

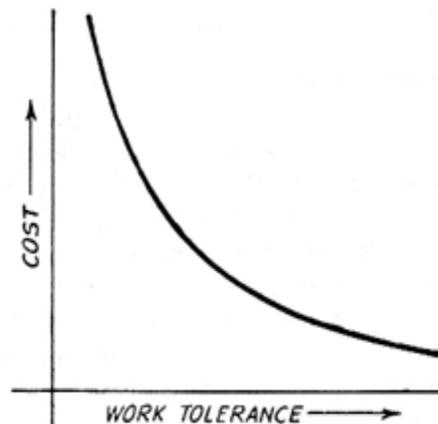
SYSTEM OF LIMITS, FITS, TOLERANCE AND GAUGING

INTRODUCTION:

It is well known fact that no two things in the nature can be identical, they may be found to be closely similar. This is true of production of component parts in engineering also. We know that every process is a combination of three elements, man, machine and material. A change in any one of these will constitute a change in the process. All these elements are subjected to inherent and characteristic variations.

Generally, in engineering, any component manufactured is required to fit or to match with some other component.

If a machine is under control, i.e. no assignable causes of variation exist, and then the resultant frequency distribution of dimension produced will be roughly in the form of normal curve, i.e. 99.7% parts will be within $\pm 3\sigma$ limits of means setting



The value of σ depends upon the machine used to produce a component. If value of σ has to be used reduced, then precision machines have to be used produces the component having less variation in dimensions. It is thus important to note that the cost of production keeps on increasing tremendously for very precise tolerance as shown in above **fig**, as the tolerance approaches zero, the task of achieving it becomes enormous and finally impossible .in general, tolerance vs. fabrication cost is hyperbolic curve.

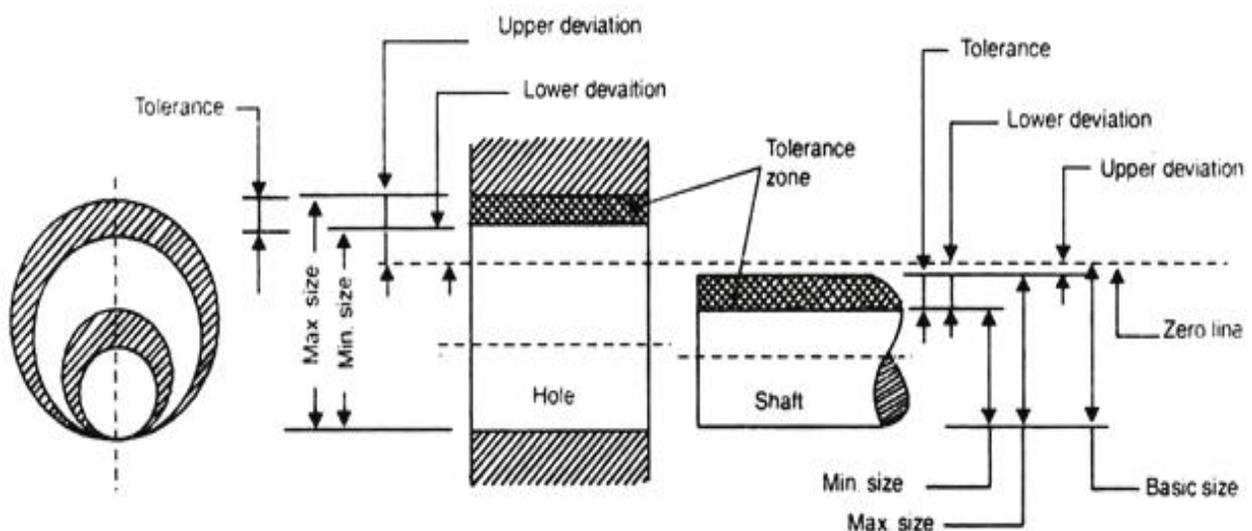
➤ LIMITS:

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

- Limits are fixed with reference to the basic size of that dimension.
- Upper limit (The high limit) for that dimension is the largest size permitted and the low limit is the smallest size permitted for that dimension.

TERMINOLOGY

The terminology used in fits and tolerances is shown in Fig below. The important terms are



Basic size: It is the exact theoretical size arrived at by design. It is also called nominal size.

Actual size: The size of a part as may be found by measurement.

Maximum limit of size: The greater of the two limits of size.

Minimum limit of size: The smaller of the two limits of size.

Allowance: It is an intentional difference between maximum material limits of mating parts. It is a minimum clearance or maximum interference between mating parts.

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

Actual deviation: The algebraic difference between the actual size and the corresponding basic size.

Upper deviation: The algebraic difference between the maximum limit of size and the corresponding basic size.

Upper deviation of hole = ES (& art Superior)

Upper deviation of shaft = es

Lower deviation: The algebraic difference between the minimum limit of size and the corresponding basic size.

Lower deviation of hole = EI (Ecart Inferior)

Lower deviation of shaft = ei

Upper deviation Lower deviation + Tolerance

Zero line: It is the line of zero deviation and represents the basic size.

Tolerance zone: It is the zone bounded by the two limits of size of the parts and defined by its magnitude, i.e. tolerance and by its position in relation to the zero line.

Fundamental deviation: That one of the two deviations which is conveniently chosen to define the position of the tolerance zone in relation to zero line, as shown in fig below.

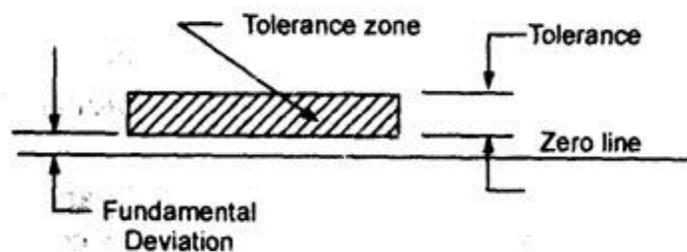


Fig: Disposition of fundamental deviation and tolerance zone with respect to the zero line

Basic shaft: A shaft whose upper deviation is zero.

Basic hole: A hole whose, lower deviation of zero.

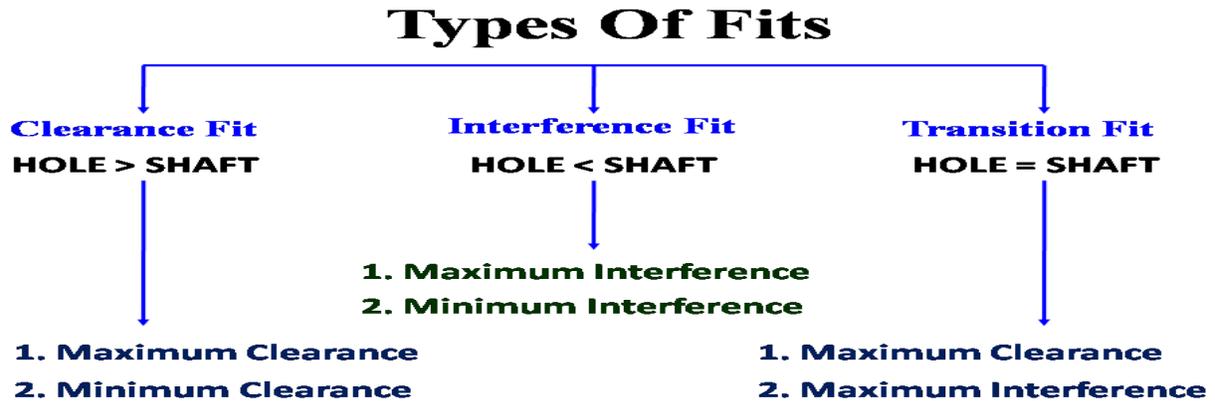
Clearance: It is the positive difference between the hole size and the shaft size.

Maximum clearance: The positive difference between the maximum size of a hole and the minimum size of a shaft.

Minimum clearance: The positive difference between the minimum size of a hole and the maximum size of a shaft.

➤ **FITS**

When two parts are to be assembled, the relation resulting from the difference between their sizes before assembly is called a fit. A fit may be defined as the degree of tightness and looseness between two mating parts.



(i) Clearance Fit:

This means there is a gap between the two mating parts. Let's see the following schematic representation of clearance fit. The diameter of the shaft is smaller than the diameter of the hole. There is a clearance between the shaft and the hole. Hence the shaft can easily slide into the hole.

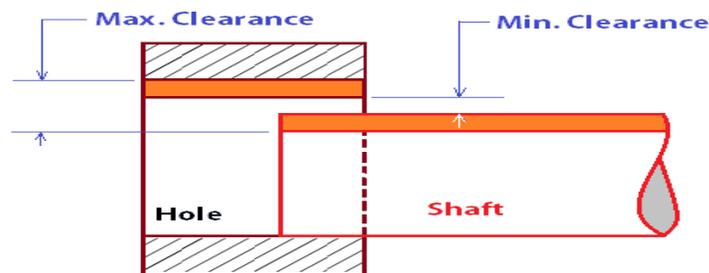


Figure: Clearance fit

In clearance fit the difference between the maximum size of the hole and the minimum size of the shaft is known as the **Maximum clearance** and the difference between the minimum size of the hole and the maximum size of the shaft is known as the **Minimum clearance**.

Clearance fit can be sub-classified as follows:

Loose Fit: 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.

Running Fit: For a running fit, the dimension of shaft should be smaller enough to maintain a film of oil for lubrication. It is used in bearing pair etc. An allowance 0.025 mm per 25 mm of diameter of boring may be used.

Slide Fit or Medium Fit: It is used on those mating parts where great precision is required. It provides medium allowance and is used in tool slides, slide valve, automobile parts, etc.

EXAMPLE:

Question: A spindle slides freely in a bush. The basic size of the fit is 50×10^{-3} mm. If the tolerances quoted are $0 +62$ for the holes and $-80 +180$ for the shaft, find the upper limit and lower limit of the shaft and the minimum clearance.

Solution: Tolerances are given in units of one thousandth of millimeter, so the upper limit of the hole will be 50.062 mm and lower limit for the hole is the same as the basic size of 50.000 mm.

The shaft upper limit will be $(50.000 - 0.080) \times 10^{-3} = 49.92 \times 10^{-3}$ m

The shaft lower limit will be $(50.000 - 0.180) \times 10^{-3} = 49.82 \times 10^{-3}$ m

The minimum clearance or allowance is $(50.000 - 49.920) \times 10^{-3} = 8 \times 10^{-3}$ mm

(ii) Interference Fit:

There is no gap between the faces and there will be an intersecting of material will occur. In the following schematic representation of the Interference fit. The diameter of the shaft is larger than the hole diameter. There will be the intersection of two mating components will be occurred. Hence the shaft will need additional force to fit into the hole.

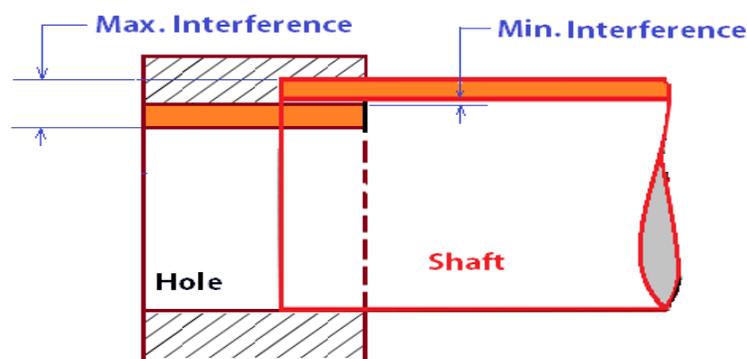


Figure: Interference Fit

In Interference fit the difference between the [maximum size](#) of the shaft and the minimum size of the hole is known as the **Maximum Interference** and the difference between the minimum size of the shaft and the maximum size of the hole is known as the **Minimum Interference**.

The interference fit can be sub-classified as follows:

Shrink Fit or Heavy Force Fit: It refers to maximum negative allowance. In assembly of the hole and the shaft, the hole is expanded by heating and then rapidly cooled in its position. It is used in fitting of rims etc.

Medium Force Fit: These fits have medium negative allowance. Considerable pressure is required to assemble the hole and the shaft. It is used in car wheels, armature of dynamos etc.

Tight Fit or Press Fit: One part can be assembled into the other with a hand hammer or by light pressure. A slight negative allowance exists between two mating parts (more than wringing fit). It gives a semi-permanent fit and is used on a keyed pulley and shaft, rocker arm, etc.

EXAMPLE

A dowel pin is required to be inserted in a base. For this application H 7 fit for hole and a p 6 fit for the shaft are chosen. The tolerance quoted is $0 +25$ for the hole and $26 +42$ for the shaft. Find the upper and lower limits of the hole and also dowel pin, and the maximum interference between dowel pin and the hole. The basic size of the fit is 50×10^{-3} m.

Solution:

The upper limit for the hole will be $(50.000 + 0.025) \times 10^{-3} = 50.025 \times 10^{-3}$ m

The lower limit for the hole will be $(50.000 + 0) \times 10^{-3} = 50 \times 10^{-3}$ m

The upper limit for dowel pin will be $(50.000 + 0.042) \times 10^{-3} = 50.042 \times 10^{-3}$ m

The lower limit for dowel pin will be $(50.000 + 0.026) \times 10^{-3} = 50.026 \times 10^{-3}$ mm

The maximum interference between dowel pin and the hole is

$$(50.042 - 50.000) \times 10^{-3} = 0.042 \times 10^{-3} \text{ m} = 42 \times 10^{-6} \text{ m}$$

(iii) Transition Fit:

Transition fit is neither loose nor tight as like clearance fit and interference fit. The [tolerance zones](#) of the shaft and the hole will be overlapped between the interference and clearance fits.

See the following schematic representation of the transition fit.

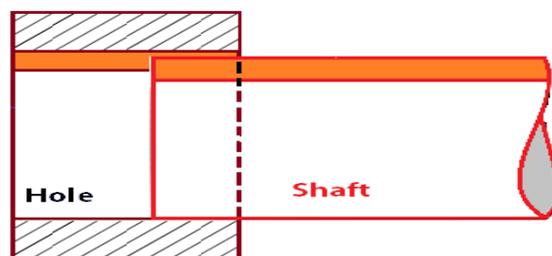


Figure: Transition Fit

Transition fit can be sub-classified as follows:

Push Fit: It refers to zero allowance and a light pressure (10 cating dowels, pins, etc.) is required in assembling the hole and the shaft. The moving parts show least vibration with this type of fit. It is also known as snug fit.

Force Fit or Shrink Fit: A force fit is used when the two mating parts are to be rigidly fixed so that one cannot move without the other. It either requires high pressure to force the shaft into the hole or the hole to be expanded by heating. It is used in railway wheels, etc.

Wringing Fit: A slight negative allowance exists between two mating parts in wringing fit. It requires pressure to force the shaft into the hole and gives a light assembly. It is used in fixing keys, pins, etc.

EXAMPLE:

For a particular application, an H 7 fit has been selected for the hole and a K 6 fit for the shaft. The tolerance quoted are $0 +25$ for the hole and $12 +18$ for the shaft. Find the upper limit and lower limit for the hole and also for bush. The basic size of fit is 50×10^{-3} m.

Solution:

The upper limit for the hole will be $(50.000 + 0.025) \times 10^{-3} = 50.025 \times 10^{-3}$ m

The lower limit for the hole will be $(50.000 + 0) \times 10^{-3} = 50.000 \times 10^{-3}$ m

The upper limit for the bush will be $(50.000 + 0.018) \times 10^{-3} = 50.018 \times 10^{-3}$ m

The lower limit for the bush will be $(50.000 + 0.002) \times 10^{-3} = 50.002 \times 10^{-3}$ m

SYSTEMS OF FITS:

A fit system is the systems of standard allowance to suit specific range of basic size. If these standard allowances are selected properly and assigned in mating parts ensures specific classes of fit.

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 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 high limit of the hole and the two limits of size for the shaft are then varied to give desired type of fit. 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.

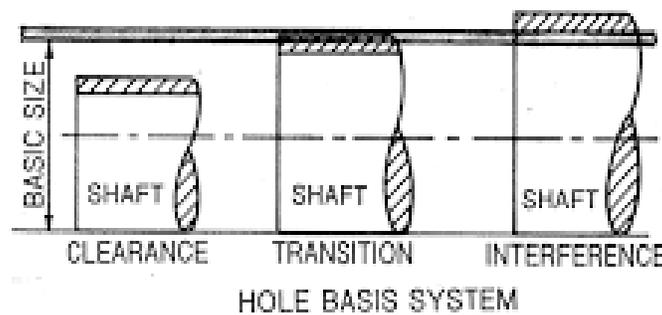


Fig: Hole basis system

2. Shaft Basis System:

In the shaft basis system, the size of the shaft is kept constant and different fits are obtained by varying the size of the hole. Shaft basis system is used when the ground bars or drawn bars are readily available. These bars do not require further machining and fits are obtained by varying the sizes of the hole. In this system, the upper deviation (fundamental deviation) of shaft is zero, i.e. the high limit of the shaft is same as basic size and the various fits are obtained by varying the low limit of shaft and both the limits of the hole.

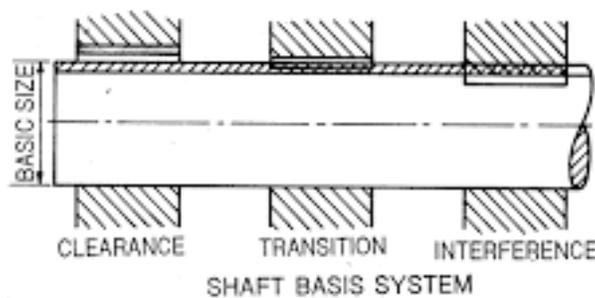


Fig: Shaft Basis System

DIFFERENCE BETWEEN HOLE BASIS & SHAFT BASIS SYSTEM:

BASIS SYSTEM	SHAFT SYSTEM
1. Size of hole whose lower deviation is zero (H.hole) is assumed as the basic size.	1. Size of shaft whose upper deviation is zero (h-shaft) assumed as basic size.
2. Limits on the hole are kept constant and those of shafts are varied to obtain desired type of fit.	2. Limits on the shaft are kept constant and those of holes 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 reamers.	3. This system is not suitable for mass production because it is inconvenient time consuming and costly to make a hole of any size w.r to field shaft size so as to obtain required fit.
4. It is much more easy to vary the shaft sizes according to the fit required.	4. It is rather difficult to vary the hole sizes according to the fit required.
5. It required less amount of capital and storage space for tools needed to produce shaft of different sizes.	5. It needs large amount of capital and storage space for large numbers of tools required to produce holes of different sizes.
6. Changing of shafts can be easily and conveniently done with suitable gap Gauges	6. Being internal measurement gauging of holes can't be easily and conveniently done.

TOLERANCES:

Tolerance is a permissible limit and variation in dimensions or in physical parameters. It is possible to achieve dimensions and physical parameter exactly but it is very time consuming and economically unjustified or costly. It is quite often not necessary to achieve exact dimensions, in such functions or cases some permissible variation or tolerance is given.

The permissible variation in size or dimension is tolerance. The difference between the upper limit (high limit) and the lower limit of a dimension represents the margin for variation to 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.

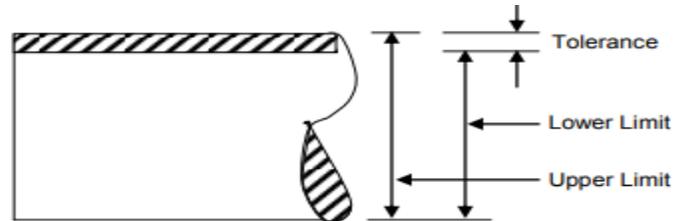


Fig: Tolerance

There are two ways of writing tolerances:

- (a) Unilateral tolerance
- (b) Bilateral tolerance.

Unilateral Tolerance:

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

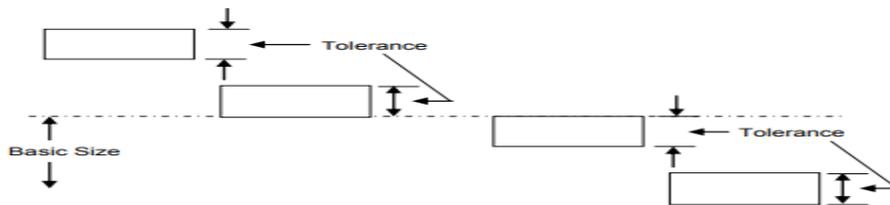


Fig: unilateral Tolerance

Examples of unilateral tolerance are :

$$25^{+0.02}_{+0.01}, 25^{-0.02}_{-0.01}, 25^{-0.01}_{-0.02}, 25^{+0.0}_{-0.02} \text{ etc.}$$

Unilateral system is preferred in interchangeable manufacture, especially when precision fits are required, because

- (a) it is easy and simple to determine deviations,
- (b) another advantage of this system is that „Go“ Gauge ends can be standardized as the holes of different tolerance grades have the same lower limit and all the shafts have same upper limit, and
- (c) This form of tolerance greatly assists the operator, when machining of mating parts. The operator machines to the upper limit of shaft (lower limit for hole) knowing full well that he still has some margin left for machining before the parts are rejected.

Bilateral Tolerance:

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 disposing about it.

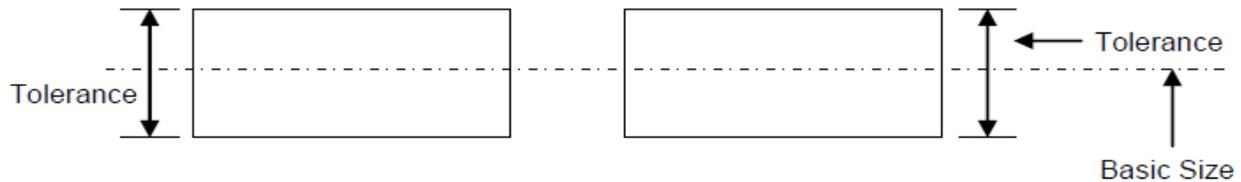


Fig: Bilateral Tolerance

Examples of bilateral tolerance are :

$$25^{\pm 0.02}, 25_{-0.01}^{+0.02} \text{ etc.}$$

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 are to be varied. This system is used in mass production when machine setting is done for the basic size.

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 0.075 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.

High limit of hole = Low limit + Tolerance

$$= 50.00 + 0.050$$

$$= 50.050 \text{ mm} = 50.050 \times 10^{-3} \text{ m}$$

High limit of shaft = Low limit of hole – Allowance

$$= 50.00 - 0.075$$

$$= 49.925 \text{ mm} = 49.925 \times 10^{-3} \text{ m}$$

Low limit of the shaft = High limit – Tolerance

$$= 49.925 - 0.050$$

$$= 49.875 \text{ mm} = 49.875 \times 10^{-3} \text{ m}$$

The dimension of the system is shown in Figure

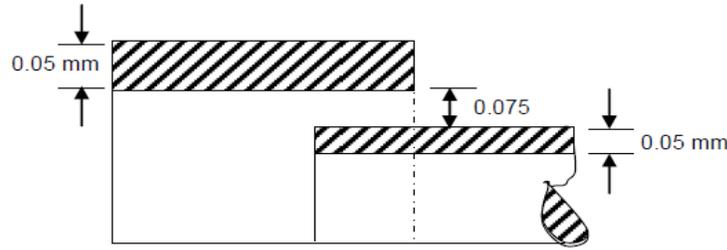


Fig: Shaft with Bush

GEOMETRICAL TOLERANCES

Geometric means geometric forms such as a plane, cylinder, square, etc. Geometrical features are: flatness, straightness, squareness etc. Geometrical tolerances refer to the shape of the surfaces (tolerance of form) as well as the relative location of one feature to another (tolerance of position). These tolerances are specified by special symbols (refer Tables 1 and 2).

Table 1 : Symbol Specifying the Shape (Tolerance of Form)

Types of Error		Symbol
Briefly	Interpretation	
Flatness	Deviation from a flat Surface	—
Straightness	Deviation from a straight line	—
Cylindricity	Deviation from true cylinder	ϕ
Circularity or roundness	Deviation from true circle	O
Accuracy of any surface	—	C

Table 2 : Symbol Specifying the Relative Location (Tolerance of Position)

Types of Error		Symbol
Briefly	Interpretation	
Parallelism	Lack of parallelism	//
Squareness and Perpendicularity	Lack of squareness	\perp
Concentricity	Lack of concentricity	\odot
Symmetry	Lack of symmetry	\equiv

SYSTEM OF LIMITS, FITS, TOLERANCES AND GAUGING

Geometrical tolerances are specified for geometrical features, in addition to linear tolerances. Data about the tolerances on the shape and location of surfaces are indicated on drawings in a rectangular box divided into two or three parts. For example “Lack parallelism between two surfaces is within 0.1 mm” can be written as

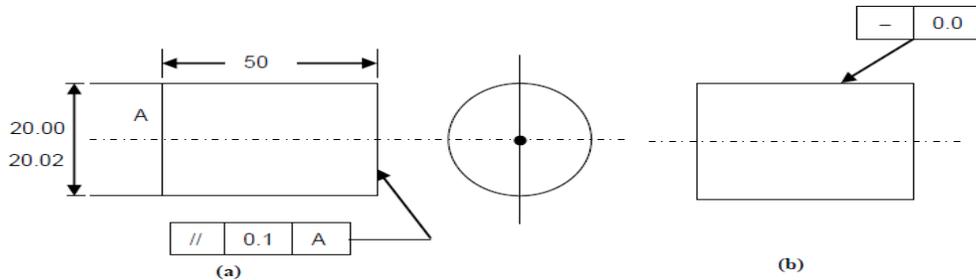


Examples of geometrical tolerances are given below :

Parallelism (Figure (a))

It indicates the requirement, “Surface A is parallel to opposite face within 0.1 mm”.

Straightness (Figure (b))

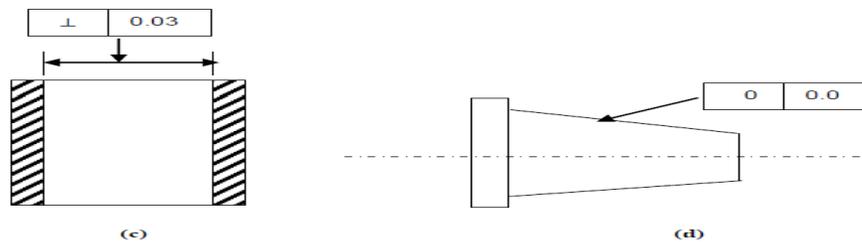


It indicates the requirement, “Straight within 0.02 mm”.

Squareness (Figure (c))

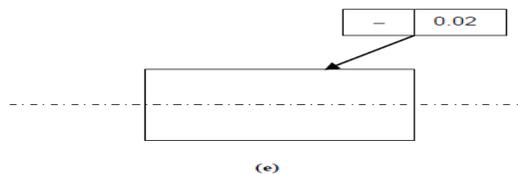
It indicates the requirement, “Square within 0.03 mm total”.

Flatness (Figure (d))



It indicates the requirement, “Flat within 0.002 mm total”.

Roundness (Figure (e))



It indicates the requirement, “Taper round within 0.01 mm”.

GAUGES:

Limit Gauges:

Two sets of limit gauges are necessary for checking the size of various parts. There are two gauges: Go limit gauge, and Not Go limit gauge.

1. **Go Limit:** The Go limit applied to that of the two limits of size corresponds to the maximum material condition, i.e. (1) an upper limit of a shaft, and (ii) the lower limit of a hole. This is checked by the Go gauge.

2. **Not Go Limit:** The Not Go limit applied to that of the two limits of size corresponds to the minimum material condition, i.e. (1) lower limit of a shaft, and (ii) the upper limit of a hole. This is checked by the Not Go gauge.

The types are:

1. Plug Gauge
3. Snap Gauge
4. Ring Gauge

1. Plug Gauge:

A plug gauge is a cylindrical type of gauge, used to check the accuracy of holes. The plug gauge checks whether the whole diameter is within specified tolerance or not. The 'Go' plug gauge is the size of the low limit of the hole while the 'Not-Go' plug gauge corresponds to the high limit of the hole.

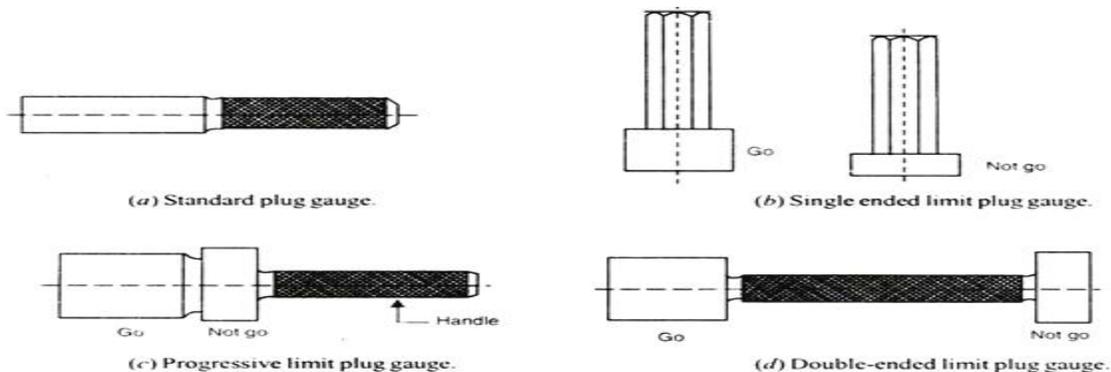


Fig: Types of Plug gauges

It should engage the hole to be checked without using pressure and should be able to stand in the hole without falling.

Snap Gauge:

A snap gauge is a U-Shaped frame having jaws, used to check the accuracy of shafts and male members. The snap gauge checks whether the shaft diameter is within specified tolerances or not.

The ‘Go’ snap gauge is the size of the high (maximum) limit of the shaft while the ‘Not-Go’ snap gauge corresponds to the low (minimum) limit of the shaft.

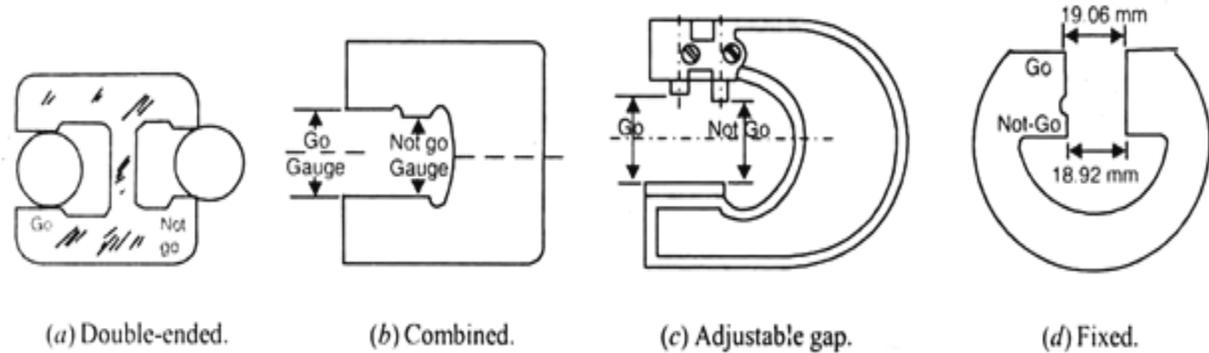


Fig: Types of Snap gauges

Ring Gauge:

A ring gauge is in the form of a ring, used to check the shafts and male members. The ‘Go’ and ‘Not Go’ members may be separate or in a single ring. The opening or hole in the Go gauge is larger than that in the Not-Go gauge.

A ring gauge with both members combined in one ring is shown in figure (a):

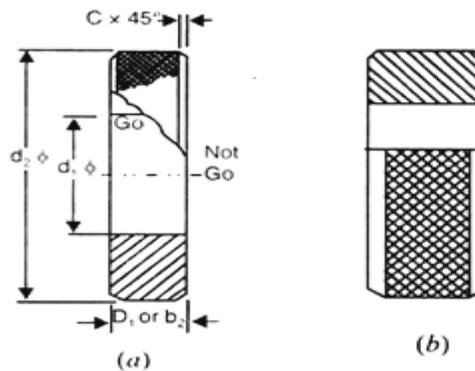


Fig: Plain ring gauge

PROPERTIES OF GAUGE MATERIAL:

The material for limit gauges should meet most of the following requirements:

(i) Optimal Hardness:

This is primary and most important property of gauge material. It is concerned with high durability, resistance to wear, and resistance to damage in use.

(ii) Stability of Dimensions:

The material should have high stability of dimensions to preserve size and form.

(iii) Proper Workability:

Proper workability, especially in manufacturing processes like grinding and polishing, to obtain required accuracy.

(iv) Wear and Corrosion Resistance:

The material should have high resistance to mechanical wear and corrosion.

(v) Low Coefficient of Linear Expansion:

The material should have low coefficient of linear expansion to avoid temperature and heating effect.

(vi) Uniformity of Structure:

The structure of gauge material should be uniform for better accuracy.

TAYLOR'S PRINCIPLE OF GAUGE DESIGN

The Taylor's Principle of gauge design gives two statements which are discussed here:

Statement 1:

The "Go" gauge should always be so designed that it will cover the maximum metal condition (MMC), whereas a "NOT-GO" gauge will cover the minimum (least) metal condition (LMC) of a feature, whether external or internal.

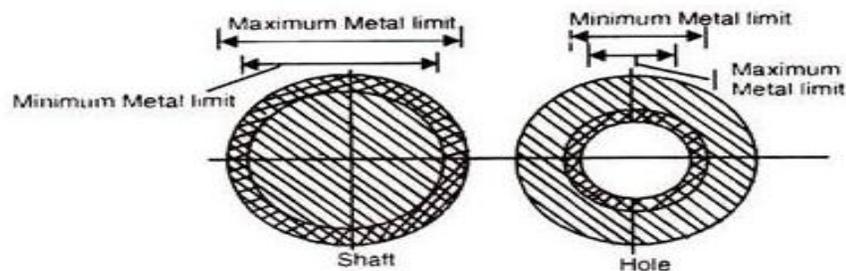


Fig. 1 . Maximum and minimum metal condition for Taylor's principle.

Statement 2:

The “Go” gauge should always be so designed that it will cover as many dimensions as possible in a single operation, whereas the “NOT-GO” gauge will cover only one dimension.

Means a Go plug gauge should have a full circular section and be of full length of the hole being checked as in shown figure 2:

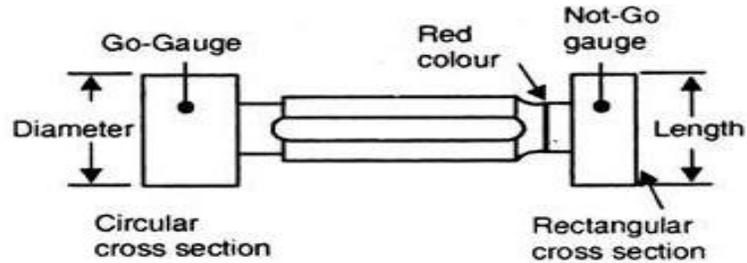


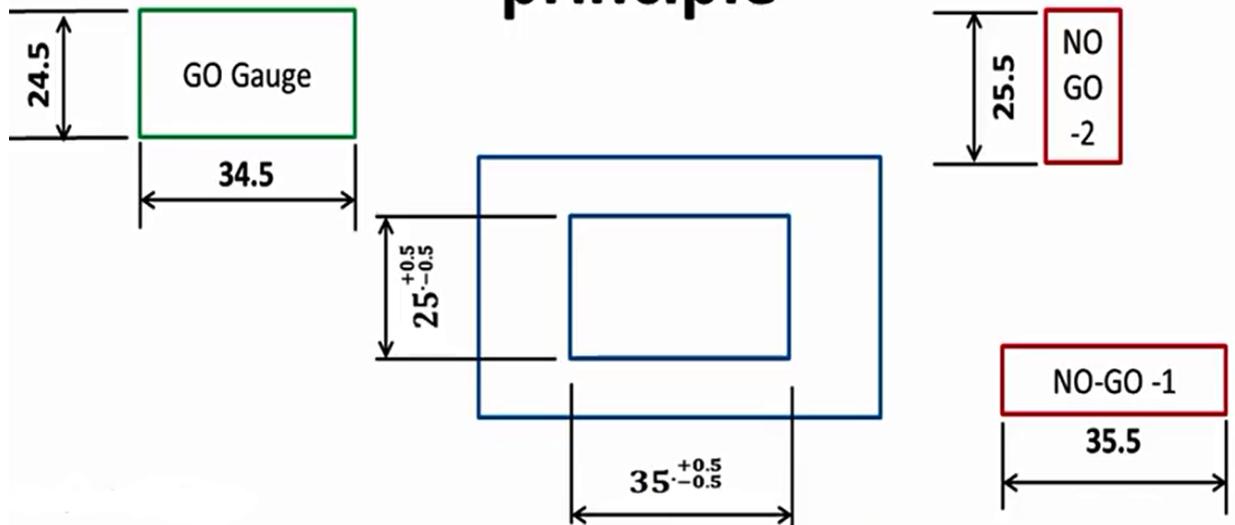
Fig. 2. Go and Not-Go gauges for hole (Taylor’s principle).

OR

i.e. According to Taylor’s principle, the GO gauge should be made for maximum material limit and it has to incorporate as many dimensions as possible to inspect in one pass and NO-GO gauge can be made for minimum material limit and separate NO GO gauge should be made for each separate dimension.

Example:

Gauge design comply with Taylor’s principle



Gauge design not comply with Taylor's principle

