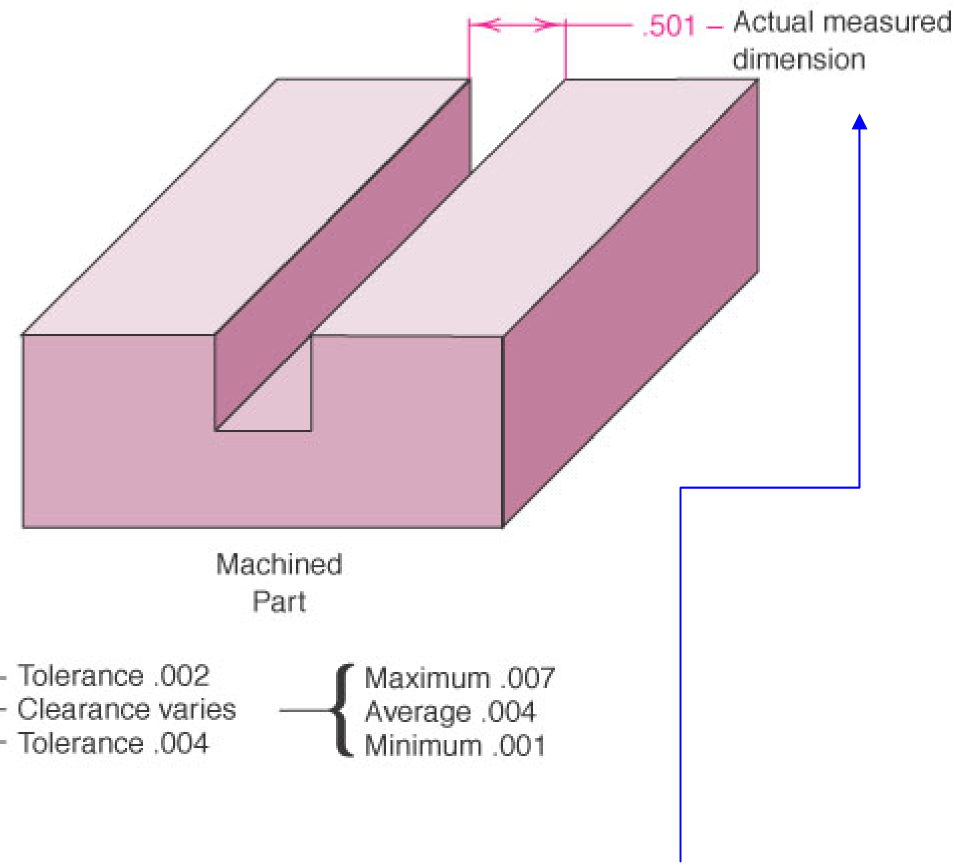


Basic size the theoretical size used as a starting point for the application of tolerances.

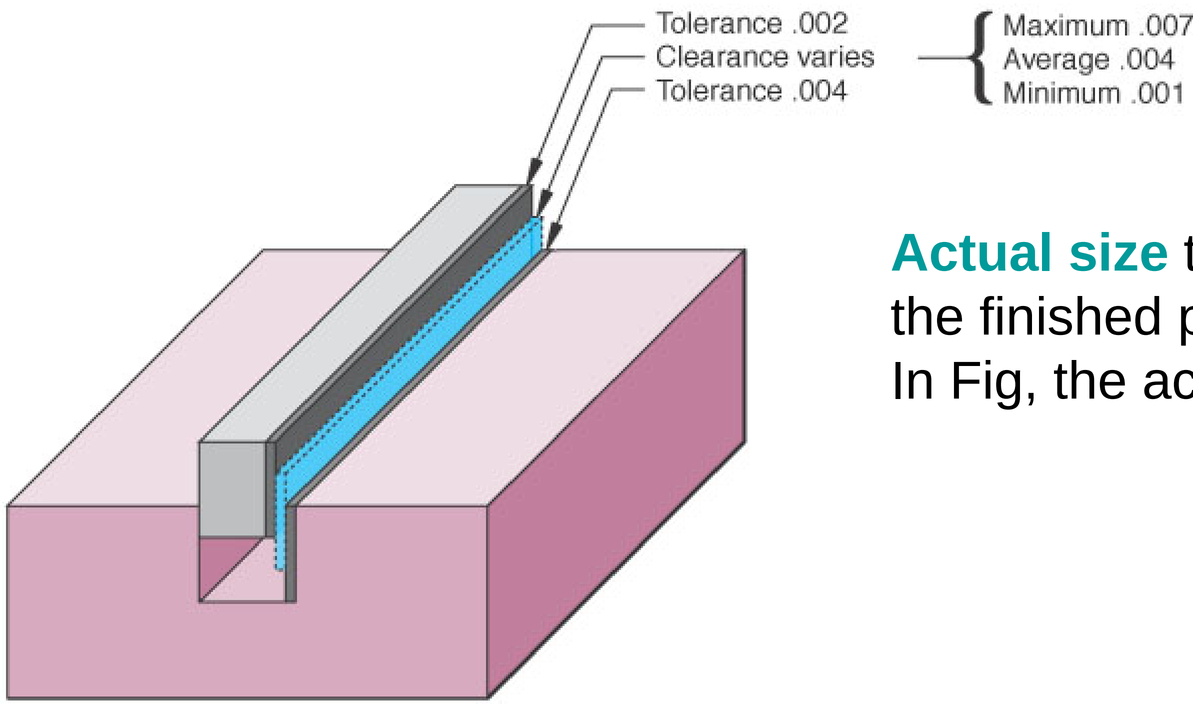
The basic size of the slot in Fig is .500”.



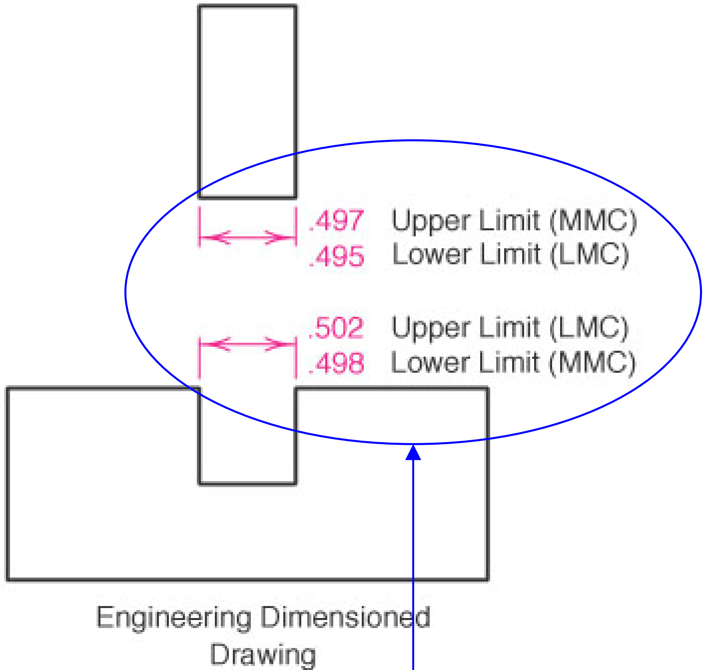
Engineering Dimensioned Drawing

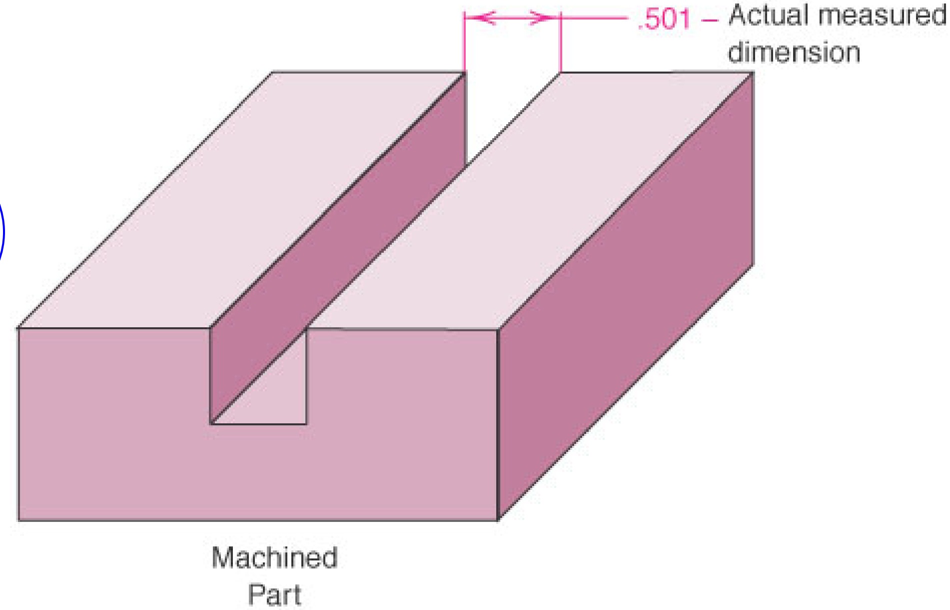


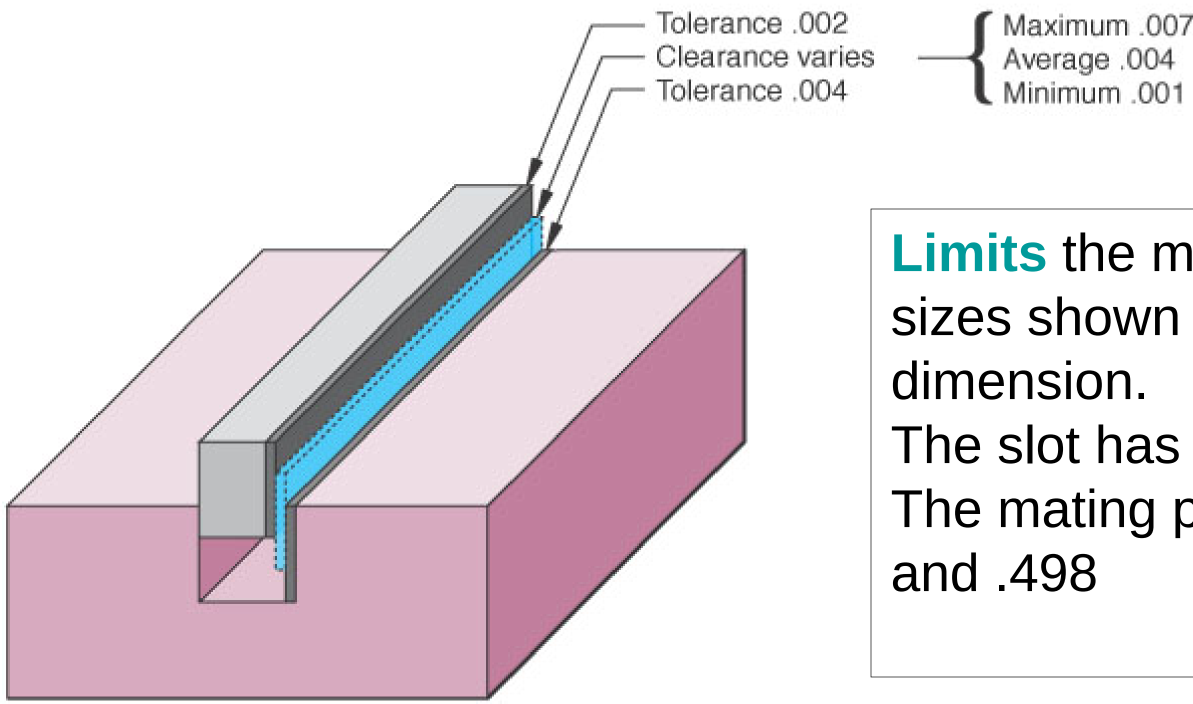
Machined Part



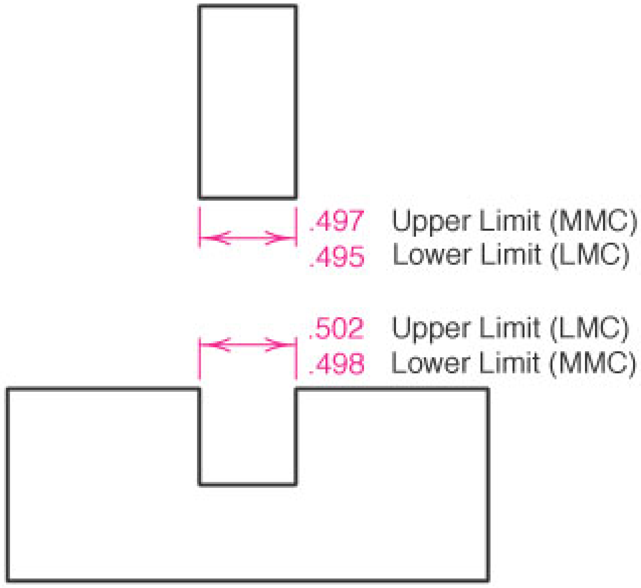
Actual size the measured size of the finished part after machining. In Fig, the actual size is .501



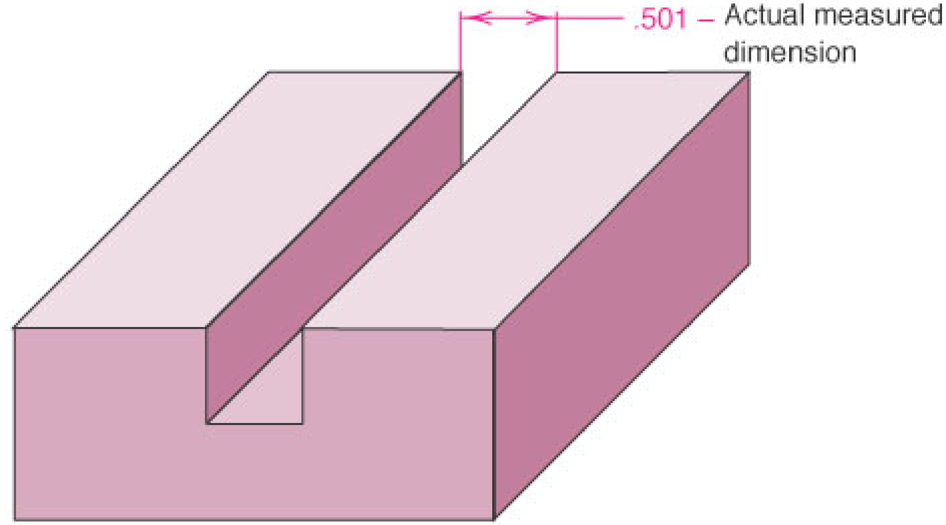




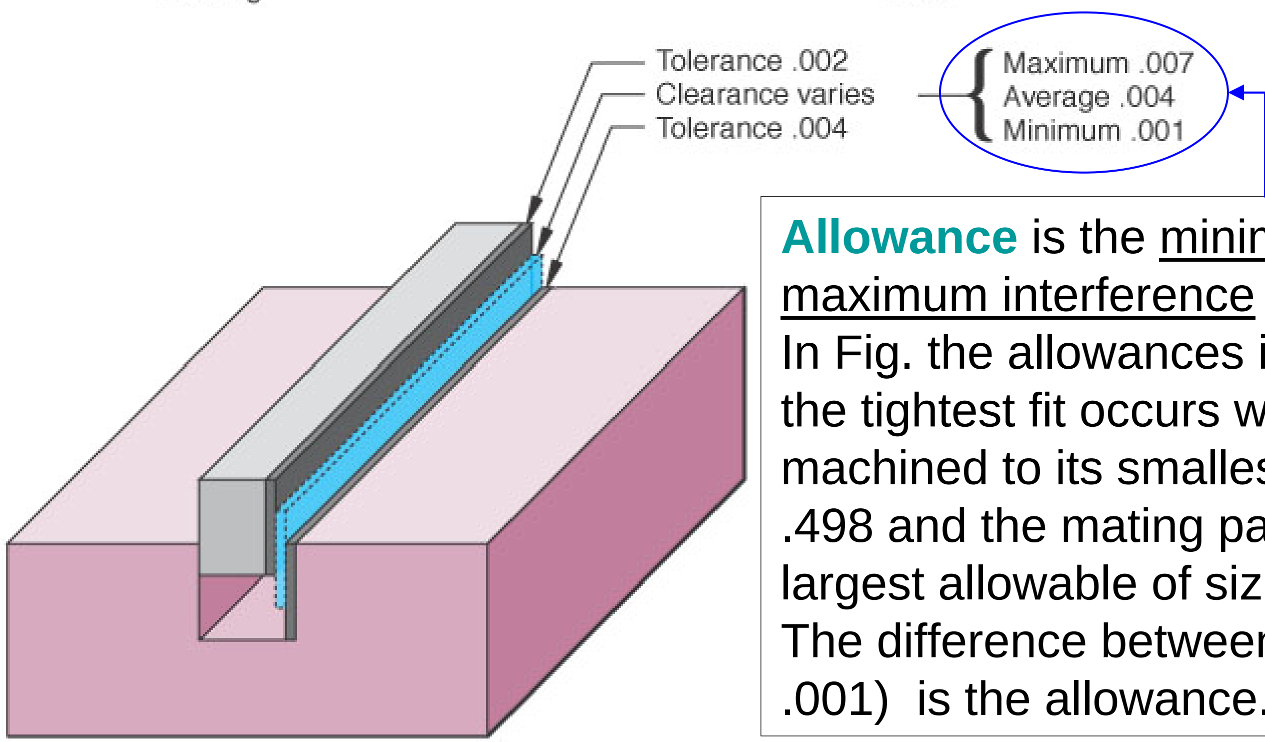
Limits the maximum and minimum sizes shown by the toleranced dimension.
The slot has limits of .502 and .498
The mating part has limits of .495 and .498



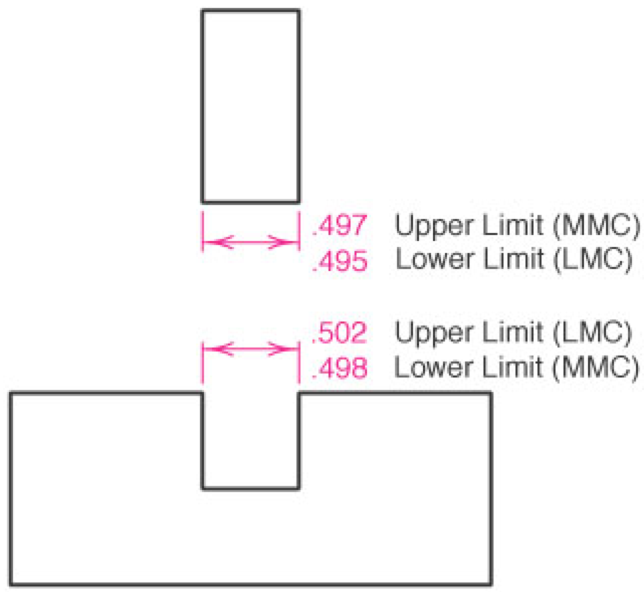
Engineering Dimensioned Drawing



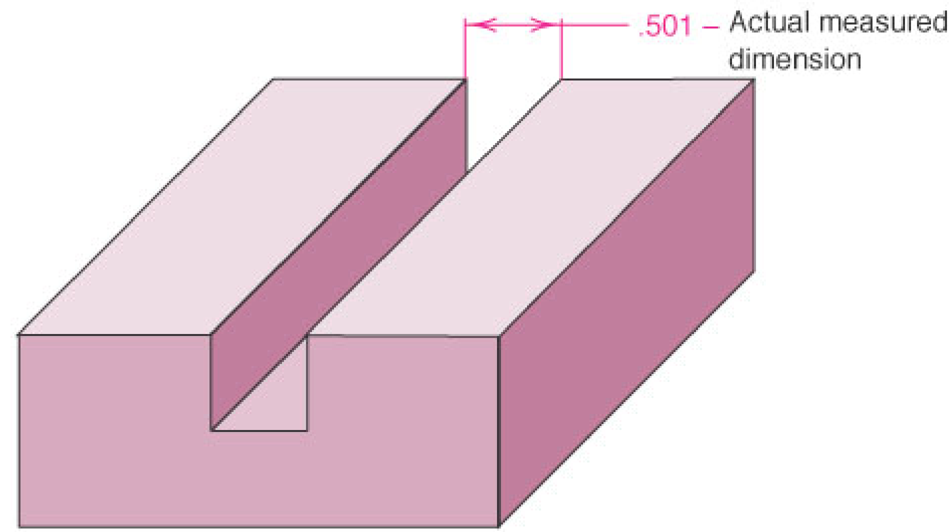
Machined Part



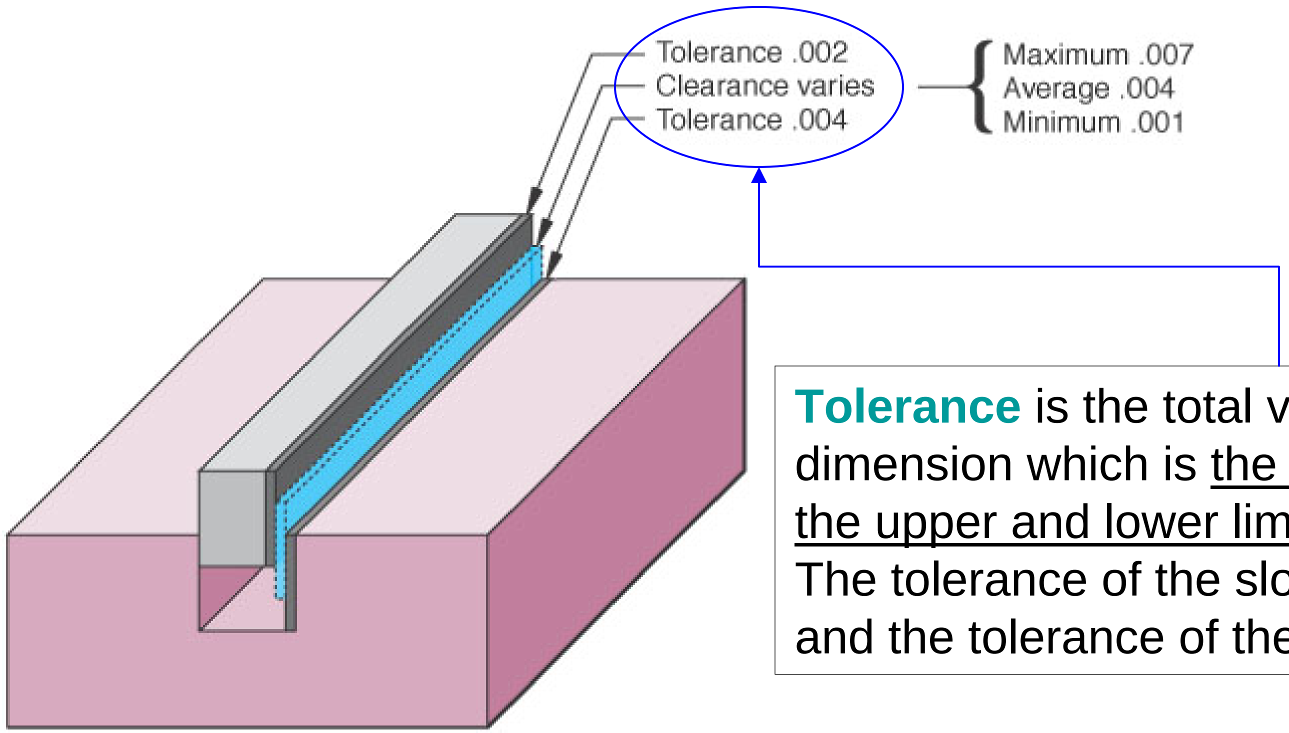
Allowance is the minimum clearance or maximum interference between parts.
In Fig. the allowances is .001, meaning that the tightest fit occurs when the slot is machined to its smallest allowable size of .498 and the mating part is machined to its largest allowable of size of .497.
The difference between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



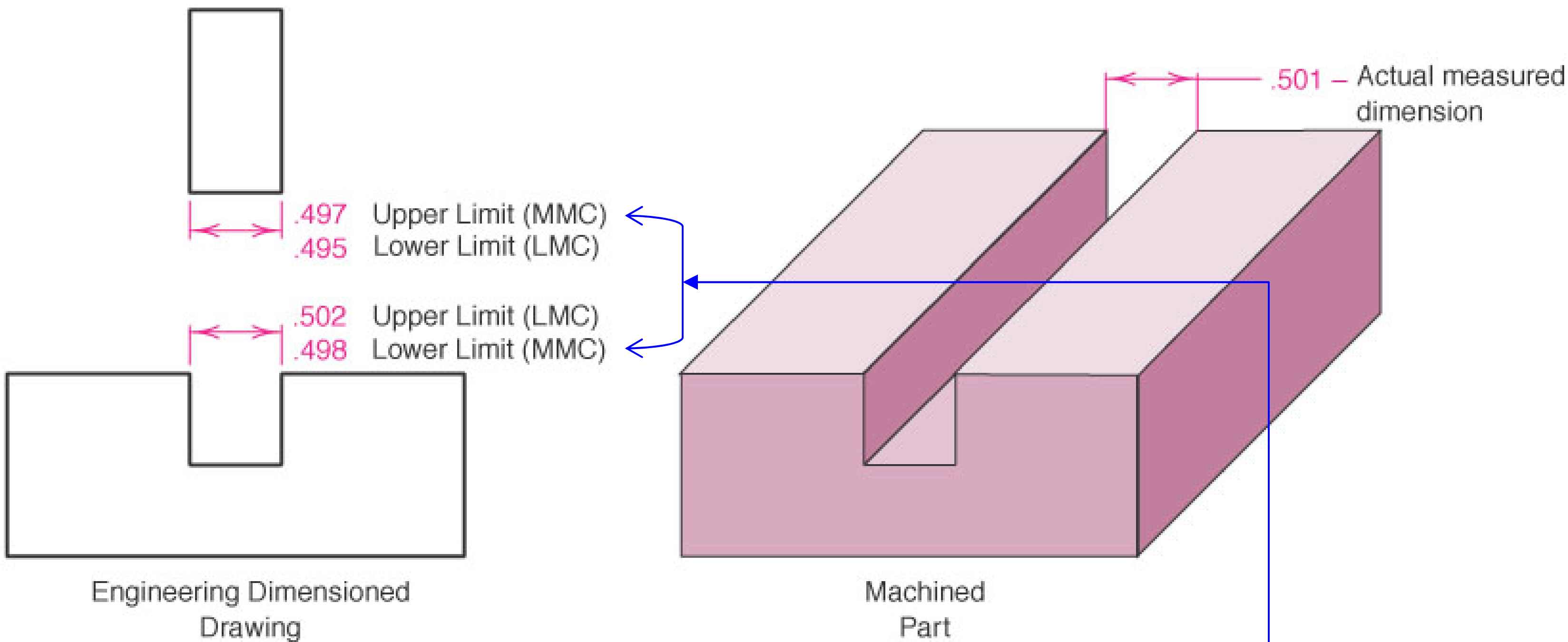
Engineering Dimensioned Drawing

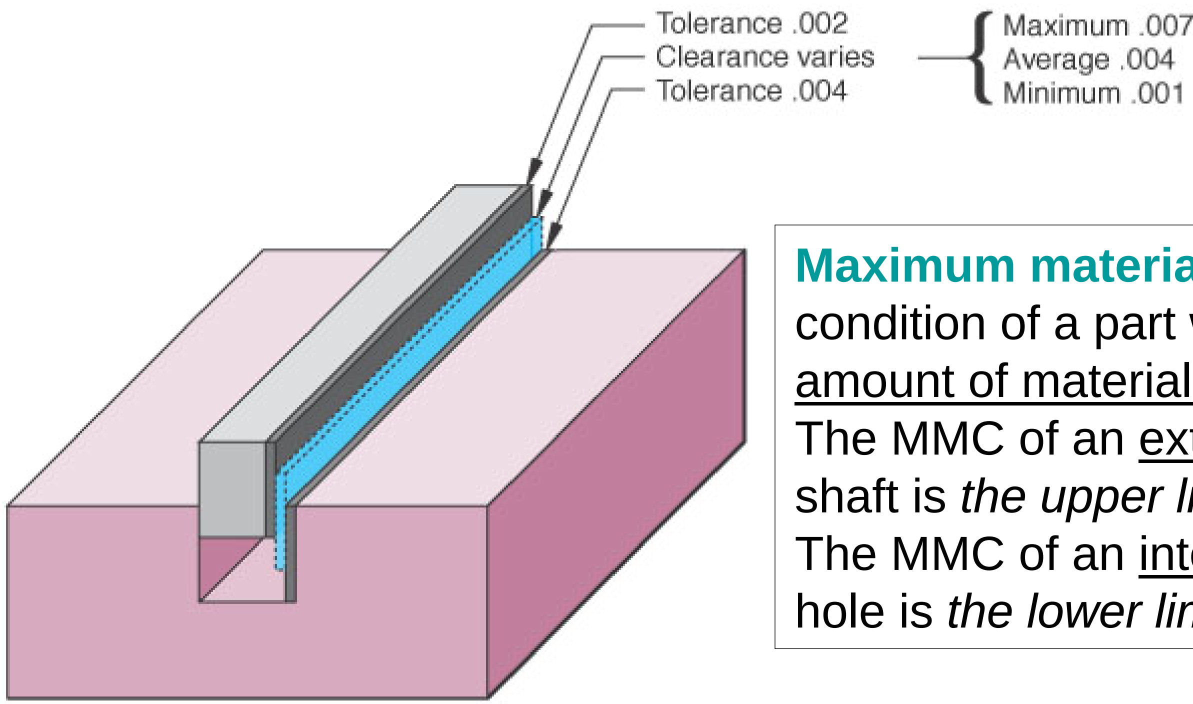


Machined Part



Tolerance is the total variance in a dimension which is <u>the difference between the upper and lower limits.</u>
The tolerance of the slot part in Fig. is .004" and the tolerance of the mating part is .002".

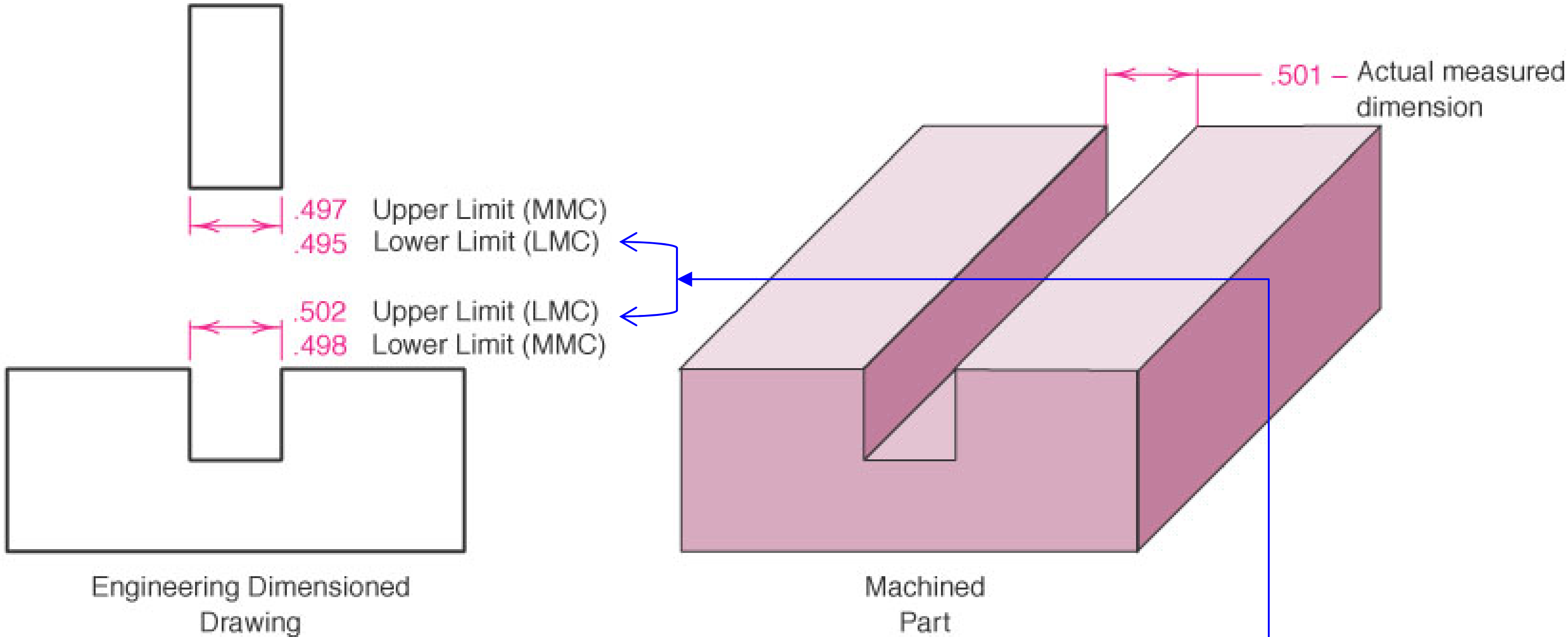


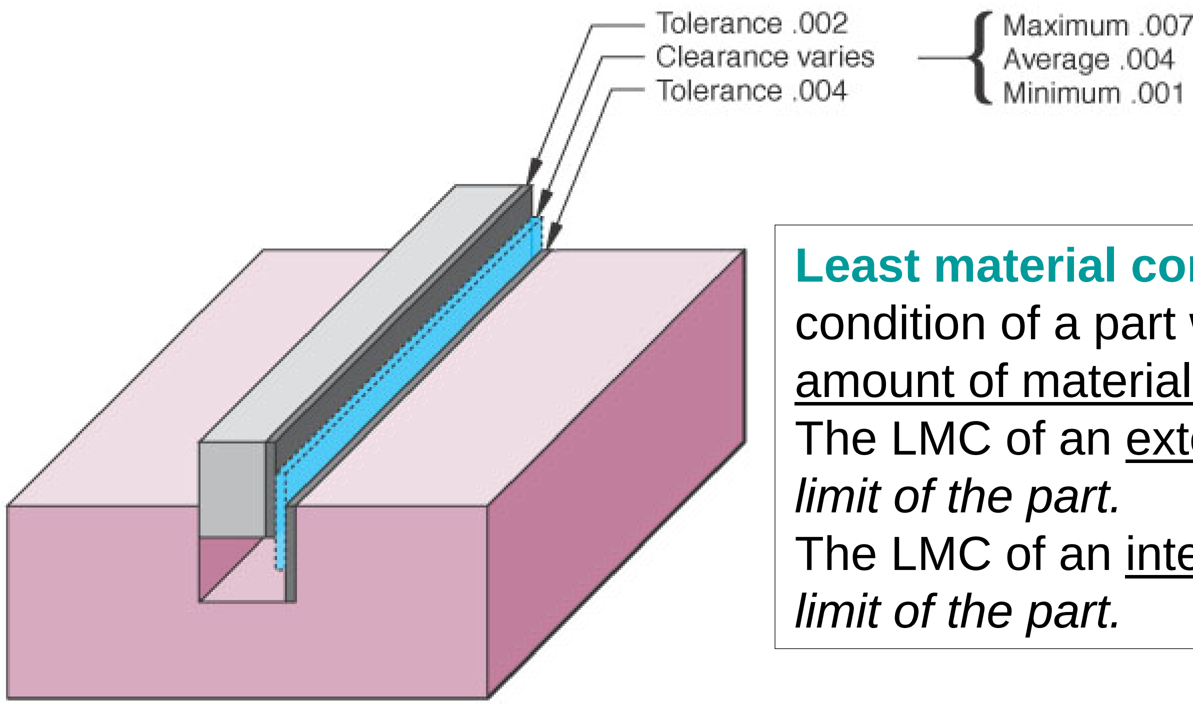


Maximum material condition (MMC) is the condition of a part when <u>it contains the most amount of material.</u>

The MMC of an <u>external feature</u> such as a shaft is *the upper limit.*

The MMC of an <u>internal feature</u> such as a hole is *the lower limit.*

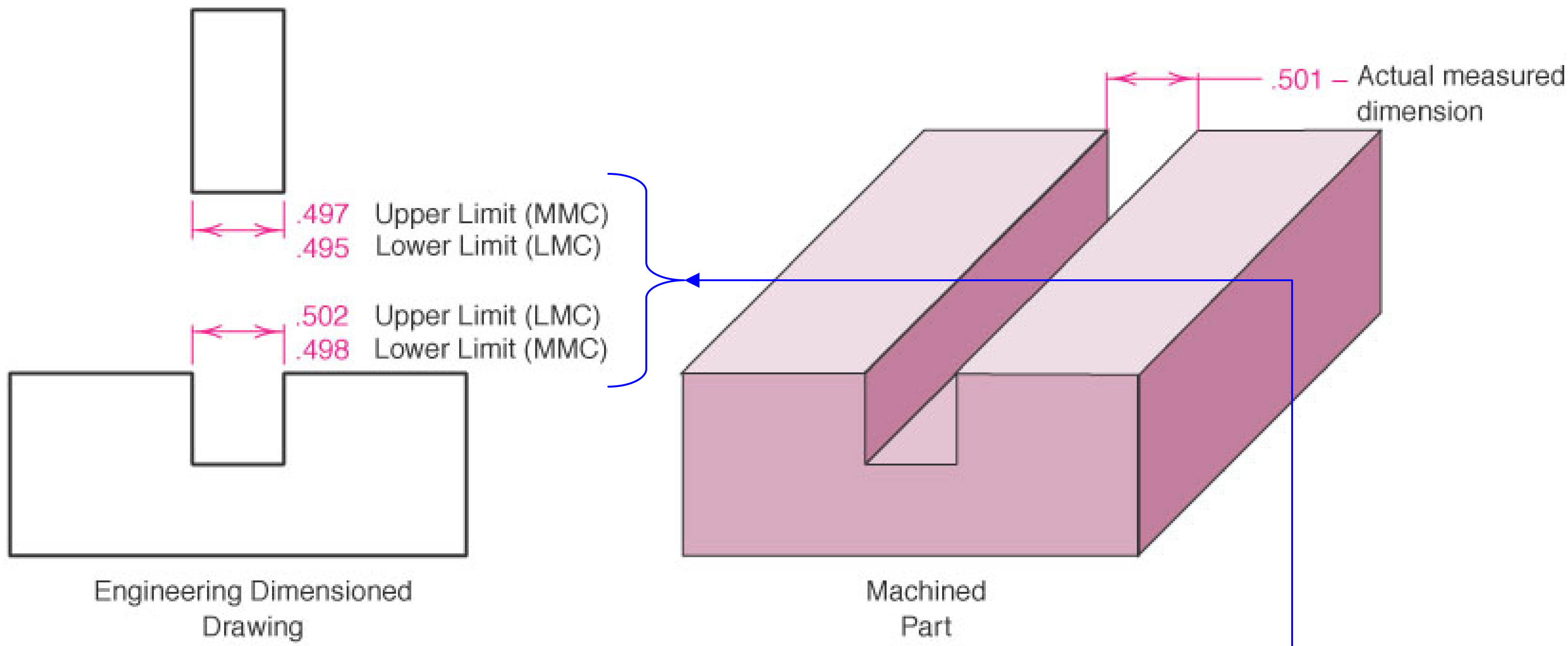


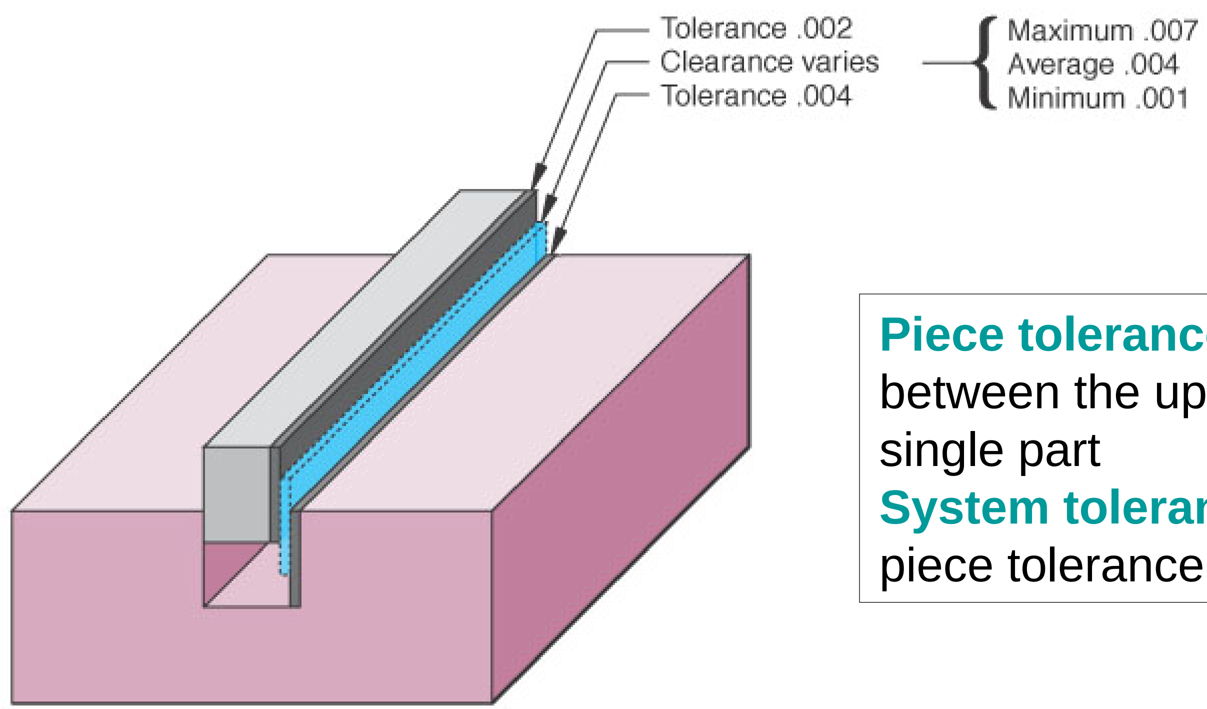


Least material condition (LMC) is the condition of a part when it contains the least amount of material possible.

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.*





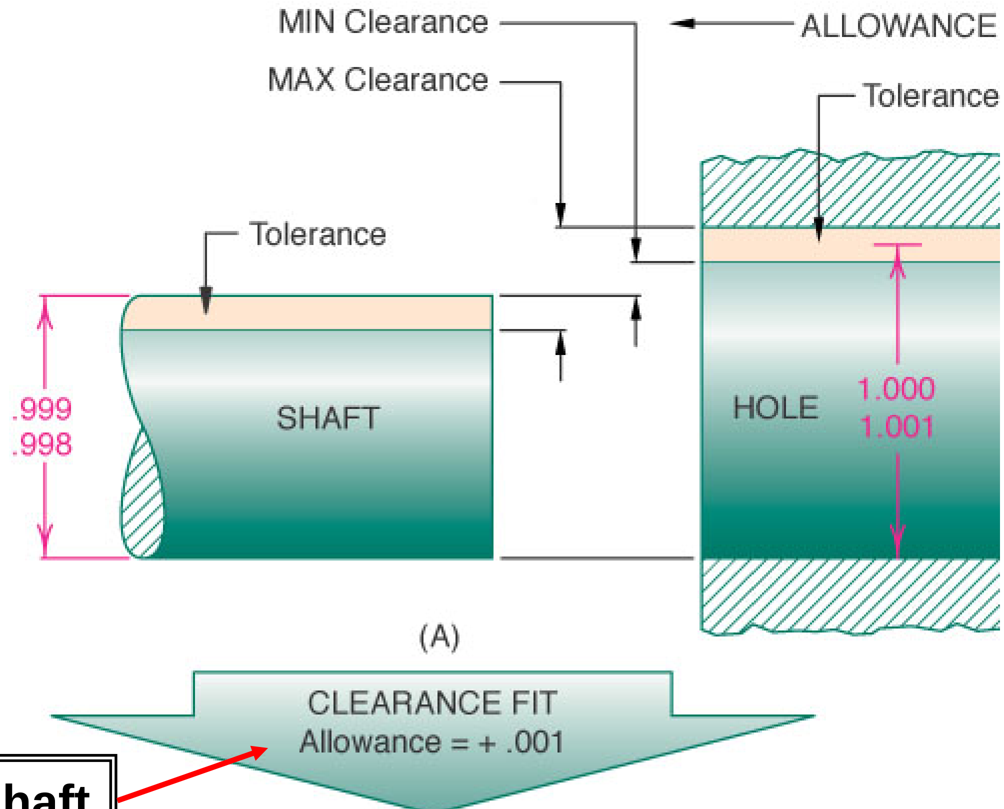
Piece tolerance – the difference between the upper and lower limits of a single part

System tolerance – the sum of all the piece tolerance

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.

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 smallest than the hole.

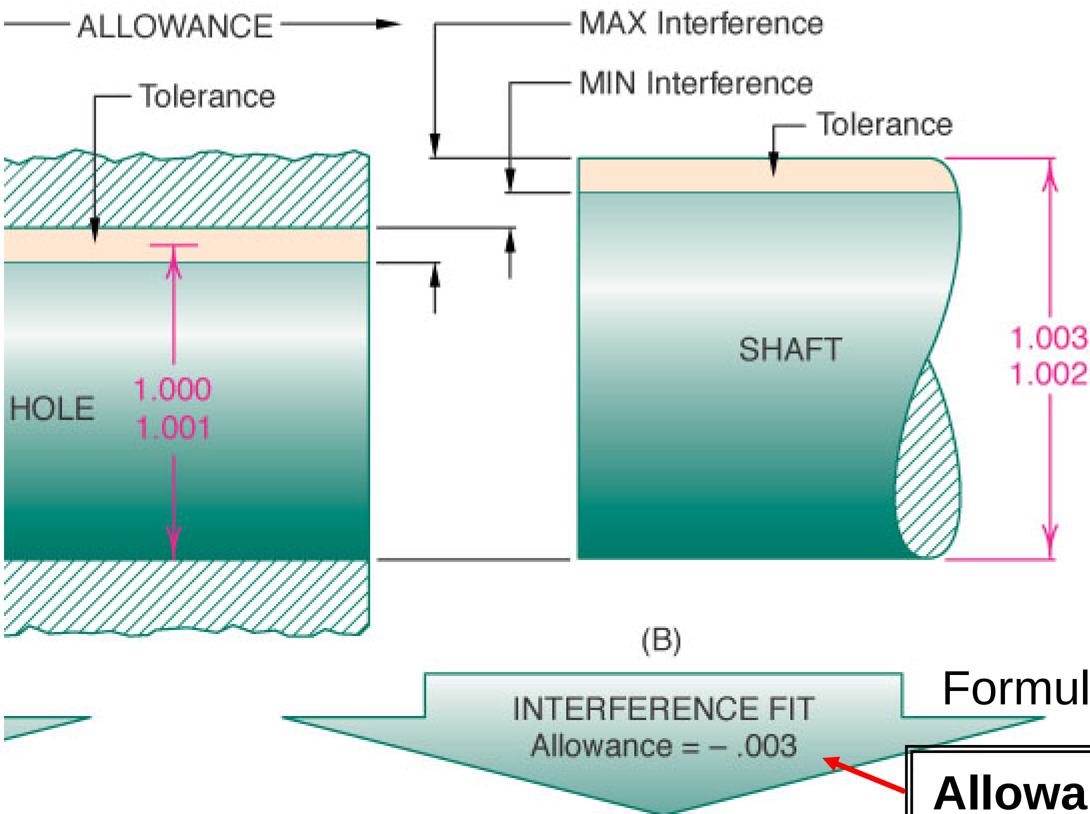


Formula:

Allowance = smallest hole – largest shaft

Fit types

- **Interference fit** occurs when two toleranced mating parts will always interfere when assembled.
- This fit type would be necessary to *stretch* the hole or *shrink* the shaft or to use force to press the shaft into the hole.



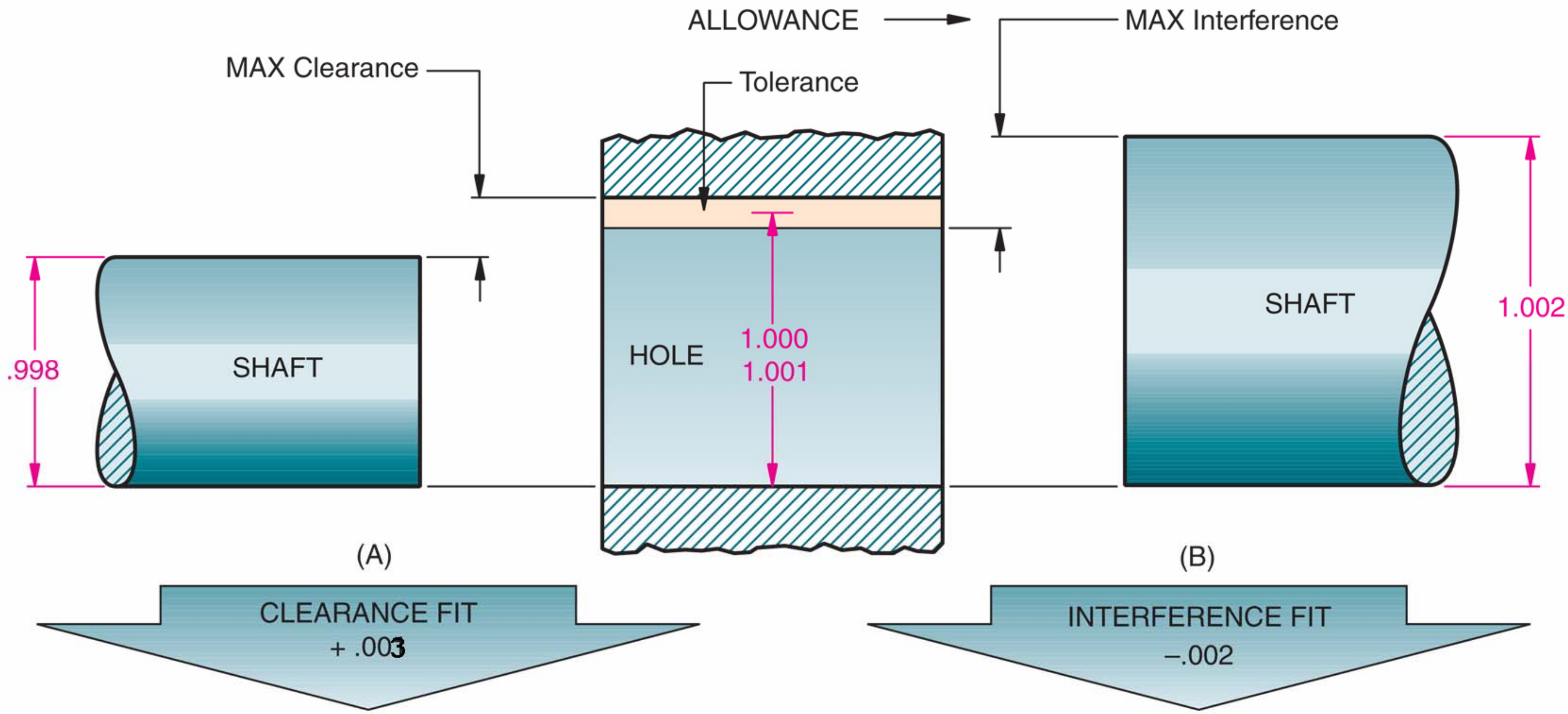
For example this fit type can be used to fasten two parts together without the use of mechanical fasteners or adhesive

Formula:

Allowance = smallest hole – largest shaft

Fit types

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

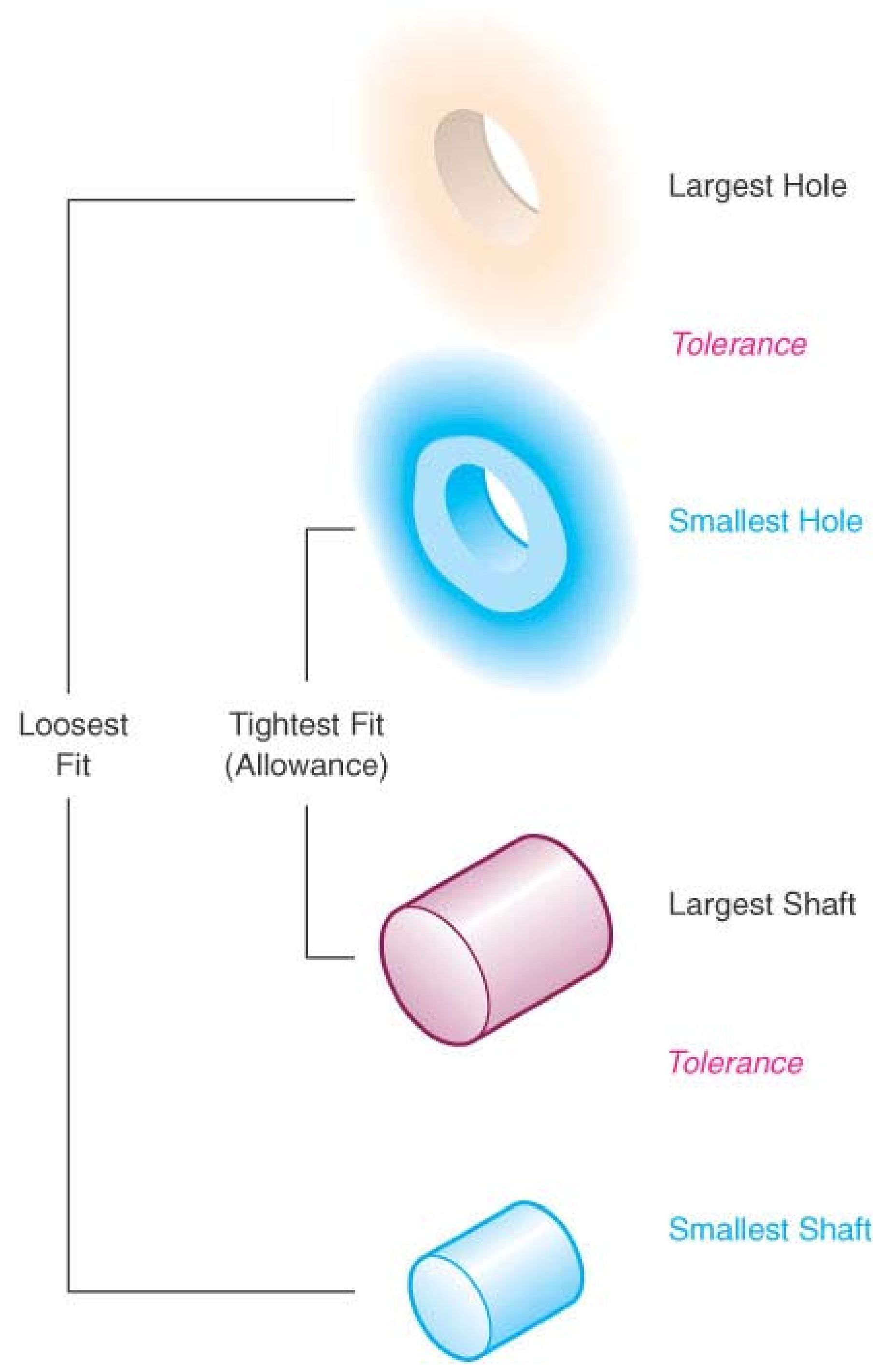


Allowance = largest hole – smallest shaft

Allowance = smallest hole – largest shaft

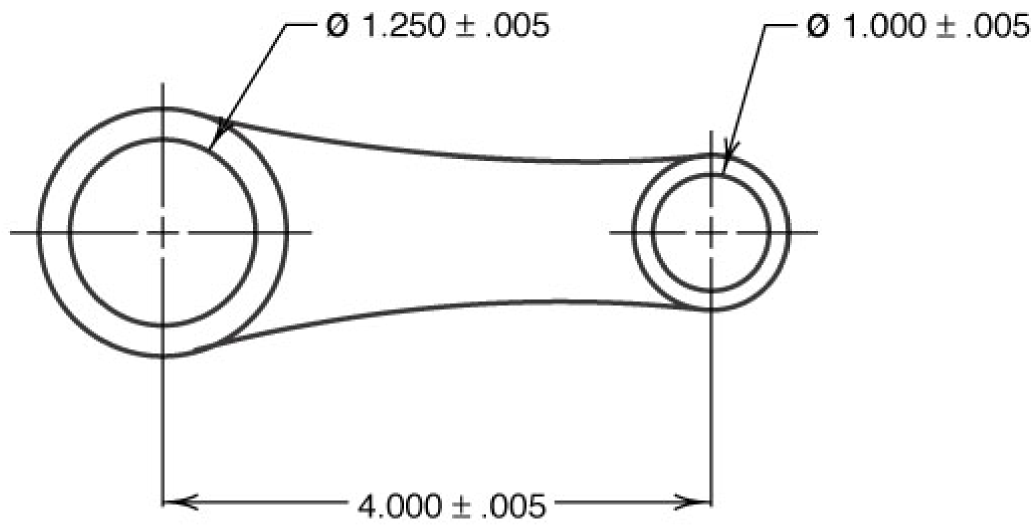
Fit type determination

- The **loosest fit** is the difference between the smallest feature A and the largest feature B.
- The **tightest fit** is the difference between the largest feature A and the smallest feature B.

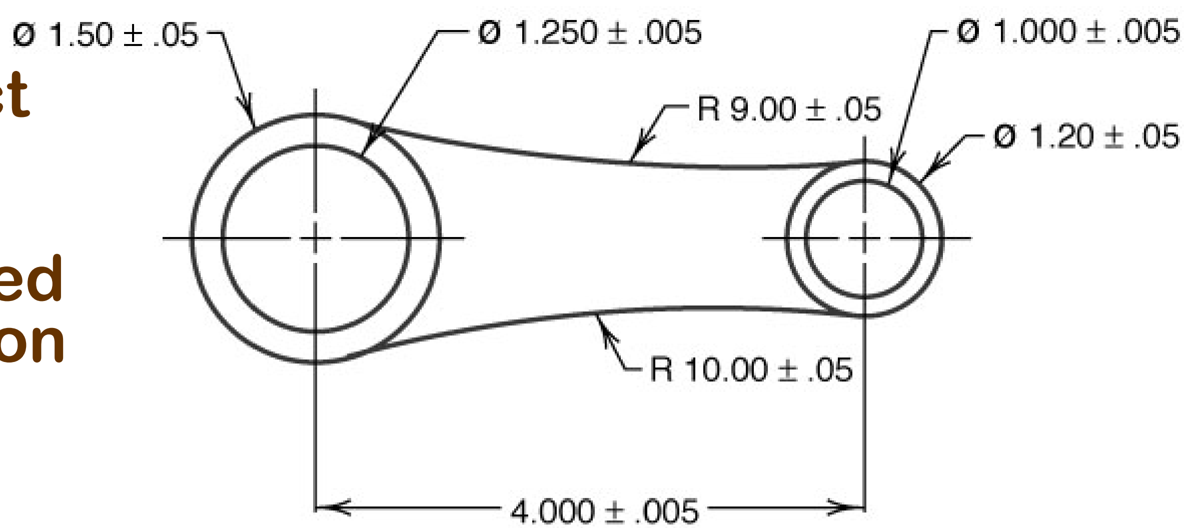


Functional dimensioning

- When dimensioning a part it is critical to start out by identifying the functional features first.
- Many times these features are holes.
- Any other features that come in contact with other parts, especially moving parts, are considered functional. Dimension these features first, then all other non-functional features can be considered.



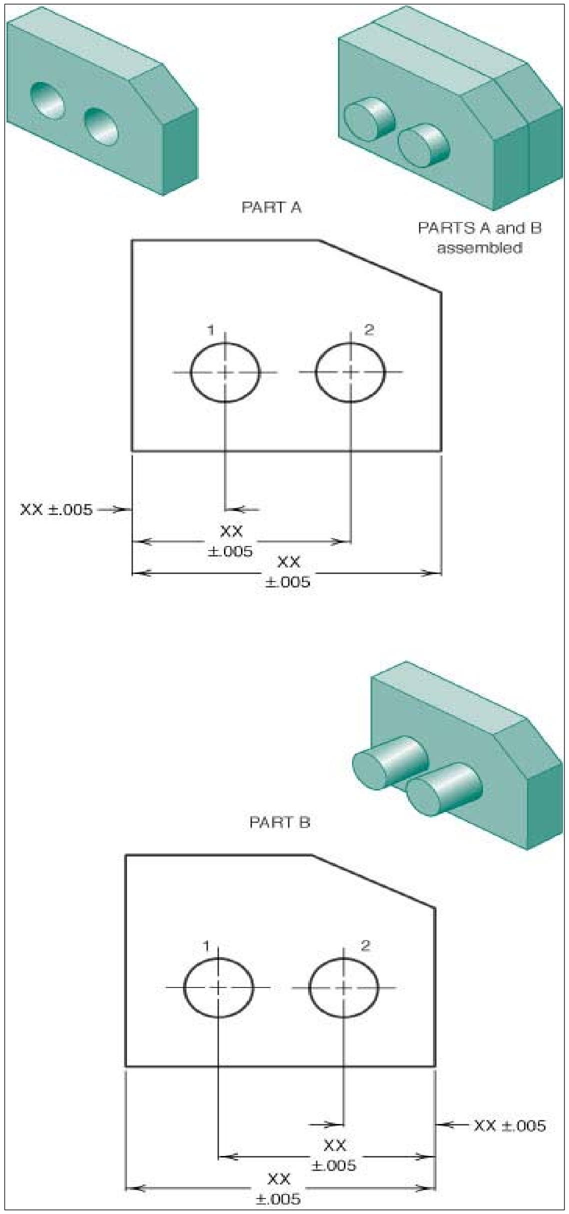
Functional dimensioning
Step 1



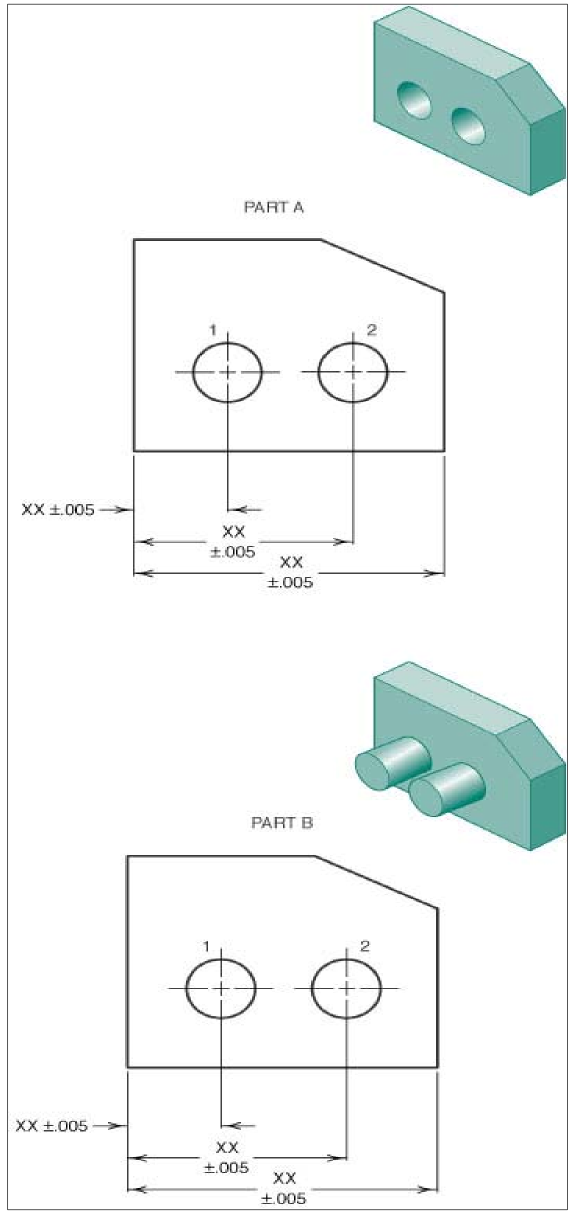
Functional dimensioning
Step 2

Tolerance Stack-Up

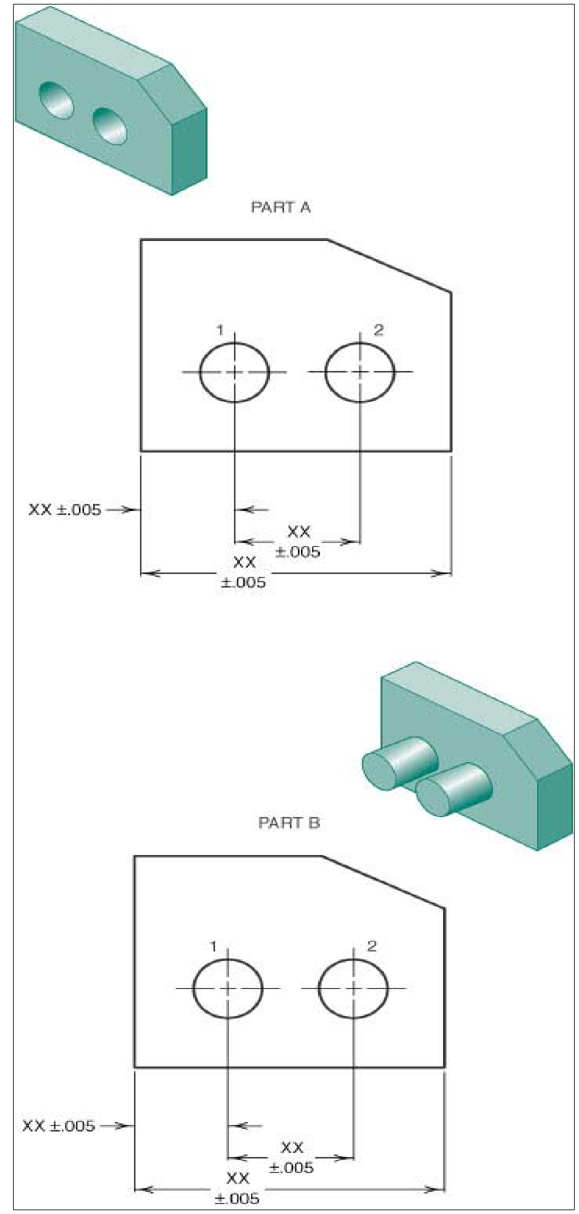
- The additive rule for tolerances is that tolerances taken in the same direction from one point of reference are additive
- The corollary is that tolerances to the same point taken from different directions becomes additive.
- The effect is called tolerance stack-up



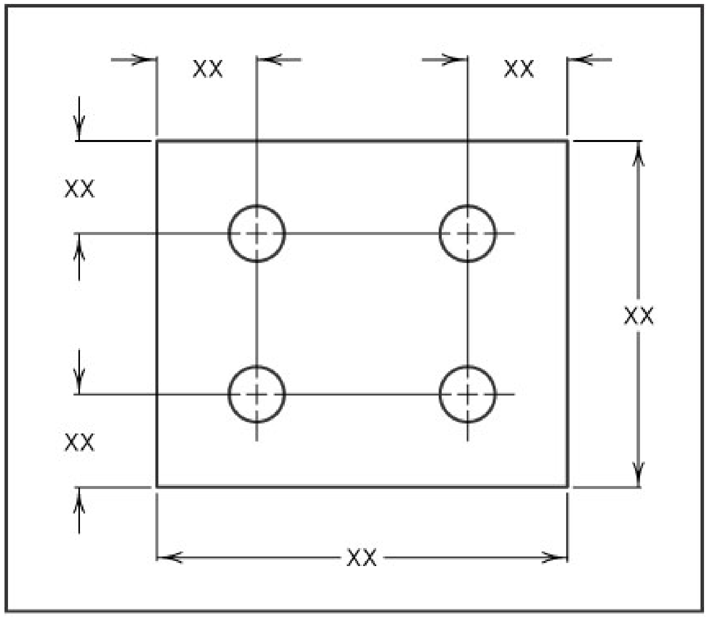
Tolerance stack-up
(cause assembly problem)



Tolerance stack-up eliminated
(used same reference)



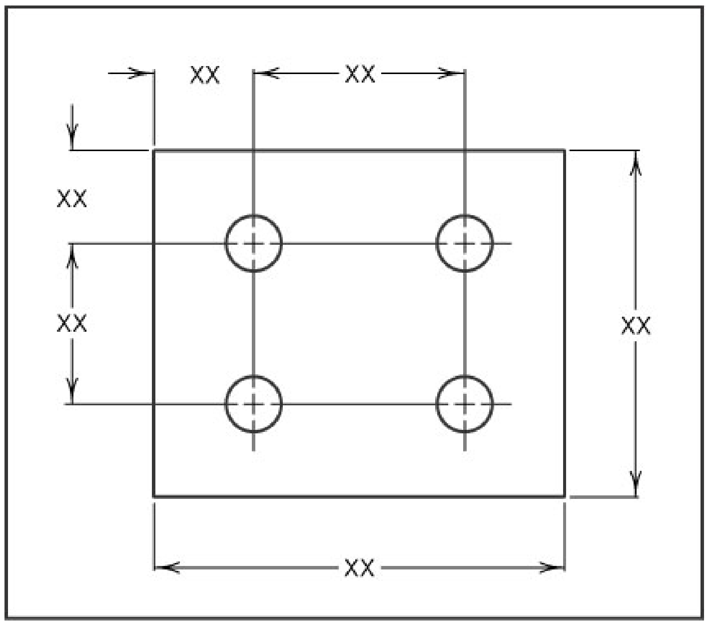
Alternate dimension
(A method locating the pattern first)



(A) No!

Coordinate dimension stack-up

Avoid coordinate dimension stack-up by using a common point and dimensioning the hole spacing directly (B)



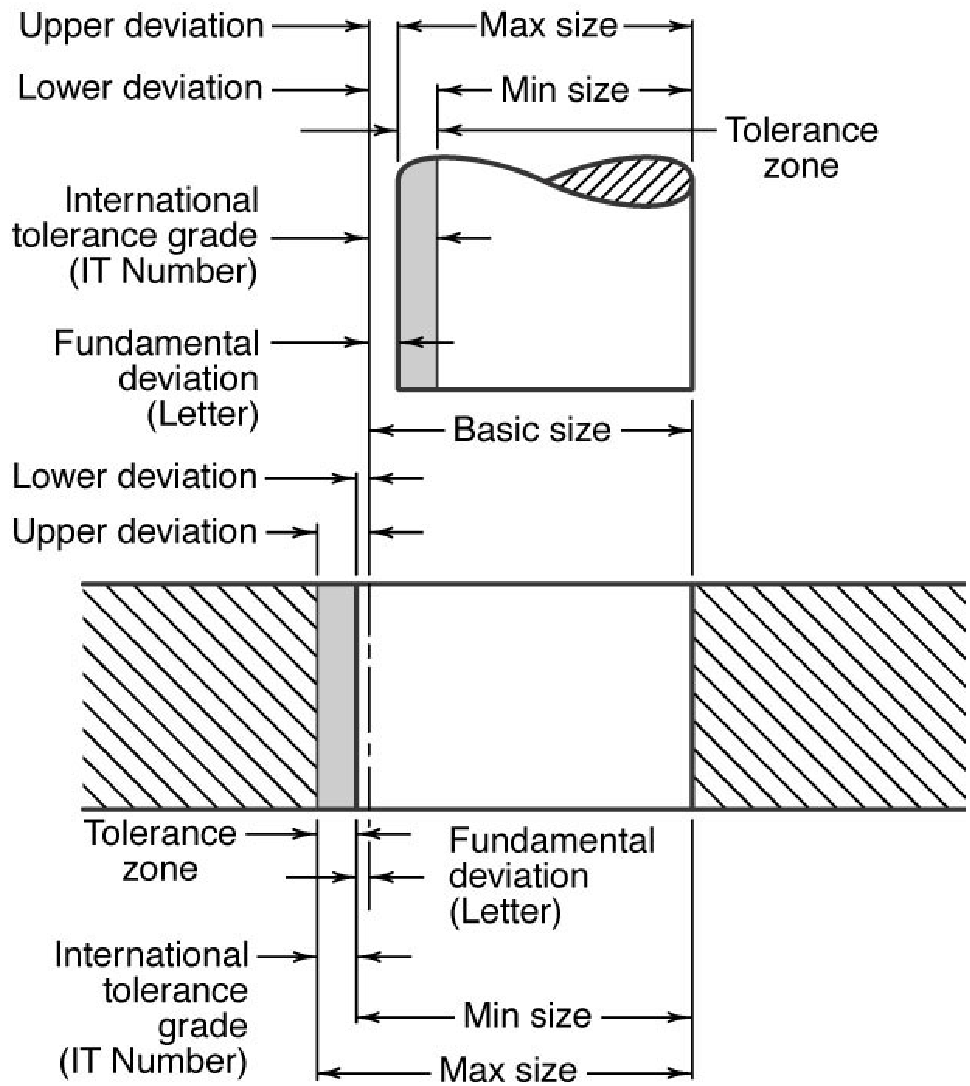
(B)

Metric limits and fits

- The standards used for metric measurements are recommended by the ISO and are given in ANSI B4.2-1978
- The terms used in metric tolerencing are follows:

- **Basic Size** is the size to which limits of deviation are assigned and are the same for both parts.

Basic Size, mm		Basic Size, mm	
1st Choice	2nd Choice	1st Choice	2nd Choice
1.0	—	—	7.0
—	1.1	8.0	—
1.2	—	—	9.0
—	1.4	10	—
1.6	—	—	11
—	1.8	12	—
2.0	—	—	13
—	2.2	14	—
2.5	—	—	15
—	2.8	16	—
3.0	—	—	17
—	3.5	18	—
4.0	—	—	19
—	4.5	20	—
5.0	—	—	21
—	5.5	22	—
6.0	—	—	23
—	6.5	—	24



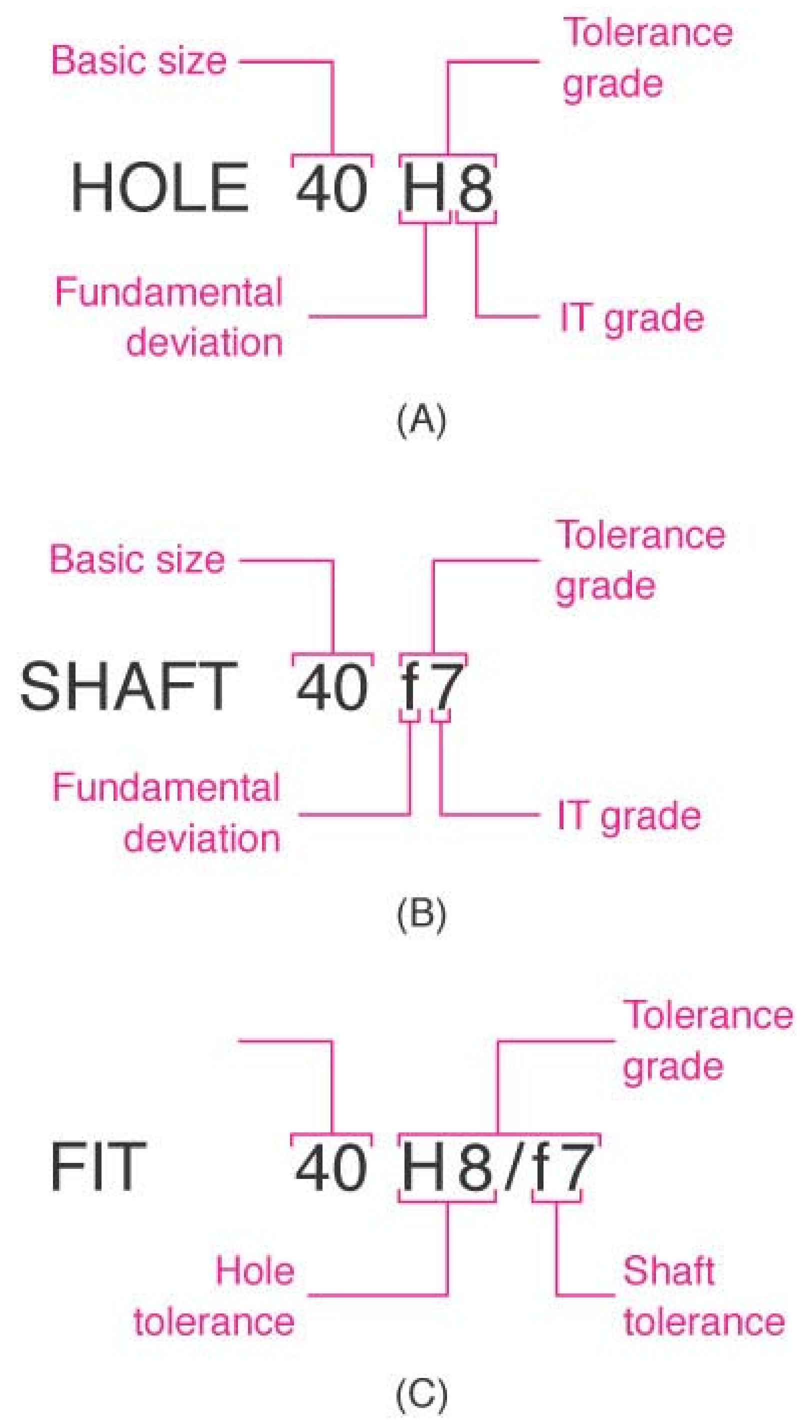
Fit category	ISO Symbol: Hole Basis	ISO Symbol: Shaft Basis	Description	
Clearance fits	H11/c11	C11/h11	Loose running fit for wide commercial tolerances or allowances on external members	More clearance ↑
	H9/d9	D9/h9	Free running fit not for use where accuracy is essential, but good for large temperature 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	
	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	More interference ↓

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

Metric symbols and their definitions

H – Hole basis is the system of fits where the minimum hole size is the basic size.

f – Shaft basis is the system of fits where the minimum shaft size is the basic size.



Three methods of showing metric tolerance symbols used for dimensions

(A)	(B)	(C)
40H8	40H8 (40.039 / 40.000)	(40.039 / 40.000) 40H8

The values in parentheses are for reference only and come from ANSI Standard B4.2-1978 tables

BASIC SIZE		LOOSE RUNNING			FREE RUNNING			CLOSE RUNNI	
		Hole H11	Shaft c11	Fit	Hole H9	Shaft d9	Fit	Hole H8	Shaft f7
40	MAX	40.160	39.880	0.440	40.062	39.920	0.204	40.039	39.975
	MIN	40.000	39.720	0.120	40.000	39.858	0.060	40.000	39.950
50	MAX	50.160	49.870	0.450	50.062	49.920	0.204	50.039	49.975
	MIN	50.000	49.710	0.130	50.000	49.858	0.080	50.000	49.950
60	MAX	60.190	59.860	0.520	60.074	59.900	0.248	60.046	59.970
	MIN	60.000	59.670	0.140	60.000	59.826	0.100	60.000	59.940

Example Determine the Tolerance using The Hole Basis System

- Given:
  - A shaft & Hole
  - The hole basis system
  - Clearance fit, and
  - A basic diameter of 41mm for the hole

Example Determine the Tolerance using The Hole Basis System

Solution:

- **STEP 1:**
  - From Table ANSI preferred metric sizes, assign the basic size of 40mm to the shaft

Basic Size, mm		Basic Size, mm	
1st Choice	2nd Choice	1st Choice	2nd Choice
—	7.0	25	—
8.0	—	—	26
—	9.0	—	28
10	—	30	—
—	11	—	32
12	—	35	—
—	13	—	38
14	—	40	—
—	15	—	42
16		45	

Example Determine the Tolerance using The Hole Basis System

Solution:

- **STEP 2:**
  - From Table ISO preferred metric fits, assign the sliding fit H7/g6. Sliding fit is defined in the table

ISO Symbol		Description	
Hole Basis	Shaft Basis		
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	
H7/g6	G7/h6	Sliding fit not intended to run freely but to move and turn freely and locate accurately	

Clearance fits

Example Determine the Tolerance using The Hole Basis System

Solution:

- **STEP 3 (For Hole):**
  - Determine the upper and lower limits of the hole from ANSI Standard B4.2-1978 tables. Using column H7 and row 40
  - From the table, the limits are 40.025 and 40.000

BASIC SIZE		LOOSE RUNNING			FREE RUNNING			CLOSE RUNNING			SLIDING		
		Hole H11	Shaft c11	Fit	Hole H9	Shaft d9	Fit	Hole H8	Shaft f7	Fit	Hole H7	Shaft g6	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
	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

Example Determine the Tolerance using The Hole Basis System

Solution:

- <u>STEP 4 (For Shaft):</u>
  - Determine the upper and lower limits of the shaft from ANSI Standard B4.2-1978 tables. Using column g6 and row 40.
  - From the table, the limits are 39.991 and 39.975

BASIC SIZE		LOOSE RUNNING			FREE RUNNING			CLOSE RUNNING			SLIDING		
		Hole H11	Shaft c11	Fit	Hole H9	Shaft d9	Fit	Hole H8	Shaft f7	Fit	Hole H7	Shaft g6	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
	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

NEXT continue to GDT 3