UNIT-3

SYSTEMS OF LIMITS AND FITS

Introduction:-

Metrology is derived from a Greek word which means "measurement". It is the science of measurement and measurement is the language of science. But, for engineering purposes it has limited to the measurement of length, angles and other quantities that can be expressed in linear and angular terms. It is concerned with the methods execution and estimation of accuracy of measurements.

Metrology plays a vital role in the field of engineering for the designing and manufacturing of various engineering products. It is used for measuring the size, shape, etc. The products obtained should be in the limits of the specification with dimensional accuracy. In order to improve the process of manufacturing, it is required to develop the means of measurement. Every type of quantity measured must be followed by the units, which gives the correct meaning to the quantity measured.

Significance of Metrology:

(a) Metrology is very helpful in the scientific investigation of our dynamic world.

(b) It plays a critical role in the fields of chemistry, nanotechnology, etc.

(c) Metrology provides an infrastructure not only for physical and natural sciences but also exceeds to comprise environment, medicine, agriculture and food.

(d) Various higher level studies demonstrate the impact of measurement to the society.

LIMITS:-

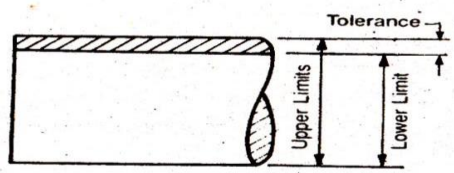
Limits can be defined as the permissible variation in dimension that is permitted to account for variability. Manufacturing process is a combination of three elements man, materials and machine. A change in any one or all of these will result in changes in sizes of manufactured parts. Usually in mass production, large number of components are to be made by different operators on different machines. So, it is impossible to make all components with exact dimensions.

The difference in dimensions vary from machine to machine, operator to operator and quality of the components. The dimension of the manufactured part can thus only be made to lie between two limits, maximum and minimum. The maximum limit is the maximum size permitted for the component whereas the minimum limit is the minimum size permitted for the component.

TOLERANCE:-

The permissible variation in size or dimension is called tolerance. Thus, the word tolerance indicates that a worker is not expected to produce the part to the exact size, but a definite small size error is permitted. The difference between the upper limit (high. limit) and the lower limit of a dimension represents the margin for variation in' workmanship, and is called a ‘tolerance Zone'.

Tolerance can also be defined as the amount by which the job is allowed to go away from accuracy and perfectness without causing any functional trouble, when assembled with its mating part and put into actual service.



Tolerance

For example, a shaft of 25 mm basic size may be written as 25 ± 0.02.

The maximum permissible size (upper limit) = 25.02 mm and the minimum permissible size (2000 limit) = 24.98 mm

Then, Tolerance = Upper limit − Lower limit

= 25.02 − 24.98 = 0.04 mm.

Systems of Writing Tolerances:-

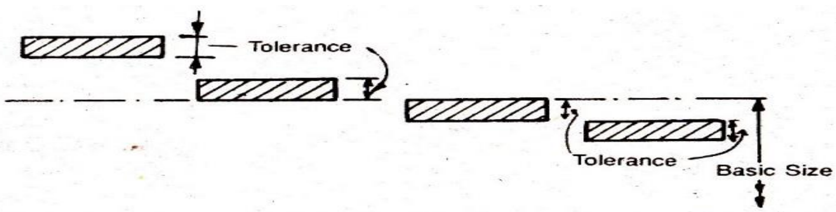
There are two systems of writing tolerances:

(i) Unilateral system

(ii) Bilateral system

(i) Unilateral System

In this system, the dimension of a part is allowed to vary only on one side of the basic size i.e., tolerance lies wholly on one side of the basic size either above or below it.



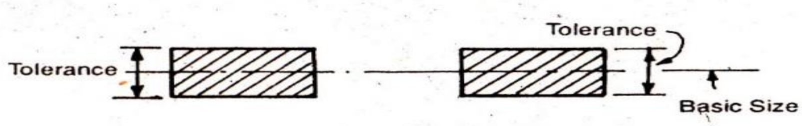
Unilateral Tolerance

Examples of unilateral tolerance are :

$25^{+0.02}_{+0.01}$, $25^{+0.02}_{-0.00}$, $25^{-0.01}_{-0.02}$, $25^{+0.00}_{-0.02}$ etc.

(ii) Bilateral system

In this system, the dimension of the part is allowed to vary on both the sides of the basic size i.e., the limits of tolerance lie on either side of the basic size; but may not be necessarily equally disposed about it.



Bi-lateral tolerance

e.g., $25^{\pm 0.02}$, $25^{+0.02}_{-0.01}$

In this system it is not possible to retain the same fit when tolerance is varied and .the basic size of one or both of the mating parts is to be varied. This system is used in mass production where machine setting is done for the basic size.

Advantages of Unilateral Dimensioning System

1. Unilateral system of dimensioning is the most easiest and simplest method to find the deviations.

2. It can standardize the 'Go' gauge ends Without any difficulty.

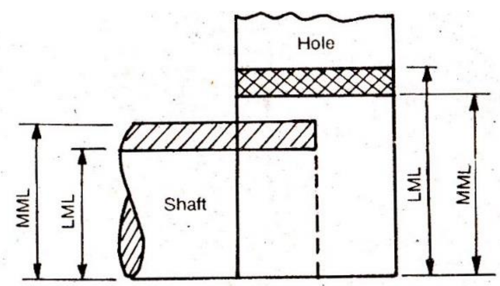
3. While machining the mating parts, the tolerance under this system facilitates the operator to a higher extent.

Advantage of Bilateral Dimensioning System

This system is used in mass production, as the setting of machine for basic size is the main criteria.

<u>Maximum and Minimum Metal Limits (or conditions):-</u>

If the tolerance for the shaft is given as $25^{\pm 0.05}$, the upper limit will be 25.05 mm and the lower limit will be 24.94 mm. The Shaft is said to have Maximum Metal Limit (MML) of 25.05 mm, since at this limit the shaft has maximum possible amount of metal. The limit of 24.95 will then be the minimum or "Least Metal Limit" (LML) because at this limit the shaft will have the least possible amount of metal.

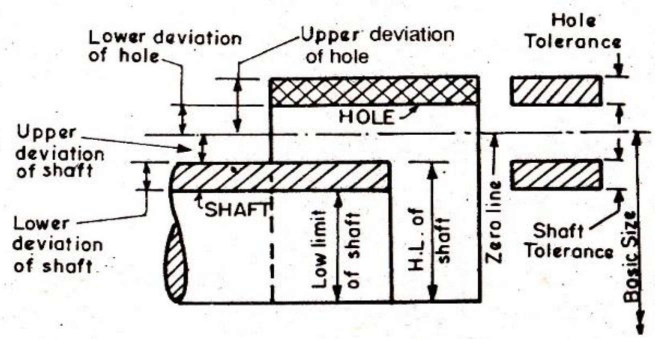


MML and LML

Similarly, if the hole is designated as $30^{\pm 0.05}$mm, the upper limit will be 30.05 mm and the lower limit will be 29.95 mm. Then, the Maximum Metal Limit (MML) of hole will be equal to 29.95, since at this lower limit the hole has the maximum possible amount of metal; while the upper limit of 30.05 mm will be the minimum of 'Least Metal Limit' (LML) of hole as, at this limit the hole will have the least possible amount of metal.

<u>Conventional Diagram of Limits and Fits:-</u>

In the system of limits and fits, we are simply interested in the tolerance on shafts and holes and not in their sizes. Therefore, in the conventional simplified diagram the shaft is shown resting on the hole to make it easy to understand.



Conventional Diagram of Limits

Terminology for Limits and Fits:-

Basic or Nominal Size: It is the standard size of a part with reference to which the limits of variation of a size are determined. It is referred to as a matter of convenience. The basic size is the same for the hole and its shaft. It is the designed size obtained by calculations for strength.

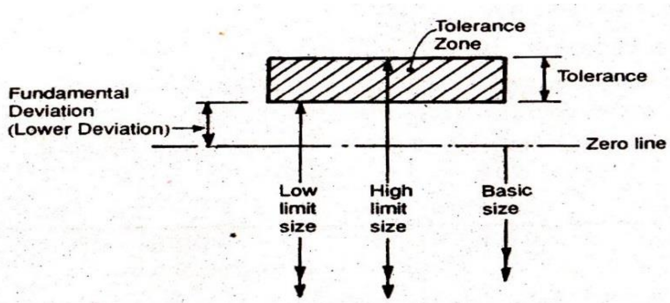
Zero line: It is a straight line drawn horizontally to represent the basic size. In the graphical representation of limits and fits, all the deviations are shown with respect to the zero line (datum line). The positive deviations are shown above the zero line and negative deviations below as shown in Fig (Conventional diagram of limits above).

Deviation: Deviation is the algebraic difference between the size (actual, maximum etc.) and the corresponding basic size.

Upper Deviation: It is the algebraic difference between the upper (maximum) limit of size and the corresponding basic size. It is a positive quantity when the maximum limit of size is greater than the basic size and a negative quantity when the upper limit of size is less than the basic size as shown in Fig. It is denoted by 'ES' for hole and 'es' for a shaft.

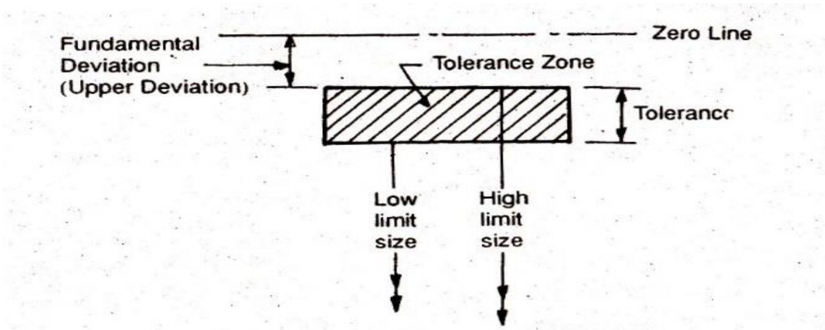
Lower Deviation: It is the algebraic difference between the lower limit of size and the corresponding basic size. It is a positive quantity when the maximum limit of size is greater than the basic size and a negative quantity when the lower limit of size is less than the basic size.

Fundamental Deviation: Fundamental deviation is that one of the two deviations (either the upper or the lower) which is the nearest to the zero line for either hole or a shaft. It fixes the position of the 'Tolerance Zone' in relation to the zero line as shown in Fig.



Lower deviation as fundamental deviation

The fundamental deviation for the hole is denoted by capital letters A, B, C, ... 2 C and the same for shaft is denoted by small letters a, b, c zc etc. as explained later.



Upper deviation as fundamental deviation

From Fig it is clear that when the tolerance zone is above the zero line, lower deviation is the fundamental deviation. While, when the tolerance zone is below the zero line, upper deviation is the fundamental deviation.

FIT:-

Fit may be defined as a degree of tightness or looseness, between two mating parts to perform a definite function when they are assembled together.

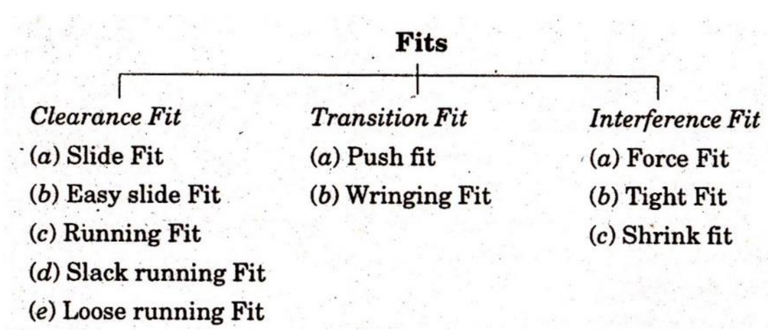
The fit given the relationship between two mating parts that is shaft and hole. A fit can either provide a fixed joint or movable joint. For example a shaft running in a bearing can move in relation to it and thus forms a movable joint, whereas, a pulley mounted on the shaft forms a fixed joint.

Types of fits:-

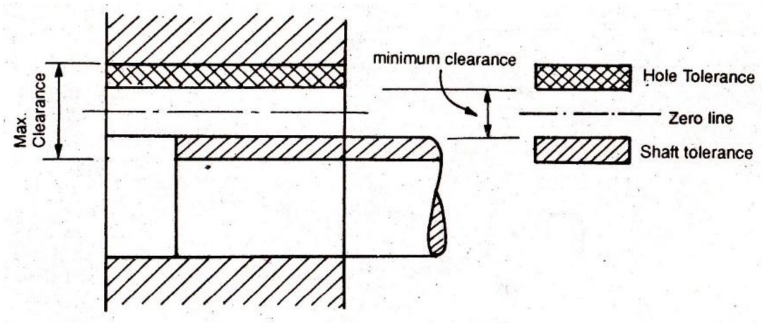
On the basis of positive, zero and negative values of Clearance, there are three basic types of fits:

(1) Clearance Fit (2) Transition Fit and, (3) Interference Fit.

These are further classified in the following manner:



(1) Clearance Fit: In this type of fit aha& is always smaller than the hole i.e., the largest permissible aha& diameter is smaller than the diameter of the smallest hole. So that the shaft can rotate or slide through with different degrees of freedom according to the purpose of mating part.



Clearance Fit

Clearance fit exists when the shaft and the hole are at their maximum metal conditions. The tolerance zone of the hole is above that of the shaft as shown in Fig.

Maximum Clearance: It is the difference between the minimum size of shaft and maximum size of hole.

Minimum Clearance: It is the difference between the maximum size of shaft and minimum size of hole.

i. Slide Fit: This type of fit has a very small clearance, the minimum clearance being zero. Sliding fits are employed when the mating parts are required to move slowly in relation to each other e.g., tailstock spindle of lathe, feed movement of the spindle quill in a drilling machine, sliding change gears in quick change gear box of a centre lathe etc. .

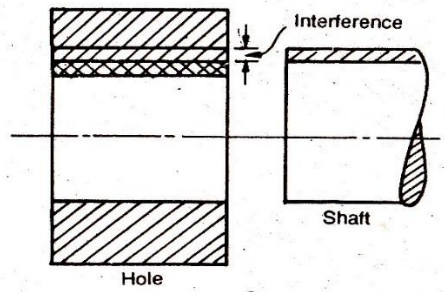
ii. Easy Slide Fit: This type of fit provides for a small guaranteed clearance. It serves to ensure alignment between the shaft and hole. It is applicable for slow and non-regular motion, for example, spindle of lathe and dividing heads, piston and slide valves, spigots etc.

iii. Running Fit: Running fit is obtained when there is an appreciable clearance between the ~mating parts. The clearance provides a sufficient space for a lubrication film between mating friction surfaces. It is employed for rotation at moderate speed, e.g., gear box bearings, shaft pulleys, crank shafts in their main bearings etc.

iv. Slack running Fit: It is obtained when there is a considerable clearance between the mating parts. This type of fit may be required as compensation for mounting errors e.g., arm shaft of I.C. engine, shaft of certifigual pump etc.

v. Loose running Fit: Loose running fit is employed for rotation at very high speed, eg., idle pulley on their shaft such as that used in quick return mechanism of a planer.

(2) Interference Fit: In this type of fit the minimum permissible diameter of the shaft is larger than the maximum allowable diameter of the hole. Thus the shaft and the hole members are intended to be attached permanently and used as a solid component.



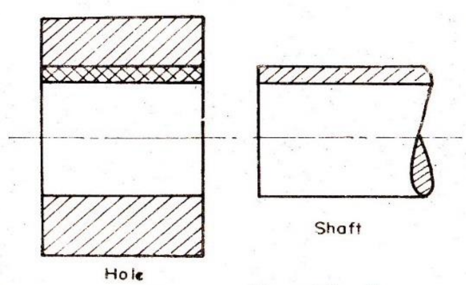
Interference fit

i. Force Fit: Force fits are employed when the mating parts are not required to be disassembled during their total service life. In this case the interference is quite appreciable and, therefore, assembly is obtained only when high pressure is applied. This fit, thus, offers a permanent type of assembly, e.g. , gears on the shaft of a concrete mixture, forging machine etc.

ii. Tight Fit: It provides less interference than force fit. Tight fits are employed for mating parts that may be replaced while overhauling of the machine, for example, stepped pulleys on the drive shaft of a conveyor, cylindrical grinding machine etc.

iii. Heavy force and Shrink Fit: It refers to maximum negative allowance. Hence considerable force is necessary for the assembly. The fitting of the frame on the rim can also be obtained first by heating the frame and then rapidly cooling it in its position.

(3) Transition Fit: Transition fit lies mid way between clearance and interference fit. In this type the size limits of mating parts (shaft and hole) are so selected that either clearance or indifference may occur depending upon the actual sizes of the parts. Push fit and wringing fit are the examples of this type of fit.



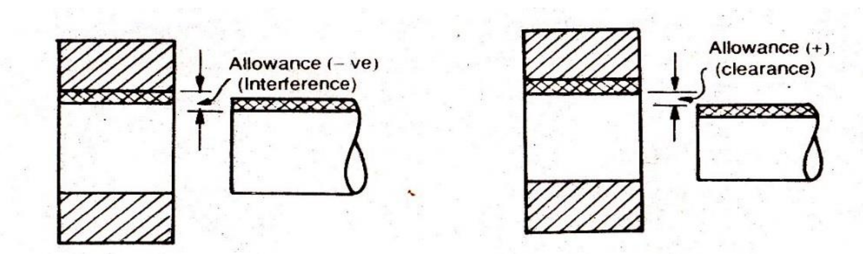
Transition fit

In this type of fit the tolerance zones of the hole and shaft overlap completely or in part.

i. Wringing Fit: A wringing fit provides either zero interference or a clearance. These are used where parts can be replaced without difficulty during minor repairs.

ii. Push Fit: The fit provides small clearance. It is employed for parts that must be disassembled during operation of a machine for example, change gears, slip bushing etc.

ALLOWANCE:-



Allowance is the prescribed difference between the dimensions of two mating parts for any type of fit.

It is the intentional difference between the lower limit of hole and higher limit of the shaft. The allowance may be positive or negative.

The positive allowance is called clearance and the negative allowance is called interference.

Difference between Tolerance and Allowance:-

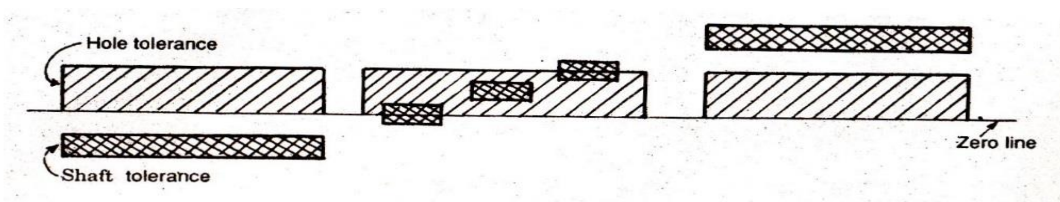
Tolerance	*Allowance*
1. It is the permissible variation in dimension of a part (either a hole or a shaft).	It is the prescribed difference between the dimensions of two mating parts (hole and shaft).
2. It is the difference between higher and lower limits of a dimension of a part.	It is the intentional difference between the lower limit of hole and higher limit of shaft.
3. The tolerance is provided on a dimension of a part as it is not possible to make a part to exact specified dimension.	Allowance is to be provided on the dimension of mating parts to obtain desired type of fit.
4. It has absolute value without sign.	Allowance may be positive (clearance) or negative (interference).

Systems of Obtaining Different Types of Fits:-

There are two systems of fit for obtaining clearance, interference or transition fit. These are:

(1) Hole basis system. (2) Shaft basis system.

(1) Hole basis system: In the hole basis system the hole is kept constant and the shaft sizes are varied to give the various types of fits. In this system lower deviation of the hole is zero i.e., the low limit of hole is the same as basic size. The high limit of hole and the two limits of size for the shaft are then varied to give the desired type of fit, as shown in Fig.

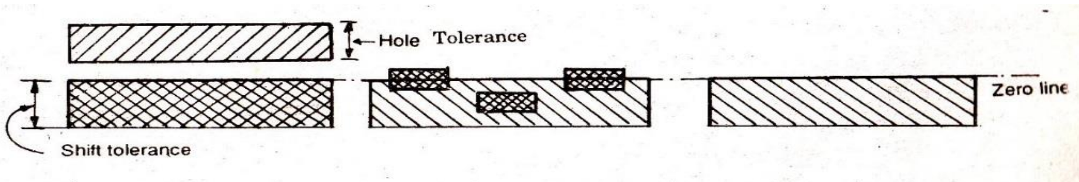


(*a*) Clearance Fit (*b*) Transition Fit (*c*) Interference Fit

Shaft basis System. In the shaft basis system the shaft is kept constant and the sizes of the hole are varied to give various types of fits.

In this system the upper deviation (fundamental deviation) of shaft is zero i.e., the high limit of shaft is the same as basic size and the various fits are obtained by varying the low limit of shaft and both the limits of hole.

(2) Shaft basis system:



(a) Clearance Fit (b) Transition Fit (c) Interference Fit

The hole basis system is most commonly used because it is more convenient to make correct holes of fixed sizes, since the standard drills, taps, reamors and broaches etc. are available for producing holes and their sizes are not adjustable. On the other hand size of shaft produced by turning, grinding etc. can be very easily varied.

Shaft basis system is used when the ground bars or drawn bars are readily available. These bars do not require further machining and fit are obtained by varying the sizes of hole.

<u>Difference between 'Hole Basis' and 'Shaft Basis' Systems:-</u>

Hole Basis System	*Shaft Basis System*
1. Size of hole whose lower deviation is zero (H-hole) is assumed as the basic size.	Size of shaft whose upper deviation is zero (*h*-shaft) is assumed as basic size.
2. Limits on the hole are kept constant and those of shaft are varied to obtain desired type of fit.	Limits on the shaft are kept constant and those on the hole are varied to have necessary fit.
3. Hole basis system is preferred in mass production, because it is convenient and less costly to make a hole of correct size due to availability of standard drills and reamers.	This system is not suitable for mass production because it is convenient, time consuming and costly to make a shaft of correct size.
4. It is much more easy to vary the shaft sizes according to the fit required.	It is rather difficult to vary the hole sizes according to the fit required.
5. It requires less amount of capital and storage space for tools needed to produce shafts of different sizes.	It needs large amounts of capital and storage space for large number of tools required to produce holes of different sizes.
6. Gauging of shafts can be easily and conveniently done with adjustable gap gauges.	Being internal measurement, gauging of holes cannot be easily and conveniently done.

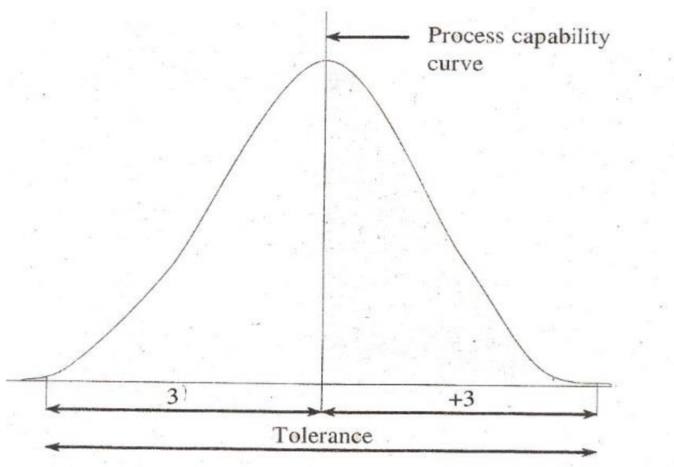
Types of Assemblies:-

There are three ways by which the mating parts can be made to fit together in the desired manner. These are:

(1) Trial and Error (2) Interchangeable Assembly (3) Selective Assembly

(1) Trial and Error: when a small number of similar assemblies are to be made by the same operator the necessary fit can be obtained by trial and error. This technique simply requires one part to be made to its nominal size as accurately as possible, the other part is then machined with a small amount at a time by trial and error until they fit in the required manner. This method may be used for "one off jobs", tool room work etc. where both parts will be replaced at once.

(2) Interchangeable Assembly:



It is a system of producing the mating parts in which large number of mating parts are produced. In earlier days, a single operator was confined with number of units and assemble it, which used to take long time and it was not economical. So to reduce the cost and time, mass production 'system was developed. In most production systems, the components are produced in one or more batches by different operations on different machines.

Advantages of Interchangeability

1. This system reduces the production cost and increases the output

2. The operator need not Waste time in assembling the parts by trial and error method.

3. Worn out parts and defective parts can be easily replaced.

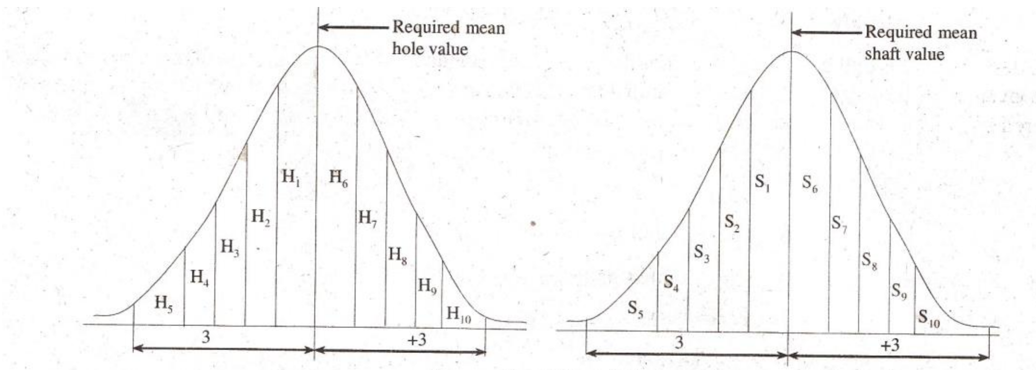
4. By this method, it is possible to produce mating parts at different places by different operators.

5. Maintenance cost and shut down period is reduced.

(3) Selective Assembly:

The need of the consumer is not only the quality, precision and trouble-free products but also the availability of products at economical prices. This is possible by automatic gauging for selective assembly. In this system, the parts are manufactured to rather wider tolerances and the products produced are classified into various groups according to their sizes by automatic gauging. Classification is made for formatting parts and only matched groups are assembled together.

If hole and shaft are to be produced with in a tolerance of 0.02 mm and both are in the curve of normal distribution, then automatic gauging divides 'them into parts with a 0.002 mm limit for selective assembly of individual pans. Consider an example of piston with cylinder. Let the size of the above be 60 mm and the clearance of 0. 12 mm is required for the assembly. Let the tolerance on bore and piston each be 0.04 mm. Then



Process Capability of Shaft and Hole

Dimension of bore diameter is $60^{\pm 0.12}$ mm and

Dimension of piston is $59.88^{\pm 0.12}$ mm

The pistons and bores may be selected to give the clearance of 0.12 as given below.

Cylinder bore	59.98	60.00	60.02
Piston	59.86	59.88	59.90

<u>What is the difference between international and British standards?</u>

There are a few different standards, British standards, European Standards, American standards, Canadian....

The International Standards (IEC) are worldwide, European ones cover European countries and country specific ones cover that country. Many countries have similar standards.

The ones that cover larger areas (International and European) are used by the countries when they write their standards. An example might be BS EN numbered standards which are British standards that cover the requirements of the European standard. Some are country specific only (BS) and some cover international standards (IEC). It can be confusing but there should be a standard in each country to cover most engineering things.

The main difference is the geographical area that they cover. If you are say working in Britain and follow the BS requirements (or BS EN, or IEC) that applies then you can say that you are working to best practices and can't be faulted for that.

There will be small differences between them based on custom and practices for the countries that have written them.

Indian Standard System of Limits and Fits (IS-919 and 2709)

The Indian standards are in line with the ISO (International Organizations for Standards) recommendations.

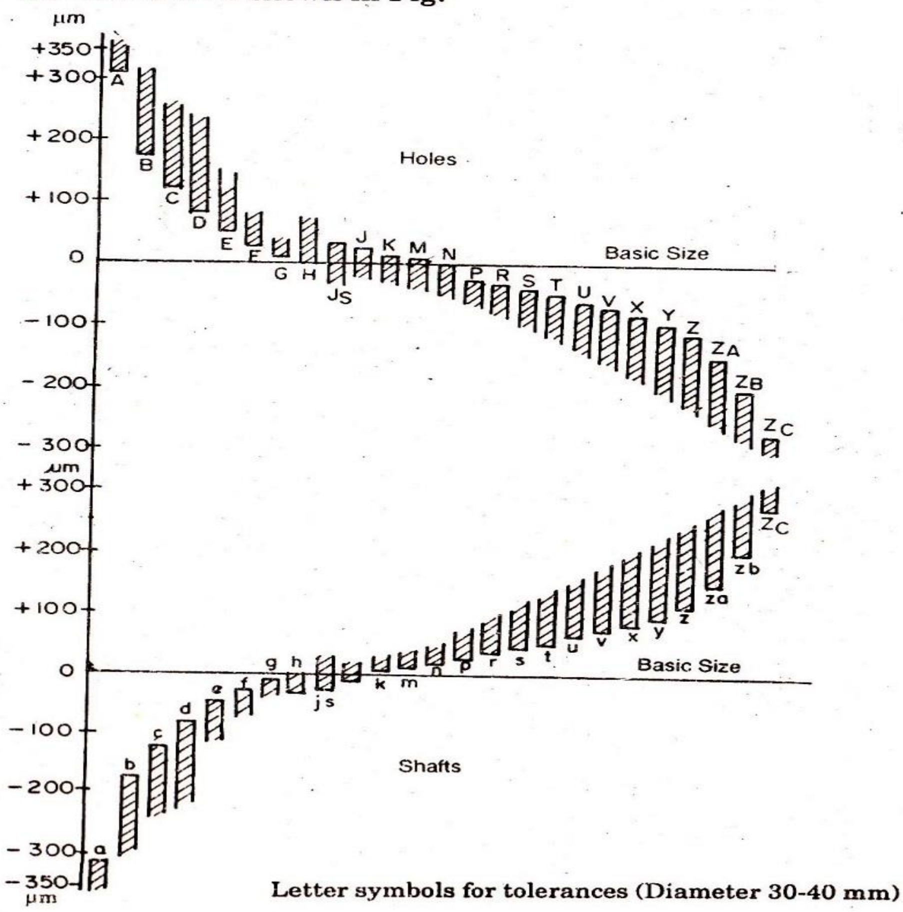
It consists of suitable combination of 18 grades of fundamental tolerances or in other words grades of accuracy for manufacture, and 25 types of fundamental deviations.

The 18 grades of fundamental tolerances are designated as IT01, IT0, IT1 to IT16. While, the fundamental deviations are indicated by letter symbols for both hole and shaft (capital letters 'A to Zc' for holes and small letters a to z_c for shafts. These are : A, B, C, D, E, F, G, H, J_S, J, K, M, N, P, R, S, T, U, V, X, Y, Z , Z_A, Z_B, Z_C).

Innumerable fits ranging from extreme clearance to those of extreme interference can be obtained by a suitable combination of fundamental tolerances and fundamental deviations. Each of 25 holes has a choice of 18 tolerances.

For shafts 'a' to 'h' the upper deviation is below the zero line and for shafts 'j' to 'Zi' it is above the zero line.

For holes 'A' to 'H' lower deviation is above the zero line and for J to Zc it is below the zero line as shown in Fig.



Letter symbols for tolerances (Diameter 30-40 mm)

Problems:-

EXAMPLE 1. Find the values of allowance, and tolerances for hole and shaft assembly for the following dimensions of mating parts :

Hole : $25^{+0.05}_{+0.00}$ Shaft : $25^{-0.02}_{-0.05}$

SOLUTION.

(*i*) Hole : Tolerance = High limit – Low limit
= 25.05 – 25 = 0.05 mm

(*ii*) Shaft : Tolerance = High limit – Low limit
Now, High limit = 25 – 0.02 = 24.98 mm
Low limit = 25 – 0.05 = 24.95 mm
∴ Tolerance = 24.98 – 24.95 = 0.03 mm

(*iii*) Allowance = Low limit of hole – High limit of shaft
= Maximum metal condition of hole – Maximum metal condition of shaft
= 25.00 – 24.98 = 0.02 mm

EXAMPLE 2. 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 the bush to give a maximum clearance of 0.075 mm with the hole basis system.

SOLUTION. In the hole basis system lower deviation of hole is zero therefore low limit of hole = 50 mm

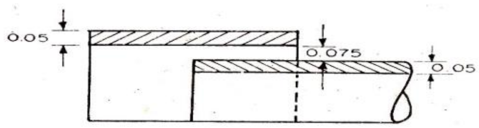


Fig. 9.44.

High limit ot nole = Low limit + Tolerance
= 50.00 + 0.050 = 50.050 mm

High limit of shaft = Low limit of hole – Allowance
= 50.00 – 0.075 = 49.025 mm

Low limit of shaft = High limit – Tolerance
= 49.025 – 0.050 = 48.975 mm

EXAMPLE 3. For each of the following hole and shaft assembly, find shaft-tolerance, hole tolerance and state whether the type of fit is (*i*) clearance (*ii*) transition or (*iii*) interference.

(*a*) Hole : $50^{+0.25}_{+0.00}$ mm Shaft : $50^{+0.05}_{+0.005}$ mm

(*b*) Hole : $30^{+0.05}_{+0.00}$ mm Shaft : $30^{-0.02}_{+0.05}$ mm

(*c*) Hole : $25^{+0.04}_{+0.00}$ mm Shaft : $25^{+0.06}_{+0.04}$ mm

SOLUTION. (*a*) Hole : High limit of hole = 50.025 mm
Low limit of hole = 50.00 mm
∴ Hole tolerance = 50.025 – 50.00 = **0.025** mm ...(*i*)
Shaft : High limit of shaft = 50.05 mm
Low limit of shaft = 50.005 mm
Shaft tolerance = 50.05 – 50.005 = **0.045** mm ...(*ii*)

If we choose high limit of hole with high limit of shaft then
Allowance = 50.025 – 50.05 = – 0.025 (Interference)
If we choose high limit of hole and low limit of shaft then
Allowance = 50.025 – 50.005 = 0.020 mm (Clearance)
Similarly, if we choose low limit of hole and either high limit or low limit of shaft it is clear that there will be interference.
Thus, we conclude that the type of fit is **Transition Fit**.

(*b*) Hole : High limit = 30.05 mm
Low limit = 30.00 mm
∴ Tolerance = 0.05 mm
Shaft : High limit = 30 – 0.02 = 29.98 mm
Low limit = 30 – 0.05 = 29.95 mm
∴ Tolerance = 29.98 – 29.95 = 0.03 mm
If we select high limit of hole and high limit of shaft then
Allowance = 30.05 – 29.98 = 0.07 mm
If we select low limit of hole and high limit of shaft then
Allowance = 30.00 – 29.98 = 0.02 mm
Thus we conclude that the type of fit is **Clearance Fit**.

(*c*) Hole : High limit = 25.04 mm
Low limit = 25.00 mm
Tolerance = 25.04 – 25.00 = 0.04 mm
Shaft : High limit = 25.06 mm
Low limit = 25.04 mm
Tolerance = 25.06 – 25.04 = 0.02 mm
If we select, H.L. of shaft and L.L. of hole then
Allowance = 25.00 – 25.06 = – 0.06 mm
It is clear that for any combination of hole and shaft the allowance will be negative.
Thus we conclude that the type of fit is **Interference Fit**.

Example 4. In a limit system, the following limits are specified to give a clearance fit between a shaft and a hole.

$$\text{Shaft } 50^{-0.006}_{-0.020} \text{ mm} \qquad \text{Hole } 50^{+0.030}_{-0.000} \text{ mm}$$

Find : (*a*) Basic size (*b*) Shaft and hole tolerances (*c*) Maximum clearance (*d*) Minimum clearance.

Solution. (*a*) Basic size (same for hole and shaft) = 50 mm

(*b*) Shaft tolerance = H.L. of shaft − L.L. of shaft

= (50 − 0.006) − (50 − 0.02) = **0.014 mm**

Hole tolerance = H.L. − L.L. = 50.030 − 50.00 = 0.030 mm

(*c*) Maximum clearance = H.L. of hole − L.L. of shaft

= 50.030 − (50 − 0.02) = 50.030 − 49.98 = 0.05 mm

(*d*) Minimum clearance = L.L. of hole − H.L. of shaft

= 50.00 − (50 − 0.006) = + 0.006 mm.

Example 5. In a hole and shaft assembly of 30 mm nominal size, the tolerances for hole and shaft are as specified below :

$$\text{Hole : } 30^{+0.02}_{-0.000} \text{ mm} \qquad \text{Shaft : } 30^{-0.040}_{-0.070} \text{ mm}$$

Determine :

(*i*) Maximum and minimum clearance obtainable
(*ii*) Allowance
(*iii*) Hole and Shaft tolerance
(*iv*) MML shaft and hole
(*v*) The type of fit.

Solution. (*i*) Maximum clearance = H.L. of hole − L.L. of shaft

= 30.02 − (30 − 0.07) = + 0.09 mm

Minimum clearance = L.L. of hole − H.L. of shaft

= 30.00 − (30 − 0.04) = + 0.04 mm

(*ii*) Allowance = L.L. of hole − H.L. of shaft

= 0.04 mm as above

(*iii*) Hole tolerance = H.L. of hole − L.L. of hole

= 30.02 − 30.00 = 0.02 mm

Shaft tolerance = H.L. of shaft − L.L. of shaft

= 29.96 − 29.93 = 0.03 mm

(*iv*) MML for shaft *i.e.* maximum metal limit for shaft

= H.L. of shaft = 29.96 mm

MML for hole = L.L. of hole = 25.00 mm

(*v*) Since the allowance is positive, it gives a clearance fit.

Example 6. A hole and mating shaft are to have a nominal assembly size of 50 mm. The assembly is to have a maximum clearance of 0.15 mm and a minimum clearance of 0.05 mm. The hole tolerance is 1.5 times the shaft tolerance. Determine the limits for both hole and shaft :

By using (*i*) Hole basis system (*ii*) shaft basis system.

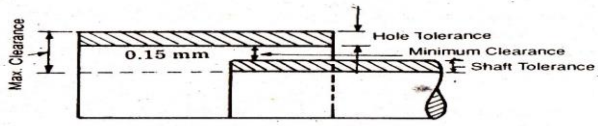


Fig. 9.45.

Solution. (*i*) **Hole Basis system**

In hole basis system lower deviation of hole is zero *i.e.*, the low limit of hole is the same as basic size from Fig. 9.45.

Max. clearance = Hole tolerance + Minimum clearance + Shaft tolerance

Therefore 0.15 = 1.5 × shaft tol. + 0.05 + shaft tol.

∴ 0.15 − 0.05 = shaft tol. (1.5 + 1)

i.e., Shaft tolerance $= \frac{0.1}{2.5} = 0.04$ mm

Hole tolerance = 0.04 × 1.5 = 0.06 mm

Now, low limit of hole = 40 mm (basic size)

∴ High Limit of hole = 40 + 0.06 = 40.06 mm

Thus hole sizes are 40 and 40.06 mm.

We know that minimum clearance = Low limit of hole − High limit of shaft

Therefore, 0.05 = 40.00 − H.L. of shaft

∴ H.L. of shaft = 40 − 0.05 = 39.95 mm

L.L. of shaft = H.L. − Tolerance

= 39.95 − 0.04 = 39.91 mm

Thus, shaft limits are 39.95 mm, and 39.91 mm.

(*ii*) **Shaft Basis system**

In shaft basis system upper deviation of shaft is zero *i.e.*, H.L. of shaft is the same as basis size = 40.00 mm

L.L. of shaft = H.L. − Tolerance

= 40.00 − 0.05 = 39.95 mm

Max. clearance = H.L. of hole − Low limit of shaft

∴ 0.15 = H.L. of hole − 39.95

∴ H.L. of hole = 39.95 + 0.15 = 40.10 mm

L.L. of hole = H.L. − Tolerance

= 40.10 − 0.06 = 40.04 mm

Example 7. In an assembly of two parts 50 mm nominal diameter, the lower deviation of the hole is zero and the higher is 5 microns ; while that of shaft is – 4 and – 8 microns respectively. Estimate the allowance and state the type of fit of the assembly.

Solution. *Hole size :* H.L. of hole = 50.005 mm

L.L. of hole = 50.000 mm

Shaft size : H.L. of shaft = 50 – 0.004 = 49.996 mm

L.L. of shaft = 50 – 0.008 = 49.992 mm

Minimum allowance = Lower limit of hole – Higher limit of shaft

= 50.000 – 49.996 = 10.004 mm

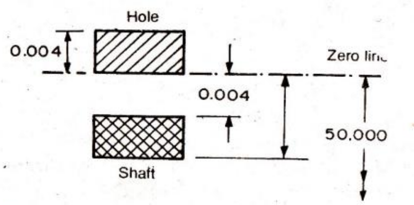


Fig. 7.46.

Example 8. A 20 mm diameter shaft and bearing are to be assembled with a clearance fit. The tolerance and allowances are as under :

Allowance = 0.002 mm

Tolerance on hole = 0.005 mm

Tolerance on shaft = 0.003 mm

Find the limits of size for the hole and shaft if :

(*a*) the hole basis system is used (*b*) shaft basis system is used. The tolerances are disposed of unilaterally.

Solution. *For Hole Basis System :*

Hole size :

Higher limit of hole = 20.005 mm

Lower limit of hole = 20.000 mm

Now, allowance given is + 0.002 mm

Therefore, Higher limit of shaft = Lower limit of hole – Allowance

= 20.000 – 0.002 = 19.998 mm

and, lower limit of shaft = Higher limit of shaft – Tolerance

= 19.998 – 0.003 mm = 19.995 mm

For Shaft Basis System :

Shaft size ; High limit = 20.000 mm and

Lower limit = 20.000 – 0.003 = 19.997 mm

Allowance = + 0.002 (given)

Therefore, Low limit of hole = High limit of size + Allowance

= 20.000 + 0.002 = 20.002 mm

and High limit of hole = 20.002 + 0.005 = 20.007 mm

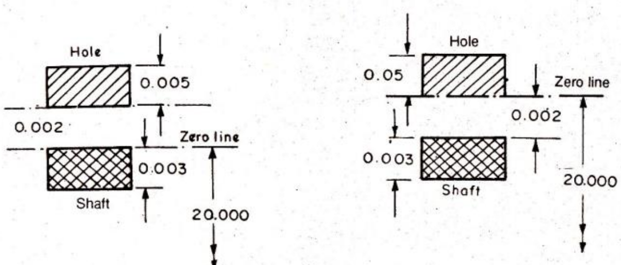


Fig. 7.47.

-----------------------THE END-----------------------

Taylor's principle – Design of GO and NO GO Gauges:

Taylor's principle of Gauge design:

This theory is the key to the design of limit gauges and defines the function and hence the form of most limit gauge. It states that,

1. The "GO" gauge should be designed to check the maximum material condition and should check as many dimensions as possible.
2. The "NOT GO" or "No GO" gauge should be designed to check the minimum materials limit and should only check one dimension at a time.

Thus a separate "No GO" gauge is required for each individual dimension. consider a system of limit gauge for a rectangular hole as shown in figure.

The "GO" gauge is used to ensure that the maximum metal condition is not exceeded and that metal does not encroach in to the minimum allowable hole space. It shoulc

Rectangular hole oversize in one Direction (31)

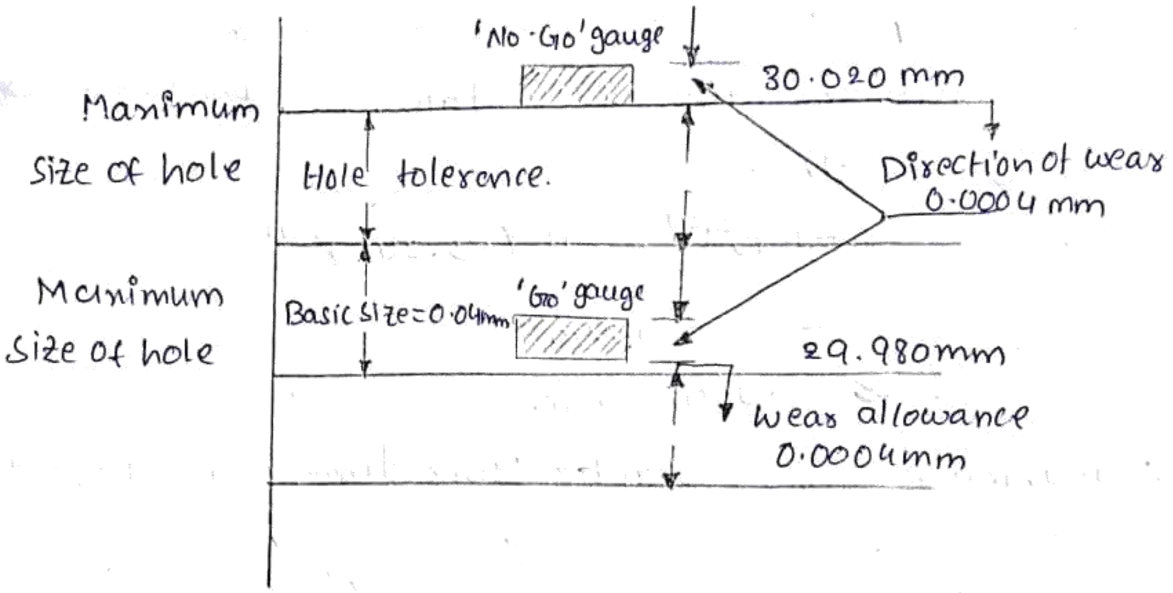
This system should be applied to all systems of limit gauges. The 'Go' gauge should be made equal in length to about three or four times of diameter, while the "No Go" gauge is always relatively short and approximately equal in length to the hole diameter, in case of circular hole.

In case of curved holes, if the length of the gauge is made shorter, then it will get pay through the obstruction in the hole and gives a wrong reading. But if the length of the gauge is made considerably long, it will not pass through the curved holes and error will be detected. A long gauges will thus check the surface not in one direction but in a number of sections simultaneously. The length of the "Go" gauge should not be less than 1.5 times the length of the hole to be checked.

Limitations of Taylor's principle:

1. Taylor's principle for 'Go' gauge is designed to have its length equal to the engagement length of fit. But in actual practice, it is not always possible. For instance, if Go gauge passes through a hole or shaft very easily so that it does not affect the required fit of assembly, then the length of 'Go' gauge (i.e., ring or cylindrical type gauge) might be less than engagement length of fit. — Similarly for big holes, the full form gauge will be much heavy and unsuitable to use. Hence, if 'Go' gauge does not gives roundness error to a affect the engagement fit than other gauges like spherical gauge, segmental cylindrical bar etc, should be used.

(b) 'NO - GO' gauge = $30.020^{+0.0004}_{+0.000}$ mm. (92)



Q. Design a typical 'Go' and 'No Go' gauge for a shaft of 80 mm.

Answer:

Let us design the Go and NO Go gauge for 80 $H_8 e_9$ fit

Nominal diameter of shaft and hole is 80 mm

Since 80 mm diameter lies in the diameter steps of 50 and 80 mm.

∴ Value of D = $\sqrt{50 \times 80}$ = 63.246 mm

we know that, the fundamental tolerance factor,

$$i = 0.45\sqrt[3]{D} + 0.001D$$

$$\Rightarrow i = 0.45\sqrt[3]{63.246} + 0.001 \times 63.246$$

$$= 1.856\, \mu m$$

$$\cong 0.00186 \text{ mm}$$

Hole type is H_8, and for H-hole, fundamental deviation = 0, ∴ minimum size of hole = 80.000 mm

For H_8 i.e., IT_8, the value of tolerance

$$= 25i$$

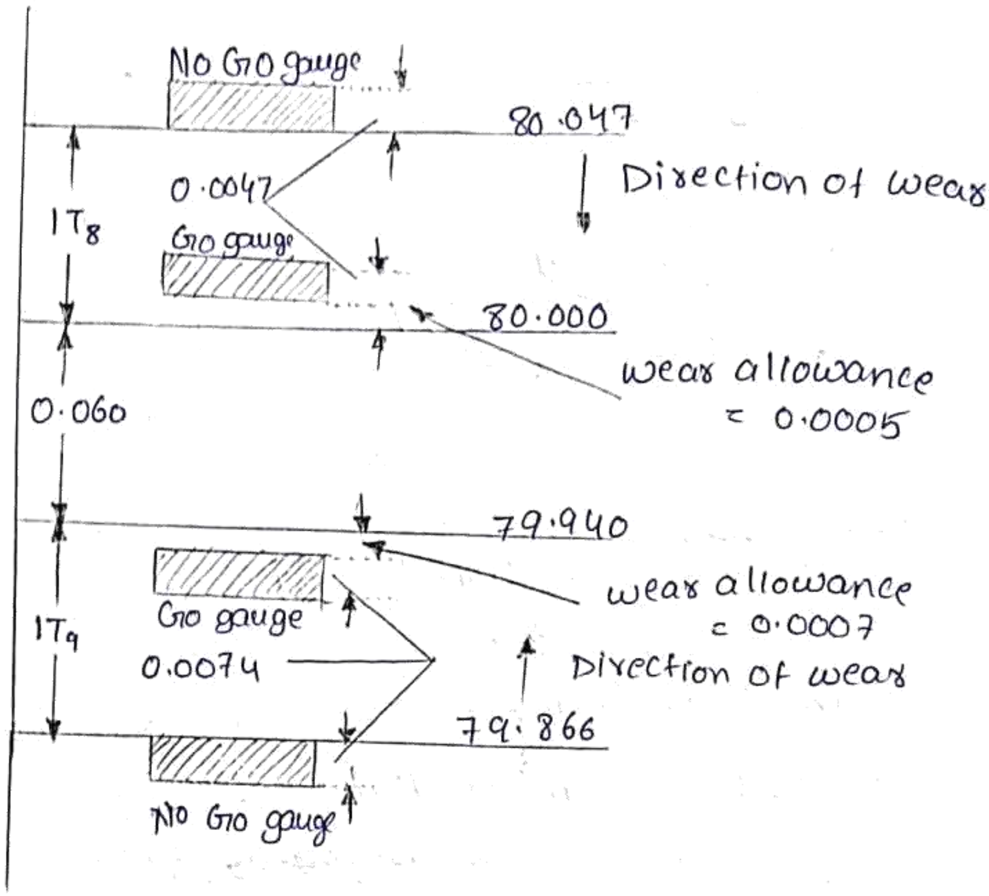
$$= 25 \times 0.00186$$

$$= 0.047 \text{ mm}$$

For shaft,

Go gauge $= 79.940^{-0.0007}_{-0.0007-0.0074} = 79.940^{-0.0007}_{-0.0081} = 80^{-0.0607}_{-0.0681}$ mm

No Go gauge $= 79.866^{-0.0000}_{-0.0074} = 80^{-0.1340}_{-0.1414}$ mm



Snap gauges:

These gauges are used for gauging the shafts and male components. The 'Go' snap gauge is of a size corresponding to the high (maximum) limit of the shaft, while the 'NOT Go' gauge corresponds to the low (minimum limit). It is a plain gauge, category of form of tested surface with single ended and double ended gauge. Its ranges from 3mm to 100mm – with double end and 100 to 250 mm as single ended type. The gauging surfaces of the snap gauge is hardened and suitably grounded and lapped. Snap gauge have 'Go' and 'No-Go' checking facility. The various snap, gap and ring gauges are shown in figure.

checked is parallel to the axis, as it normally is the gauge must be positioned parallel to the shaft axis also.

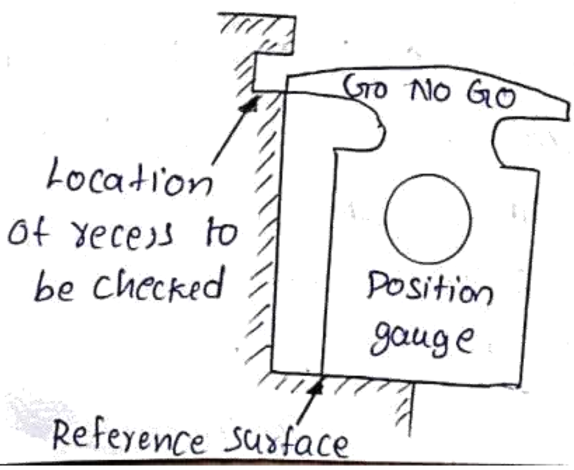
Profile gauges are used to check the form of the components. profiles are difficult to be checked by limit gauges and it is used practice to use fixed gauges mated to profile for checking profiles. There are two method of tolerance the form of profile (consisting of straight lines and curves).

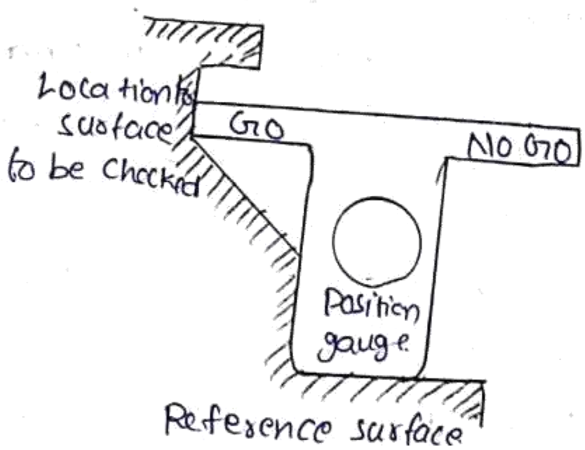
(i) It provide a tolerance zone within which the finished profile must lie. This method provides a uniform metal tolerance normal to the profile.

(ii) To use ordinates which are provided with individual tolerances. In this method the tolerance is normal to the surfaces will vary with the form of the profile.

Position Gauge:-

The position gauges are employed for checking the position of some features as the work in relation to other surface or reference point. They are available to avoid variety and their design is based on the principle of sighting the gauge and work or on the method of feel. Practically, different position gauges are required for different works.

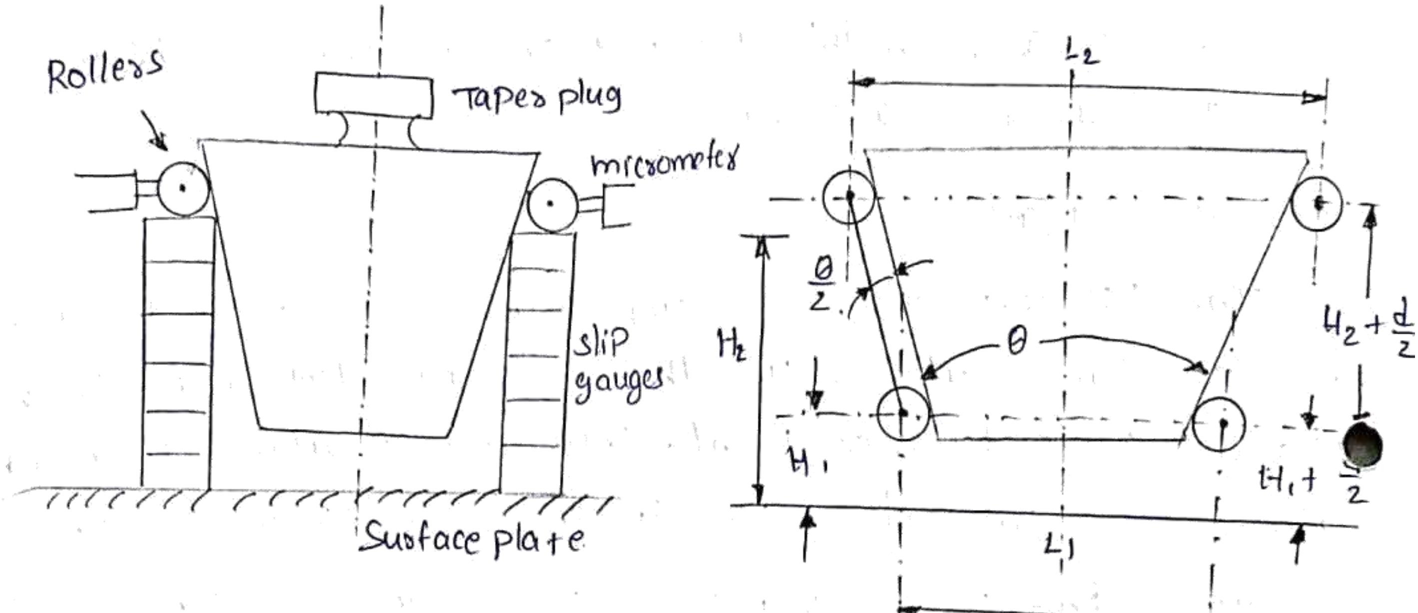




From the figure it may be noted that no light will pass between the reference surface and gauge surface in contact with "Go" side and light will pass "No Go" side. Figure shows checking to location of a recess in relation to a flat surface and figure shows the surface parallel to the reference surface to be located.

Checking the angle of taper using rollers, Micrometer and Slip gauges.:

In this method, measurements are made over two equal rollers standing on slip gauges at each side of the taper plug gauge at two positions (one near the lower end and one near the upper end) are shown in figure (1).



(1): Measurement of Taper plug gauge.

At first the taper plug is placed on the surface plate. Two equal rollers are placed on the slip gauges on either sides of the taper plug at a height of H_2 and two equal rollers are placed on lower sides of the plug at a height of H_1. Now, the distance between the ends of the rollers are measured by a micrometer. Let 'L_1' be the distance between the two upper position rollers and 'L_2' be the distance between the two lower position rollers. Also "d' be the diameter of the roller then,

$$\tan\left(\frac{\theta}{2}\right)\left[\frac{\left[\frac{L_2-d}{2}\right]-\left[\frac{L_1-d}{2}\right]}{\left[H_2+\frac{d}{2}\right]-\left[H_1+\frac{d}{2}\right]}\right]$$

$$\therefore \tan\left(\frac{\theta}{2}\right)=\left[\frac{\left(\frac{L_2-d}{2}\right)-\left(\frac{L_1-d}{2}\right)}{\left[\frac{2H_2+d}{2}\right]-\left[\frac{2H_1+d}{2}\right]}\right]$$

$$=\frac{\left[\frac{L_2-d-L_1+d}{2}\right]}{\left(\frac{2H_2+d-2H_1-d}{2}\right)}$$

$$\tan\left(\frac{\theta}{2}\right)=\frac{L_2-L_1}{2(H_2-H_1)}$$

where, $\frac{\theta}{2}$ - Half the taper angle of the plug.

Q Calculate the angle of taper and minimum diameter of an internal taper from the following readings,
Diameter of bigger ball - 10.25 mm
Diameter of smaller ball - 6.07 mm
Height of top of bigger ball from datum - 30.13 mm
Height of top of smaller ball from datum - 10.08 mm.

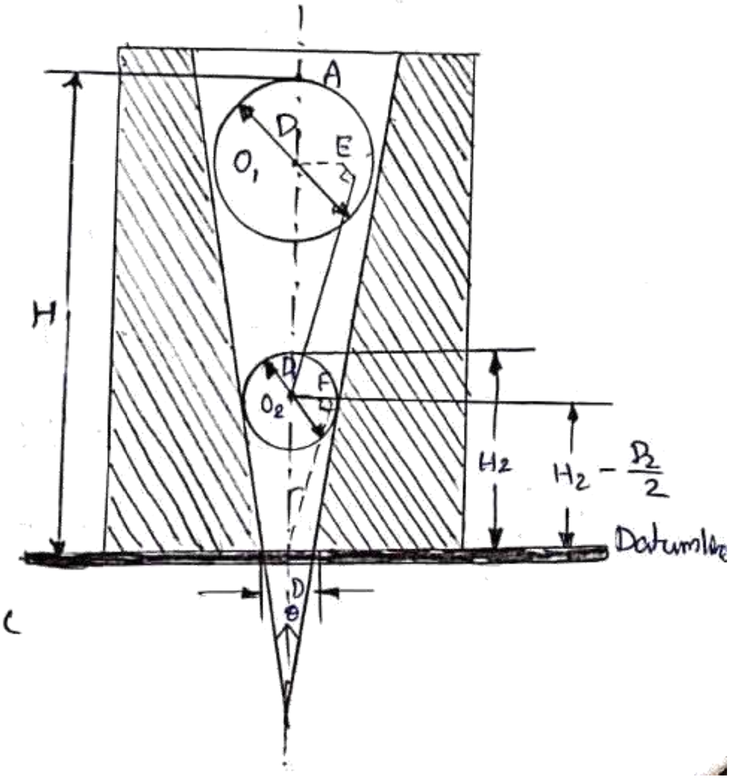
Answer: Given that

$D_1 = 10.25\text{mm}$

$D_2 = 6.07\text{mm}$

$H_1 = 30.13\text{ mm}$

$H_2 = 10.08\text{ mm}$



From triangle $O_1 O_2 D$, we have,

$$\sin\frac{\theta}{2} = \frac{O_1E}{O_1O_2} = \frac{O_1E}{AC - O_1A - O_2C}$$

$$\sin\frac{\theta}{2} = \frac{\frac{D_1}{2}-\frac{D_2}{2}}{H_1-\frac{D_1}{2}-\left(H_2-\frac{D_2}{2}\right)}$$

$$\sin\frac{\theta}{2} = \frac{D_1 - D_2}{2H_1 - D_1 - 2H_2 + D_2}$$

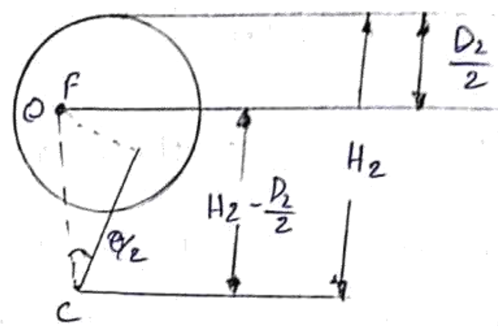
$$= \frac{10.25 - 6.07}{2(30.13) - 10.25 - 2(10.08) + 6.07}$$

$$\sin\frac{\theta}{2} = \frac{4.18}{35.92} = 0.11637$$

$$\frac{\theta}{2} = \sin^{-1}(0.11637) = 6.68265^{\circ}$$

$\therefore$ Taper angle, $\theta = 13.3653^{\circ}$

For finding minimum diameter (D) of an internal taper.



Let us consider the triangle O_2CF i.e.,

$$\sin\frac{\theta}{2} = \frac{O_2F}{O_2C}$$

$$\sin\frac{\theta}{2} = \frac{\frac{D_2}{2} - \frac{D}{2}}{H_2 - \frac{D_2}{2}} = \frac{D_2 - D}{2H_2 - D_2}$$

$$0.11637 = \frac{6.07 - D}{2(10.08) - 6.07}$$

$$1.63965 = 6.07 - D$$

$$D = 6.07 - 1.63965 = 4.43\ mm$$

$\therefore$ minimum diameter of taper = 4.43 mm.

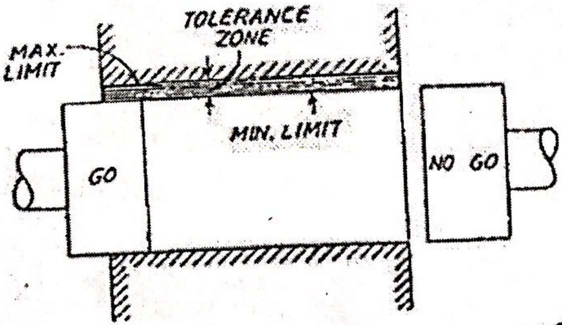
2.17 Gauges:

Before accepting any component that is manufactured, need to be inspected for high accuracy. In mass production several components will be manufactured checking each part requires high cost and time. Conformations for the parts can be done easily, by measuring the required tolerances of parts using gauges. Gauges are scale less rigid instruments, these gauges are made to specify the limits of the component. Unskilled persons can check the work piece. No need to do calculations for measurement. They give quick result.

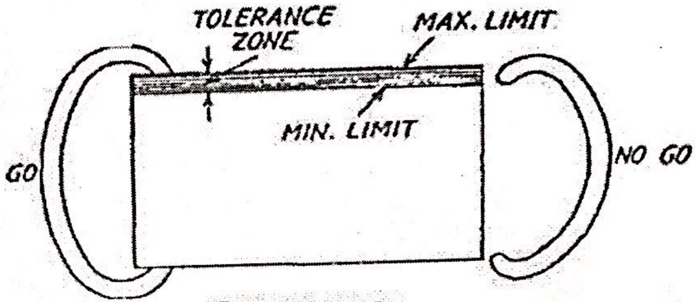
Gauges are of two types:

A **Go-No gauge** (or Go/no go) refers to an inspection tool used to check a workpiece against its allowed tolerances. Its name derives 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.



Plug gauges

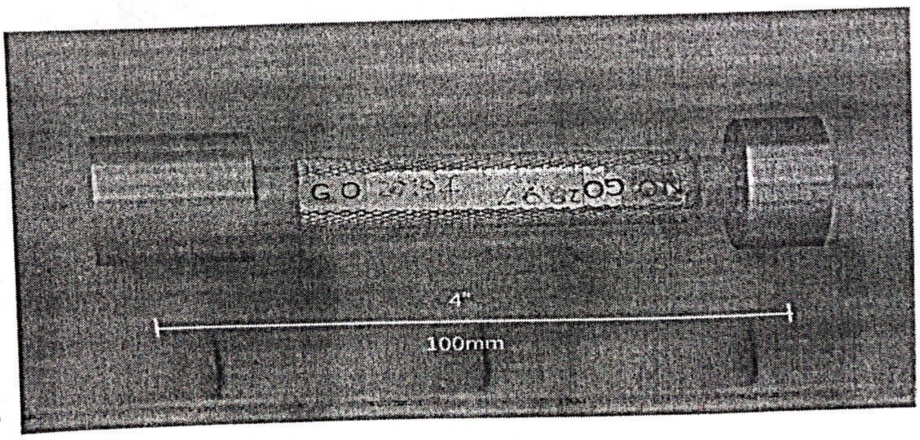


Snap gauges

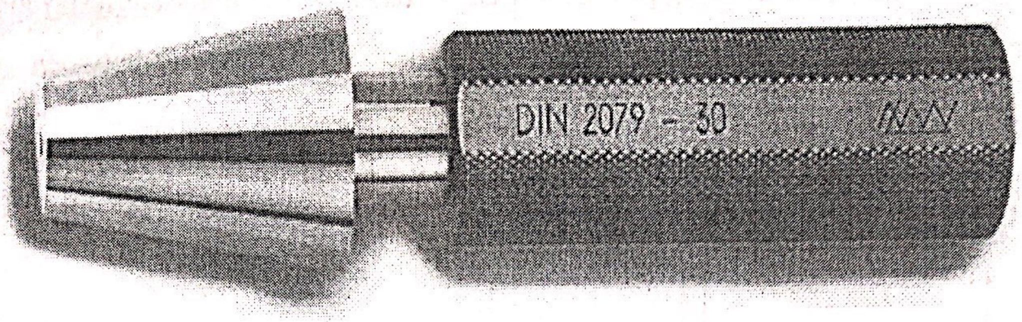
Types of gauges:

a.) PLUG GAUGE

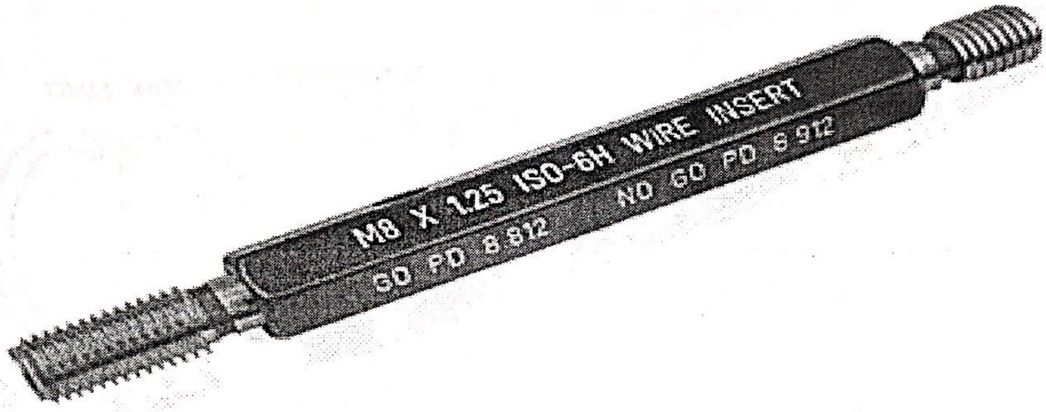
i. SOLID TYPE DOUBLE ENDED



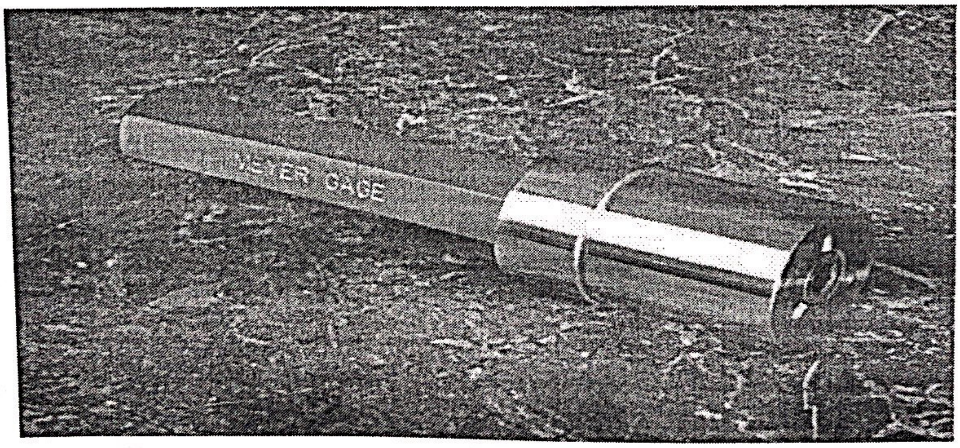
ii. TAPER INSERTED TYPE



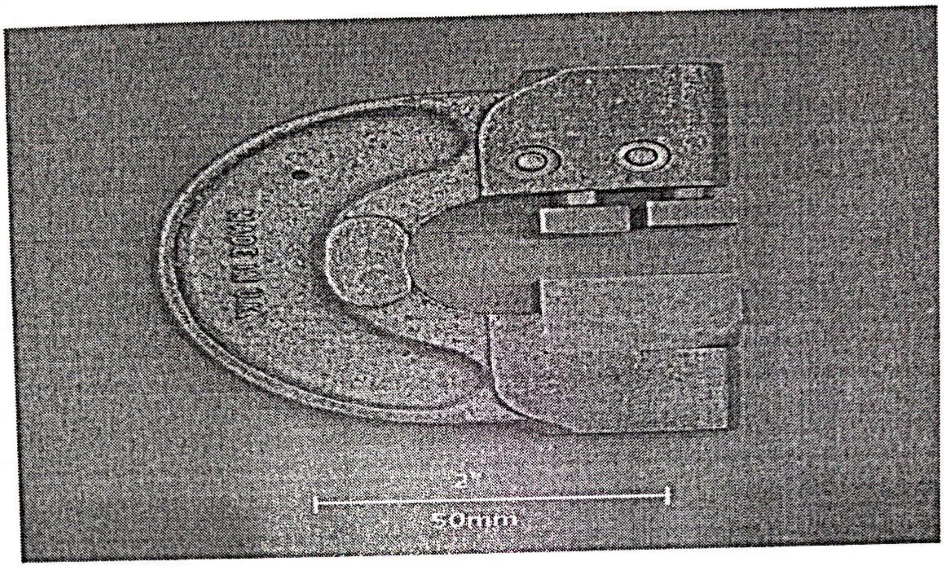
iii. FASTENED TYPE DOUBLE ENDED



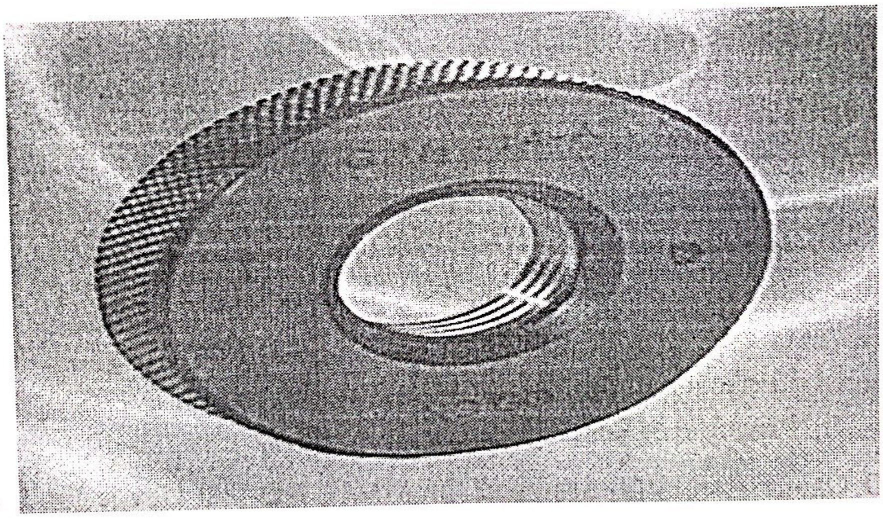
iv. PROGRESSIVE TYPE PLUG GAUGE



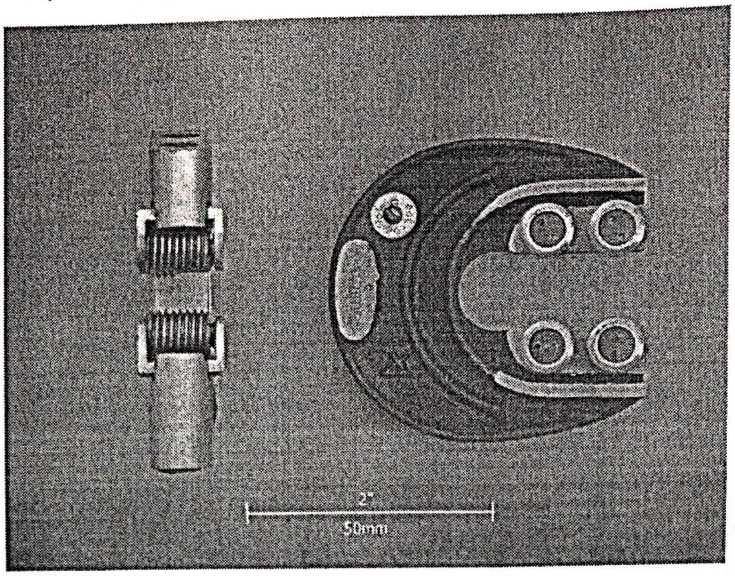
b.) SNAP GAUGE



c.) RING GAUGE



d.) THREAD SNAP GAUGE

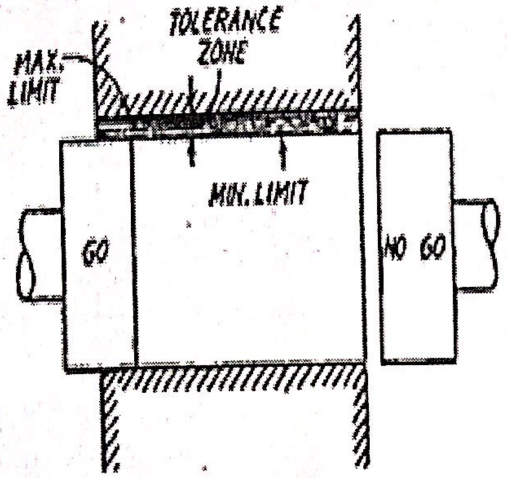


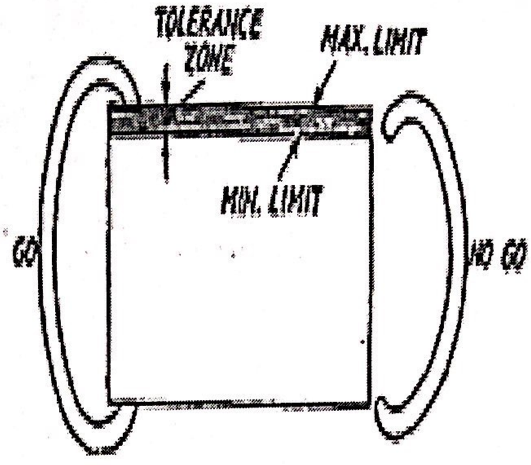
Material for gauge:

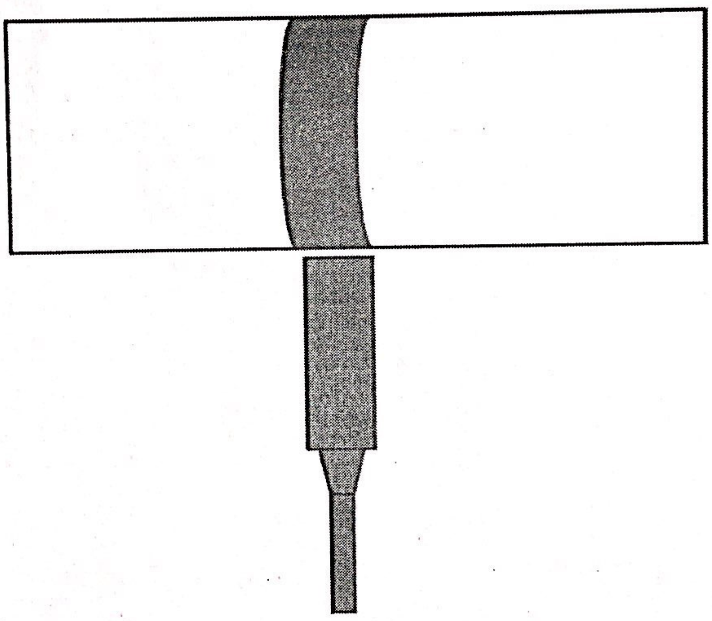
- Hardness to resist wear, Should be made with high carbon steel and heat treated to impart hardness
- Stability so that they won't change size and shape
- Corrosion resistant
- High degree of accuracy
- Low coefficient of thermal expansion
- Low thermal conductivity
- Coated products will be beneficiary

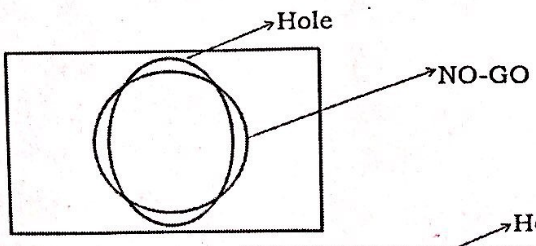
2.18 Taylors Principle of Gauge Design:

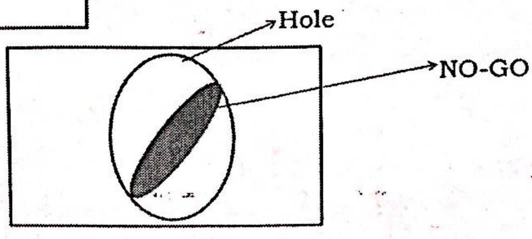
1. GO Gauge should be designed for MMC, NO-GO Gauge for LMC. If the plug gauge is check the hole, size of Go should be to lower limit and NO-GO to Upper limit. Similarly if snap gauge is used to check the shaft, size of GO should be to upper limit of shaft and NO-GO to lower limit.
2. Taylor's principle states that the "GO" gauge should check all the possible elements of dimensions at a time (Roundness, size, location, etc.) the no gauge should check only one element of the dimension at a time.
3. Holes that need to be inspected will in oval shape usually. So NO-GO will not enter and we wrongly prospect that NO-GO is not entering in hole so it's perfect.


TOLERANCE ZONE
MAX. LIMIT
MIN. LIMIT
GO
NO GO


TOLERANCE ZONE
MAX. LIMIT
MIN. LIMIT
GO
NO GO




Hole
NO-GO


Hole
NO-GO