Chapter 2

SYSTEM OF LIMITS, FITS, TOLERANCE AND GAUGING

Limits & Fits: Why study Limits & Fits?

- Exact size is impossible to achieve.
- Establish boundaries within which deviation from perfect form is allowed but still the design intent is fulfilled.
- Enable interchangeability of components during assembly

Definition of Limits:

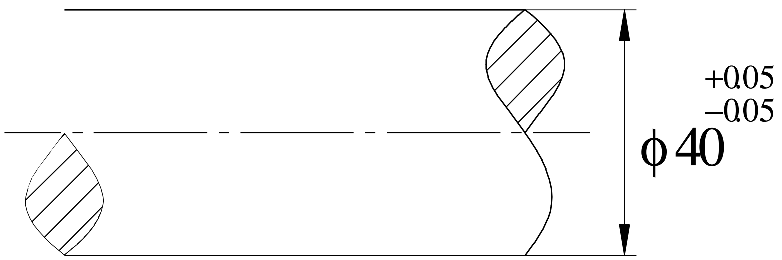
The maximum and minimum permissible sizes within which the actual size of a component lies are called Limits.

Tolerance:

It is impossible to make anything to an exact size, therefore it is essential to allow a definite tolerance or permissible variation on every specified dimension.

Why Tolerances are specified?

- Variations in properties of the material being machined introduce errors.
- The production machines themselves may have some inherent inaccuracies.
- It is impossible for an operator to make perfect settings. While setting up the tools and workpiece on the machine, some errors are likely to creep in.



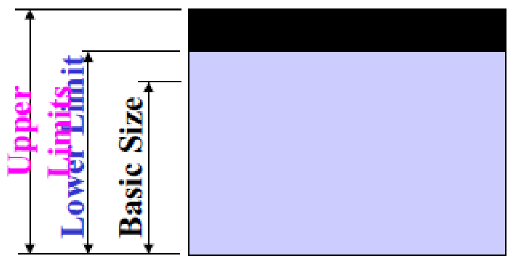
Consider the dimension shown in fig. When trying to achieve a diameter of 40 mm (Basic or Nominal diameter), a variation of 0.05 mm on either side may result.

If the shaft is satisfactory even if its diameter lies between 40.05 mm & 39.95 mm, the dimension 40.05 mm is known as Upper limit and the dimension 39.95 mm is known as Lower limit of size. Tolerance in the above example is (40.05-39.95) =0.10 mm

Tolerance is always a positive quantitative number.

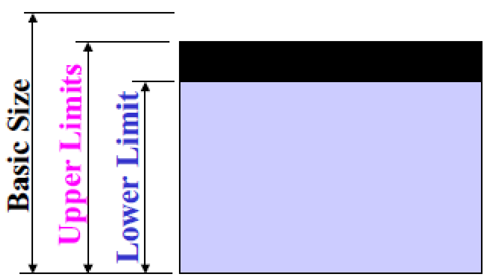
Unilateral Tolerance:

- Tolerances on a dimension may either be unilateral or bilateral.
- When the two limit dimensions are only on one side of the nominal size, (either above or below) the tolerances are said to be unilateral.
- For unilateral tolerances, a case may occur when one of the limits coincide with the basic size.



e.g. $\varnothing 25^{+0.18}_{+0.10}$

Basic Size = 25.00 mm
Upper Limit = 25.18 mm
Lower Limit = 25.10 mm
Tolerance = **0.08 mm**



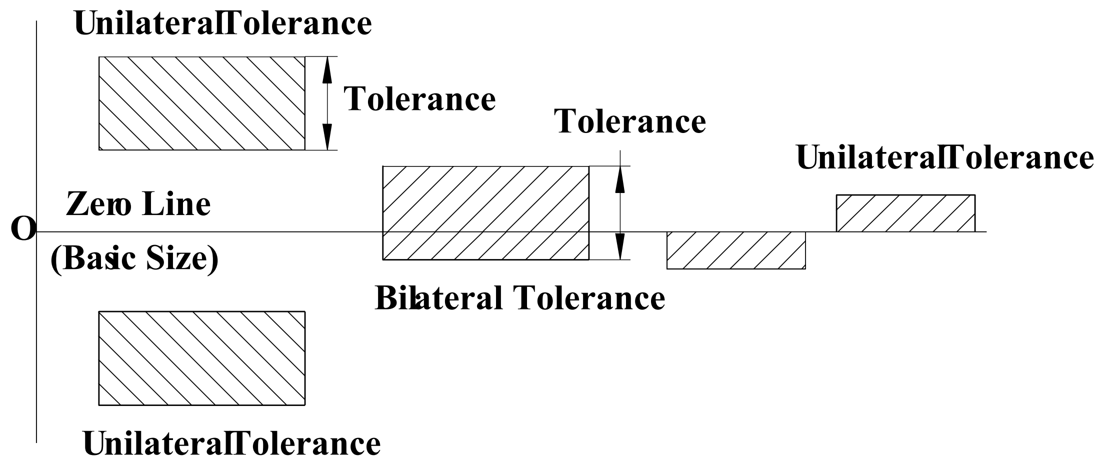
e.g. $\varnothing 25^{-0.10}_{-0.20}$

Basic Size = 25.00 mm
Upper Limit = 24.90 mm
Lower Limit = 24.80 mm
Tolerance = **0.10 mm**

Bilateral Tolerance: When the two limit dimensions are above and below nominal size, (i.e. on either side of the nominal size) the tolerances are said to be bilateral.

Unilateral tolerances, are preferred over bilateral because the operator can machine to the upper limit of the shaft (or lower limit of a hole) still having the whole tolerance left for machining to avoid rejection of parts.

Schematic representation of tolerances:



Tolerance Accumulation (or) Tolerance Build up:

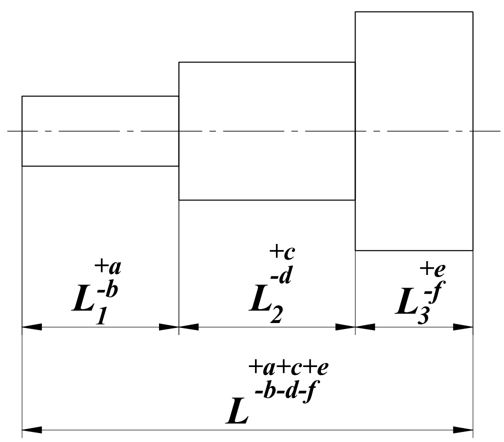
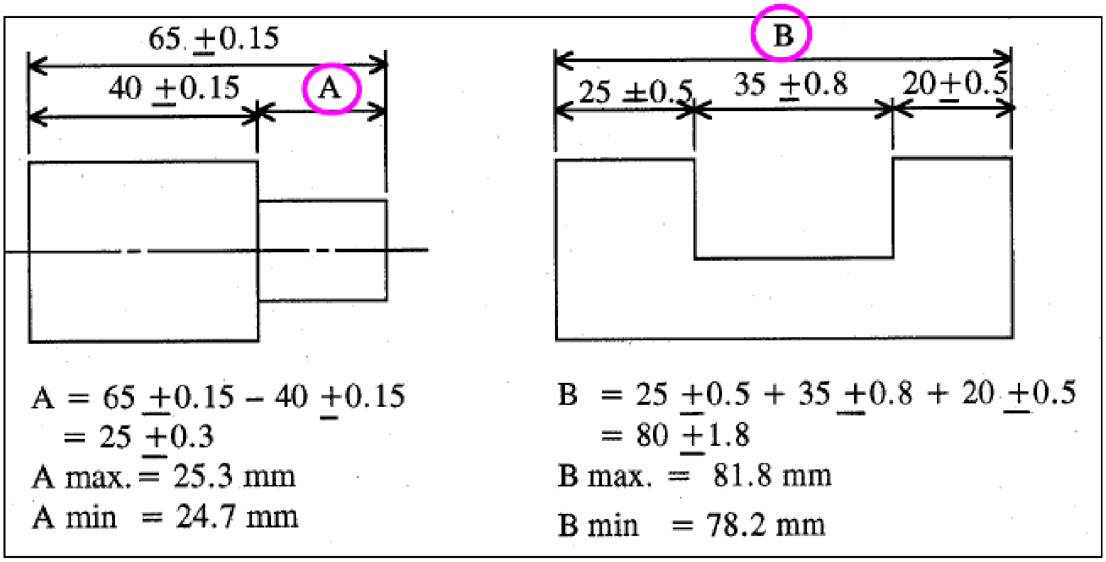


Fig (b)

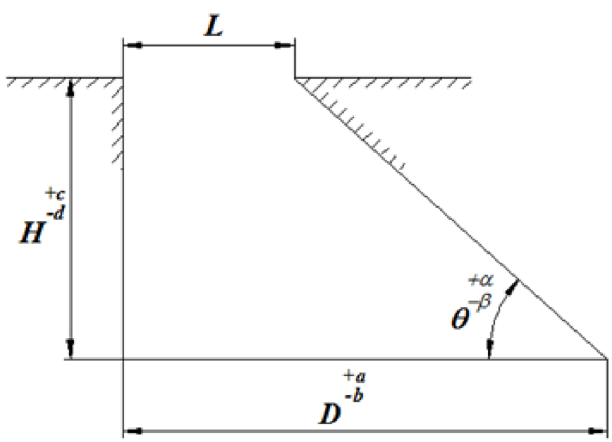
If a part comprises of several steps, each step having some tolerance specified over its length, then the overall tolerance on the complete length will be the sum of tolerances on individual lengths as shown in fig (a).

The effect of accumulation of tolerances can be minimized by adopting progressive dimensioning from a common datum as shown in fig (b).

Another example of tolerance build up is shown below.



Compound Tolerances: A compound tolerance is one which is derived by considering the effect of tolerances on more than one dimension.



For ex, the tolerance on the dimension L is dependent on the tolerances on D, H & θ.

The dimension L will be maximum when the base dimension is (D+a), the angle is (θ+a), and the vertical dimension is (H-d).

The dimension L will be minimum when the base dimension is (D-b), the angle is (θ-b), and the vertical dimension is (H+c).

LIMITS OF SIZE & TOLERANCE

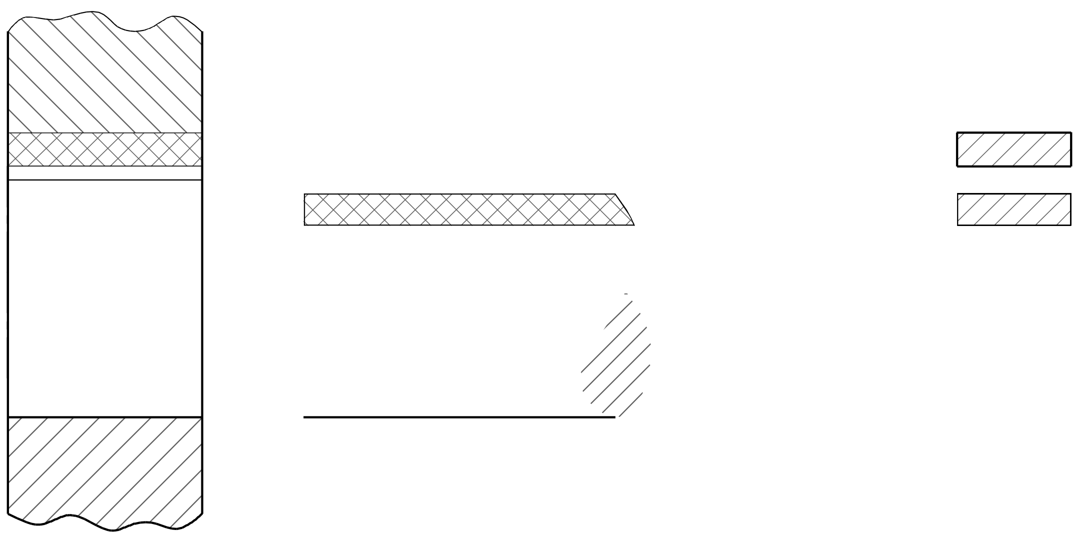
Terminology of limit systems:

Limits of size: The two extreme permissible sizes of a component between which the actual size should lie including the maximum and minimum sizes of the component.

Nominal size: It is the size of the component by which it is referred to as a matter of convenience.

Basic size: It is the size of a part in relation to which all limits of variation are determined.

Zero Line: It is the line w.r.t which the positions of tolerance zones are shown**.**



Deviation: It is the algebraic difference between a limit of size and the corresponding basic size.

Upper Deviation: It is the algebraic difference between the maximum limit of size and the corresponding basic size. It is denoted by letters ***'ES'*** for a hole and ***'es'*** for a shaft.

Lower Deviation: It is the algebraic difference between the minimum limit of size and the corresponding basic size. It is denoted by letters ***'EI'*** for a hole and ***'ei'*** for a shaft.

Fundamental Deviation: It is the deviation, either upper or lower deviation, which is nearest to the zero line for either a hole or a shaft. It fixes the position of the tolerance zone in relation to the zero line.

Allowance: It is the intentional difference between the hole dimensions and shaft dimension for any type of fit.

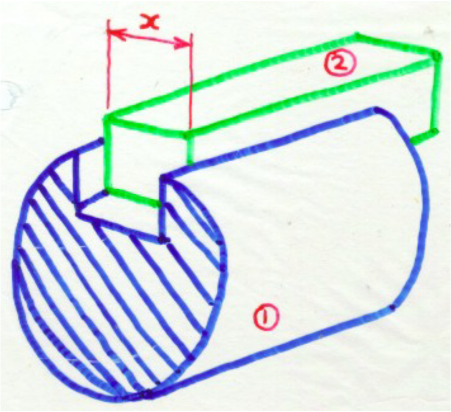
Size of tolerance: It is the difference between the maximum and minimum limits of size.

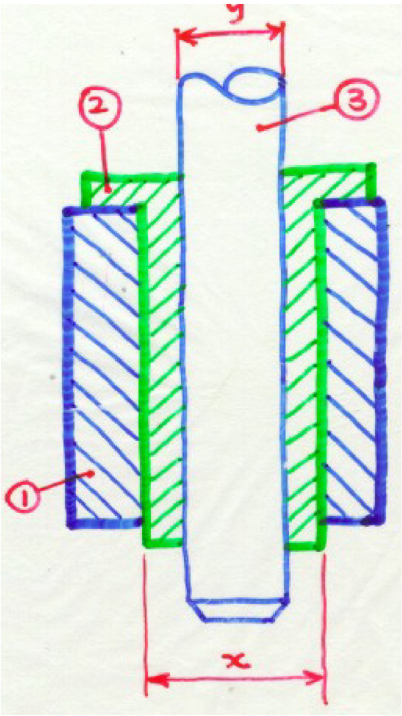
SYSTEM OF FITS

Fit is an assembly condition between 'Hole' & 'Shaft'

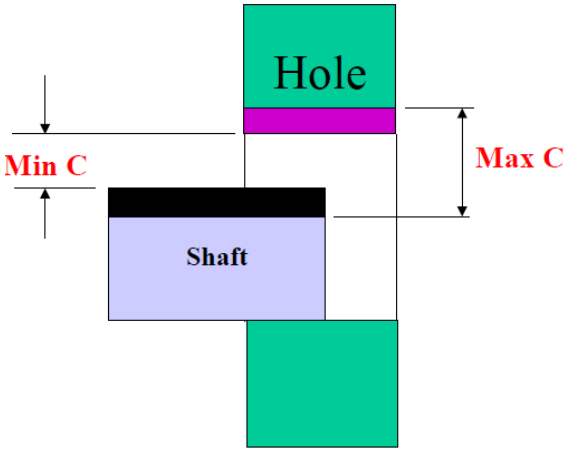
Hole: A feature engulfing a component.

Shaft: A feature being engulfed by a component.





Clearance fit: In this type of fit, the largest permitted shaft diameter is less than the smallest hole diameter so that the shaft can rotate or slide according to the purpose of the assembly.



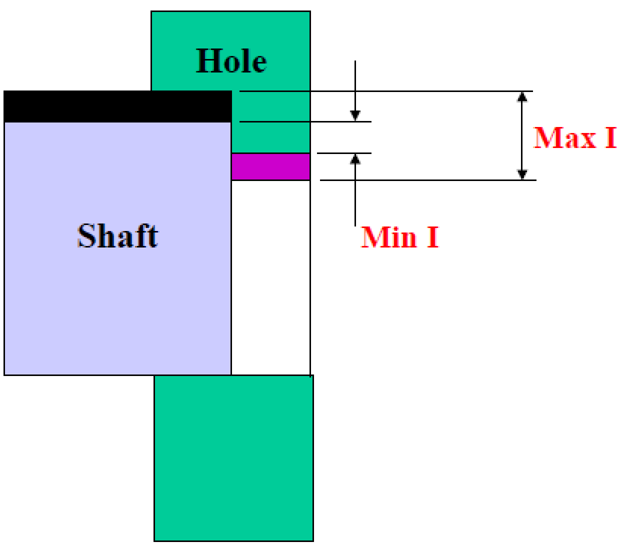
Tolerance zones never meet

Max. C = UL of hole - LL of shaft
Min. C = LL of hole - UL of shaft

Interference Fit:

It is defined as the fit established when a negative clearance exists between the sizes of holes and the shaft. In this type of fit, the minimum permitted diameter of the shaft is larger than the maximum allowable diameter of the hole. In case of this type of fit, the members are intended to be permanently attached.

Ex: Bearing bushes, Keys & key ways

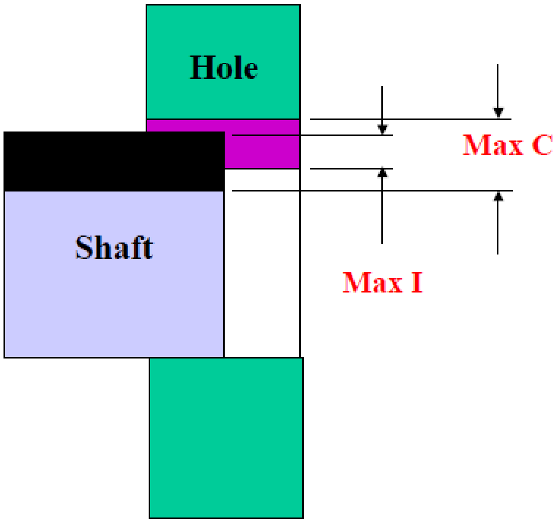


Tolerance zones never meet but crosses each other

Max. I = LL of hole - UL of shaft
Min. I = UL of hole - LL of shaft

Transition Fit: In this type of fit, the diameter of the largest allowable hole is greater than the smallest shaft, but the smallest hole is smaller than the largest shaft, such that a small positive or negative clearance exists between the shaft & hole.

Ex: Coupling rings, Spigot in mating holes, etc.



Tolerance zones always overlap

Max. C = UL of hole - LL of shaft
Max. I = LL of hole - UL of shaft

Interchangeability:

Interchangeability occurs when one part in an assembly can be substituted for a similar part which has been made to the same drawing. Interchangeability is possible only when certain standards are strictly followed.

Universal interchangeability means the parts to be assembled are from two different manufacturing sources.

Local interchangeability means all the parts to be assembled are made in the same manufacturing unit.

Selective Assembly:

In selective assembly, the parts are graded according to the size and only matched grades of mating parts are assembled. This technique is most suitable where close fit of two components assembled is required.

Selective assembly provides complete protection against non-conforming assemblies and reduces machining costs as close tolerances can be maintained.

Suppose some parts (shafts & holes) are manufactured to a tolerance of 0.01 mm, then an automatic gauge can separate them into ten different groups of 0.001 mm limit for selective assembly of the individual parts. Thus high quality and low cost can be achieved.

Selective assembly is used in aircraft, automobile industries where tolerances are very narrow and not possible to manufacture at reasonable costs.

Geometrical Tolerances:

It is necessary to specify and control the geometric features of a component, such as straightness, flatness, roundness, etc. in addition to linear dimensions. Geometric tolerance is concerned with the accuracy of relationship of one component to another and should be specified separately.

Geometrical tolerance may be defined as the maximum possible variation of ***form***, or ***position of form*** or ***position of a feature.***

Geometric tolerances define the shape of a feature as opposed to its size. There are three basic types of geometric tolerances:

Form tolerances:

Straightness, flatness, roundness, cylindricity

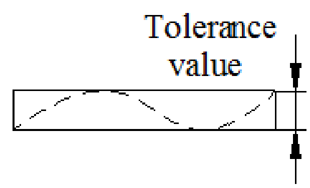
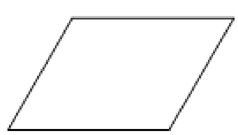
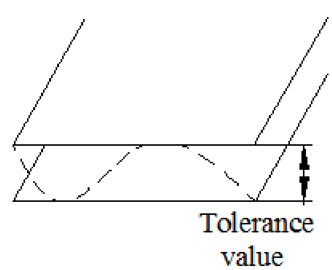
Orientation tolerances:

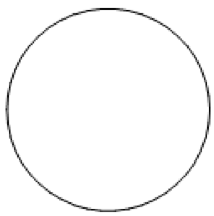
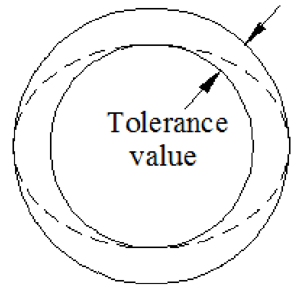
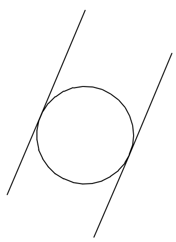
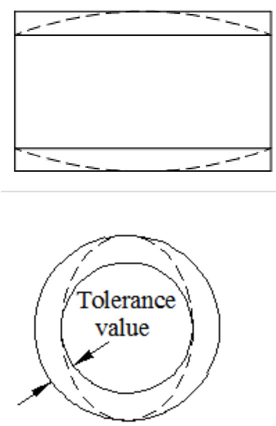
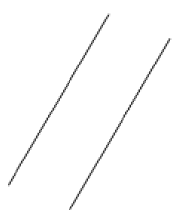
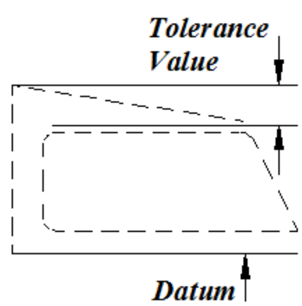
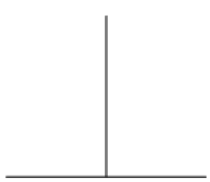
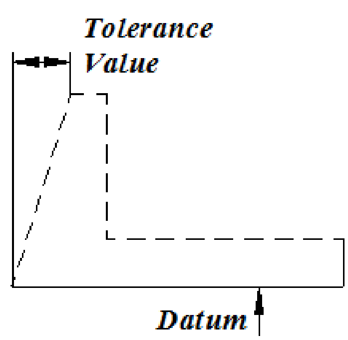
Perpendicularity, parallelism, angularity

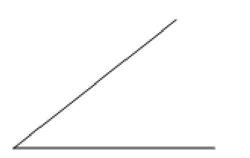
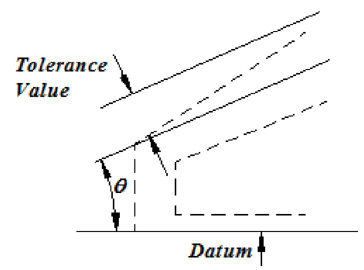
Position tolerances:

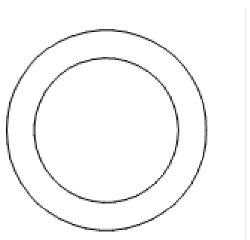
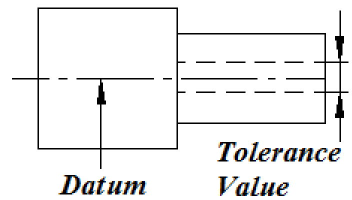
Position, symmetry, concentricity

FORM TOLERANCES

Characteristic or symbol	Function of geometric tolerance	Tolerance zone	Typical example
Straightness	To control the straightness of the line on a surface.	Area between two parallel straight lines in the plane containing the considered line or axis, Tolerance value is the distance between them.	
Flatness 	To control the flatness of a surface.	Area between two planes. Tolerance value is the distance between them.	

Roundness 	To control the errors of roundness of a circle in the plane in which it lies.	Area between two concentric circles. Tolerance value is the radial distance between them.	
Cylindricity 	To control combination of roundness, straightness, and parallelism of a cylindrical surface.	Annular space between two cylinders that are co axial. Tolerance value is the radial distance between them.	
ORIENTATION TOLERANCES			
Parallelism 	To control the parallelism of a line or surface w.r.t some datum.	Area between two parallel lines or space between two parallel lines which are parallel to the datum	
Squareness 	To control the perpendicularity of a line or surface w.r.t a datum.	Area between two parallel lines or space between two parallel lines which are perpendicular to the datum.	

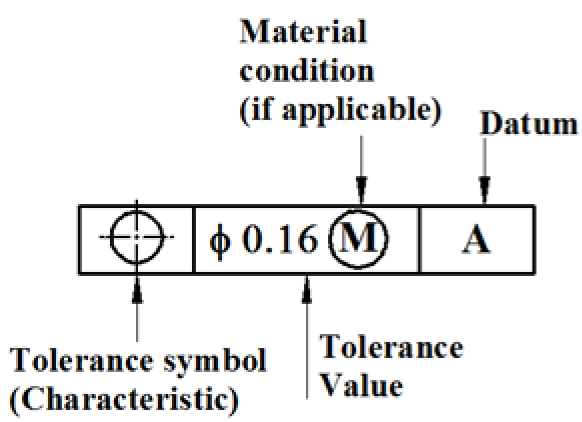
Angularity 	To control the inclination of a line or surface w.r.t a datum.	Area between two parallel lines or space between two parallel lines which are inclined at a specified angle to the datum.	

POSITIONAL TOLERANCES			
Concentricity 	To control the deviation of the position of the position of the center or axis of the toleranced circles or cylinders.	Center or axis to lie within the circle or cylinder. Tolerance value is the diameter of such a circle or cylinder.	

Feature Control Frame:

A geometric tolerance is prescribed using a feature control frame. It has three components:

- The tolerance symbol,
- The tolerance value,
- The datum labels for the reference frame.



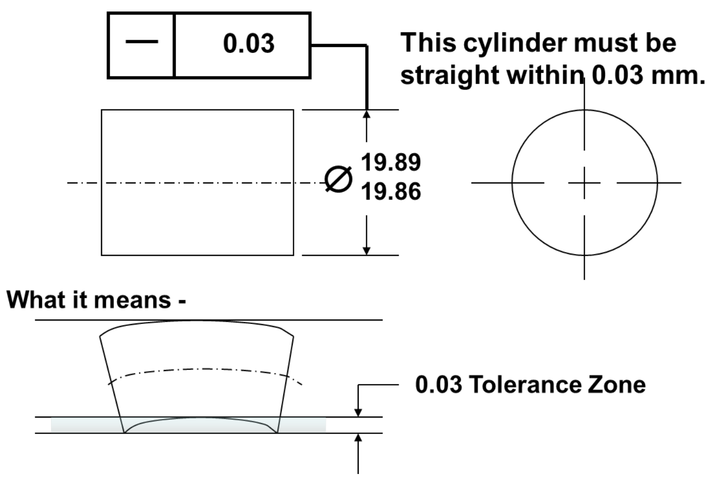
Material Conditions:

Maximum Material Condition (MMC): The condition in which a feature contains the maximum amount of material within the stated limits. e.g. minimum hole diameter, maximum shaft diameter.

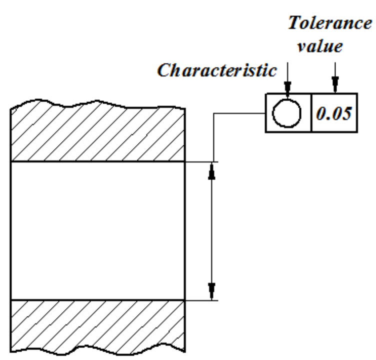
Least Material Condition (LMC): The condition in which a feature contains the least amount of material within the stated limits. e.g. maximum hole diameter, minimum shaft diameter.

Regardless of Feature Size (RFS): This is the default condition for all geometric tolerances.

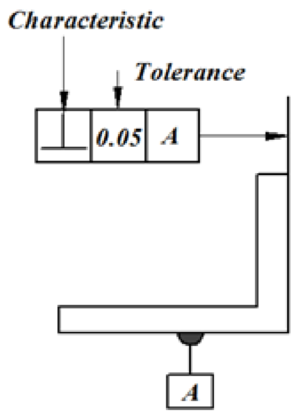
***Example*: STRAIGHTNESS**



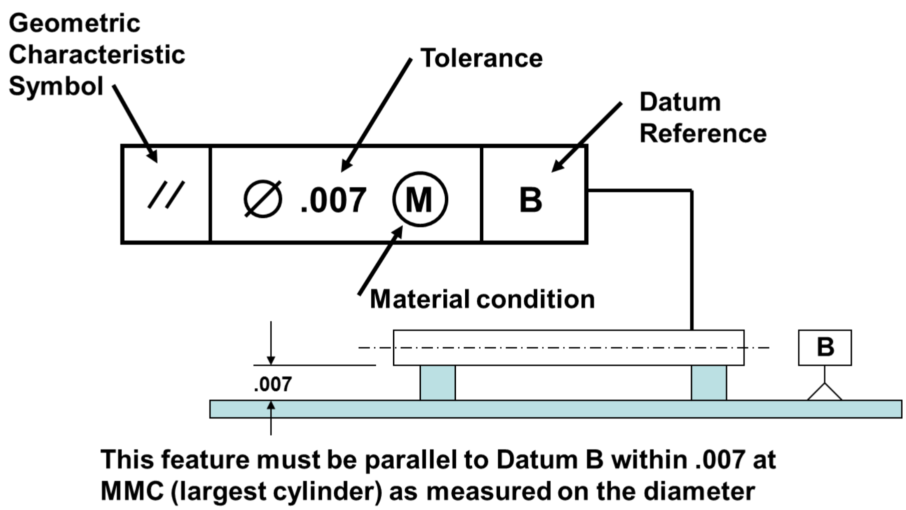
ROUNDNESS:



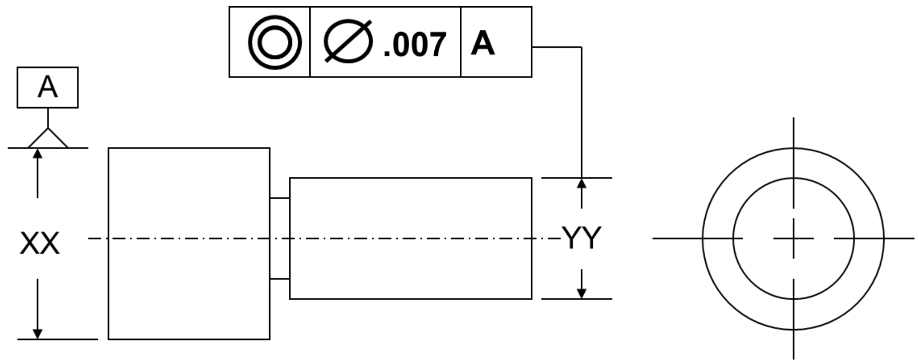
SQUARENESS:



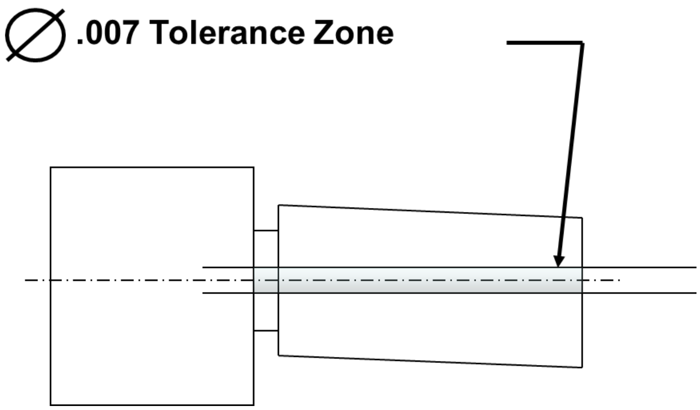
PARALLELISM:



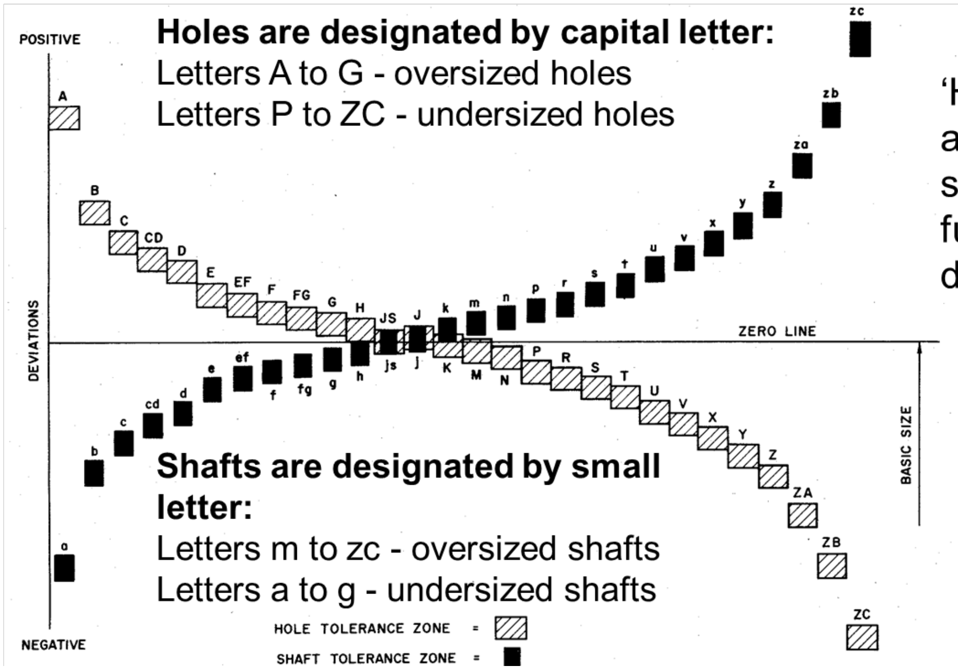
CONCENTRICITY:



This cylinder (the right cylinder) must be concentric within .007 with the Datum A (the left cylinder) as measured on the diameter



IS 919-1965 SYSTEM OF TOLERANCES



'H' is used for holes and 'h' is used for shafts whose fundamental deviation is zero.

Terms & symbols used:

Basic shaft: It is a shaft whose upper deviation is zero. i.e. the maximum limit of shaft coincides with the nominal size.(zero line). *Eg:* shaft 'h'

Basic hole: It is a hole whose lower deviation is zero. i.e. the minimum limit of hole coincides with the nominal size.(zero line). *Eg*: shaft 'H'

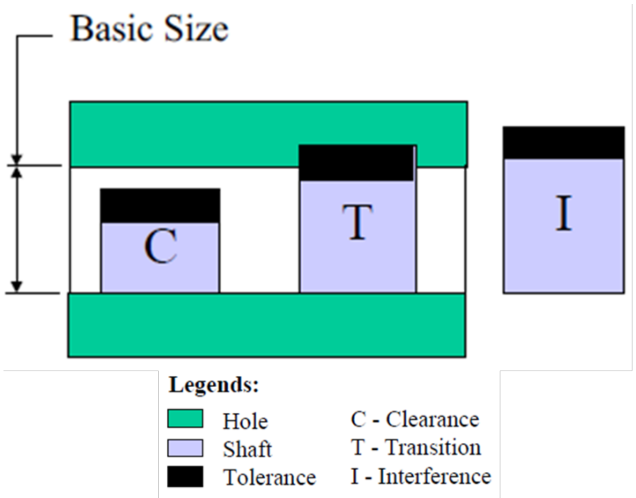
Basis of Fits

Hole Basis: In this system, the basic diameter of the hole is constant while the shaft size is varied according to the type of fit.

Significance of Hole basis system: The bureau of Indian Standards (BIS) recommends both hole basis and shaft basis systems, but their selection depends on the production methods. Generally, holes are produced by drilling, boring, reaming, broaching, etc. whereas shafts are either turned or ground.

If the shaft basis system is used to specify the limit dimensions to obtain various types of fits, number of holes of different sizes are required, which in turn requires tools of different sizes.

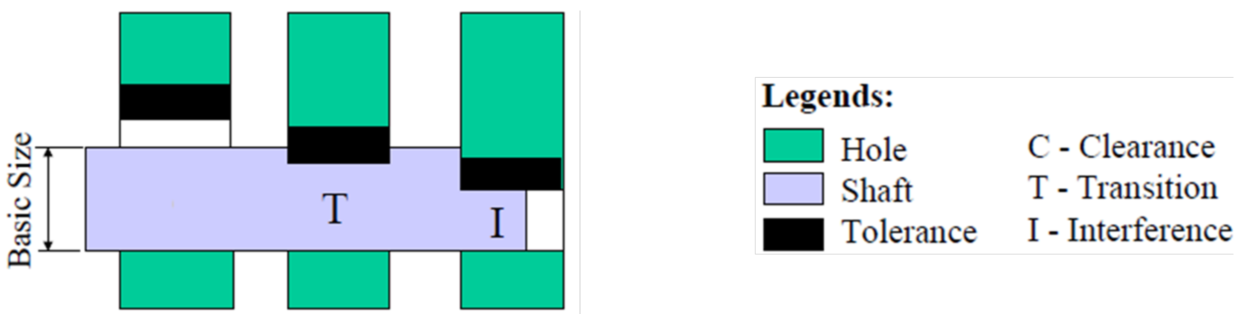
HOLE BASIS SYSTEM OF FITS



If the hole basis system is used, there will be reduction in production costs as only one tool is required to produce the ole and the shaft can be easily machined to any desired size. Hence hole basis system is preferred over shaft basis system.

Shaft Basis system:

In this system, the basic diameter of the shaft is constant while the hole size is varied according to the type of fit.



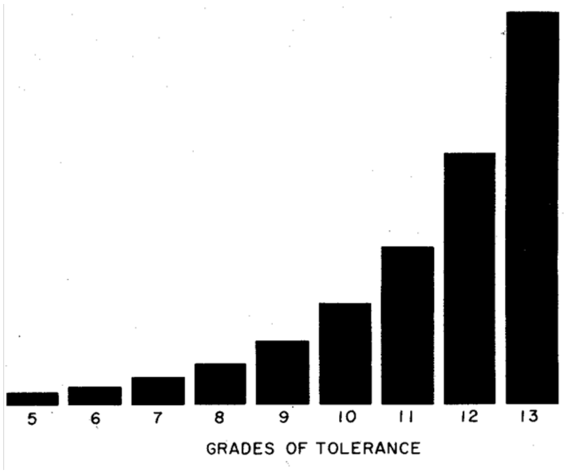
It may, however, be necessary to use shaft basis system where different fits are required along a long shaft.

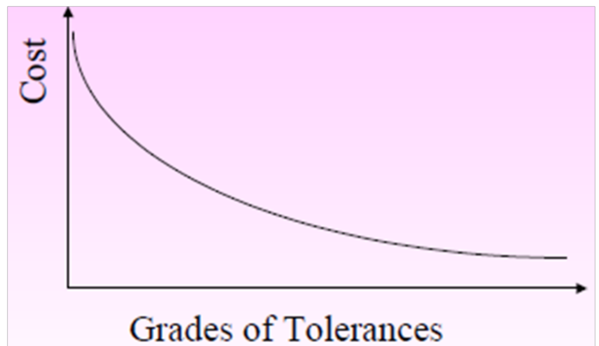
For example, in the case of driving shafts where a single shaft may have to accommodate to a variety of accessories such as couplings, bearings, collars, etc., it is preferable to maintain a constant diameter for the permanent member, which is the shaft, and vary the bore of the accessories.

GRADES OF TOLERANCES

Grade is a measure of the magnitude of the tolerance. Lower the grade the finer the tolerance. There are total of 18 grades which are allocated the numbers IT01, IT0, IT1, IT2..... IT16.

Fine grades are referred to by the first few numbers. As the numbers get larger, so the tolerance zone becomes progressively wider. Selection of grade should depend on the circumstances. As the grades get finer, the cost of production increases at a sharper rate.





TOLERANCE GRADE

The tolerance grades may be numerically determined in terms of the standard tolerance unit '***i***' where ***i*** in microns is given by (for basic size upto and including 500 mm) and (for basic size above 500 mm upto and including 3150 mm), where D is in mm and it is the geometric mean of the lower and upper diameters of a particular step in which the component lies.

The above formula is empirical and is based on the fact that the tolerance varies more or less parabolically in terms of diameter for the same manufacturing conditions. This is so because manufacture and measurement of higher sizes are relatively difficult.

The various diameter steps specified by ISI are:

1-3, 3-6, 6-10, 10-18, 18-30, 30-50, 50-80, 80-120,180-250, 250-315, 315-400, and 400-500 mm. The value of 'D' is taken as the geometric mean for a particular range of size to avoid continuous variation of tolerance with size.

The fundamental deviation of type d,e,f,g shafts are respectively $-16D^{0.44}$, $-11D^{0.41}$ $-5.5D^{0.41}$ & $-2.5D^{0.34}$

The fundamental deviation of type D,E,F,G shafts are respectively $+16D^{0.44}$, $+11D^{0.41}$ $+5.5D^{0.41}$ & $+2.5D^{0.34}$.

The relative magnitude of each grade is shown in the table below;

Tol. Grade	IT 5	IT 6	IT 7	IT 8	IT 9	IT 10	IT 11	IT 12	IT 13	IT 14	IT 15	IT 16
	7i	*10i*	*16i*	*25 i*	*40 i*	*64 i*	*100 i*	*160 i*	*250 i*	*400 i*	*640 i*	*1000 i*

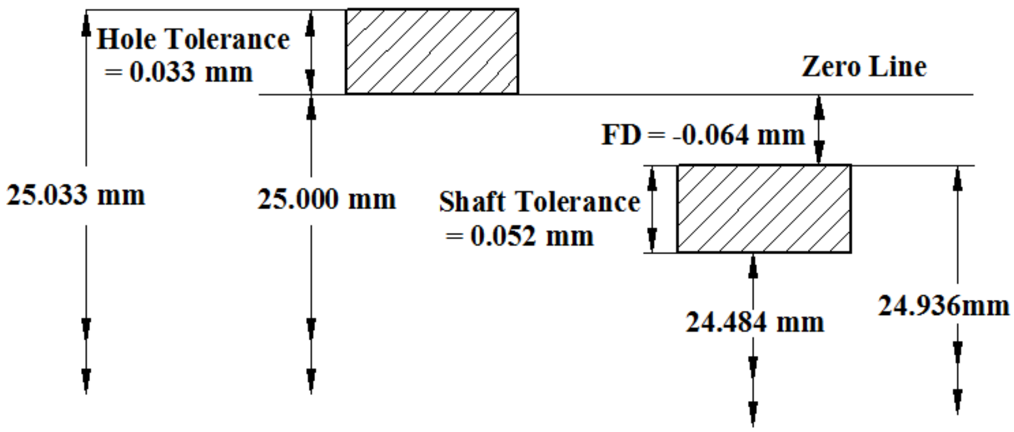
It may be noted that from IT 6 onwards, every 5th step is 10 times the respective grade.

i.e. IT 11=10xIT6=10x10i=100 i, IT12=10xIT7=10x16i=160 i, etc.

Numerical Problem 1:

Calculate the limits of tolerance and allowance for a 25 mm shaft and hole pair designated by H_8d_9. Take the fundamental deviation for'd' shaft is $-16D^{0.44}$.

Solution:



Numerical Problem 2

Determine the tolerances on the hole and the shaft for a precision running fit designated by 50 H7g6, given;

50 mm lies between 30-50 mm

i (in microns)=0.45(D)$^{1/3}$+0.001D

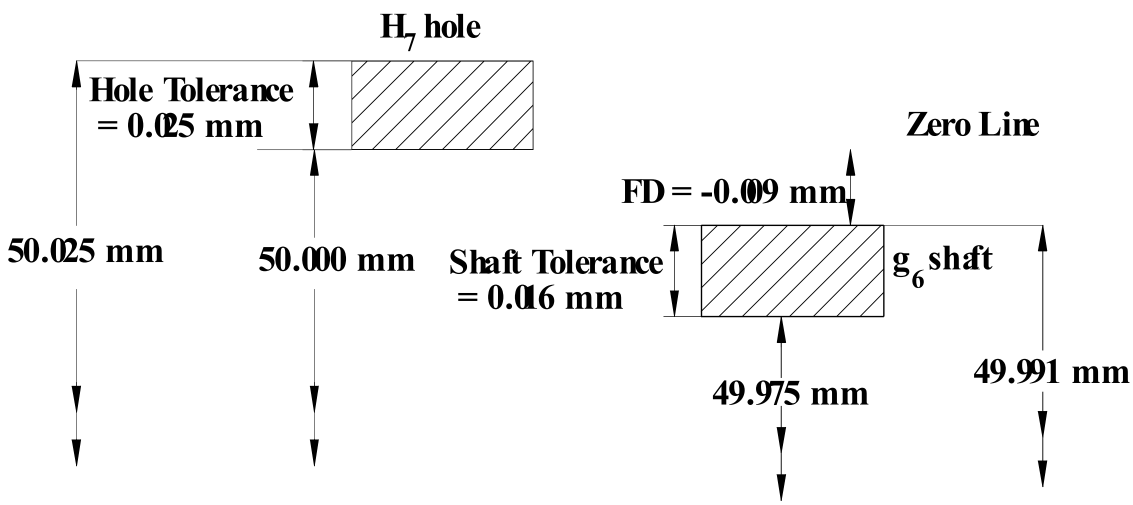
Fundamental deviation for 'H' hole=0

Fundamental deviation for g shaft =-2.5D$^{0.34}$

IT7=16i and IT6=10i

State the actual maximum and minimum sizes of the hole and shaft and maximum and minimum clearances.

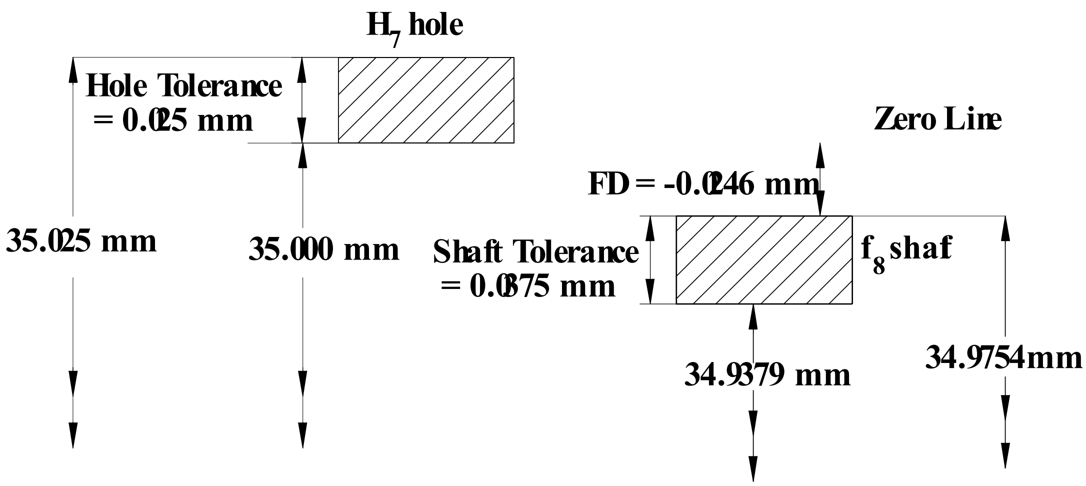
Solution:



Numerical Problem 3:

Calculate all the relevant dimensions of 35H7/f8 fit, dimension 35 mm falls in the step of 30-50 mm. The fundamental deviation for f shaft is – 5.5D0.41. i (in microns) =0.45(D)1/3+0.001D, IT7=16i and IT8=25i.

Solution:



LIMIT GAUGES

A ***Go-No GO*** gauge refers to an inspection tool used to check a workpiece against its allowed tolerances. It derives its name from its use: the gauge has two tests; the check involves the workpiece having to pass one test (Go) and fail the other (No Go).

It is an integral part of the quality process that is used in the manufacturing industry to ensure interchangeability of parts between processes, or even between different manufacturers.

A Go - No Go gauge is a measuring tool that does not return a size in the conventional sense, but instead returns a state. The state is either acceptable (the part is within tolerance and may be used) or it is unacceptable (and must be rejected).

They are well suited for use in the production area of the factory as they require little skill or interpretation to use effectively and have few, if any, moving parts to be damaged in the often hostile production environment.

PLAIN GAUGES

Gauges are inspection tools which serve to check the dimensions of the manufactured parts. Limit gauges ensure the size of the component lies within the specified limits. They are non-recording and do not determine the size of the part. Plain gauges are used for checking plain (Unthreaded) holes and shafts.

Plain gauges may be classified as follows;

According to their type:

(a) **Standard gauges** are made to the nominal size of the part to be tested and have the measuring member equal in size to the mean permissible dimension of the part to be checked. A standard gauge should mate with some snugness.

(b) ***Limit Gauges*** These are also called **'go'** and **'no go'** gauges. These are made to the limit sizes of the work to be measured. One of the sides or ends of the gauge is made to correspond to maximum and the other end to the minimum permissible size. The function of limit gauges is to determine whether the actual dimensions of the work are within or outside the specified limits.

According to their purpose:

(a) Work shop gauges: Working gauges are those used at the bench or machine in gauging the work as it being made.

(b) Inspection gauges: These gauges are used by the inspection personnel to inspect manufactured parts when finished.

(c) Reference or Master Gauges: These are used only for checking the size or condition of other gauges.

According to the form of tested surface:

Plug gauges: They check the dimensions of a hole

Snap & Ring gauges: They check the dimensions of a shaft.

According to their design:

Single limit & double limit gauges

Single ended and double ended gauges

Fixed & adjustable gauges

LIMIT GAUGING

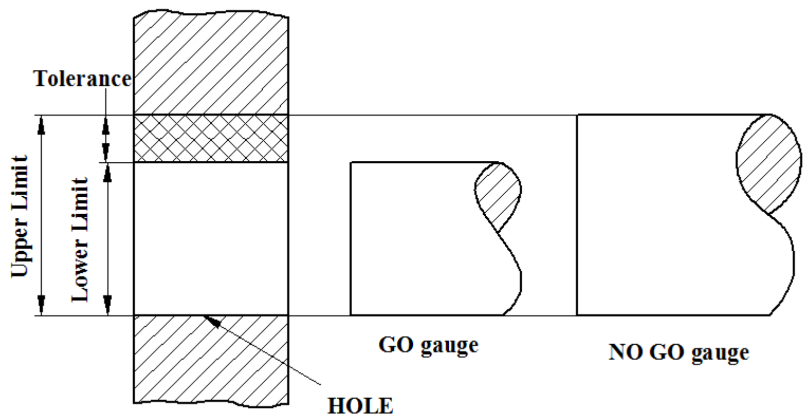
Limit gauging is adopted for checking parts produced by mass production. It has the advantage that they can be used by unskilled persons.

Instead of measuring actual dimensions, the conformance of product with tolerance specifications can be checked by a 'GO' and 'NO GO' gauges.

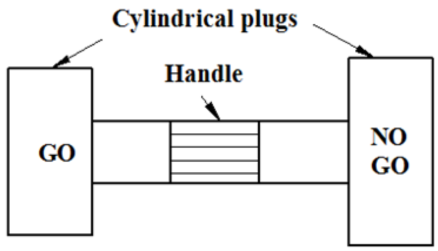
A 'GO' gauge represents the maximum material condition of the product (i.e. minimum hole size or maximum shaft size) and conversely a 'NO GO' represents the minimum material condition (i.e. maximum hole size or minimum shaft size)

Plug gauges:

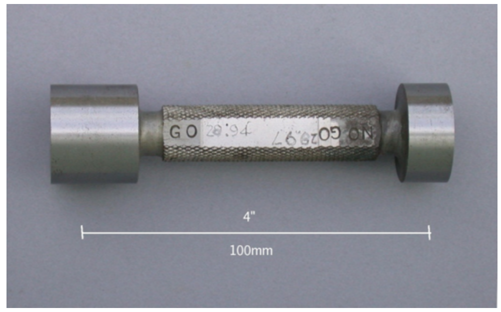
Plug gauges are the limit gauges used for checking holes and consist of two cylindrical wear resistant plugs. The plug made to the lower limit of the hole is known as 'GO' end and this will enter any hole which is not smaller than the lower limit allowed. The plug made to the upper limit of the hole is known as 'NO GO' end and this will not enter any hole which is smaller than the upper limit allowed. The plugs are arranged on either ends of a common handle.

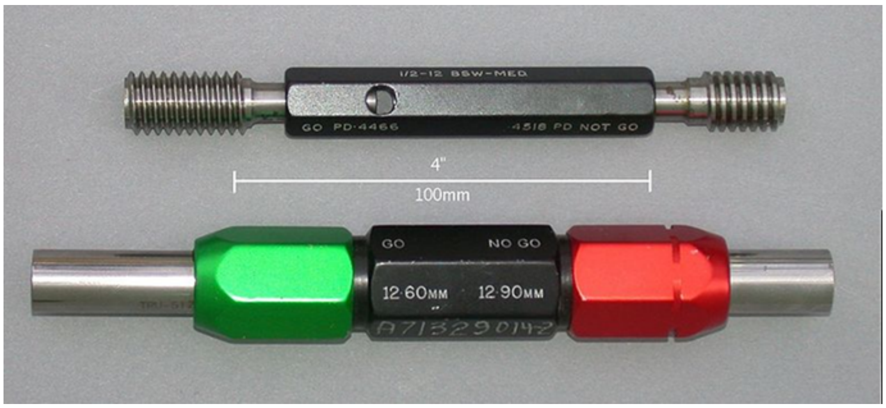


Plug gauges are normally double ended for sizes upto 63 mm and for sizes above 63 mm they are single ended type.



Double ended Plug Gauge

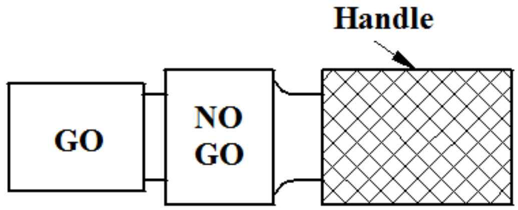




The handles of heavy plug gauges are made of light metal alloys while the handles of small plug gauges can be made of some nonmetallic materials.

Progressive plug gauges:

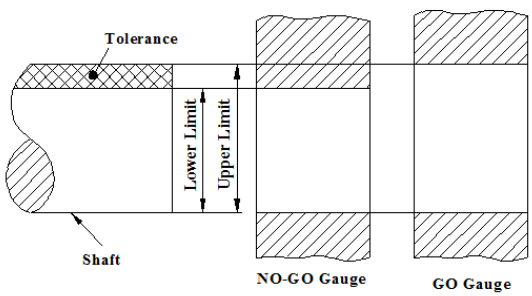
For smaller through holes, both GO & NO GO gauges are on the same side separated by a small distance. After the full length of GO portion enters the hole, further entry is obstructed by the NO GO portion if the hole is within the tolerance limits.



Progressive Plug Gauge

Ring gauges:

Ring gauges are used for gauging shafts. They are used in a similar manner to that of GO & NO GO plug gauges. A ring gauge consists of a piece of metal in which a hole of required size is bored.

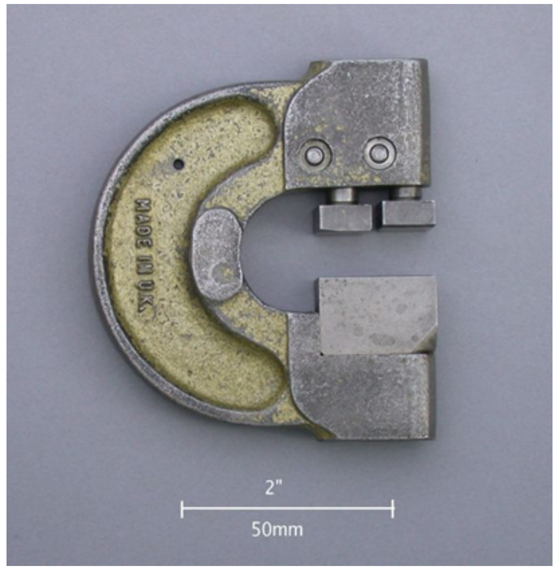


SNAP (or) GAP GAUGES:

A snap gauge usually consists of a plate or frame with a parallel faced gap of the required dimension. Snap gauges can be used for both cylindrical as well as non cylindrical work as compared to ring gauges which are conveniently used only for cylindrical work.

Double ended snap gauges can be used for sizes ranging from 3 to 100 mm.

For sizes above 100 mm upto 250 mm a single ended progressive gauge may be used.



Double Ended gap gauge Progressive gap gauge

Desirable properties of Gauge Materials:

The essential considerations in the selection of material of gauges are;

1. Hardness to resist wear.
2. Stability to preserve size and shape
3. Corrosion resistance
4. Machinability for obtaining the required degree of accuracy.
5. Low coefficient of friction of expansion to avoid temperature effects.

Materials used for gauges:

High carbon steel: Heat treated Cast steel (0.8-1% carbon) is commonly used for most gauges.

Mild Steel: Case hardened on the working surface. It is stable and easily machinable.

Case hardened steel: Used for small & medium sized gauges.

Chromium plated & Hard alloys: Chromium plating imparts hardness, resistance to abrasion & corrosion. Hard alloys of tungsten carbide may also be used.

Cast Iron: Used for bodies of frames of large gauges whose working surfaces are hard inserts of tool steel or cemented carbides.

Glass: They are free from corrosive effects due to perspiration from hands. Also they are not affected by temperature changes.

Invar: It is a nickel-iron alloy (36% nickel) which has low coefficient of expansion but not suitable for usage over long periods.

(The name, Invar, comes from the word invariable, referring to its lack of expansion or contraction with temperature changes. It was invented in 1896 by Swiss scientist Charles Eduard Guillaume. He received the Nobel Prize in Physics in 1920 for this discovery, which enabled improvements in scientific instruments.)

Taylor's Principle of Gauge Design:

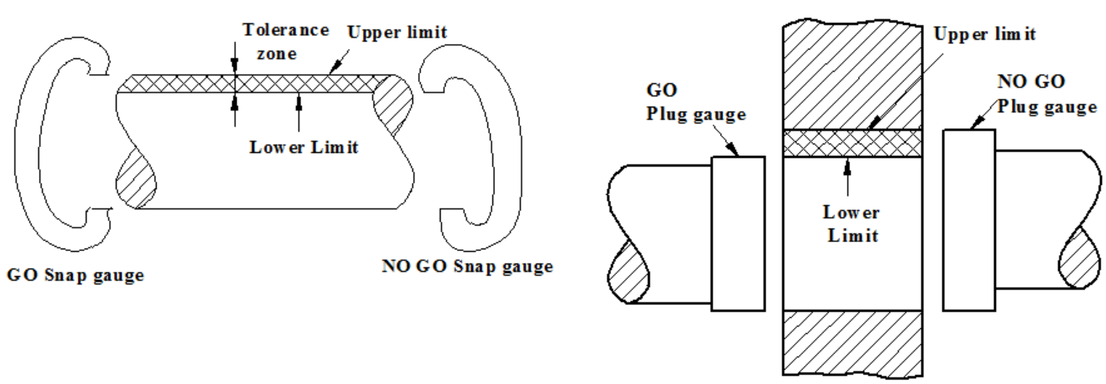
According to Taylor, 'Go' and 'No Go' gauges should be designed to check maximum and minimum material limits which are checked as below;

'GO' Limit. This designation is applied to that limit of the two limits of size which corresponds to the maximum material limit considerations, i.e. upper limit of a shaft and lower limit of a hole.

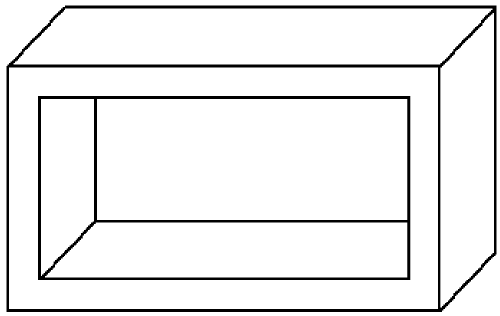
The GO gauges should be of full form, i.e. they should check shape as well as size.

No Go' Limit:

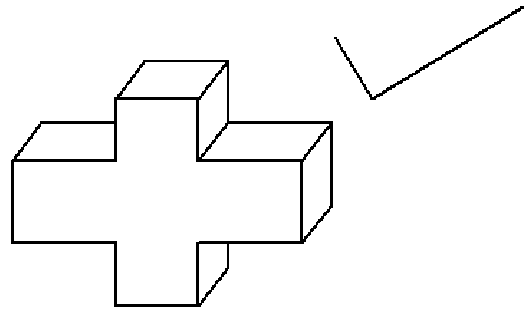
This designation is applied to that limit of the two limits of size which corresponds to the minimum material condition. i.e. the lower limit of a shaft and the upper limit of a hole. 'No Go' gauge should check only one part or feature of the component at a time, so that specific discrepancies in shape or size can be detected. Thus a separate 'No Go' gauge is required for each different individual dimension.



Example to illustrate Taylor's Principle of Gauge Design:



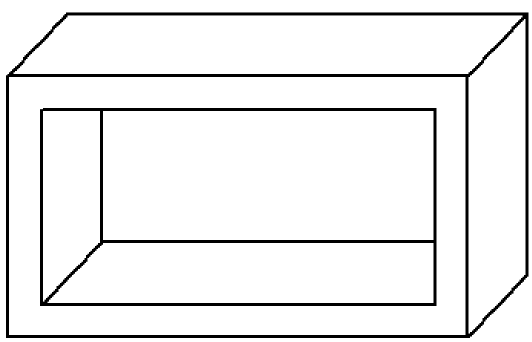
The slot is to be checked for height & Width



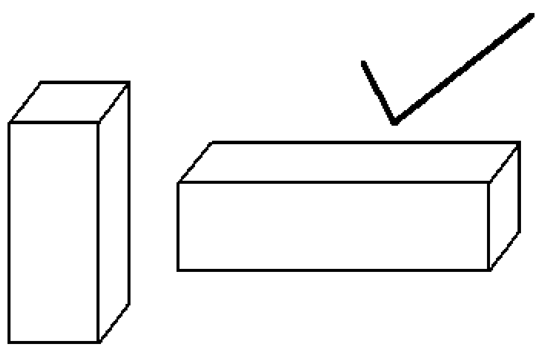
GO Gauge

A GO gauge must check the dimensions as well as form (perpendicularity) of the slot at a time. Hence the GO gauge must be as shown in fig on the right.

A NO GO gauge must check the dimensions of the slot one at a time and hence two separate gauges must be used.

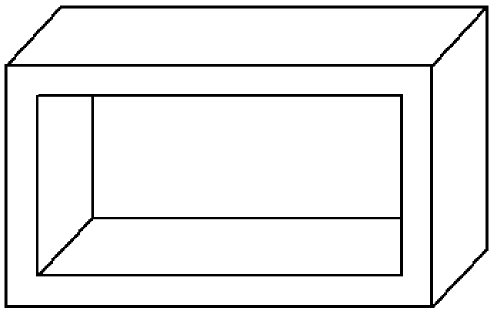


The slot is to be checked for height & Width

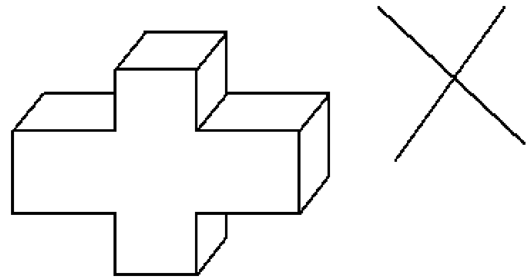


NO GO Gauge

If the single gauge as shown is used, the gage is likely to pass a component even if one of the dimensions is less than desirable limit because it gets stuck due to the other dimension which is within correct limit.



The slot is to be checked for height & Width



NO GO Gauge

Gauge Tolerance:

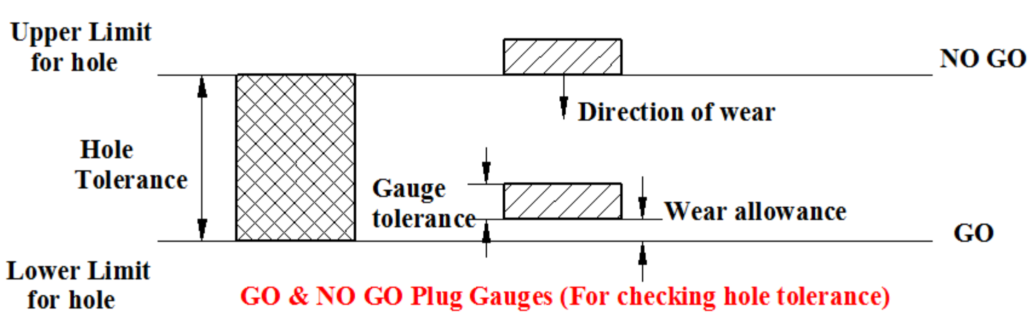
Gauges, like any other jobs require a manufacturing tolerance due to reasonable imperfections in the workmanship of the gauge maker. The gauge tolerance should be kept as minimum as possible though high costs are involved to do so. The tolerance on the GO & NO GO gauges is usually 10% of the work tolerance.

Wear Allowance:

The GO gauges only are subjected to wear due to rubbing against the parts during inspection and hence a provision has to be made for the wear allowance. Wear allowance is taken as 10% of gauge tolerance and is allowed between the tolerance zone of the gauge and the maximum material condition. *(i.e.* lower limit of a hole & upper limit of a shaft). If the work tolerance is less than 0.09 mm, wear allowance need not be given unless otherwise stated.

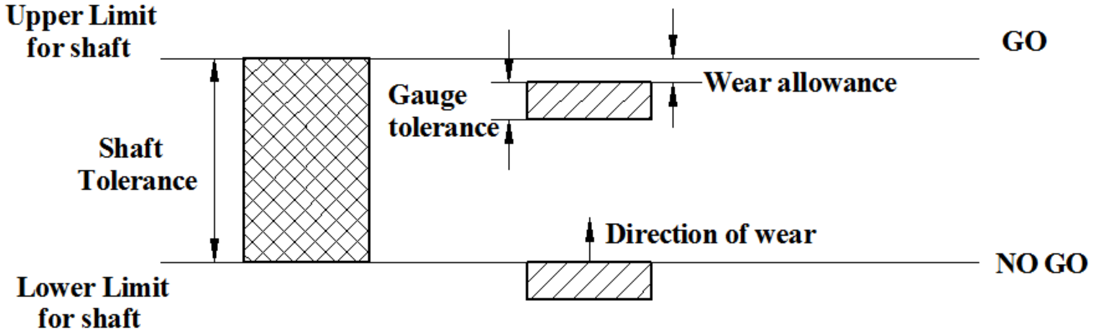
Present British System of Gauge & Wear Tolerance:

PLUG GAUGES: (For checking tolerances on holes)



GO & NO GO Plug Gauges (For checking hole tolerance)

RING/SNAP GAUGES: (For checking tolerances on shafts)



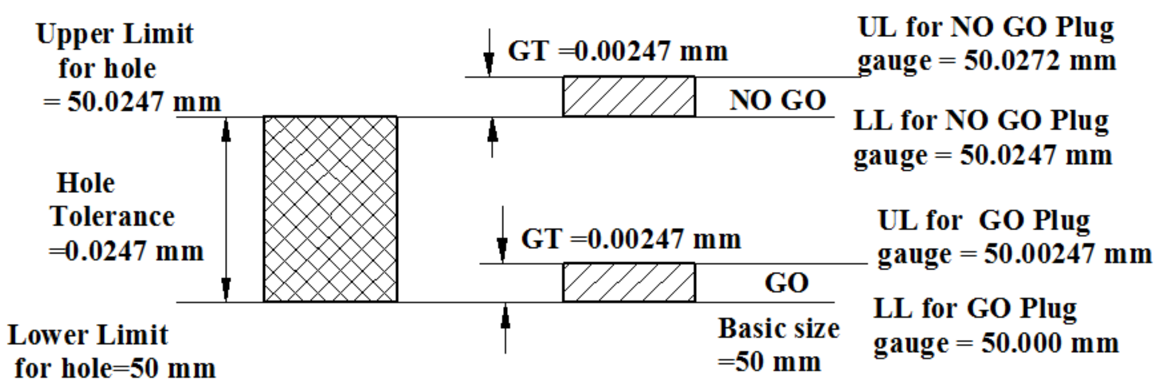
GO & NO GO Ring Gauges (For checking shaft tolerance)

Numerical Problem 1:

Calculate the dimensions of plug & ring gauges to control the production of 50 mm shaft & hole pair of H_7d_8 as per IS specifications. The following assumptions may be made: 50 mm lies in diameter step of 30-50 mm. Upper deviation for 'd' shaft is $-16D^{0.44}$ and lower deviation for hole H is zero. Tolerance unit in 'i' in microns is $=0.45\sqrt[3]{}\quad +0.001D$ and IT6=10i and above IT6 grade, the tolerance is multiplied by 10 at each 5th step.

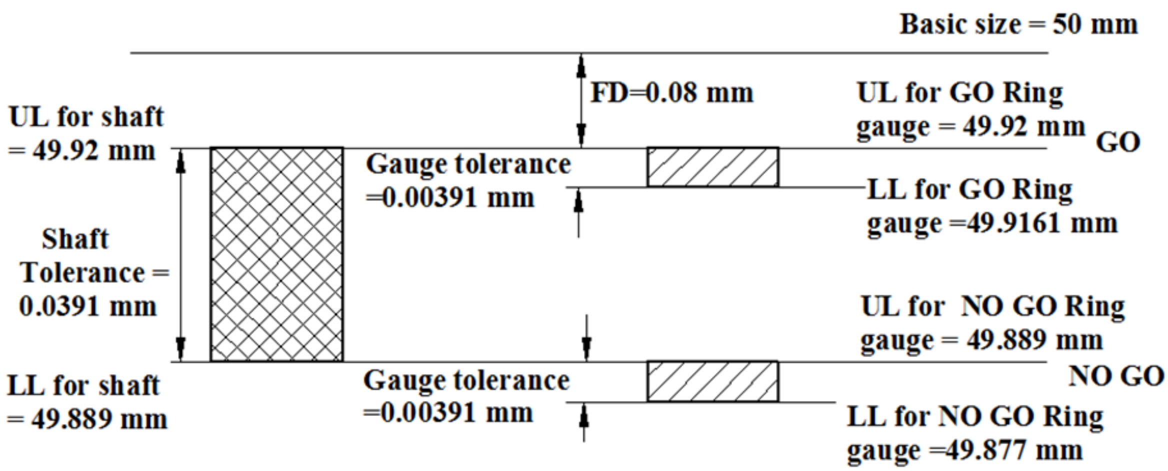
Solution:

Design of Plug gauge (for checking limits of hole):



GO & NO GO Plug Gauges (For checking hole tolerance)

Design of Ring gauge (for checking limits of Shaft):



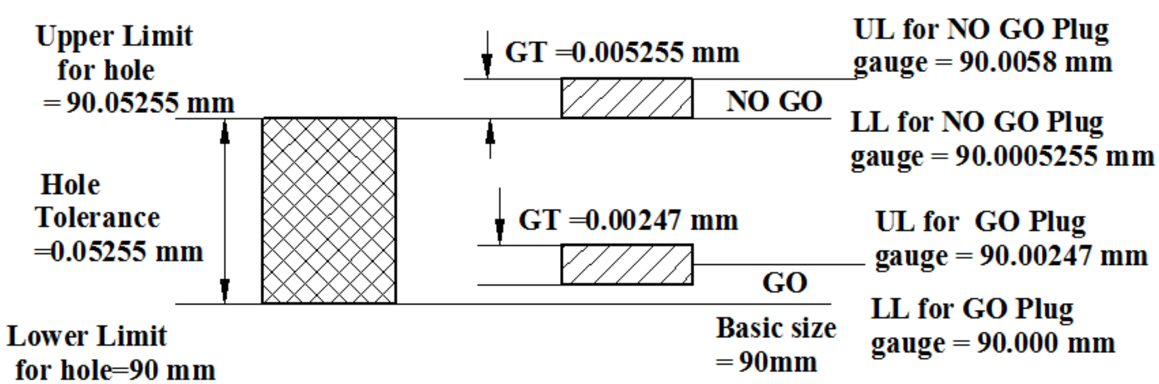
GO & NO GO Ring Gauges (For checking shaft tolerance)

Numerical Problem 2

Determine the actual dimensions to be provided for a shaft and hole 90 mm size for H_8e_9 type clearance fit. Size 90 mm falls in the diameter step of 80-100 mm. Value of standard tolerance unit $=0.45\sqrt[3]{\ }$ $+0.001$. The values of tolerances for IT8 & IT9 grades are 25i & 40i respectively. Value of fundamental deviation for 'e' type shaft is - $11D^{0.41}$. Also design the GO & NO GO gauges considering wear allowance as 10% of gauge tolerance.

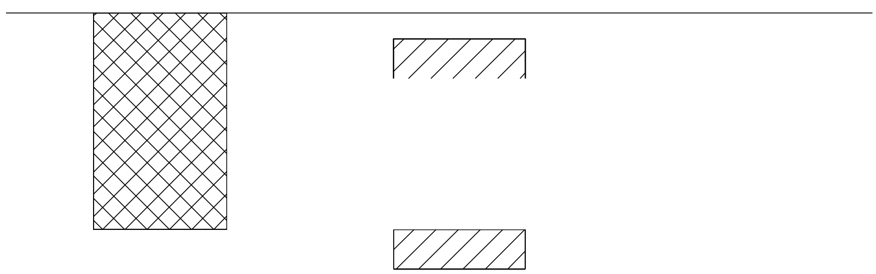
Solution:

Design of Plug gauge (for checking limits of hole):



GO & NO GO Plug Gauges (For checking hole tolerance)

Design of Ring gauges (for checking limits of shaft)



****