ROHINI COLLEGE OF ENGINEERING AND TECHNOLOGY

ACCREDITED WITH NAAC A+ GRADE

DEPARTMENT OF MECHANICAL ENGINEERING

ME 3592 – METROLOGY AND MEASUREMENTS

SEMESTER: 05

III YEAR

REGULATION 2021

PREPARED BY

MR.P. NAVIN JASS, M.E, (Ph.D).,

AP/MECH,

RCET.

UNIT – III TOLERANCE ANALYSIS

Tolerancing– Interchangeability, Selective assembly, Tolerance representation, Terminology, Limits and Fits, Problems (using tables IS919); Design of Limit gauges, Problems. Tolerance analysis in manufacturing, Process capability, tolerance stackup, tolerance charting.

Prepared By

P. Navin Jass,

Assistant Professor,

Department of Mechanical Engineering,

Rohini College of Engineering & Technology.

3.1 TOLERANCING

Tolerancing is the process of defining the permissible variation in the dimensions of a part or assembly. It is used to ensure that the parts will fit together and function properly. Tolerances are typically expressed as a range of values, with a lower limit and an upper limit. The difference between the two limits is the tolerance.

Tolerancing is an important part of engineering design. It helps to ensure that the parts of an assembly will be compatible and that the assembly will function as intended. Tolerances are also used to control the cost of manufacturing, as tighter tolerances can lead to higher costs.

There are two main types of tolerancing: geometric tolerancing and dimensional tolerancing. Geometric tolerancing is used to control the shape and orientation of features on a part. Dimensional tolerancing is used to control the size and location of features on a part.

3.1.1 Need of Tolerance in Measurements:

Certain amount of variation is inherent in all manufactured parts. Tolerances are used to control those variations, ensuring greater consistency and proper performance from the parts. Because different materials and manufacturing methods can affect tolerances, it is important to choose an experienced plastics manufacturer who understands the different tolerances each plastic can sustain.

3.2 Interchangeability

An interchangeable part is one which can be substituted for similar part manufactured to the same drawing. In earlier times production used to be confined to small number of units and the same operator could adjust the mating components to obtain desired fit. With time the concept of manufacturing techniques kept on changing and today the same operator is no more responsible for manufacture and assembly too. With economic oriented approach, mass production techniques were inevitable, that led to breaking up of a complete process into several smaller activities and this led to specialisation. As a result various mating components will come from several shops, even a small component would undergo production on several machines. Under such conditions it becomes absolutely essential to have strict control over the dimensions of portions which have to match with other parts. Any one component selected at random should assemble correctly with any other mating component, that too selected at random. When a system of this kind is ensured, it is known as interchangeable system.

Interchangeability ensures increased output with reduced production cost. In interchangeable system, every operator being concerned only with a limited portion of overall work, he can easily specialise himself in that work and give best results leading to superior quality. He need not waste his skill in fitting the components by hit and trial and assembly time is reduced considerably. In the case of big assemblies, several units to manufacture individual parts can be located in different parts of country depending on availability of specialised labour, raw material, power, water and other facilities and final assembly of all individual components manufactured in several units can be done at one place.

The replacement of worn out or defective parts and repairs is rendered very easy and the cost of maintenance is very much reduced and shut down time also reduced to minimum. Interchangeability is possible only when certain standards are strictly followed. Universal interchangeability (i.e. parts drawn from any two altogether different manufacturing sources for mating purposes) is desirable and for this it is essential that common standards be followed by all, and all standards used by various manufacturing units should be traceable to a single source, i.e. international standards.

When all parts to be assembled are made in the same manufacturing unit, local standards may be followed (condition being known as local interchangeability) but for reasons of obtaining spares from any other source it is again desirable that these local standards be also traceable to international standards. The required fit in an assembly can be obtained in two ways, namely

(i) universal or full interchangeability, and

(ii) selective assembly.

i. Universal Or Full Interchangeability

Full interchangeability means that any component will mat with any other mating component without classifying manufactured components in subgroup or without carrying out any minor alterations for mating purposes. This type of interchangeability is not must for interchangeable production and many times not feasible also as it requires machines capable of maintaining high process capability and very high accuracy, and very close supervision on production from time to time. (Process capability of a machine is defined as its ± 3o spread of dimensions of components produced by it.

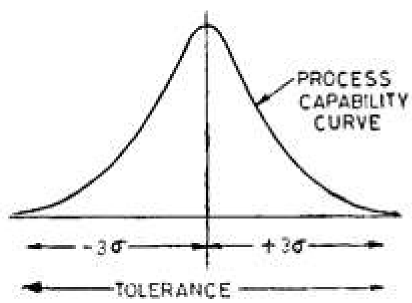

Fig 3.1 Process Capability Curve

If a plot is drawn of the actual dimensions of the similar components produced by a machine, it is found to follow natural law of distribution, i.e. having mean of all the components at central value with a spread of ± 3o value, a being known as standard deviation, and ±3 the process capability of machine). For full interchangeable assembly it is essential that only such machines be selected for manufacturing whose process capability is equal to or less than the manufacturing tolerance allowed for that part. Only then every component will be within desired tolerance and capable of mating with any other mating component.

3.3 Selective Assembly.

The consumer not only wants quality, precision and trouble-free products but also he wants them at attractive prices. This has become possible only by adopting automatic gauging for selective assembly whereby parts manufactured to rather wide tolerances fit and function as though they were precisely manufactured in precision laboratory to very close tolerances. This is a concept which does away with old idea of inspection in which part is identified as 'good' or 'bad'; good part being used for assembly and bad used to be scrapped. In selective assembly the components produced by a machine are classified into several groups according to size. This is done both for hole and shaft and then the corresponding groups will match properly. If some parts (shaft and holes) to be assembled are manufactured to normal tolerances of 0.01 mm (and both are within the curve of normal distribution), an automatic gauge can segregate them into ten different groups with a 0.001 mm limit for selective assembly of the individual parts.

Thus parts with tolerances of 0.0001 mm are obtained (due to segregation) and both the conditions of high quality and low cost can be served by selective assembly technique. However, it is very important that the two component parts to be fitted together must be kept within the normal distribution, i.e. the central or mean value should be at desired calculated value and the process capability of two machines producing shafts and holes must be identical otherwise for some components the mating components will not be available.

The process capability of both shaft and hole producing machines is same but tolerances on parts are desired as one-tenth of process capability of machines. In such a case the parts are segregated by automatic inspection into ten groups and parts in shaft region Si are matched with parts in hole region Hh S2 with H2 and so on. This results in matching of parts having tolerances l/10th of machine capability. In this case as the process capability of both machines is same, equal number of parts are available in each segregated zone and no wastage will be there. the process capability of hole making machine is much wider than the tolerance of part but shaft making machine can produce component to the desired tolerance. In such a case the parts with hole are segregated into adequate number of groups depending on the desired tolerance.

$$\text{No. of groups} = \frac{\text{Process capability}}{\text{Tolerance desired}}$$

This curve when broken into several groups enables to determine the number of components likely to be produced in each group. The shafts are ‘therefore’ so produced that their setting needs to be changed as many numbers of times as there are groups formed for the holes and at the mean value of each sub-group. The number of shafts to be produced for each setting is determined by earlier curve for holes.

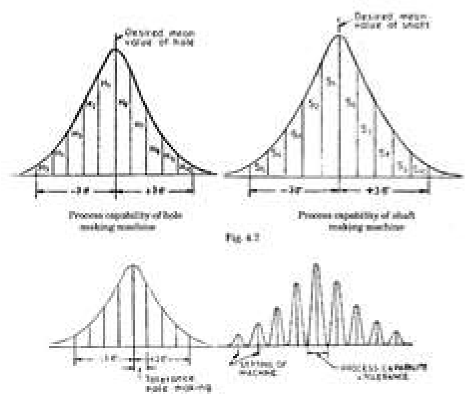

Fig 3.2 Shaft production (Area under curve dictates number of parts to be produced for each setting)

Advantages of selective assembly in Manufacture:

- Selective assembly is the fair, clear and low cost method in manufacturing.
- It increases the efficiency
- This gives high quality in assembly.
- Cost of manufacturing is reduced.
- Scrab rate in manufacturing reduces by this method.
- It reduces the machining cost. and increases the efficiency of fit without reducing the tolerance zone.

Disadvantages of selective wave soldering

(1) The purchase cost is too high, many small and medium manufacturers do not use the conditions of selective wave soldering Selective wave soldering is more complicated than usual wave soldering, and its manufacturing cost is relatively high. In addition, most of the machines are imported equipment, and only a few domestically produced machines keep the price of equipment high.

(2) **Inefficiency** Selective wave soldering is much better than ordinary wave soldering in controlling the quality of solder joints, but the manufacturing efficiency is very low, which is also the Achilles heel of selective wave soldering.

3.4 Tolerance Representation

Tolerances are assigned to mating parts in an assembly. For example, in case, when the slot in the part must accommodate another part. One of the great advantages of using tolerances is that it allows for interchangeable parts, thus permitting the replacement of individual parts.

Tolerances can be expressed in several ways:

1. Direct limits, or Limits of size.
2. Geometric tolerances, indicated by special symbols related to part surfaces
3. single limit tolerance

3.4.1 Direct limits, or Limits of size.

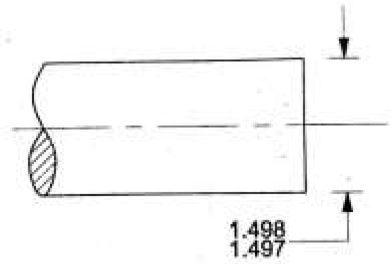

Fig 3.3 Representation of Direct limits

Tolerances can be applied directly to dimensioned features using limit It. the American Society of Mechanical Engineers (ASME) preferred method such as m and minimum sizes specified as part of the dimension.

Either the upper limit is placed above the lower limit or when the dimension is a single line, the lower limit precedes the upper limit, and they are separated by a dash or slash. For example, 1.498 — 1.497 or 1.498/1.497.

3.4.2 Geometric tolerances

Geometric tolerances specify the maximum variation that is allowed in form or position from true geometry. They are indicated by special symbols related to part surfaces. It is usually placed next to the corresponding dimensions. For example, the diameter of tolerance zone within which a surface or axis of hole or cylinder can lie which results in resulting feature being acceptable for proper function and interchangeability.

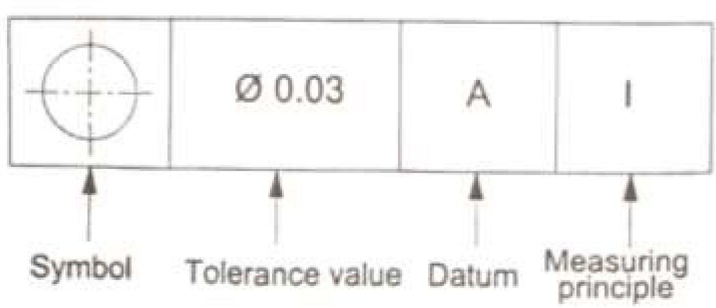

Fig 3.4 Representation of Geometric tolerances

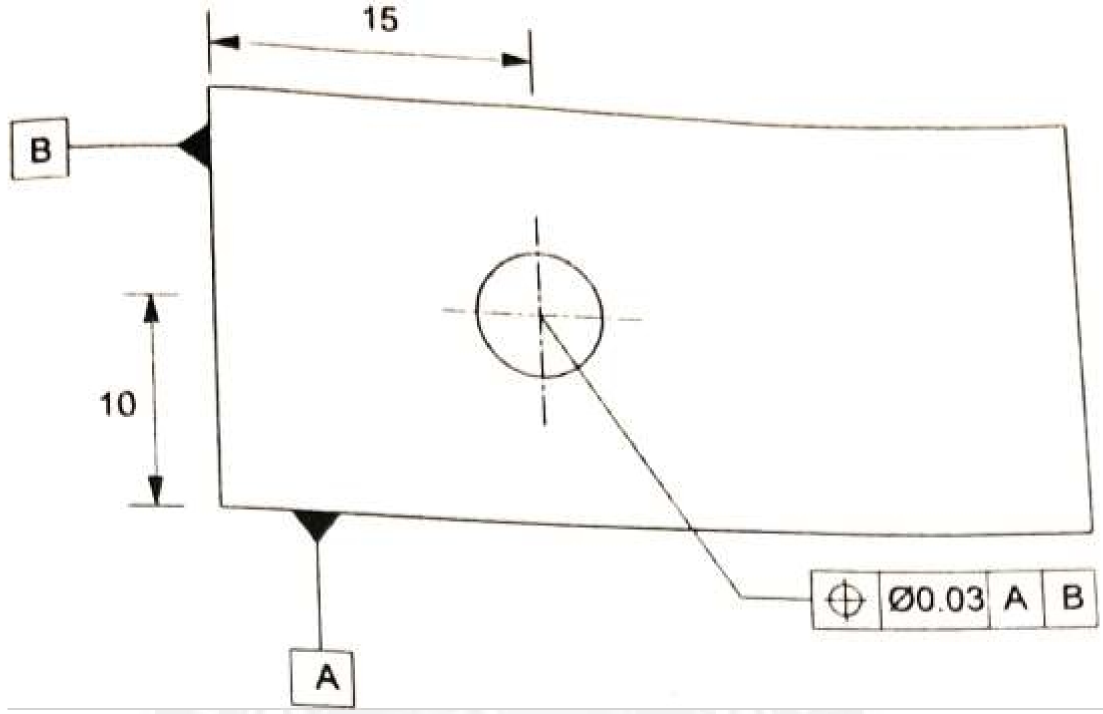

Fig 3.5 Example of Geometric tolerances

3.4.3 Single Limit Tolerance

A **Single Limit Tolerance** only defines one limit dimension, normally either the maximum or minimum value for a feature or dimension.

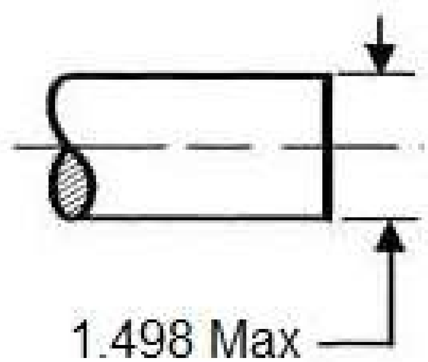

Fig 3.6 Example of Single Limit tolerances

3.4.4 Unilateral Tolerance

The **Unilateral Tolerance** shows the nominal dimension (1.498) and a tolerance in only one direction -.001.

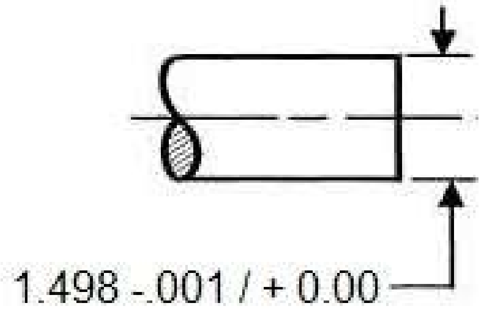

Fig 3.7 Example of Unilateral Tolerance

Unilateral tolerance is indicated by setting the limit either the maximum or minimum size of a feature or a space just by leaving the other limit of size unspecified. When tolerance is specified on one side of the basic size (either positive or negative), it is called Unilateral tolerance.

3.4.5 Bilateral Tolerance

The **Bilateral Tolerance** shows the nominal dimension (1.498) and the allowable tolerance in either direction + .001.

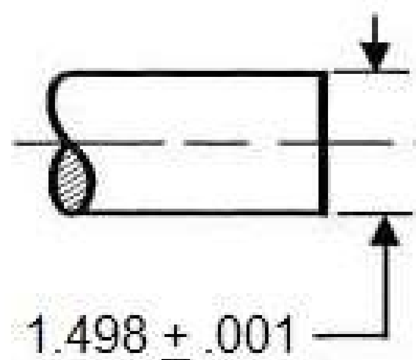

Fig 3.8 Example of Bilateral Tolerance

A bilateral tolerance is a type of tolerance that specifies the allowable variation in both the positive and negative directions from the nominal size. Bilateral tolerances are typically expressed as a plus-minus sign, followed by the amount of tolerance. For example, a bilateral tolerance of 0.010 might be expressed as "0.010 +/-". This means that the part can be any size between 0.010 inches smaller than the nominal size and 0.010 inches larger than the nominal size.

3.4.6 Representation of tolerances in angular dimensions

For angular dimensions also, the above methods are used. But in angular dimensions, the measured unit is degree or minutes or seconds.

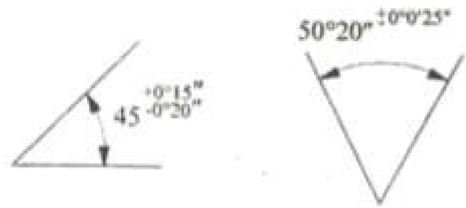

Fig 3.9 Indicating tolerances in angular dimensions

3.4.7 General tolerance

An engineering drawing may include general tolerances in the form of a table or just a little note somewhere on the drawing. They can be applied to several conditions, including linear dimensions, angular dimensions, external radius, chamfer heights, etc

ISO 2768-1 stands for the general tolerances for linear and angular dimensions without individual tolerance indications, ISO 2768-1 indicates the linear dimensions and angular dimensions such as external sizes, internal sizes, step sizes, diameters, radii, distances, external radii' and chamfer heights for broken edges. This standard covers general tolerances in three 4 classes of tolerance:

- M - Medium tolerances
- F - Fine tolerances
- C - Coarse tolerances
- V - Very coarse tolerances.

3.5 Terminologies used in TOLERANCES

Hole:

It is a term used to specify the internal features of parts.

Shaft:

It is a term used to specify the external features of parts.

Size:

It is the numerical value of a linear dimension in a particular unit.

Basic Size:

It is the standard size of a part, with reference to which, all the limits of variations of size are determined.

Actual Size:

It is the measured size of job. The difference between the basic size and the actual size should not exceed a certain limit.

Zero Line:

The line corresponding to basic size is called as zero line. It is the line of zero deviation.

Basic Shaft:

It is the shaft, whose upper deviation is zero or whose maximum limit of size is equal to basic size.

Basic hole:

It is the hole, whose lower deviation is zero or whose minimum limit of size is equal to basic size.

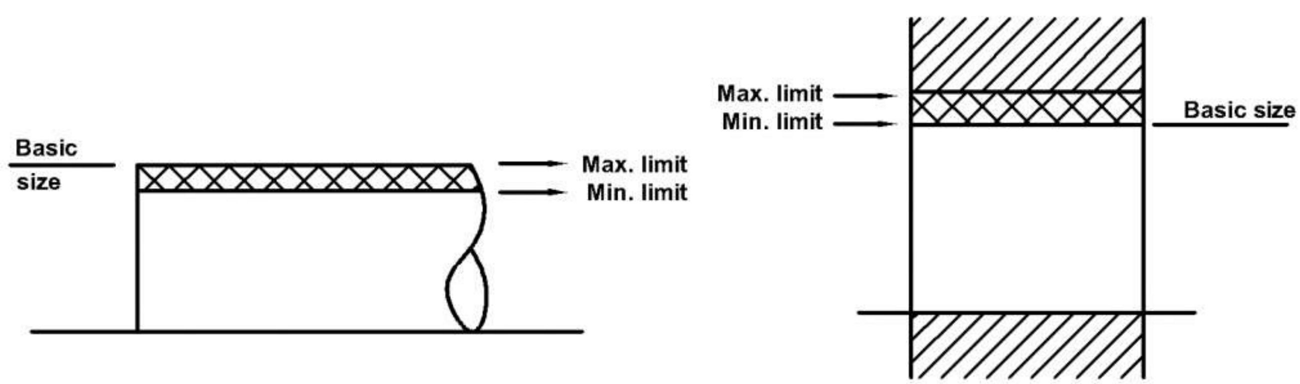

Fig 3.10 Basic Shaft and Basic Hole

Maximum Limit of Size:

It is the term maximum limit of size referred to the maximum or greatest permissible size of a feature.

Minimum Limit of Size:

It is the term minimum limit of size referred to the minimum or smallest permissible size of a feature.

Deviation:

It is the amount by which the size of a part deviates from its basic size. Hence, it is the algebraic difference between actual size and basic size.

The upper deviation is the algebraic difference between the maximum limit and the basic size.

The lower deviation is the algebraic difference between the minimum limit and the basic size.

The mean deviation is the arithmetical mean between the upper and the lower deviations.

The fundamental deviation is either upper deviation or lower deviation, which is nearest one to the zero line for either hole or a shaft.

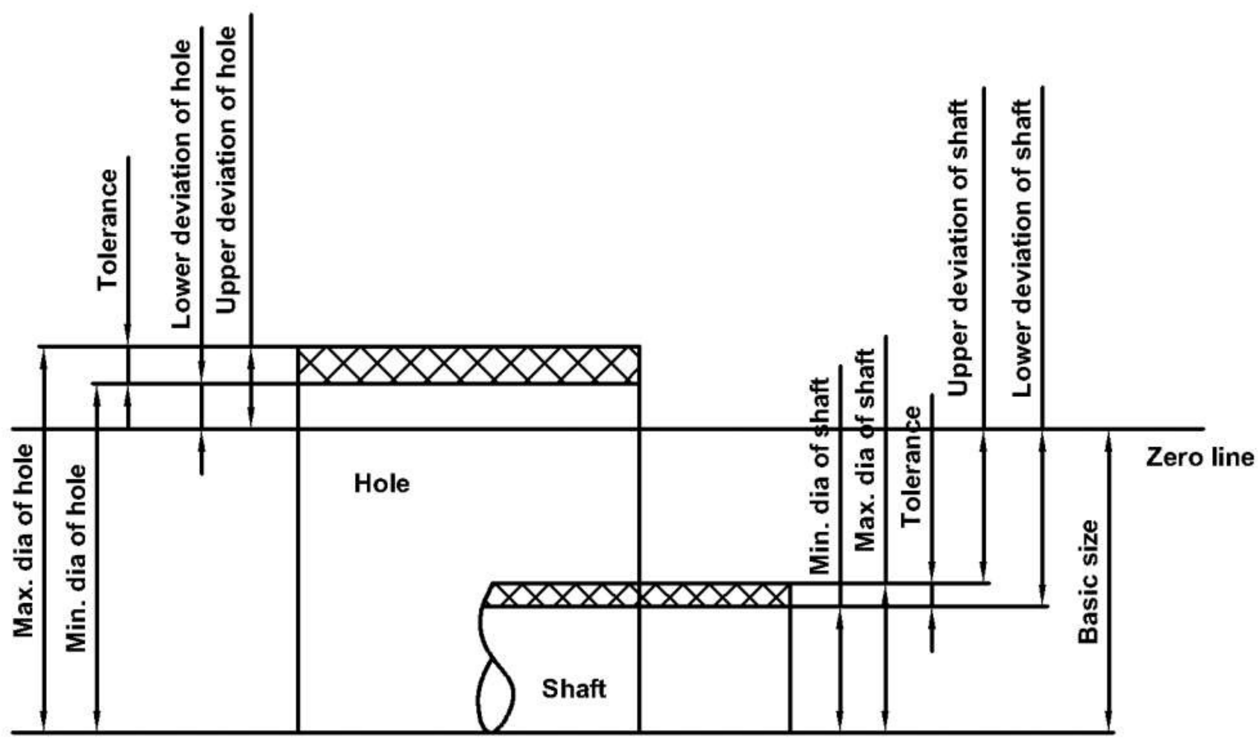

Fig 3.11 Deviation

Allowance:

Allowance is the prescribed difference between the hole dimension and shaft dimension for any type of fit. It is the intentional difference between the lower limit of the hole and higher limit of the shaft.

An allowance can be either positive or negative according to the type of fit required. If the condition in which, the shaft is smaller than the hole is called positive allowance. And if the shaft is larger than the hole is called negative allowance. The positive allowance is also called as clearance and negative allowance is called as interference.

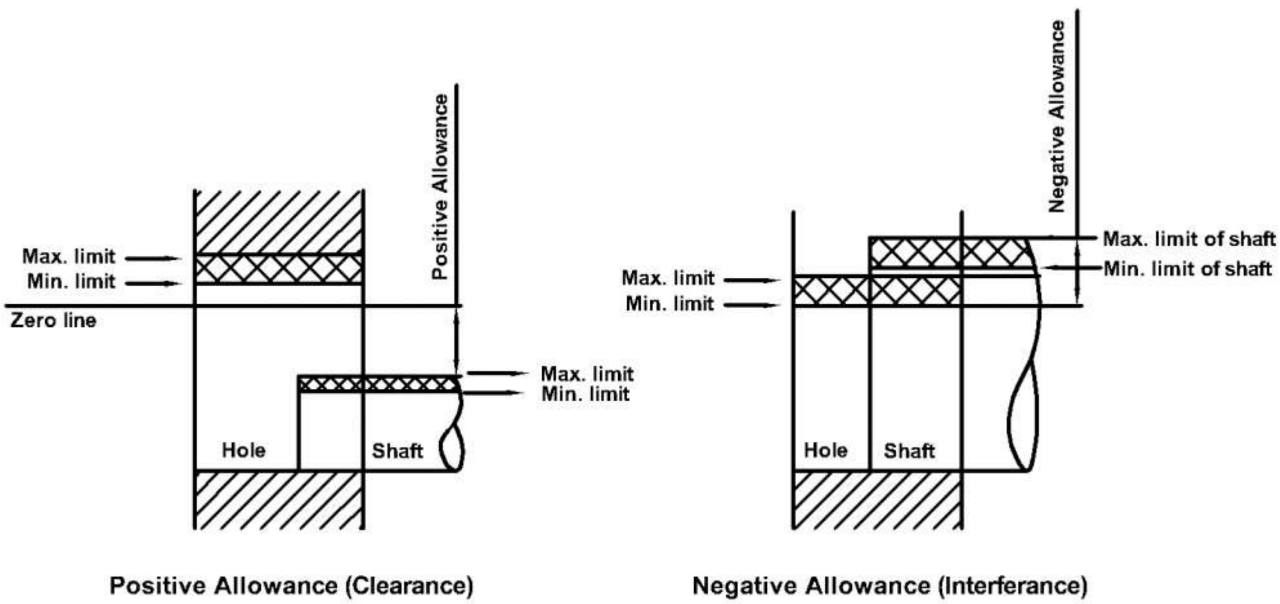

Fig 3.12 Allowance

5.2.1 Tolerance class

A tolerance class shall be designated by the letter(s) representing the fundamental deviation followed by the number representing the standard tolerance grade.

Examples: H7 (holes) f6 (shafts)

The tolerance designation consists of two components such as

i. Fundamental deviation

ii. Magnitude of tolerances or grade of tolerances (IT Grade)

Fundamental deviation

Fundamental deviation is an allowance rather than tolerance. The allowance is simply the deviation between the hole minimum diameter to the shaft maximum diameter. Where the tolerance refer to either hole (*Max. dia of hole-Min. Dia. of the hole*) or shaft (*Max. Dia. of the shaft- Min. Dia. of the Shaft*). There are 25 fundamental deviations are there to fit the shaft in a hole with these 25 allowable deviations.

All these 25 deviations are represented with the alphabets. These alphabets are case-sensitive. Block letters are used to represent the hole and small letters are used to represent the shaft deviations. all these deviations are in micrometres (Microns)

Holes deviations: A, B, C, D, E, F, G, H, JS, J, K, M, N, P, R, S, T, U, W, X, Y, Z, ZA, ZB, ZC

Shafts deviations: a, b, c, d, e, f, g, h, js, j, k, m, n, p, r, s, t, u, w, x, y, z, za, zb, zc

See the following schematic representation of the Fundamental deviation of Holes and shafts

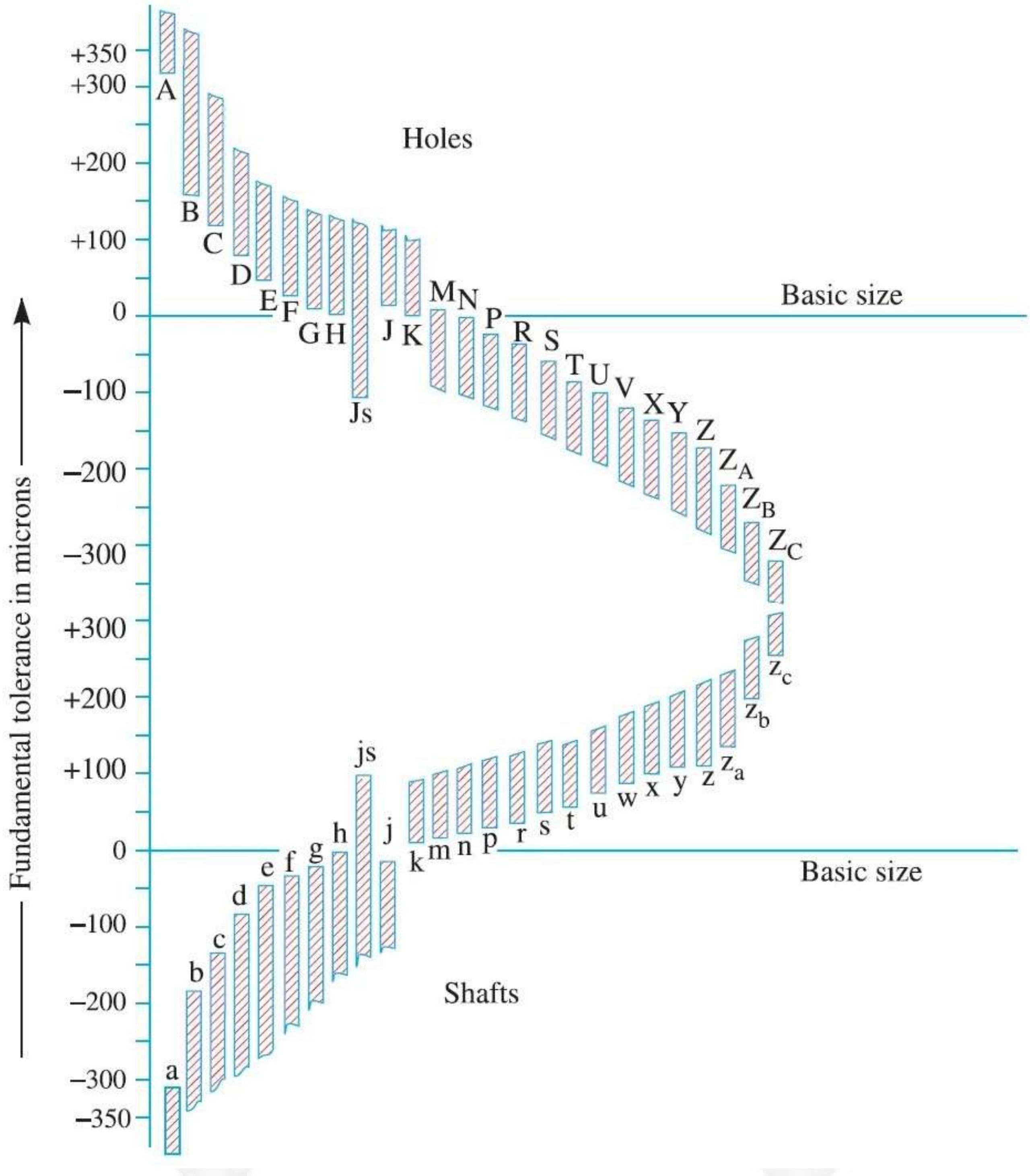

Fig 3.13 Alphabets representing fundamental deviations

Grade of tolerance

We have 18 International Tolerance (IT) grades. These grades determine the tolerances for the given designated shaft value.

IT01, IT0, IT1, IT2, IT3, IT4, IT5, IT6, IT7, IT8, IT9, IT10, IT11, IT12, IT13, IT14, IT15, IT16.

The following list will demonstrate where these grades are applied according to their tolerance range in a particular application.

IT01 to IT4 – For the production of gauges, plug gauges, measuring instruments

IT5 to IT 7 – For fits in precision engineering applications

IT8 to IT11 – For General Engineering

IT12 to IT14 – For Sheet metal working or press working

IT15 to IT16 – For processes like casting, general cutting work

CALCULATION OF TOLERANCES

Tolerance grades may be obtained either

i. Directly from tables in PSG BOOK PG.NO 3.3
ii. By using formulae

FITS

Fits are assembly conditions between Hole and Shaft. With fits, we understand how tight and loose the two components are connected. So it is a mating relationship between two components.

When a part and various other components are looking to be designed then engineering fits are taken as geometrical dimensions where it is the clearance between two mating parts in which it depends on the requirement of the size of clearance.

In engineering, fits have stated the term as a “shaft and hollow”. so, it is the degree of freedom of tightness between two mating parts.

Whenever we assemble parts of a machine, we have to be fit between two parts then only he or she can assemble.

Considering two fits such as Running feet and Sliding feet: Suppose we have a hole and a shaft. What happens in the running fit is that the shaft can easily rotate inside the hole. In the same sliding feet, the shaft is slide in the hole, so we design it accordingly.

Types of Fit:

There are mainly three types of fit and those are:

i. Clearance Fit
ii. Interference Fit and
iii. Transition Fit

1. Clearance Fit

Clearance fits allow for loose mating, where free movement is important and a certain amount of play is desired. We see clearance fits called for where elements should be able to slide in and out without obstruction, and where alignment can be loosely guided but does not require tight precision. Examples of clearance fit might include bolt/shaft holes where an element will slide freely through another feature.

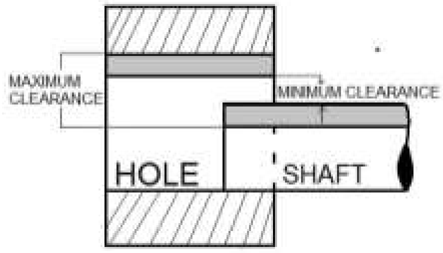

Fig 3.14 Clearance Fit

Types of Clearance Fit:

There are **five types of clearance fit:**

i. Slide fit
ii. Running Fit
iii. Loose Running Fits
iv. Easy Slide and
v. Location clearance fit

Slide fit:

It has very little clearance between two parts which we say almost to zero but it provides very greater precision and accuracy in sliding and moving parts.

Example: Sliding gears, automobile assemblies, slide valves, clutch discs, parts of machine tools, tailstock spindle of the lathe machine, guiding of shafts, etc

Running Fit:

For rotation of components at a moderate speed, the running fit has to be employed where accuracy is not necessary.

Running fit has large clearance and it involves large temperature variations, high running speeds, and heavy journal pressures. Example: Gears, coupling, etc.

It has very little clearance between two parts which we say almost to zero but it provides very greater precision and accuracy in sliding and moving parts.

Example: Sliding gears, automobile assemblies, slide valves, clutch discs, parts of machine tools, tailstock spindle of the lathe machine, guiding of shafts, etc

Loose running fits are employed for rotations at high speeds of the parts and have a larger clearance where accuracy is not important.

Example: Latches, pivots, heat, parts affected by corrosion, contamination, etc.

Easy slide:

Easy slide employed for small clearance between the hole and shaft. The easy-slide has been used for non-regular motions and for slow regular motion. Example: Piston.

Location clearance fit:

It provides very close and minimal clearance for the accurate precise requirement and with the help of lubrication, parts can be assembled without force and turn and slide free properly. Example: Guiding of shafts, roller guides, etc.

2. Interference Fit

An interference fit will be much tighter than a clearance fit. Also referred to as a press fit or friction fit, the interference fit requires some degree of force to join two components. Pressing a bushing, bearing, dowel pin, or other items into their mating components are all examples of how an interference fit can be used. Once joined, this creates a relatively solid union that would require substantial force or potential machine operations to uncouple.

Types of Interference Fit:

There are **three types of Interference fit**:

Force Fit

Tight Fit and

Shrink Fit

Force fit:

To mating the high interference fit, it requires a very high temperature for heating the part to assemble the shaft with a hole. The external force is required for the mating parts.

Example: Gears, shafts, etc.

Tight Fit:

It provides minimal interference than force fits. Example: Stepped pulley of a conveyor, cylindrical grinding of a machine, etc.

Driving fits:

It requires medium interference which can be assembled by higher forces for cold forging or hot forging. Driving fits are more reliable than tight fits.

Example: Shafts, gears, bushes, etc.

3. Transition Fit

A transition fit would fall between clearance and an interference fit. Transition fits are called for when accurate alignment is critical, and mating parts must join with greater precision. You may also see these referred to as a slip or push-fit.

There will still be a greater degree of clearance than a press/interference fit, but it will be substantially smaller and should remove excess play or movement in the joint.

Types of transition fits:

i. Similar fit and
ii. Fixed fit

Similar Fit:

It provides very negligible clearance or very small interference and assembly is achievable with using a rubber mallet. Example: Gears, Hubs, pulleys, bearing, etc.

Fixed fit:

It provides a small clearance or minimal interference fit which can be assembled or disassembled using light pressing force. Example: Driven bushes, armatures on shaft, plugs, etc.

Numerical Examples of Various Fits

1. **Clearance fit:**

Establish the nature of fit for the following hole-shaft combination.

Hole:	Φ30.05	Shaft: Φ29.90
	Φ 29.95	Φ29.85

Maximum hole size — Minimum shaft size = 30.05 — 29.85 = + 0.2

Minimum hole size — Maximum shaft size = 29.95 — 29.90 = + 0.05

As the difference is +ve, it is clearance fit

2. Interference fit

For the given dimensions, establish the fit.

Hole: 35.35	Shaft: 35.45
35.15	35.40

Maximum hole size —Minimum shaft size = 35.35 — 35.40 = 0.05

Minimum hole size — Maximum shaft size = 35.15 — 35.45 = - 0.03

As the difference is —Ve, it is interference fit.

4. **Transition fit:**

slip Find the nature of fit for the given:

Hole: 40.60	Shaft: 40.65

40.50 40.55

Maximum hole size — Minimum shaft size = 40.6 — 40.55 = + 0.05

Minimum hole size — Maximum shaft size = 40.5 40.65 =-o.15

As one difference is positive and other is negative, it is called transition fit.

BIS System of Designation of Fits

According to the Bureau of Indian Standards (BIS), fit is specified by the basic size common to two mating parts followed by the symbols for tolerance of each part.

Hole Basis and Shaft Basis System of Fits

To determine the fit, we must take one component as the constant member and the second component will have the deviations according to the type of fit chosen. By making a constant member we can classify them as hole basis system and shaft basis system. These are the two bases of the limit system.

- Hole Basis system
- Shaft Basis system

Hole Basis system

In a hole basis system, the hole is kept as the constant, and the shaft upper and lower deviation values determine the type of fit. In a hole basis system, the Lower deviation of the hole will be Zero.

Example: Nominal Size of Hole 36mm

Hole = 36.000/36.015

Where

(Clearance Fit) Shaft = 35.980/35.990 (Maximum Clearance = 0.035; Minimum Clearance = 0.010)

(Transition Fit) Shaft = 35.990/36.010 (Maximum Clearance = 0.025; Maximum Interference = 0.010)

(Interference Fit) Shaft = 36.010/36.020 (Maximum interference = 0.020; Minimum Interference = 0.005)

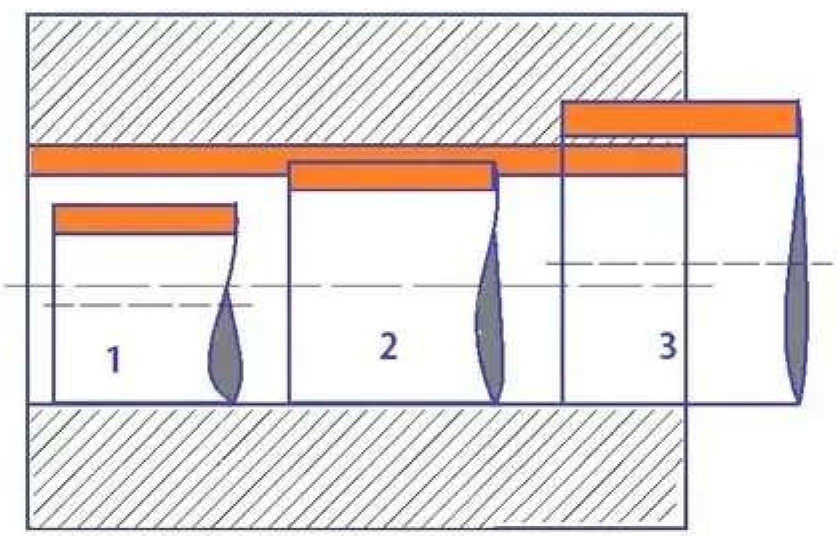

Fig 3.15 Hole Basis System

Shaft Basis system

In the Shaft basis system, the shaft is kept as the constant, and hole upper and lower deviation values determine the type of fit. In the shaft basis system, the upper deviation of the shaft will be Zero.

Example: Nominal Size of shaft 25mm

Shaft = 24.985/25.000

Where

(Clearance Fit) Hole = 25.010/25.020 (Maximum Clearance = 0.035; Minimum Clearance = 0.010)

(Transition Fit) Hole = 24.990/25.010 (Maximum Clearance = 0.025; Maximum Interference = 0.010)

(Interference Fit) Hole = 24.980/24.990 (Maximum interference = 0.020; Minimum Interference = 0.005)

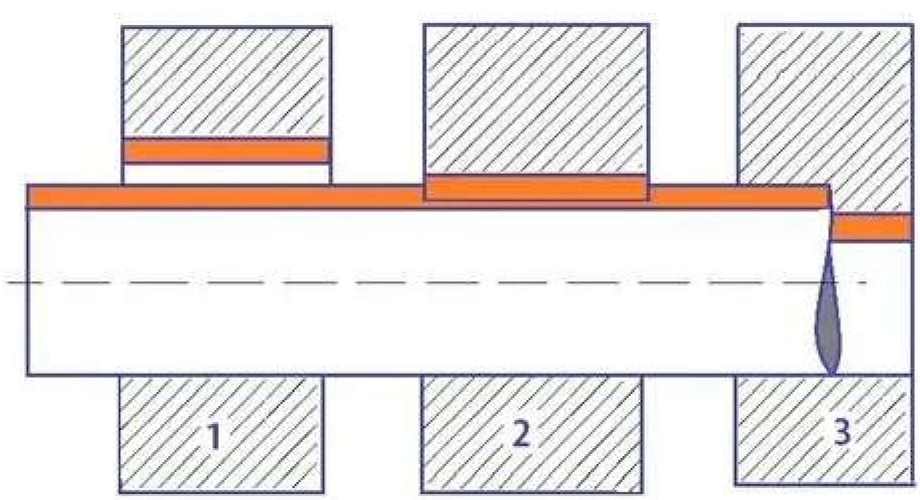

Fig 3.16 Shaft Basis System

2.6 LIMIT GAUGES

Gauging, done in manufacturing processes, refers to the method by which it is determined quickly whether or not the dimensions of the checking parts in production, are within their specified limits. It is done with the help of some tools called gauges. A gauge does not reveal the actual size of dimension.

A clear distinction between measuring instruments and gauges is not always observed. Some tools that are called gauges are used largely for measuring or layout work. Even some are used principally for gauging give definite measurement.

High carbon and alloy steels have been the principal material used for many years. Objections to steel gauges are that they are subjected to some distortion because of the heat-treating operations and that their surface hardness is limited. These objections are largely overcome by the use of chrome plating or cemented carbides as the surface material. Some gauges are made entirely of cemented carbides or they have cemented carbides inserted at certain wear points.

2.6.1 GAUGES AND THEIR CLASSIFICATIONS

Gauges are the tools which are used for checking the size, shape and relative positions of various parts but not provided with graduated adjustable members. Gauges are, therefore, understood to be single-size fixed-type measuring tools.

Classifications of Gauges

(a) Based on the standard and limit

(i) Standard gauges

(ii) Limit gauges or “go” and “not go” gauges

(b) Based on the consistency in manufacturing and inspection

(i) Working gauges

(ii) Inspection gauges

(iii) Reference or master gauges

(c) Depending on the elements to be checked

(i) Gauges for checking holes

(ii) Gauges for checking shafts

(iii) Gauges for checking tapers