ME338 – Manufacturing Process II

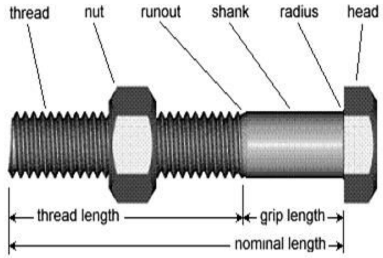
Lecture 13 – Limits & Fits

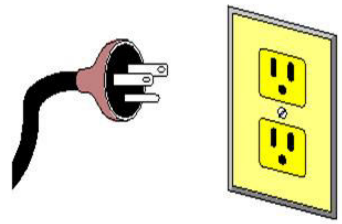
Pradeep Dixit
Associate Professor,
Department of Mechanical Engineering,
Indian Institute of Technology Bombay
Email: Pradeep.Dixit@iitb.ac.in, Tel: 25767393

R. K. Jain, Engineering Metrology (chapter 4) , Manufacturing Technology Vol. 2 – PN Rao (Chapter 15)

Quality and Interchangeability of parts

- Quality is deeply affected by variations
  - More variation >>> Poor quality
- Interchangeability: A component in an assembly made by one maker to fit its mating part made by another.
  - Nut and Bolts,
  - Chords and sockets used in computers
- Interchangeability permits some degree of allowed inaccuracy in manufacturing.
  - Reduces production cost
  - This permits the products to be manufactured in in large batches.
  - This also simplifies assembly and maintenance as components can be replaced readily.





Tolerances

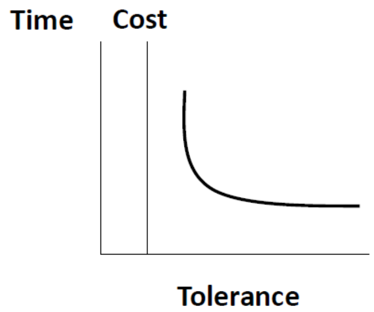
- No component can be manufactured precisely to a given dimension repeatedly in a cost-effective way.
  - This necessitates the manufacturing to be carried out with some deviation from desired size
  - This permissible deviation of dimension from the desired size is known as 'Tolerance'
- Tolerance is the acceptable error in manufacturing*
  - It is not wiser to have closer tolerances than necessary as this results in increase of manufacturing cost.
- In mass production of components, physical dimensions lie between two limits, upper (maximum) and lower (minimum)
  - Nominal size, upper limit, lower limit
- The difference between these limits is called Tolerance limit

$25^{+0.015}_{-0.010}$

ANSI (American National Standards Institute)

How to choose Tolerance?

- Two types:
  - Dimensional Tolerance (DT)
  - Geometrical Tolerance (GT)
- What are applications area:
  - Defence/medical/Space: tight
  - Kids toys: loose tolerance
- Cost of Production
  - Smaller tolerance range means higher cost
- Productivity (Time)
  - Smaller tolerance means lower productivity
- Tolerance representation
  - Numeric
  - Alphanumeric (ISO)
    - Hole: 25H7
    - Shaft: 25g6
    - Assembly: 25H7g6



$25^{+0.020}_{+0.015}$ $25^{+0.015}_{-0.010}$

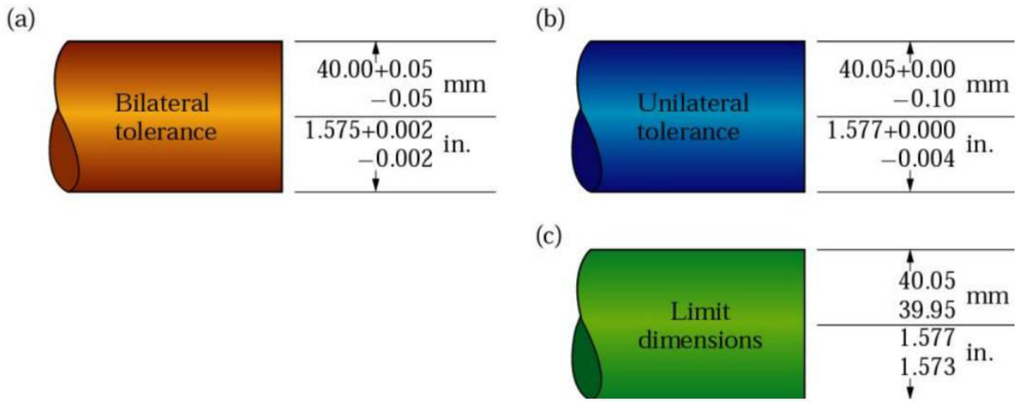
$25\ ^{0.010}$

Base size 25H7g6

Hole fit

Shaft fit

Dimensional tolerance: Unilateral and Bilateral



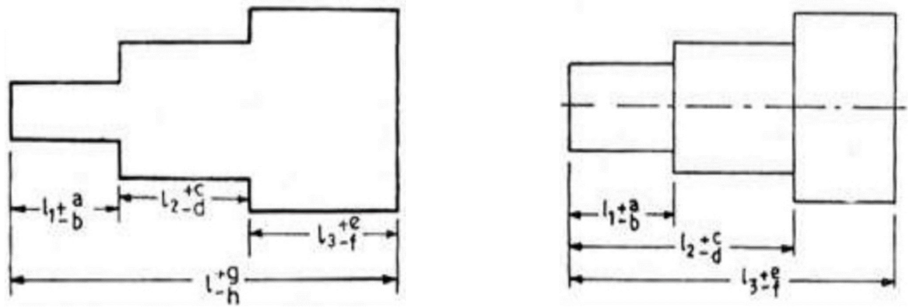
• Bilateral tolerance: The variation is permitted in both positive and negative directions from the nominal dimension

• Unilateral tolerance: The variation from the specified dimension is permitted in only one direction, either positive or negative

Tolerance zone: Graphical representation of tolerance. It is bounded by two limits.

Tolerance accumulation

- If a part comprises of several steps, each step having some tolerance over its length, then overall tolerance on complete length will be sum of the tolerance on individual length



$$l^{+g}_{-h} = l^{+a}_{1-b} + l^{+c}_{2-d} + l^{+e}_{3-f}$$

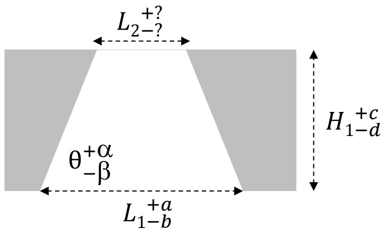
$$g = a + c + e$$

$$h = b + d + f$$

Effect of tolerance accumulation can be minimized by adopting progressive dimensioning from a datum

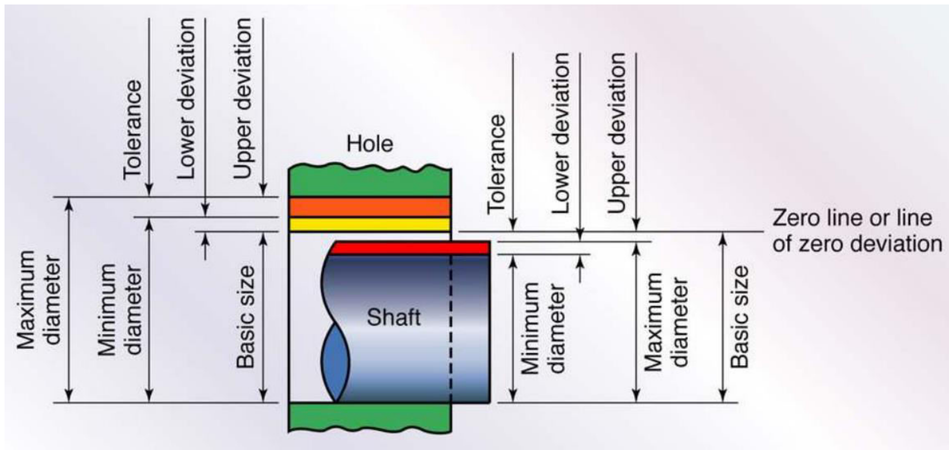
Compound tolerance

- Compound tolerance are derived by considering the effect of tolerance of more than one dimension.
- Tolerance on dimension L_2 are dependent on tolerance on L_1, H_1, and angle θ.
- Compound tolerance of 'L_2' is the combined effect of these three tolerances.
  - Minimum tolerance of L_2 will be corresponding to L_1-b, $\theta+\alpha$ and H_1+c.



$$L_2 = L_1 - 2\left(\frac{H_1}{tan\theta}\right)$$

Basic Terms used in fitting

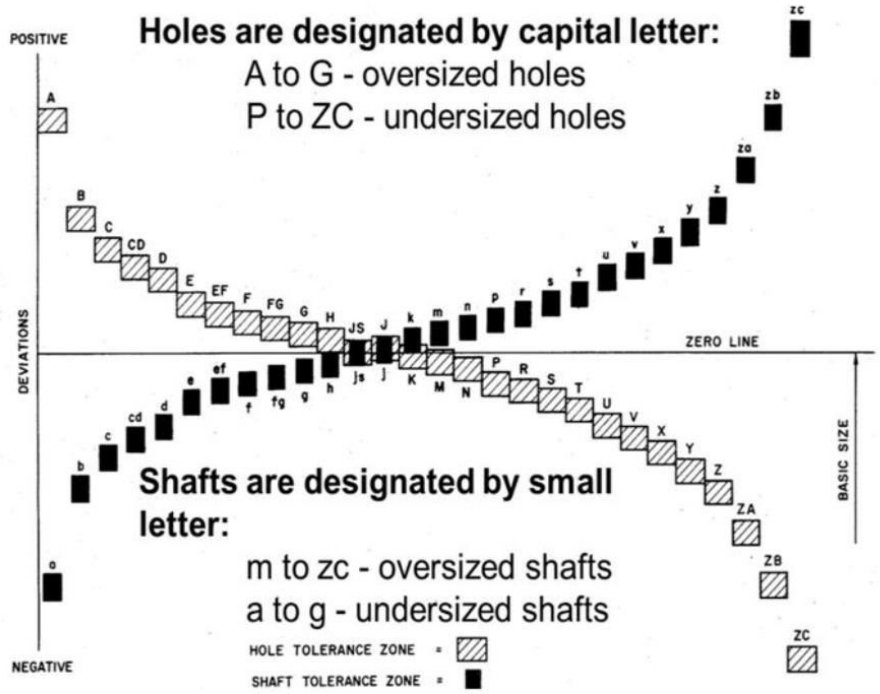


- Shaft : external dimensions, Hole : internal dimension
- Fits: The relationship resulting from the difference between the sizes of the two features (the hole and the shaft) which are to be assembled.
  - Clearance fit (positive allowance)
  - Interference (negative allowance)
  - Transit (allowance may + or -)

Basic Terms

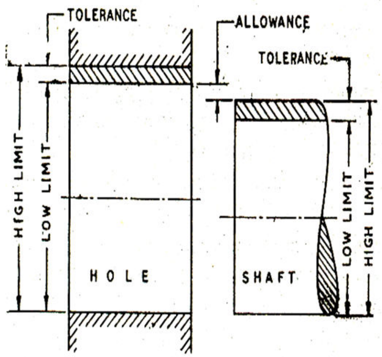
- Upper Deviation: The difference between the maximum limit and the basic size of a component
  - Expressed as ES for hole and es for shaft respectively
- Lower Deviation: The difference between the minimum limit and the basic size of a component
  - Expressed as EI and ei for holes and shaft respectively
- Fundamental Deviation:
  - The fundamental deviation is the one nearest the zero line.
  - May be either the upper or lower deviation,
  - It defines the location of the tolerance zone with respect to the nominal size (basic size)
  - 25 types each: A- ZC (For holes) and a- zc (For shafts)
- Basic Shaft (h) and Basic Holes (H) have their fundamental deviation equal to zero

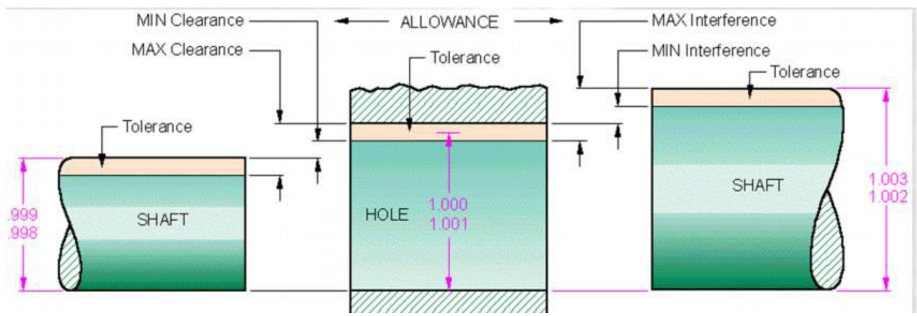
Tolerance Grades for Shafts and Holes



Allowance

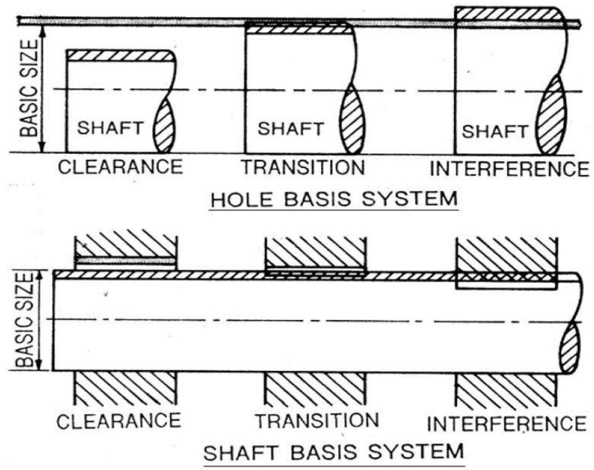
- The minimum space between two mating parts
- Based on the largest shaft (MMC for shaft and the smallest hole (MMC for hole)
- Allowance: MMC for hole – MMC for shaft
  - Positive allowance : clearance fit
  - Negative allowance: interference fit
  - A negative allowance indicates that the parts must be forced together





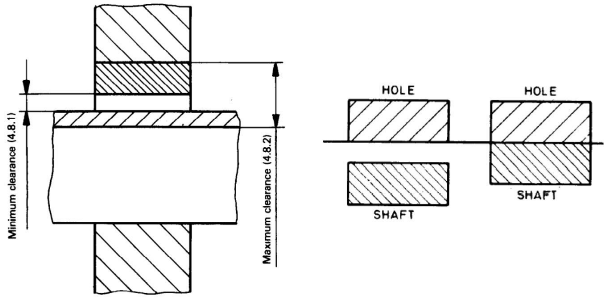
Fits in Assembly

- The relationship existing between two mating parts in an assembly with respect to the amount of play or interference between them when assembled together
- Fit may result in movable joint or permanent joint
- The nature of assembly of two mating parts is defined by three types of fit system:
  - Clearance Fit,
  - Transition Fit and
  - Interference Fit.



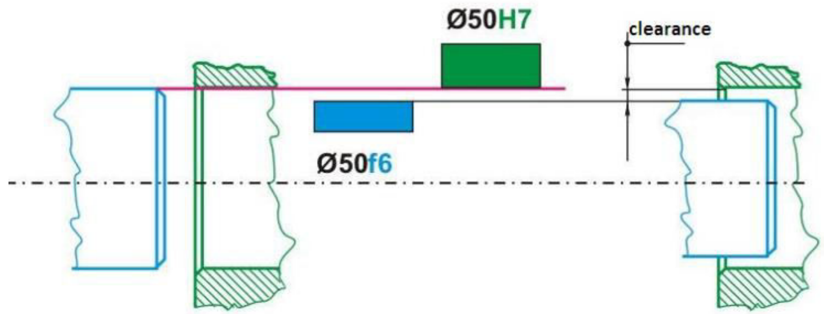
Fits: Clearance Fit

- The shaft of largest possible diameter can also be fitted easily even in the hole of smallest possible diameter
- A clearance fit has positive allowance, i.e. there is minimum positive clearance between high limit of the shaft and low limit of the hole
- <u>Clearance fit: D_{min} (hole) > D_{max} (shaft)</u>



- <u>Loose fit</u> : It is used between those mating parts where no precision is required. It provides minimum allowance and is used on loose pulleys, agricultural machineries etc.
- <u>Running fit</u>: dimension of shaft is smaller enough to maintain lubrication
- <u>Slid fit or medium fit</u>: used where precision is required: tool slides

Example of clearance fit – ϕ50H7f6



- The mating parts have such limits that the lowest hole diameter is larger than the largest shat diameter.

Selection of Fits: Clearance fit

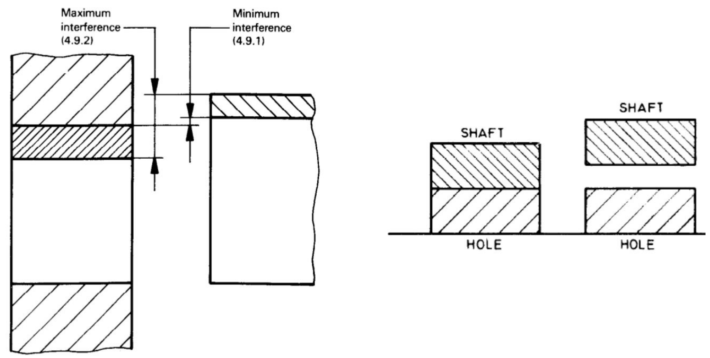
Clearance Fits (Hole Basis System):

Shafts	Grades	Description of fit	Application
a, b, c	11	Very large clearance	Generally not used
d	8, 9, 10	Loose running	Loose pulleys
e	7, 8, 9	Loose clearance	Electric motor bearings, heavily loaded bearing
f	6, 7, 8	Normal running	Lubricated bearings (with oil or grease), pumps and smaller motors, gear boxes
g	5, 6	Precision running	Lightly loaded shafts, sliding spools, accurate bearings
h	5 to 11	Extreme clearance (preferably for non-running parts)	Sockets and spigots of joints

Preferred Clearance fits (in practice) : H11/c11, H9/d9, H8/f7, H7/g6 (Guide Fit), H7/h6, C11/h11, D9/h9, F8/h7, G7/h6

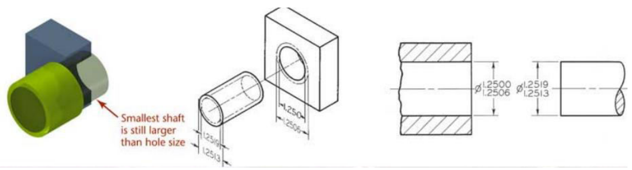
Fits: Interference Fit

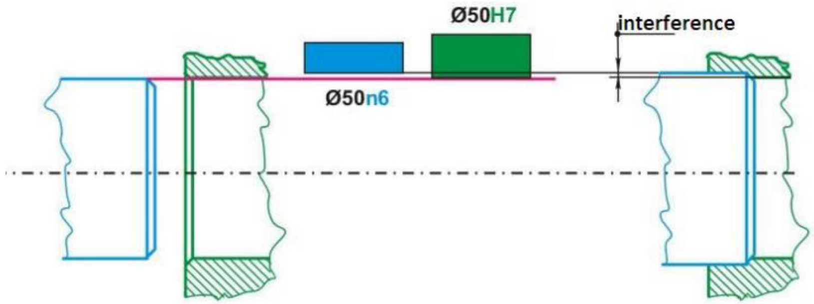
- There is always a overlapping of the matting parts. Hence, for fit, they need to pressed together
- <u>Interference fit: D_{max} (hole) < D_{min} (shaft)</u>



- <u>Shrink or heavy force fit</u>: It refers to maximum negative allowance.
- <u>Medium force fit</u>: These fits have medium negative allowance. used in car wheels
- <u>Tight or force fit</u>: used where precision is required: tool slides

Example of interference fit – ϕ50H7n6

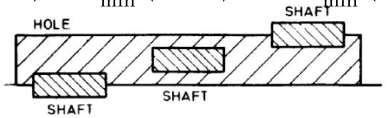


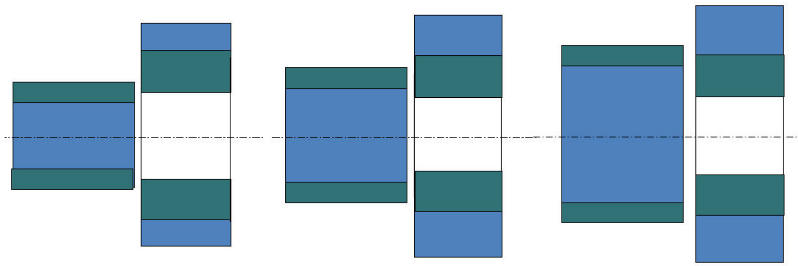


- The mating parts have such limits that the lowest shaft diameter is larger than the largest hole diameter..

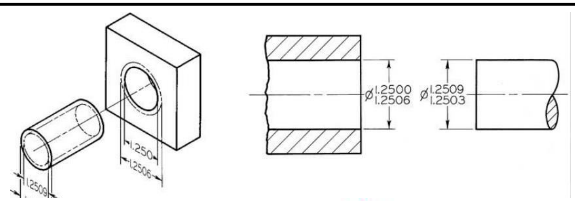
Transition Fit –Hole Basis

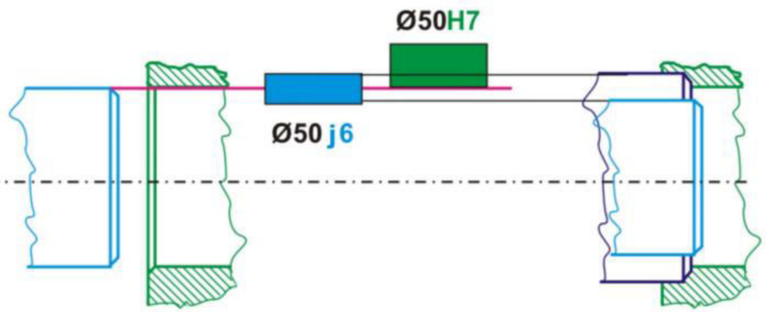
- It may result in either clearance fit or interference fit depending on the actual value of the individual tolerances of the mating components.
- Transition fits are a compromise between clearance and interference fits.
- They are used for applications where accurate location is important but either a small amount of clearance or interference is permissible.
- Transit fit: D_{max} (hole) > D_{min} (shaft) and D_{min} (hole) < D_{max} (shaft)





Example of Trasition fit – ϕ50H7j6





- Either a clearance or an interference may result depending on the exact value of the dimensions of the machined shaft and hole within the specified tolerance zones.

Typical Recommended Fits

- Clearance Fits
  - H7/h6 : Sealing rings, bearing covers
  - H7/g6 : Sleeve shafts, clutches
  - H7/f7 : High speed bearings, machine tool spindles
- Transition Fits
  - H7/n6 : Gears and bearing bushes, shaft and wheel
  - H7/m6 : Gears belt pulleys, couplings
- Interference Fits
  - H8/u8 : Worm wheel hubs, couplings
  - H7/r6 : Coupling of shaft ends, valve seats, gear wheels

Hole basis : hole dimensions is kept close to base size (i.e., 'H'

Typical Fits

Description	Hole	Shaft
Loose Running	H11	c11
Free Running	H9	d9
Close Running – Accurate machines	H8	f8
Sliding – Not to run freely	H7	g6
Close Clearance - Spigots and locations	H8	f7
Location/Clearance	H7	h6
Location- Transition – Accurate location	H7	k6
Location/Transition - More accurate location	H7	n6
Location/Interference- Press fit which can be separated	H7	p6
Medium Drive	H7	s6
Force	H7	u6

Hole basis : hole dimensions is kept close to base size (i.e., 'H')

Example

- A 50 mm diameter shaft is made to rotate in the bush. The tolerances for both shaft and bush are 0.050 mm. Determine the dimension of the shaft and bush to give a maximum clearance of 0.125 mm with the hole basis system.
- For each of the following hole and shaft assembly, find shaft-tolerance, hole tolerance and state what type of fit is.

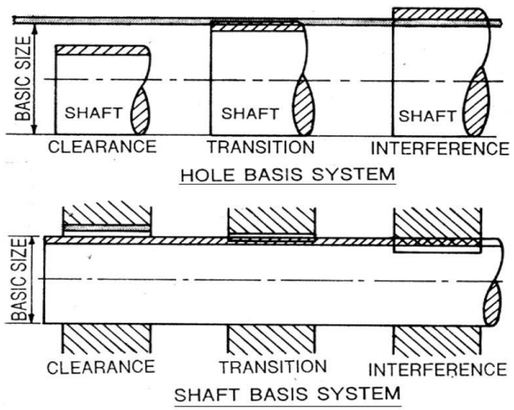
$$\text{(i)} \quad \text{Hole : } 50^{+0.25}_{+0.00}\,\text{mm, Shaft : } 50^{+0.05}_{+0.005}\,\text{mm}$$

$$\text{(ii)} \quad \text{Hole : } 30^{+0.05}_{+0.00}\,\text{mm, Shaft : } 30^{-0.02}_{+0.05}\,\text{mm}$$

$$\text{(iii)} \quad \text{Hole : } 25^{+0.04}_{+0.00}\,\text{mm, Shaft : } 25^{+0.06}_{+0.04}\,\text{mm}$$

Fit Systems: Hole Basis & Shaft basis

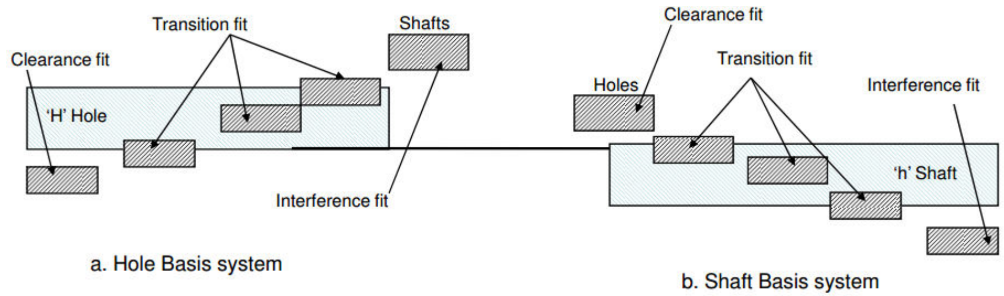
- In hole basis system: Hole dimensions fixed, shaft dimensions are varied
- In shaft basis system: Shaft dimension fixed (upper deviation of shaft is zero (shaft h), hole dimensions are varied
- Hole basis system is more popular !
  - Since the holes are produced by drilling, reaming, their size is not easily adjustable
  - Shaft dimensions can be easily changed by turning, milling etc



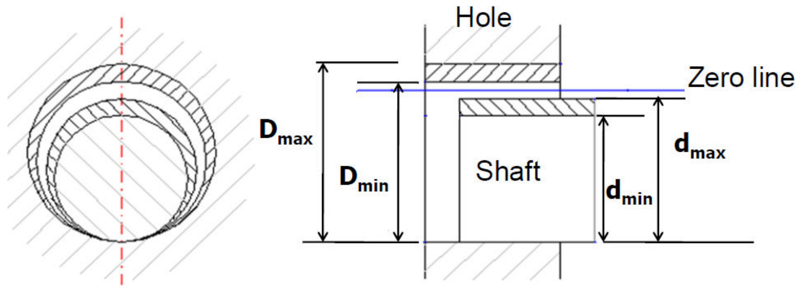
One can make 5 mm dia hole easily, but making 5.05 or 5.15 mm is challenging. Drill bits of standard size are available
Shaft dimensions can be easily changed by choosing proper depth of cut in turning, so dia. 5.05 or 5.15 mm is ok!!

Hole Basis System

- In the hole basis system, the size of the hole is kept constant and shaft sizes are varied to obtain various types of fits
- In this system, lower deviation of hole is zero, i.e. the low limit of hole is same as basic size.
- The hole basis system is commonly used because it is more convenient to make correct holes of fixed sizes, since the standard drills, taps, reamers and branches etc. are available for producing holes and their sizes are not adjustable.
- On the other hand, size of the shaft produced by turning, grinding, etc. can be very easily varied.

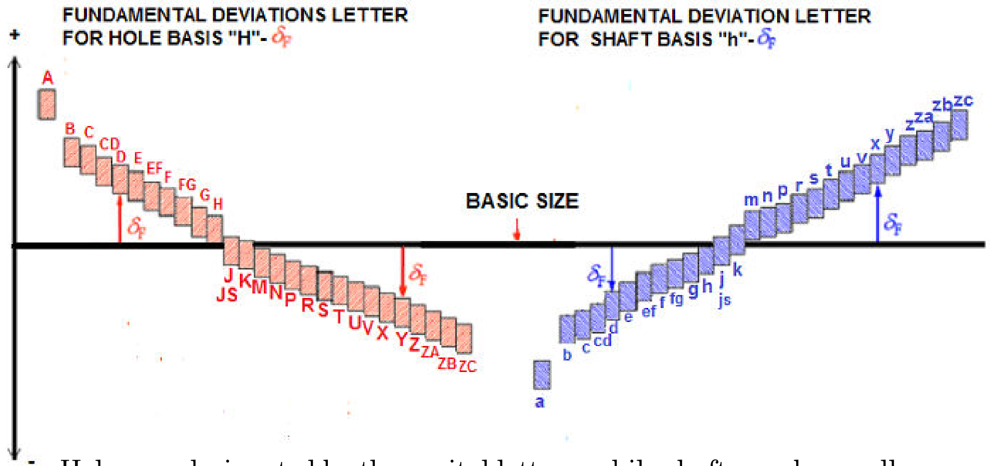


Tolerance Zones – Shaft and Hole- Basic size D_0



- Basic size D_0
- Allowance = $D_{min} - d_{max}$
- Upper deviation for hole = D_{max}-D_0 Lower deviation for hole = D_{min}-D_0
- Upper deviation for shaft = D_0-d_{max} Lower deviation for shaft = D_0-d_{min}
- Allowance = $D_{min} - d_{max}$ = Lower deviation for hole + upper deviation for shaft
- Tolerance in hole and shaft are designated in various alphabets
  - Hole : A, B, C, CD, D. E, EF, F ……Z, ZA, ZB, ZC – (total 28 Nos)
  - Shaft : a, b, c, cd, d, e, ef, f……z, za, zb, zc – (total 28 Nos)
- Letters I, L, O, Q, W are not used to avoid confusion

ISO Tolerance Grades for Shafts and Holes



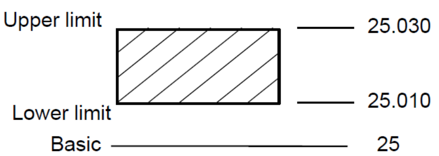
Holes are designated by the capital letters, while shafts are by small letters
A-G : oversized holes, P-ZC : undersized holes
a-g : under-sized shafts, p-zc : oversized holes

Basic shaft is a shaft whose upper deviation is zero, e.g., shaft 'h'
Basic hole is one whose lower deviation is zero, e.g., hole 'H'

Example: Tolerance Zone Location

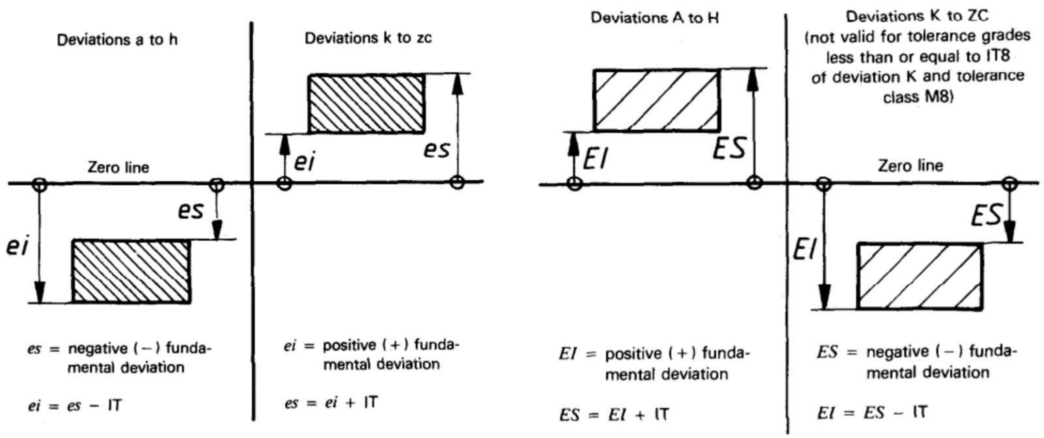
- Hole : $25.00^{+0.030}_{+0.010}$
- Upper Deviation : Upper limit – Basic size : 25.030-25.0 = 0.030 mm
- Lower Deviation : Lower limit – Basic size : 25.010-25.0 = 0.010 mm
- Fundamental Deviation = 0.010 mm
- Locates Tolerance zone
- Basic Shaft : h Fundamental Deviation
- Basic Hole : H is zero

<u>Hole</u> : $25^{+0.030}_{+0.010}$



Tolerance = 0.020 mm
= 20 μm

Deviations and Fundamental Deviations



Deviations for Shafts

Upper deviation for shaft: es
Lower deviation for shaft: ei

$es = ei + IT$

Deviations for Holes

Upper deviation for hole: ES
Lower deviation for hole: EI

$ES = EI + IT$

IT : International Tolerance

Tolerance Grade (IT)

	FOR MEASURING TOOLS																	
IT GRADES	01	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16

FOR MATERIAL

FOR FITS

FOR LARGE MANUFACTURING TOLERANCES

20 grades of tolerances: IT01, IT0, IT1, IT2, IT3………IT17, IT18

Tolerance grade	Manufacturing process and applications	Machine required
IT01, IT0 IT1 to IT5	Super finishing process, such as lapping, diamond boring etc. Use: Gauges	Super finishing machines
IT6	Grinding	Grinding machines
IT7	Precision turning, broaching, honing	Boring machine, honing machine
IT8	Turning, boring and reaming	Lathes, capstan and automats
IT9	Boring	Boring machines
IT10	Milling, slotting, planing, rolling and extrusion	Milling machine, slotting machine, planing machine and extruders
IT11	Drilling, rough turning	Drilling machine, lathes
IT12, IT13, IT14	Metal forming processes	Presses
IT15	Die casting, stamping	Die casting machine, hammer machine
IT16	Sand casting	—

Grades of Tolerance (IT)

- Standard Tolerance (i): Various grades of tolerances are defined using the 'standard tolerance unit', (i) in µm, which is a function of basic size. $i(microns) = 0.45\sqrt[3]{D} + 0.001D$
- D (mm) is the geometric mean of the lower and upper diameters of a particular diameter steps within which the chosen the diameter D lies.
- Diameter steps in I.S.I are:

1-3	3-6	6-10	10-18	18-30
30-50	50-80	80-120	120-180	180-250
250-315	315-400	400-500		

Tolerance Values (basic size < 500 mm)

- Tolerance values are D in mm, i in µms.
  - IT01: 0.3 + 0.008D, IT0 : 0.5 + 0.012D
  - IT1: 0.8 + 0.020D, IT2 - IT4:values of tolerance grades are placed geometrically between the tolerance grades of IT1 and IT5
  - IT5: $7i$

IT6	IT7	IT8	IT9	IT10	IT11	IT12	IT13	IT14	IT15	IT16
10i	16i	25i	40i	64i	100i	160i	250i	400i	640i	1000i

Standard tolerance factor I (microns)=0.004D +2.1

Tolerance Values (basic size > 500 – 3150 mm)

IT1	IT2	IT3	IT4	IT5	IT6	IT7	IT8	IT9	IT10
2I	2.7I	3.7I	5I	7I	10I	16I	25I	40I	64I

IT11	IT12	IT13	IT14	IT15	IT16	IT17	IT18		
100I	160I	250I	400I	640I	1000I	1600I	2500I		

Fundamental deviation for shafts upto 500 mm

UPPER DEVIATION (es)		LOWER DEVIATION (ei)	
Shaft Designation	In Microns (for D in mm)	Shaft Designation	In microns (for D in mm)
a	= -(265 + 1.32D) for D ≤ 120 ;and = – 3.52D for D > 120	j5 to j8	No formula
		js	ITx1/2
		k4 to k7	=+ 0.6x $\sqrt[3]{D}$
b	= – (140 + 0.852D); for D <160 ;And = – 1.82D for D > 160	k for Grade ≤3 and ≥4	=0
		m	= + (T7-IT6)
c	= – $52D^{0.2}$ for D ≤ 40 = -(95 + 0.82) for D> 40	n	= + $5D^{0.34}$
		P	= + IT7 + 0 to 5
cd	G.M. of values for c and d	r	= GM of values for p and s
d	= – $16D^{0.44}$		
e	= $-11D^{0.41}$	s	= IT8 + 1 to 4; for D ≤50 = + 7T7 to + 0.4D; for D > 50
ef	G.M. of values for e and f		
f	= $-5.5D^{0.41}$	t	= + IT7 + 0.63D
fg	G.M. of values for f and g	u	= + IT7 + D
g	= $-2.5D^{0.34}$	V	= + IT1 + 1.2525D
h	= 0	X	= + IT7 + 1.62D
		y	= + IT7 + 2D
		z	= + IT7 + 2.52D
		za	= IT8 + 3 + 3.152D
		zb	= + IT9 + 4D
		zc	= + IT10 + 4D

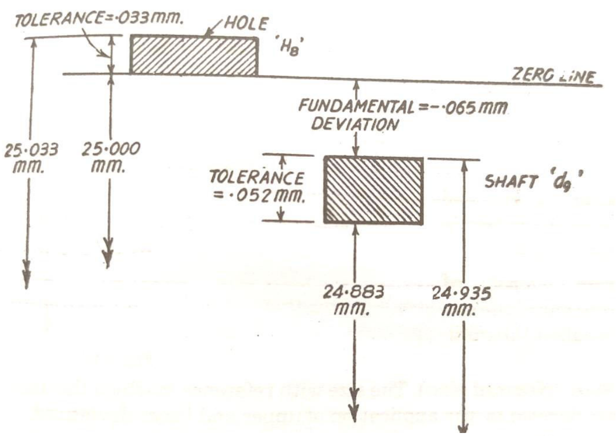
Table 4.5
Formulae for Shaft and Hole Deviations for sizes above 500 mm and upto 3150 mm

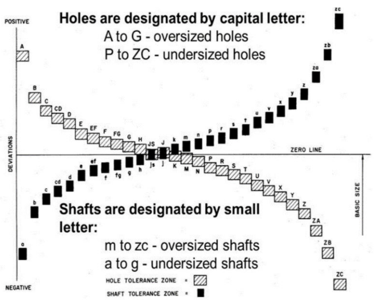
Shafts			Holes			Formula for deviations in μ
Type	Fund Deviation	Sign	Type	Fund Deviation	Sign	(For D in mm)
d	s	—	D	EI	+	$16D^{0.44}$
e	"	—	E	"	+	$11D^{0.41}$
f	"	—	F	"	+	$5.5D^{0.41}$
g	"	—	G	"	+	$2.5D^{0.34}$
h	"	No sign	H	"	No sign	0
js	ei	—	JS	ES	+	$0.5ITn$
k	"	+	K	"	—	0
m	"	+	M	"	—	$0.024D + 12.6$
n	"	+	N	"	—	$0.04D + 21$
p	"	+	P	"	—	$0.072D + 37.8$
r	"	+	R	"	—	Geometric mean of the values for p and s or P and S
s	"	+	S	"	—	$IT7 + 0.4D$
t	"	+	T	"	—	$IT7 + 0.63D$
u	"	+	U	"	—	$IT7 + D$

Example

- Calculate the limits of tolerance and allowance for a 25 mm shaft and hole designated H_8d_9.
- Steps: Base size : 25 mm (lying between 18 and 30 mm), Hole 'H8', shaft 'd9'
- 1 – find mean diameter ($D_{mean} = \sqrt{D_1D_2}$) = 23.2 mm
- 2 – find fundamental tolerance factor i ($0.45\sqrt[3]{D} + 0.001D$) = 1.3 μm
- 3 – for hole H8, find tolerance grade IT (25i) = 33 μm
- 4 – For 'H' hole, fundamental deviation (lower deviation = 0)
- Lower limit for 'H' hole = base size = 25 mm
- Upper limit for 'H' hole = ES=EI+IT = 25.033 mm
- For shaft 'd9': tolerance grade IT (40i) = 52 μm = 0.052 mm
- For 'd' shaft, fundamental deviation : $-16D^{0.44}$ = -65 μm = 0.065mm
- Therefore, for 'd' shaft, upper limit: 25 mm – 0.065 = 24.935 mm
- Lower limit : upper limit – IT = 24.935 – 0.052 = 24.833 mm

Example





1-3	3-6	6-10	10-18	18-30
30-50	50-80	80-120	120-180	180-250
250-315	315-400	400-500		

IT6	IT7	IT8	IT9	IT10	IT11	IT12	IT13	IT14	IT15	IT16
10i	16i	25i	40i	64i	100i	160i	250i	400i	640i	1000i

Example

- For a particular application, an H7 fit has been selected for the hole and a K6 fit for the shaft. The tolerance quoted are 0 to +25, for the hole and 12 to +18 for the shaft. Find the upper limit and lower limit for the hole and also for bush. The basic size of fit is 50 x 10^{-3} m.

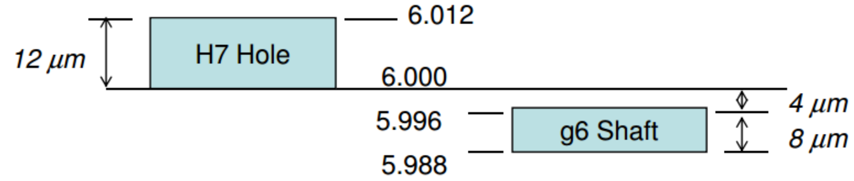
- **Solution:**
- The upper limit for the hole will be (50.000 + 0.025) x 10^{-3}
- = 50.025 x 10^{-3} m
- The lower limit for the hole will be (50.000 + 0) x 10– 3 = 50.000 x 10– 3 m.
- The upper limit for the bush will be (50.000 + 0.018) x 10^{-3} = 50.018 x 10^{-3} m.
- The lower limit for the bush will be (50.000 + 0.002) x 10^{-3} = 50.002x 10^{-3} m

Example 1

- Evaluate limits and fits for a pair of – Diameter 6 H7/g6
- Solution steps:
  - First find out the diameter D
  - Find out fundamental tolerance unit (i)
  - Find out limits of tolerance for hole H7 and g6 s $i = 0.45\sqrt[3]{D} + 0.001D$
    - for H7, tolerance limit 16i, for g6 shaft 10i
  - Find out fundamental deviation for H hole

- The size 6 mm lies in the diametral step of 3-6, so Diameter D would be geometric mean D = 4.24 mm
- fundamental tolerance unit (i) = 0.7327 µm
- The standard tolerance for hole H7 : – 16 i = 16x0.7327 = 11.72 = 12 µm
- The standard tolerance for shaft g6 : – 10 i = 10x0.7327 = 7.327 = 8 µm
- The fundamental deviation (lower deviation) for 'H' hole is 0
- Fundamental deviation (upper deviation) for g shaft : $-2.5D^{0.34}$ = -4.085 µm

Example 1

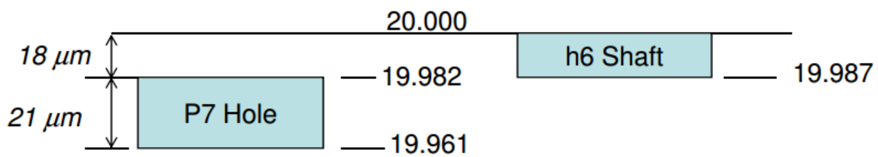


Disposition of tolerance zone around the zero line

- Fit:
- Maximum clearance = Maximum size of hole - Minimum size of shaft
- = 6.012 – 6.988 = 0.024 mm = 24 µm
- Minimum clearance = Minimum size of hole - Maximum size of shaft
- = 6.000 - 6.996 = 0.004 mm = 4 µm
- The type of fit is Clearance

Example 2

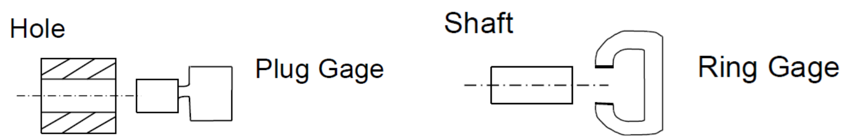
- Calculate the limits of sizes for φ 20 P7/h6 and identify the fit
- Solution:
  - D = 20.78 mm, fundamental tolerance unit (i) = 1.258 µm
  - fundamental deviation for p shaft is – IT7:



Disposition of tolerance zone around the zero line

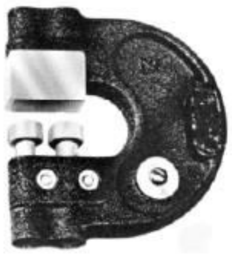
Limit Gauges

- Gauging is defined as the acceptability of a given dimension whether it lies in its specified or allowable limits or not
- Actual size measurement of mating parts is not important as long as it falls within the tolerance zone
- Go-NoGo Limit Gauges
- Objective of limit gauges:
  - Check if the part size is within the Upper and Lower size Limits
  - Go gage should always go (into the part) – min dimension of hole.
  - NoGo gauge should NOT go. - maximum dimension of hole
- Go gauge should check both part Form (shape) and Dimension.



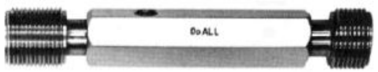
Various Types of Limit Gauges

Cylindrical Pin Gauge



Adjustable Snap Gauge

Plain Ring Gauge



Thread Plug Gauge

Cylindrical Plug Gauge

Progressive Cylindrical Plug Gauge

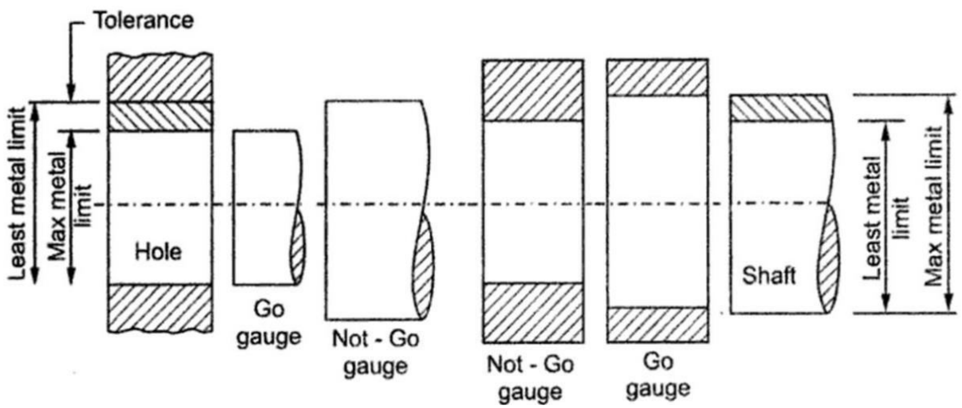
Adjustable Thread Ring Gauge

MMC and LMC/ Taylor's principle

- Maximum Material Conditions (MMC) : GO gauge follows MMC
  - Upper limit of the shaft
  - Lower limit of the hole
- Least Material Conditions (LMC): No-GO gauge follows LMC
  - Lower limit of the shaft
  - Upper limit of the hole
- Taylor's First consideration:
  - "The maximum material limit of as many related dimensions as possible should be incorporated in the GO gauge; whereas the minimum material limits of these dimension should be gauged by separate NOT GO gauges"
- Second consideration:
  - GO gauges should be of Full Form i.e. should check all related dimensions at a time (roundness, size, location)
  - NO-GO gauge should check only one dimension at a time

GO Limit and NO GO Limit

- GO Limit – Limit between the two size limit which corresponds to the Maximum Material Limit condition
- NO GO Limit – Limit between the two size limit which corresponds to the Minimum Material Limit condition

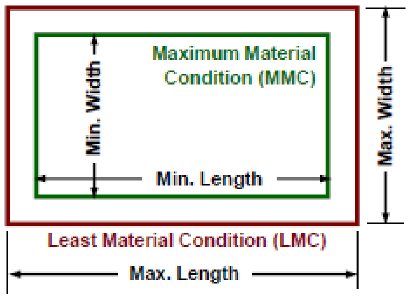


Taylor's Principle of Gauging

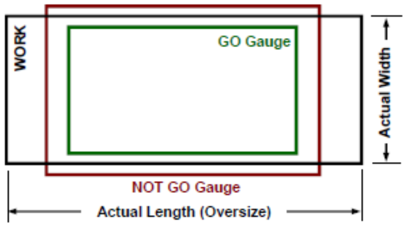
- The go gauge should be designed to check the form (roundness, size, location etc)
- The no-go gauge checks the individual sections
- Go gauge:
  - Corresponds to the Maximum Material Condition (MMC) of the Part.
    - Upper size limit for Shaft
    - Lower size limit for Hole
  - GO gauge is used to ensure that MMC is not exceeded.
  - GO gauge should be made to MMC based on wear and gauge tolerances
    - Plug gauge for maximum material dimension, i.e., smallest hole
    - Testing the function, i.e., shape and pairing dimension
- No Go gauge
  - For minimum material dimension, biggest hole
  - Testing the maximum dimension

Taylor's Theory of Gauging

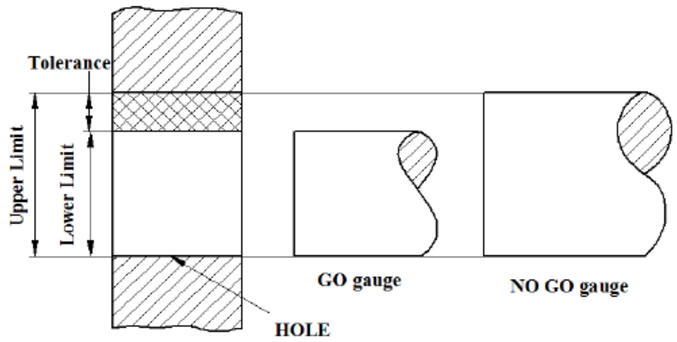
- GO gauge checks the Maximum Material Condition (MMC),
- Should check as many dimensions as possible.
- GO gauge is used to ensure that MMC is not exceeded.
- GO gauge should be made to MMC based on wear and gauge tolerances

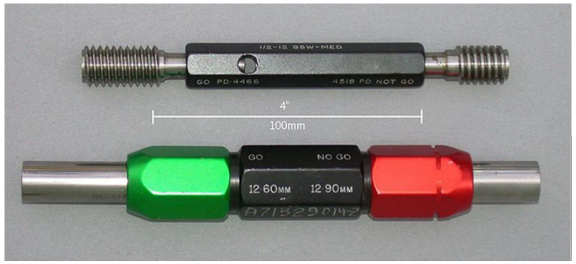


- NOT-GO gauge checks the Least Material Condition (LMC), and it should only check one dimension
- Separate NOT-GO gauge is required for each individual dimension

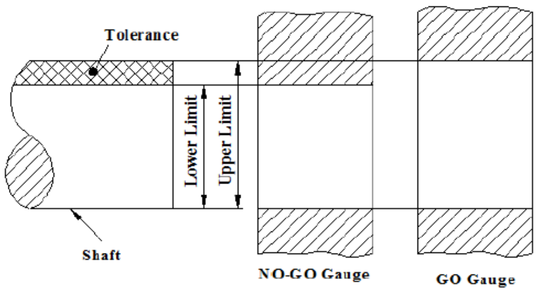


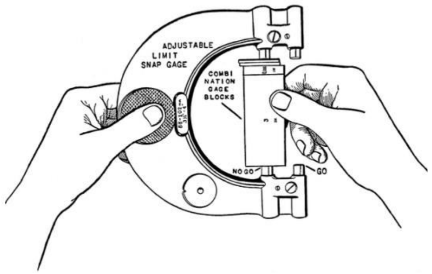
Plug Gauges : Go – NoGo





Ring Gauges : Go – NoGo

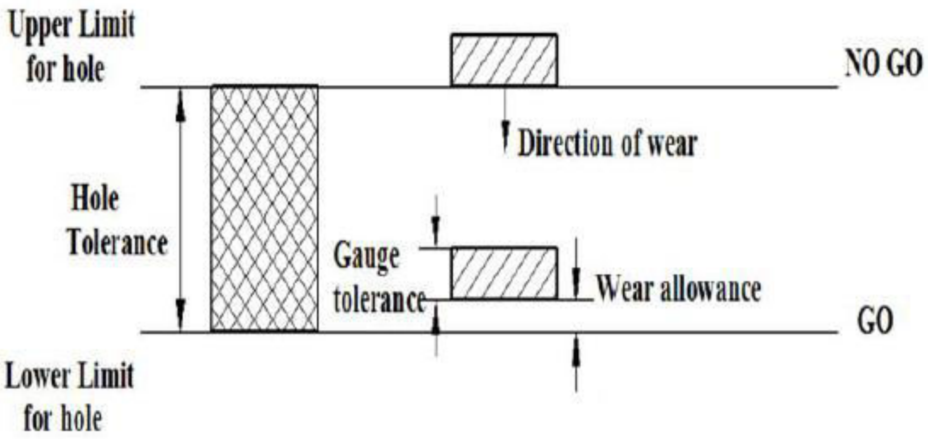




Gauge Tolerances

- Gauge Maker's allowance or Gauge Tolerance
  - Limit gauges are made nearly 10 times more accurate than the tolerances they supposed to inspect
  - Gauge tolerances are 1/10th of the work tolerances
- Wear Allowance
  - GO gauges encounter more rubbing with the surfaces , so wear allowance is applied as 5% of work tolerance or 10% of gauge tolerance
  - It is applied in the opposite direction of the wear i.e. in case of plug gauge, wear allowance is added and ring or snap gauge, it is subtracted

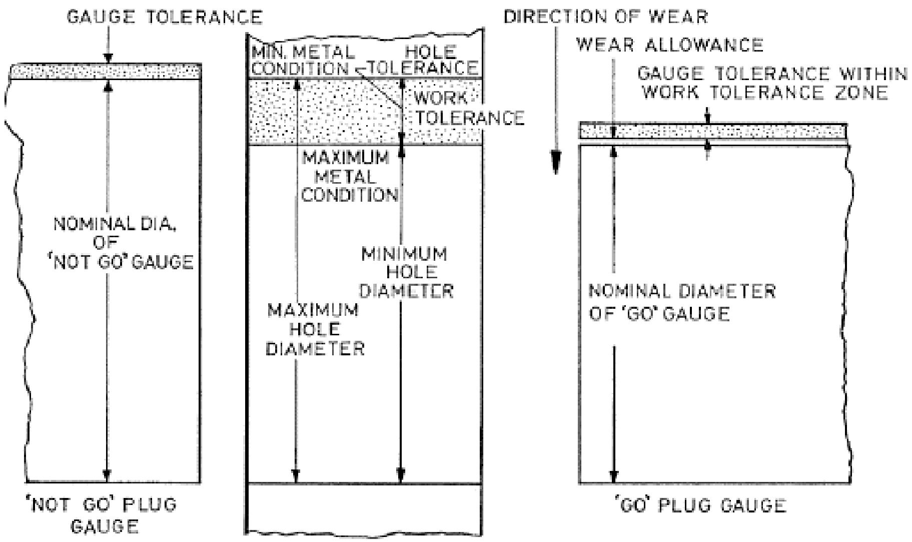
Gauge Tolerances



Tolerances in Limit Gauges

- For a hole to be checked:
  - GO gauge is a cylinder with diameter equal to the minimum hole size
  - NOT-GO gauge is a cylinder with diameter equal to the maximum hole size.
  - It is vice versa for a shaft to be checked.
- The gauge maker needs a tolerance to which gauges may work.
  - Defining the gauge tolerance relative to the nominal gauge size is critical.
- The tolerance on GO gauge shall be within the work tolerance zone.
- The tolerance on NOT GO gauge shall be outside the work tolerance zone
- Allowances must also be made for the initial wear which occurs on a new gauge.
- Thus, the tolerances shall be:
  - 1) Gauge tolerance = 10% of work tolerance
  - 2) Wear allowance = 5% of work tolerance

Tolerances in Limit Gauges

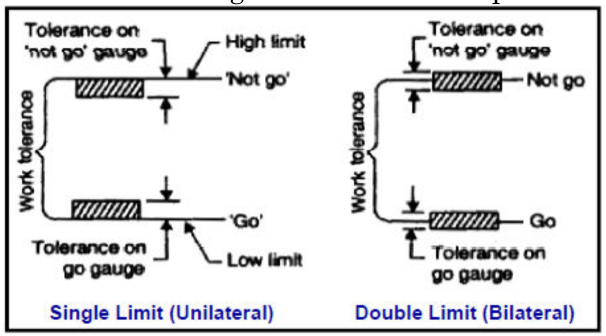


No-Go plug : only gauge tolerance

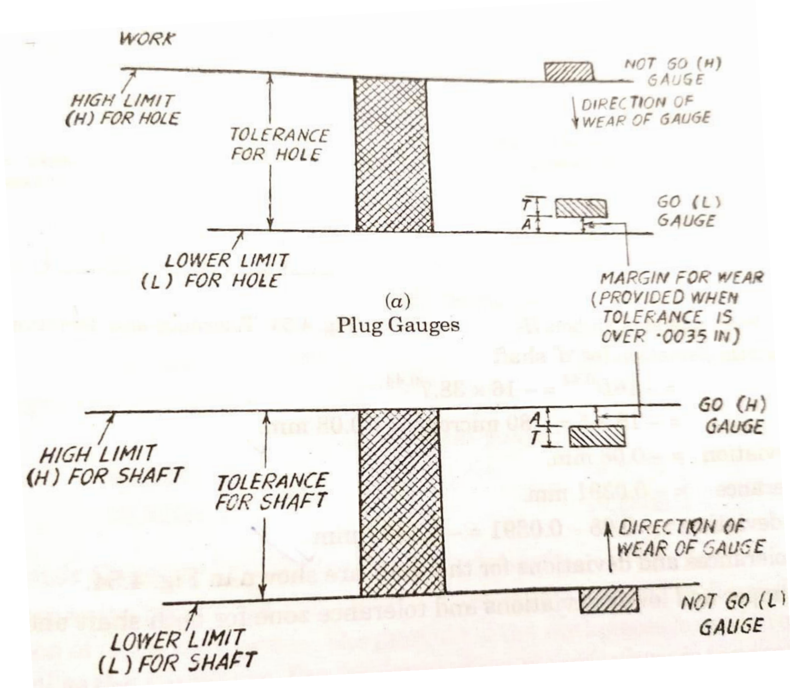
Go plug : gauge tolerance + wear tolerance

Tolerancing System and Wear Allowance

- Two systems for gauge tolerances: unilateral and bilateral
- Unilateral system (preferred method): the gauge tolerance zones lie entirely within the work tolerance zone.
  - Disadvantage: Certain parts may be rejected as if they were outside the limits
- Bilateral system: Gauge tolerance zones are bisected by high and low limits of work tolerance zone
  - Disadvantage: Parts within the working limits can be rejected and parts outside the working limits can be accepted.

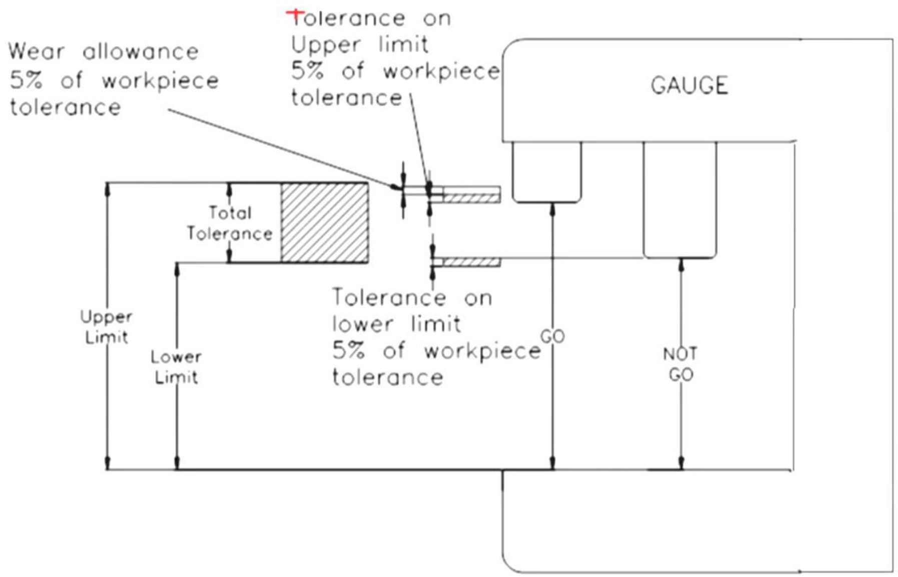


Tolerance in Gauge design



(a)
Plug Gauges

Snap gauge



Example: Gauge design

Example. *Calculate the dimensions of plug and ring gauges to control the production of 50 mm shaft and hole pair of H_7d_8 as per I.S. specification. The following assumptions may be made : 50 mm lies in diameter step of 30 and 50 mm and the upper deviation for 'd' shaft is given by — $-16D^{0.44}$ and lower deviation for hole H is zero. Tolerance factor i (microns) $= 0.45\sqrt[3]{D} + 0.001D$ and IT6 = 10i and above IT6 grade the tolerance magnitude is multiplied by 10 at each fifth step.*

Solution. For calculation of tolerance, value of diameter is taken as the mean of range in which it lies.

$$D = \sqrt{30 \times 50} = \sqrt{1500} = 10\sqrt{15} = 38.68 \text{ mm} \simeq 38.7 \text{ mm}.$$

Value of tolerance unit $i = 0.45\sqrt[3]{D} + 0.001D$

$$= 0.45\sqrt[3]{38.7} + 0.001 \times 38.7 = 0.45(3.38) + 0.0387$$
$$= 1.521 + 0.0387 = 1.5597 \text{ microns} \simeq 0.00156 \text{ mm}.$$

Now hole is of type H and grade 7

$$IT7 = IT6 \times 10^{0.2} = 10i \times 10^{0.2} = 10i \times 1.585 = 15.85i$$

and for H hole, fundamental deviation = 0

and value of tolerance = 15.85 × 0.00156 = 0.02475 mm.

∴ For hole $H7$ disposition of work tolerance will be as shown in Fig. 4.53.

For shaft d_8

Tolerance $= IT8 = 10^{0.2} \times IT7$
$= 1.585 \times 15.85i = 25i$ (approx.)
$= 25 \times 0.00156 = 0.0391$ mm.

Example: Gauge design

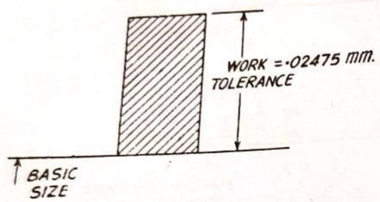


Fig. 4.53. Work tolerance for hole H_7.

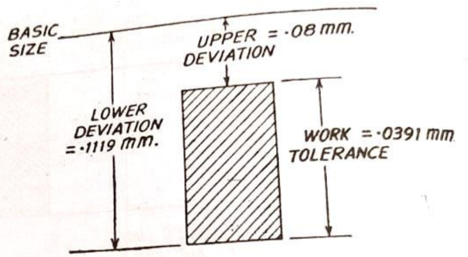


Fig. 4.54. Tolerance and deviations for shaft.

Fundamental deviation for 'd' shaft

$$= -16D^{0.44} = -16 \times 38.7^{0.44}$$
$$= -16 \times 5 = -80 \text{ microns} = -0.08 \text{ mm}.$$

Upper deviation $= -0.08$ mm.

and Work tolerance $= -0.0391$ mm.

∴ Lower deviation $= -0.08 - 0.0391 = -0.1191$ mm.

Example: Gauge design

∴ Effect of wear in go gauges is not ...

Work tolerance = 0.02475 mm. (FOR HOLE)

∴ Gauge tolerance = 0.002475 mm.

∴ Limits for Go' gauges are $\left.\begin{matrix} 50.000 \\ 50.0025 \end{matrix}\right\}$ mm.

and for 'No Go' gauge, dimensions are

$$\left.\begin{matrix} 50 + 0.0248 = 50.0248 \\ 50.0248 + 0.0025 = 50.0273 \end{matrix}\right\} \text{mm.}$$

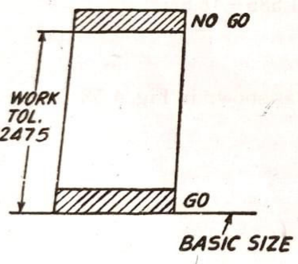


Fig. 4.55. Limits for plug gauge.

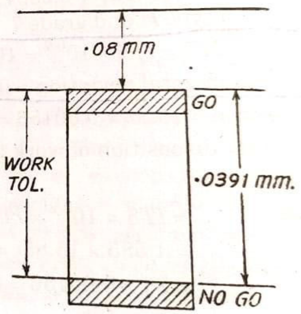


Fig. 4.56. Limits for ring gauge.

Example: Gauge design

(2) *Ring gauges.* The various dimensions of Gap gauges are shown, in Fig. 4.56 and calculations made in a similar manner as for plug gauges.

Dimensions for 'Go' gauge are

$$50 - 0.08 = 42.92 \text{ mm.}$$

and

$$49.92 - 0.0039 = 49.9161 \text{ mm.}$$

Dimensions for 'No Go' gauge are

$$50 - (0.38 + 0.391) = 49.8809 \text{ mm}$$

and

$$49.8809 - 0.0039 = 49.8770 \text{ mm.}$$

Answers : Plug Gauge. Dimensions for

'Go' Gauge : $\left.\begin{matrix} 50.000 \\ 50.0025 \end{matrix}\right\}$ mm.

'No Go' gauge : $\left.\begin{matrix} 50.0248 \\ 50.0273 \end{matrix}\right\}$ mm.

Ring Gauge : Dimensions for

'Go' gauge : $\left.\begin{matrix} 49.9161 \\ 49.9200 \end{matrix}\right\}$ mm.

'No Go' gauge : $\left.\begin{matrix} 49.8770 \\ 49.8809 \end{matrix}\right\}$ mm.

Case Study on Gauge Design

- Suppose that size of the hole to be tested is: 25 ± 0.02 mm
  - Highest limit of hole = 25.02 mm
  - Lowest limit of hole = 24.98 mm
  - Work tolerance = 25.02 - 24.98 = 0.04 mm
  - Gauge tolerance = 10% of work tolerance = 0.00
  - Wear allowance = 5% of work tolerance = 0.002
  - Nominal size of GO plug gauge = 24.98 + 0.002 = 24.982 mm
  - Nominal size of NOT-GO plug gauge = 25.02 mm

Dimension of GO plug gauge: $24.982^{+0.004}_{-0.000}$

Dimension of NOT-GO plug gauge: $25.02^{+0.000}_{-0.004}$

- If bilateral system was used, the gauge dimensions would be as follows:

Dimension of GO plug gauge: $24.982^{+0.002}_{-0.002}$

Dimension of NOT-GO plug gauge: $25.02^{+0.002}_{-0.002}$