METROLOGY

UNIT -1

Limits, Fits and Tolerances

Course Outcome :

CO1

Apply the knowledge of limits, fits, tolerances to a machine component and able to select a suitable gauge for inspection during mass production.

UNIT -1 Limits Fits and Tolerances

UNIT -2 Measurements of screw threads and Gears

UNIT -3 Geometrical Feature Measurements, Angular Measurements and Comparators

UNIT - 4 Surface Texture

UNIT - 5 Acceptance tests on machine tools and CMM

METROLOGY

As per U.P. Technical University

A TEXTBOOK OF

MEASUREMENT AND METROLOGY

A.K. SAWHNEY
M. MAHAJAN

DHANPAT RAI & Co.

Scheme of valuation	
1st mid exam	30
2nd mid exam	30
Average for 20 Marks	
Assignment (or) Surprise test	10 Marks
Final Exam	70 marks
Total	100 Marks

UNIT -1

Introduction to Metrology, Need of Inspection, Accuracy and Precision

Definition of tolerance, Specification in assembly,

Principle of interchangeability and selective assembly,

concept of limits and terminology.

hole basis and shaft basis system, types of fits Clearance, Transition, Interference.

Tolerances – Unilateral, bilateral tolerances, geometric and position-tolerances.

Classification of gauges, brief concept of design of gauges (Taylor's principle),

Wear allowance on gauges, Types of gauges.

Introduction :

Metrology is the science of measurement.

Metrology may be divided depending upon the quantity to be measured like metrology of length, metrology of time.

But for engineering purposes, it is restricted to measurement of length and angles and other qualities which are expressed in linear or angular terms.

In the broader sense it is not limited to length measurement but is also concerned with industrial inspection and its various techniques.

Metrology is mainly concerned with:

- Establishing the units of measurements, ensuring the uniformity of measurements.
- Developing methods of measurement.
- Errors of measurement.
- Accuracy of measuring instruments and their care.
- Industrial inspection and its various techniques.

Legal Metrology

Legal metrology is directed by a national organization which is called national service of Legal metrology. It includes a no. of international organization whose ultimate object is to maintain uniformity of measurement throughout the world.

The activities of legal metrology are:

(1) Control of measuring instruments.

(2) Testing of prototype/models of measuring instruments.

(3) Examination of measuring instrument to verify its conformity.

Objectives of Metrology

- The basic objective of a measurement is to provide the required accuracy at a minimum cost.
- Complete evaluation of newly developed products.
- Determination of Process Capabilities.
- Determination of the measuring instrument capabilities and ensure that they are quite sufficient for their respective measurements.
- Minimizing the cost of inspection by effective and efficient use of available facilities.

- Reducing the cost of rejects and rework through application of statistical quality control techniques.
- To maintain the accuracies of measurement.
- To prepare design for all gauges and special inspection fixtures.

Process of measurement:

- The sequence of operations necessary for the execution of measurement is called process of measurement.

There are main three important elements of measurement,

1) Measuring is the physical quantity or property like length, angle, diameter, thickness etc. to be measured.

2) Reference: - It is the physical quantity or property to which quantitative comparisons are made.

(3) Comparator: - It is the means of comparing measuring measurand with some reference.

Suppose a fitter has to measure the length of M.S. plate- he first lays his rule along the flat. He then carefully aligns the zero end of his rule with one end of M.S. flat and finally compares the length of M.S. flat with the graduations on his rule by his eyes.

In this example,

the length of M.S. flat is a measurand,

Steel rule is the reference and **eye** can be considered as a comparator.

Methods of Measurement:

The methods of measurement can be classified as:

(1) Direct method:

This is a simple method of measurement, in which the value of the quantity to be measured is obtained directly without the calculations.

For example, measurements by scales, Vernier calipers, micrometers, bevel protector etc.

This method is most widely used in production. This method is not very accurate because it depends on human judgment.

(2) Indirect method:

In indirect method the value of quantity to be measured is obtained by measuring other quantities which are functionally related to required value.

for example, angle measurement by sine bar, measurement of shaft power by dynamometer etc.

Measuring system:

A measuring system is made of five elements:

These are: (1) Standard

(2) Work piece

(3) Instrument

(4) Person

(5) Environment

- The most basic element of measurement is a standard without which no measurement is possible.
- Once the standard is chosen select a work piece on which measurement will be performed.
- Then select a instrument with the help of which measurement will be done.
- The measurement should be performed under standard environment.
- And lastly there must be some person or mechanism to carry out the measurement.

Accuracy:

Accuracy is defined as the closeness of the measured value with true value.

OR

Accuracy is defined as the degree to which the measured value agrees with the true value.

Practically it is very difficult to measure the true value and therefore a set of observations is made whose mean value is taken as the true value of the quantity measured.

Precision:

A measure of how close repeated trials are to each other.

OR

The closeness of repeated measurements.

- Precision is the repeatability of the measuring process. It refers to the group of measurements for the same characteristics taken under identical conditions.

- It indicated to what extent the identically performed measurements agree with each other.

If the instrument is not precise it will give different results for the same dimension when measured again and again.

Difference between Precision and Accuracy

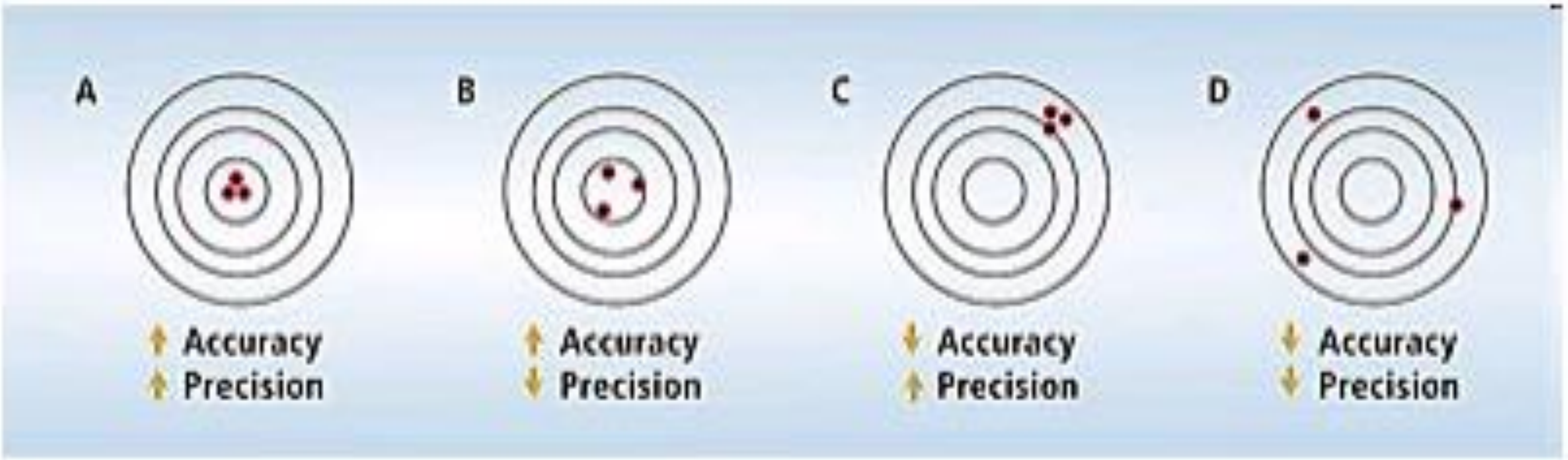
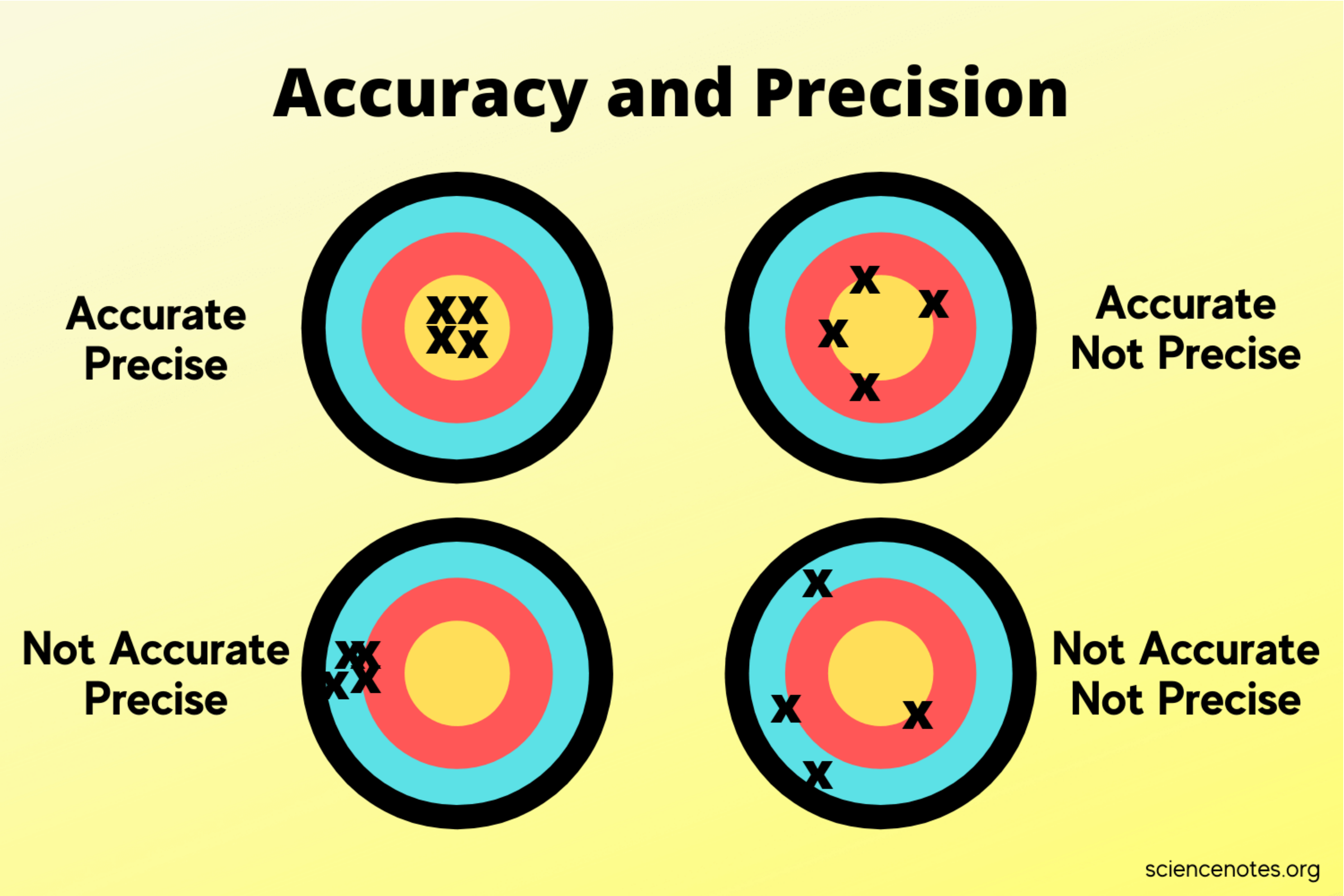


Figure shows the difference between the concepts of accuracy versus precision using a dartboard analogy that shows four different scenarios that contrast the two terms.



Accuracy and Precision
Accurate
Precise
Accurate
Not Precise
Not Accurate
Precise
Not Accurate
Not Precise
sciencenotes.org

Metrology

Accuracy	Precision
Accuracy is closeness with the true value of the quantity being measured	Precision is a measure of the reproducibility of the measurement
Measurement can be accurate but not necessarily precise	Measurement can be precise but not necessarily accurate
It can be determined with a single measurement	It needs several measurement need to be determined
Accuracy may be affected with systematic error	Precision may be affected with random error
Accurate values have to be precise in most cases	Precise values may or may not be accurate
Degree of conformability	Degree of reproducibility

Tolerances

- When the tolerance allowed is sufficiently greater than the process variation, no difficulty arises.
- The difference between the upper and lower limits is termed ***permissive tolerance.***
- For example, a shaft has to be manufactured to a diameter of 40 ± 0.02 mm.
- This means that the shaft, which has a basic size of 40 mm, will be acceptable if its diameter lies anywhere between the limits of sizes, that is, an upper limit of 40.02 mm and a lower limit of 39.98 mm.

Then permissive tolerance is equal to $40.02 - 39.98 = 0.04$.

Tolerances

- Tolerance can be defined as the magnitude of permissible variation of a dimension or other measured value from the specified value.

- It can also be defined as the total variation permitted in the size of a dimension, and is the algebraic difference between the upper and lower acceptable dimensions. It is an absolute value.

- The basic purpose of providing tolerances is to permit dimensional variations in the manufacture of components, adhering to the performance criterion as established by the specification and design.

Manufacturing Cost and Work Tolerance

- It is very pertinent to relate the production of components within the specified tolerance zone to its associated manufacturing cost.
- As the permissive tolerance goes on decreasing, the manufacturing cost incurred to achieve it goes on increasing exponentially.

Fig. 3.2 Relationship between work tolerance and manufacturing cost

- When the permissive tolerance limits are relaxed without degrading the functional requirements, the manufacturing cost decreases.

Classification of Tolerance:

1. Unilateral tolerance
2. Bilateral tolerance
3. Compound tolerance
4. Geometric tolerance

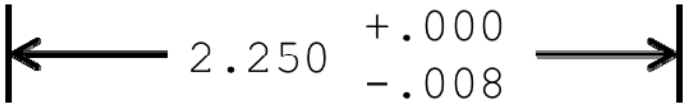
1.Unilateral Tolerance :

When the tolerance distribution is only on one side of the basic size, it is known as unilateral tolerance.

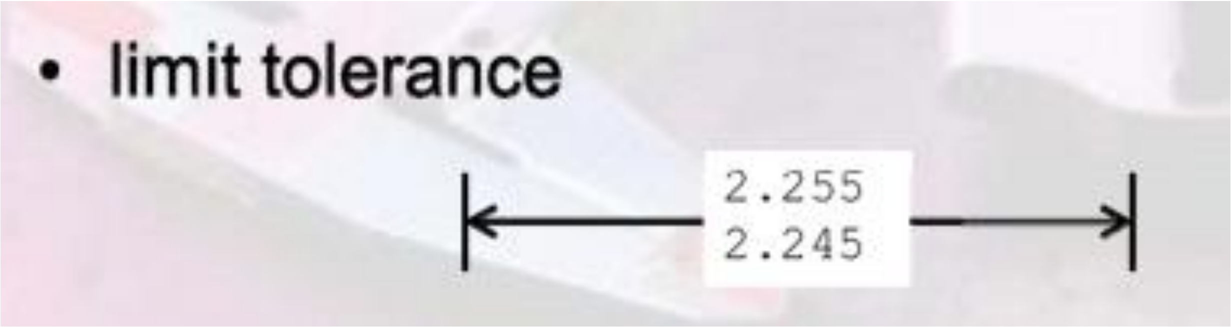
In other words, tolerance limits lie wholly on one side of the basic size, either above or below it.

Example: $40^{+0.02}_{+0.01}$, $40^{+0.01}_{-0.00}$, $40^{-0.01}_{-0.02}$, $40^{+0.00}_{-0.02}$,

- unilateral tolerance



Tolerances



Bilateral Tolerance

When the tolerance distribution lies on either side of the basic size, it is known as bilateral tolerance.

In other words, the dimension of the part is allowed to vary on both sides of the basic size but may not be necessarily equally disposed about it.

Example:

$40^{\pm 0.02}$, $40^{+0.02}_{-0.01}$

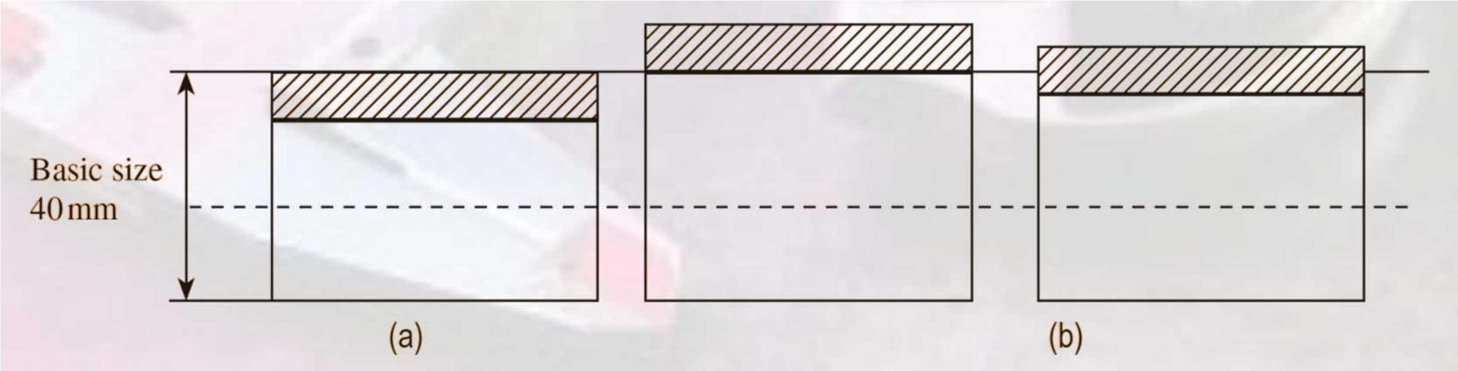


Fig. 3.3 Tolerances (a) Unilateral (b) Bilateral

Compound Tolerance

When tolerance is determined by established tolerances on more than one dimension, it is known as compound tolerance.

For example, tolerance for the dimension R is determined by the combined effects of tolerance on 40 mm dimension, on 60°, and on 20 mm dimension. The tolerance obtained for dimension R is known as compound tolerance (Fig. 3.4). In practice, compound tolerance should be avoided as far as possible.

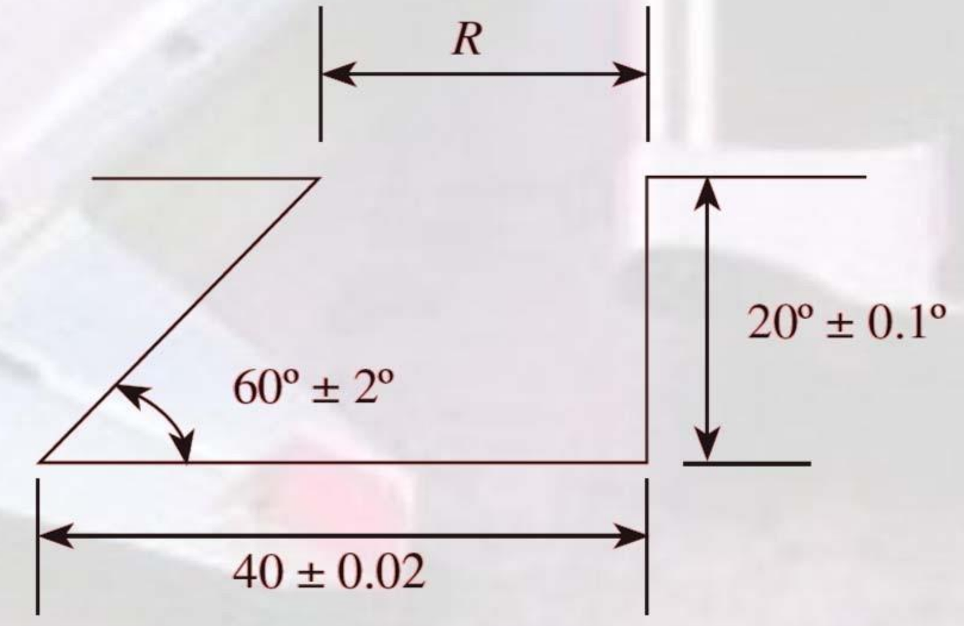
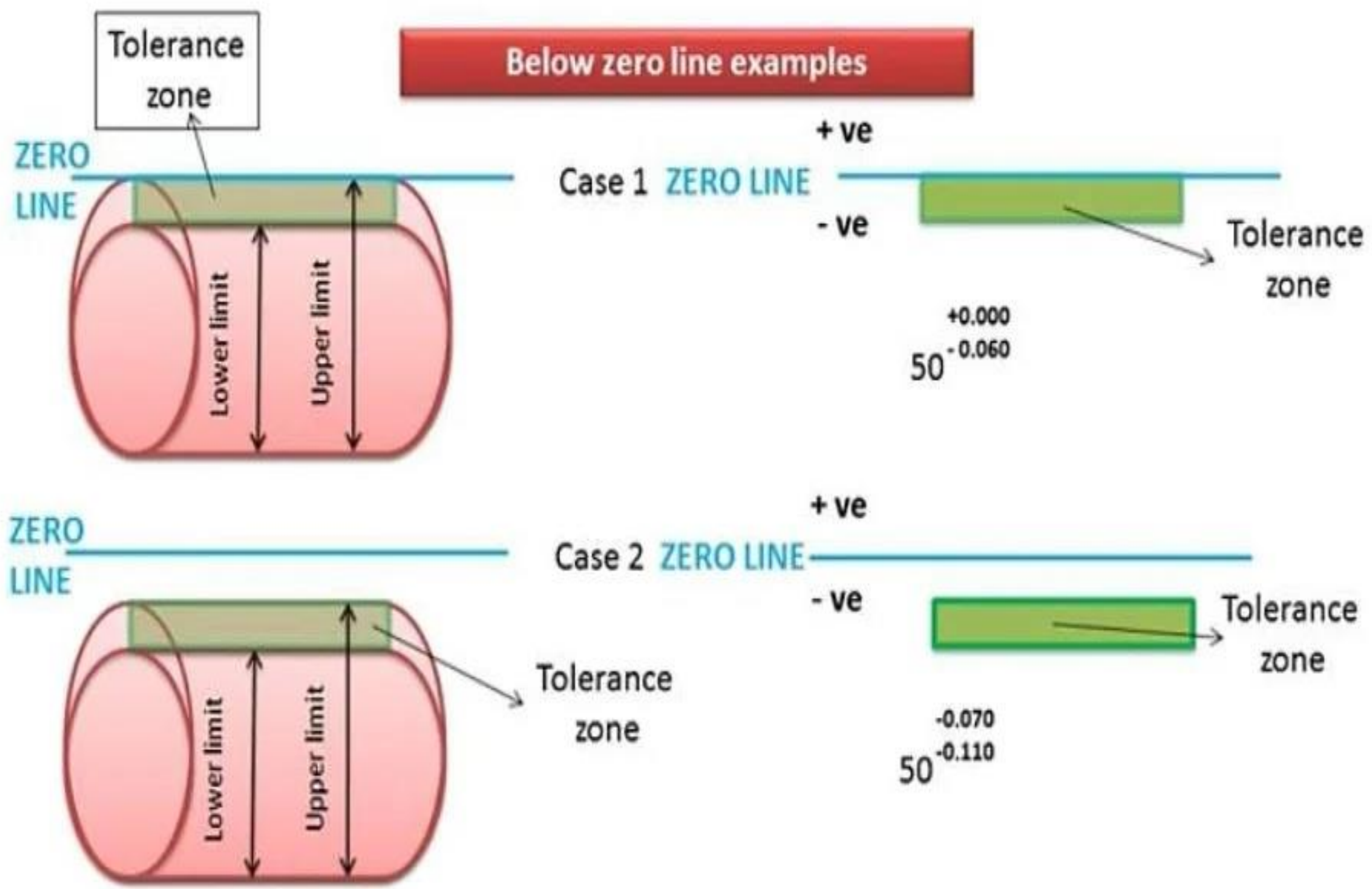
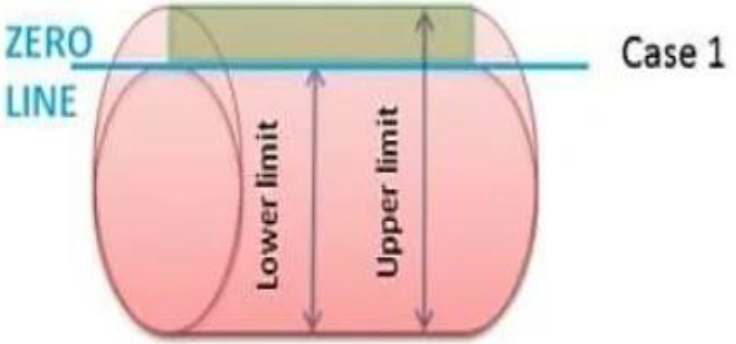


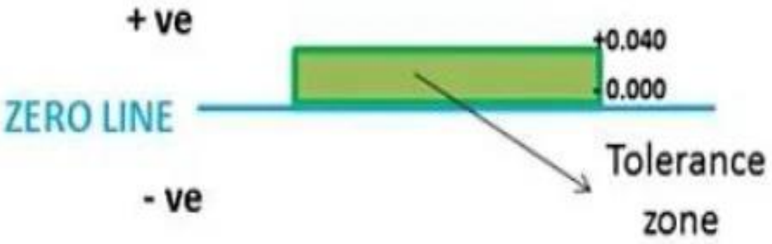
Fig. 3.4 Compound tolerance

1. Unilateral tolerance: Below zero line: Negative

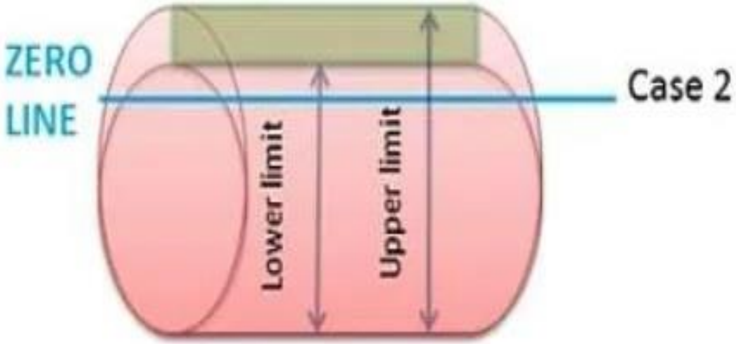


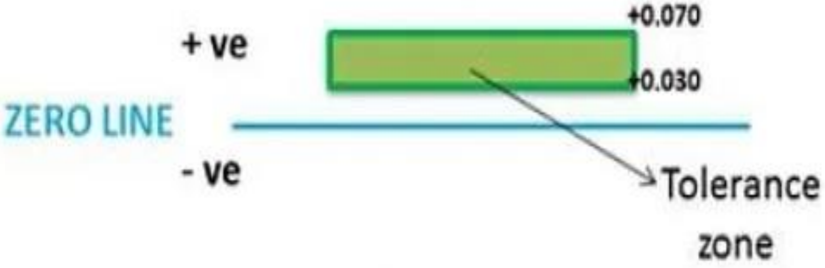
1. Unilateral tolerance: **Above zero line**: Positive





eg. $50^{+0.040}_{-0.000}$

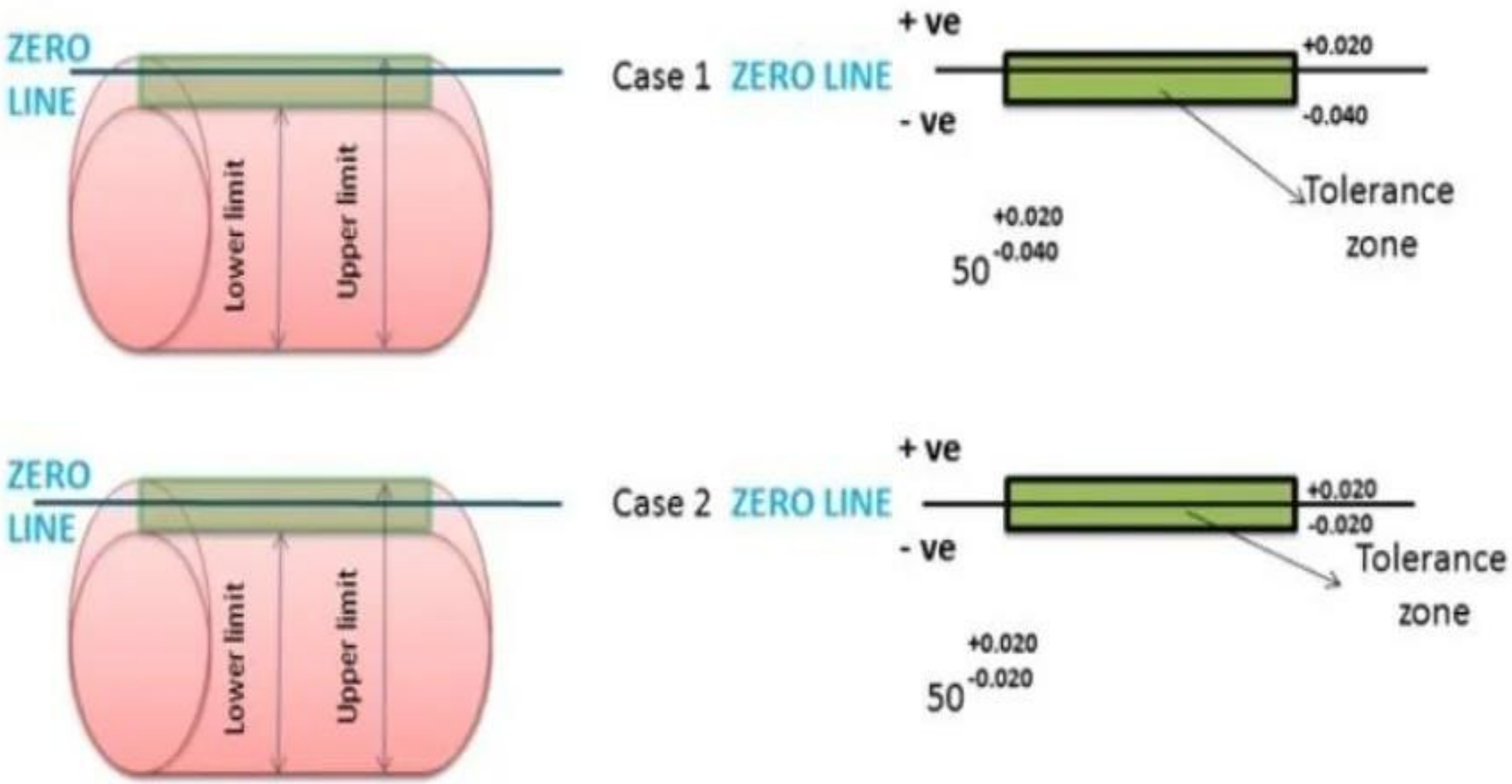




eg. $50^{+0.070}_{+0.030}$

2. Bilateral tolerance

When the tolerance distribution lies on either side of the basic size.



- It is not necessary that Zero line will divide the tolerance zone equally on both sides.
- It may be equal or unequal

✓ **Geometric tolerances** :Geometric tolerances are used to indicate the relationship of one part of an object with another. Consider the example shown in Fig. 3.5.

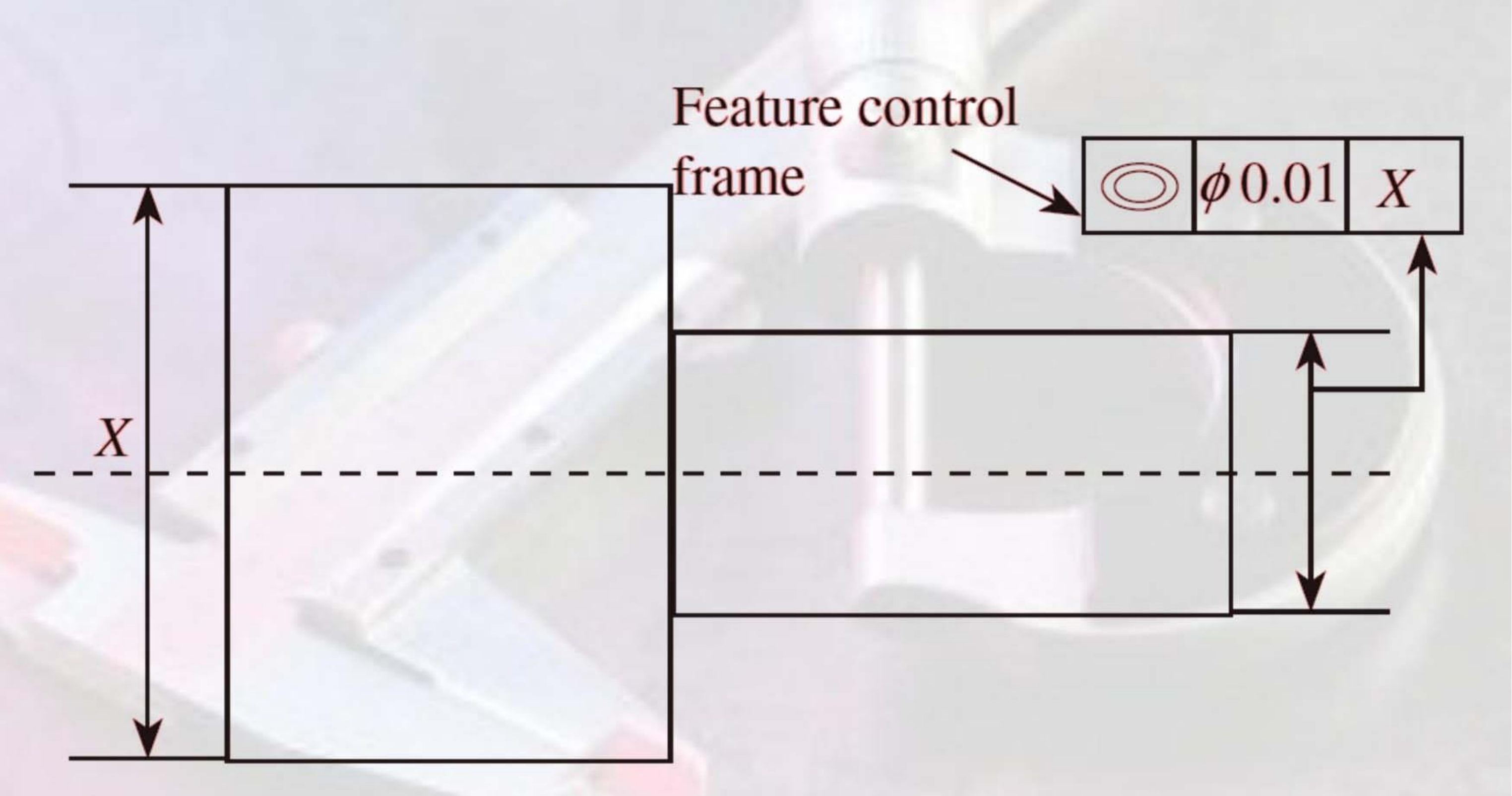


Fig. 3.5 Representation of geometric tolerance

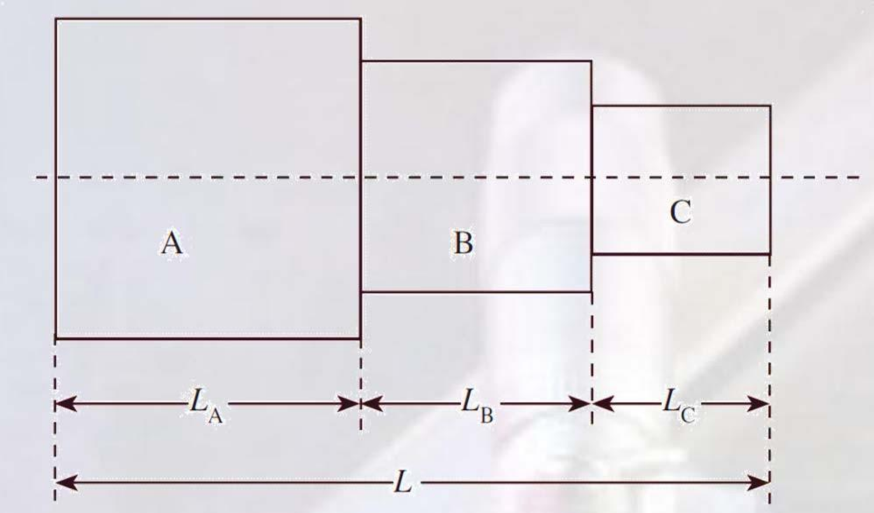


Fig. 3.6 Accumulation of tolerances

Consider the example shown if figure 3.6.

Let $L_A = 30^{+0.02}_{-0.01}$ mm, $L_B = 20^{+0.02}_{-0.01}$ mm and $L_C = 10^{+0.02}_{-0.01}$ mm

The overall length of the assembly is the sum of the individual length of components given as

$L = L_A + L_B + L_C$

$L = 30 + 20 + 10 = 60$ mm

Then, cumulative upper tolerance limit is 0.02 + 0.02 + 0.02 =0.06 mm
and cumulative lower limit = – 0.01 – 0.01 – 0.01 = – 0.03 mm

Therefore dimension of the assembled length will be = $60^{+0.06}_{-0.03}$ mm

It is essential to avoid or minimize the cumulative effect of tolerance build-up, as it leads to a high tolerance on overall length, which is undesirable.

If progressive dimensioning from a common reference line or a baseline dimensioning is adopted, then tolerance accumulation effect can be minimized. This is clearly illustrated in Fig. 3.7.

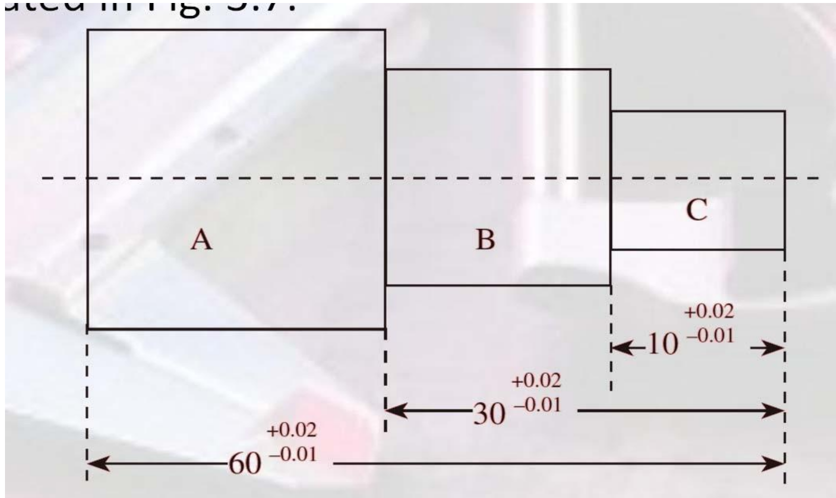


Fig. 3.7 Progressive dimensioning

Maximum and Minimum Metal Conditions

- Let us consider a shaft having a dimension of 40 ± 0.05 mm.
- The maximum metal limit (MML) of the shaft will have a dimension of 40.05 mm because at this higher limit, the shaft will have the maximum possible amount of metal.
- The shaft will have the least possible amount of metal at a lower limit of 39.95 mm, and this limit of the shaft is known as minimum or least metal limit (LML).
- Similarly, consider a hole having a dimension of 45 ± 0.05 mm.
- The hole will have a maximum possible amount of metal at a lower limit of 44.95 mm and the lower limit of the hole is designated as MML.

Maximum and Minimum Metal Conditions (contd.)

- For example, when a hole is drilled in a component, minimum amount of material is removed at the lower limit size of the hole. This lower limit of the hole is known as **MML**.
- The higher limit of the hole will be the LML. At a high limit of 45.05 mm, the hole will have the least possible amount of metal.
- The maximum and minimum metal conditions are shown in Fig. 3.8

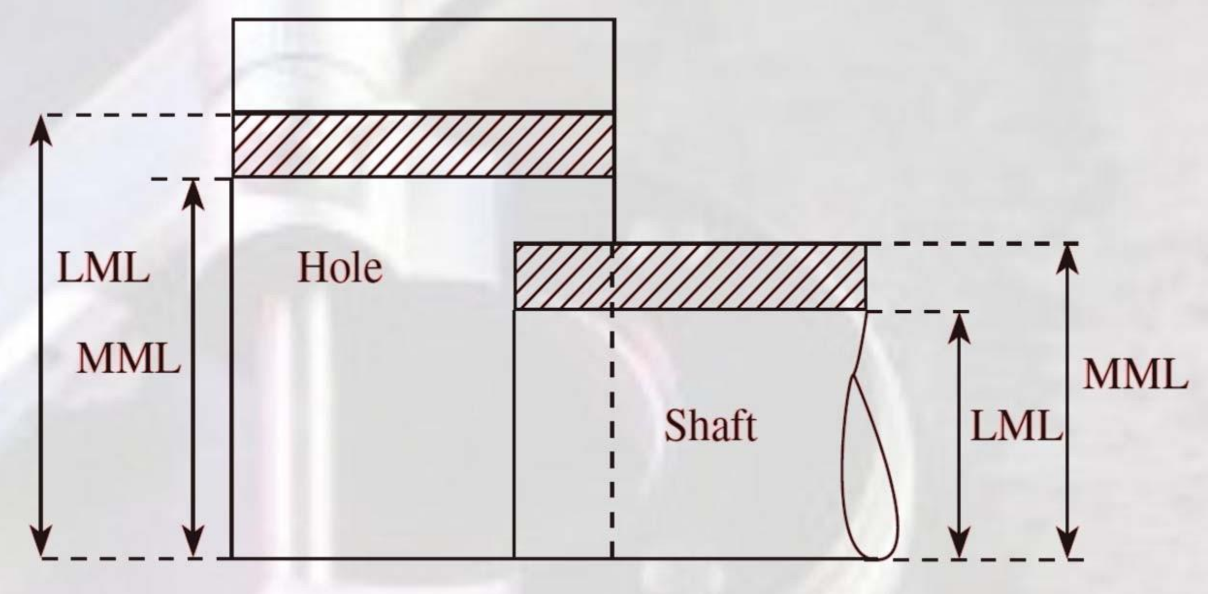


Fig. 3.8 MML and LML

Fits

- Manufactured parts are required to mate with one another during assembly.
- The relationship between the two mating parts that are to be assembled, that is, the hole and the shaft, with respect to the difference in their dimensions before assembly is called a *fit*.

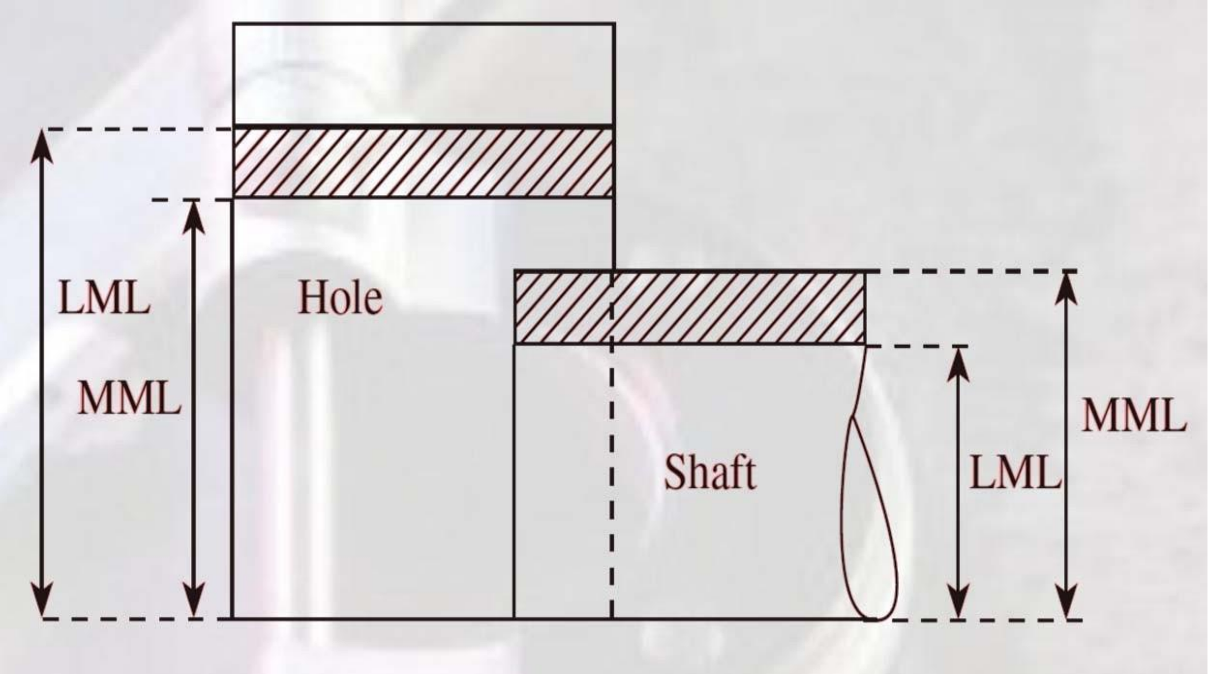


Fig. 3.8 MML and LML

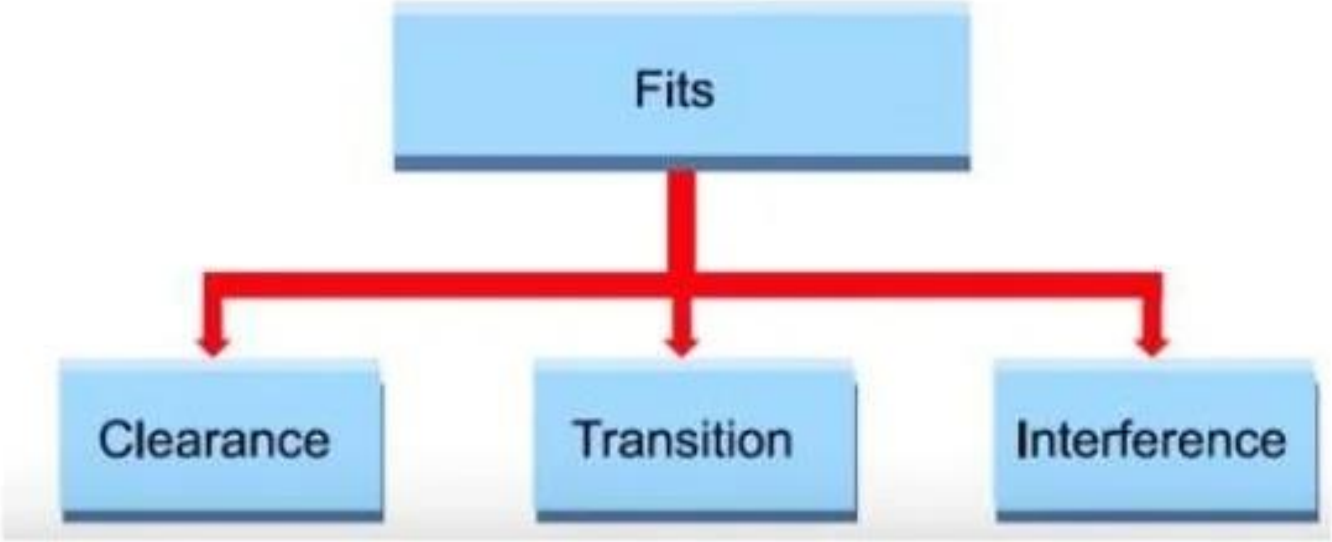
- An ideal fit is required for proper functioning of the mating parts.

FITS

- The degree of tightness and or looseness between the two mating parts.

Three basic types of fits can be identified, depending on the actual limits of the hole or shaft.

1. Clearance fit
2. Interference fit
3. Transition fit



Three basic types of fits can be identified, depending on the actual limits of the hole or shaft:

1. Clearance fit

- The largest permissible diameter of the shaft is smaller than the diameter of the smallest hole.
- This type of fit always provides clearance Small clearances are provided for a precise fit that can easily be assembled without the assistance of tools. When relative motions are required, large clearances can be provided, for example, a shaft rotating in a bush.

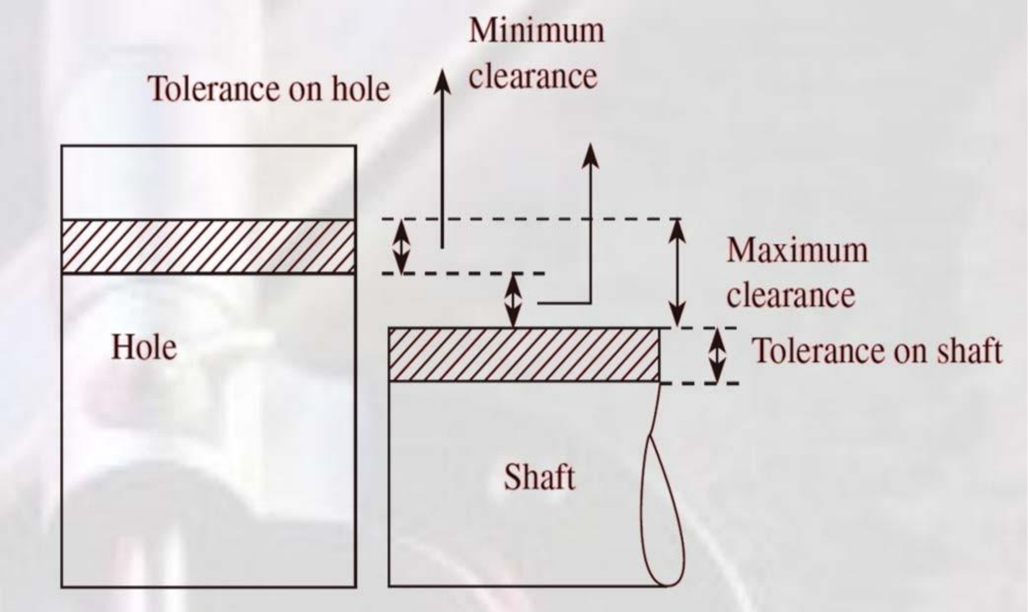


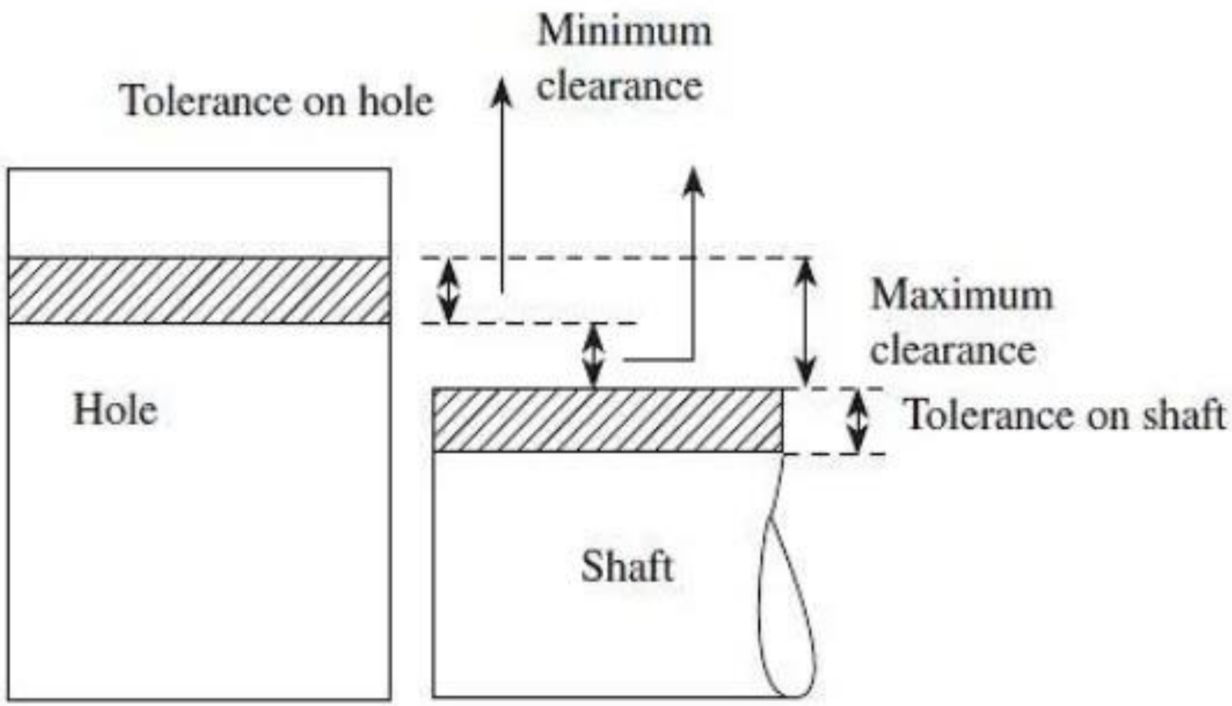
Fig. 3.9 Clearance fit

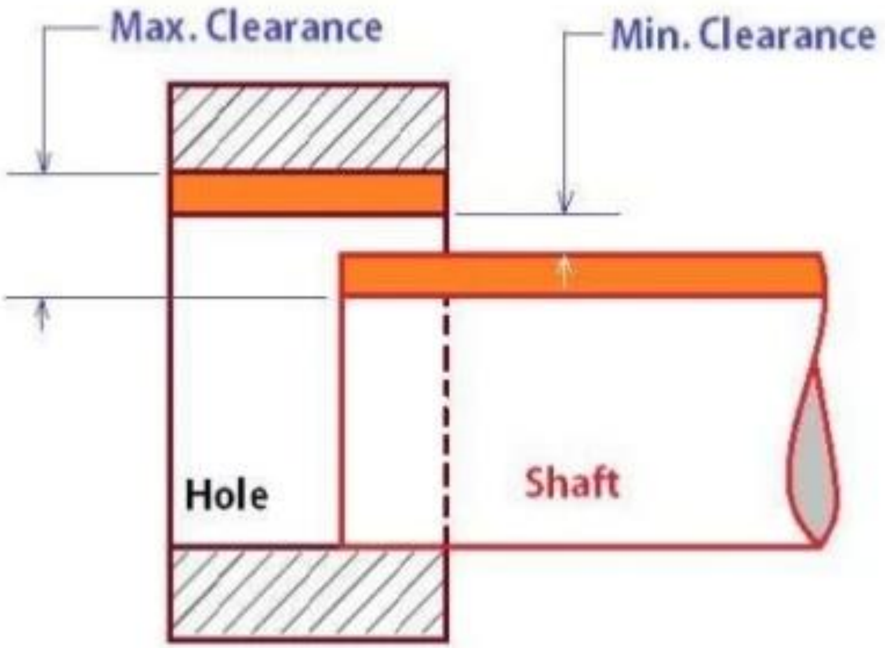
- In case of clearance fit, the difference between the sizes is always positive. The clearance fit is described in Fig. 3.9.

FITS

1. Clearance fit

Upper limit of shaft is less than the lower limit of the hole.





The largest permissible dia. of the shaft is smaller than the dia. of the smallest hole.

E.g.: Shaft rotating in a bush

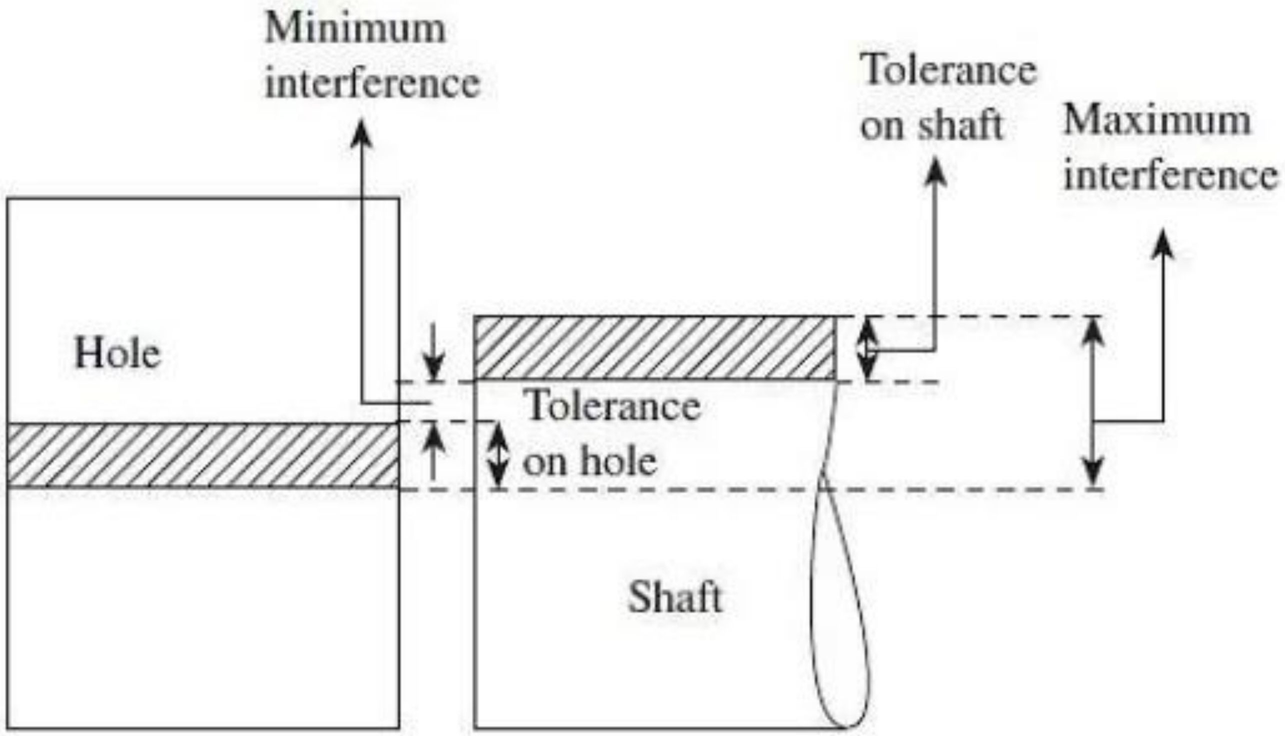
2.Interference fit:

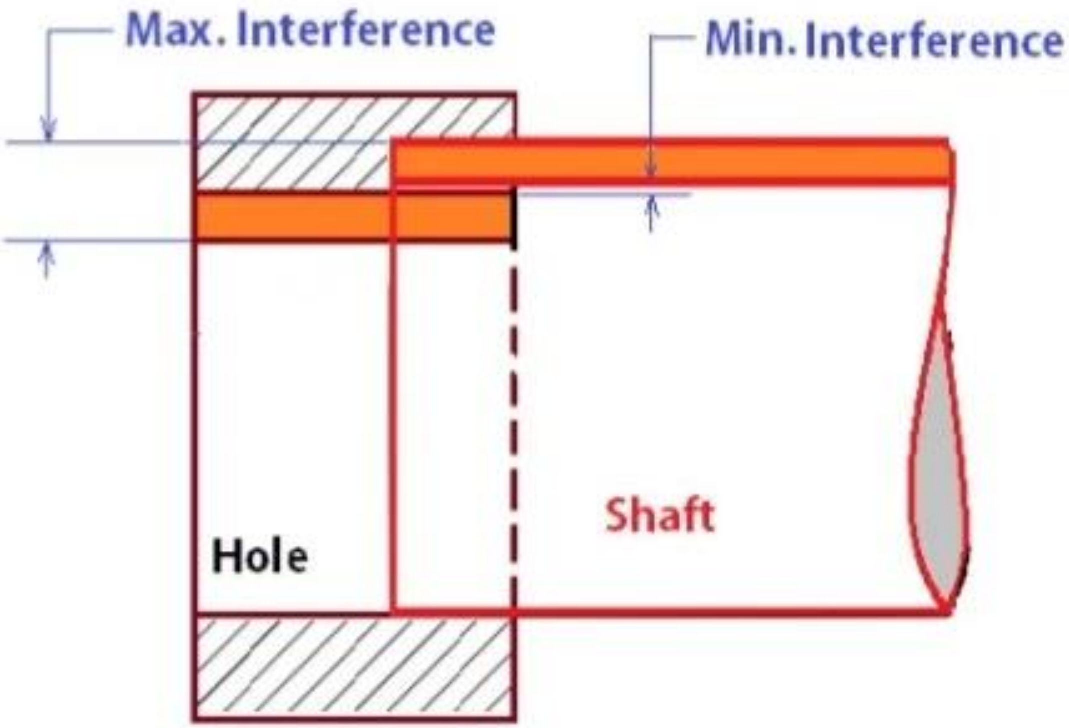
The minimum permissible diameter of the shaft exceeds the maximum allowable diameter of the hole.

- This type of fit always provides interference. Interference fit is a form of a tight fit. Tools are required for the precise assembly of two parts with an interference fit.
- When two mating parts are assembled with an interference fit, it will be an almost permanent assembly, that is, the parts will not come apart or move during use. To assemble the parts with interference, heating or cooling may be required.
- In an interference fit, the difference between the sizes is always negative.

2. Interference fit

Upper limit of the hole is less than the lower limit of shaft.

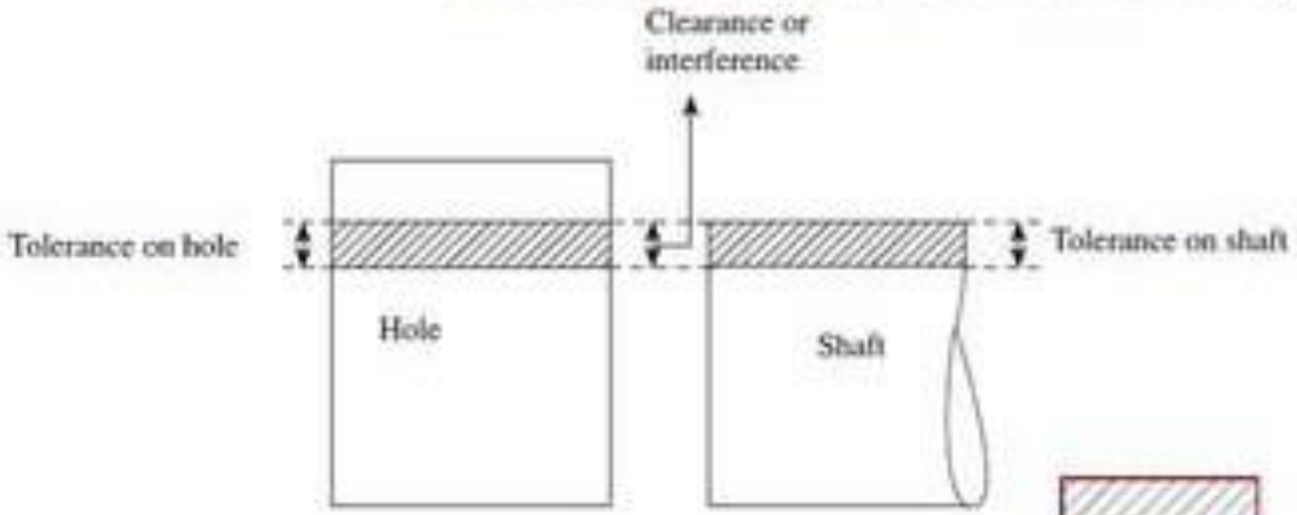


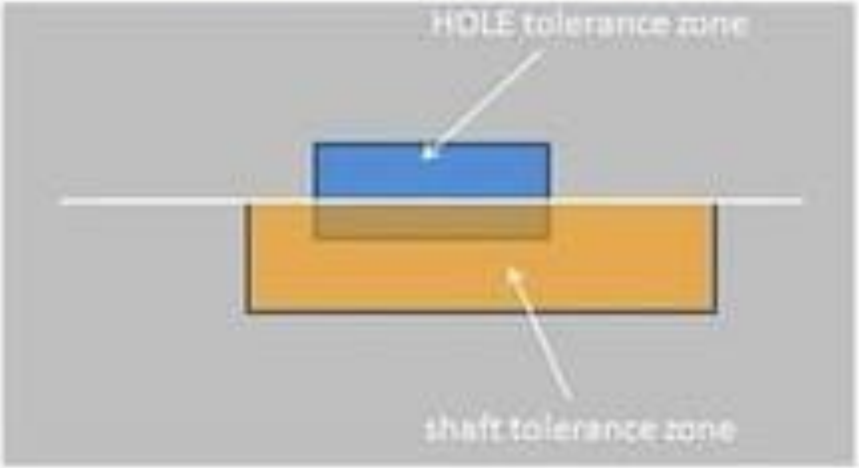


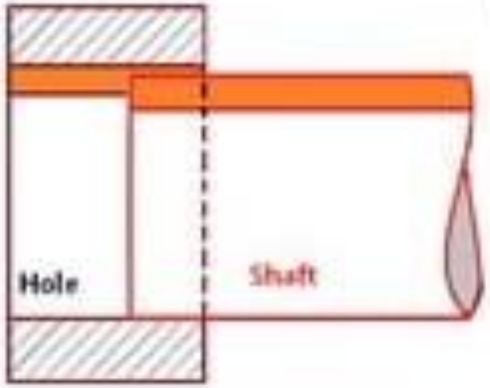
- No gap between the faces and intersecting of material will occur.
- Shaft need additional force to fit into the hole.

3. Transition fit

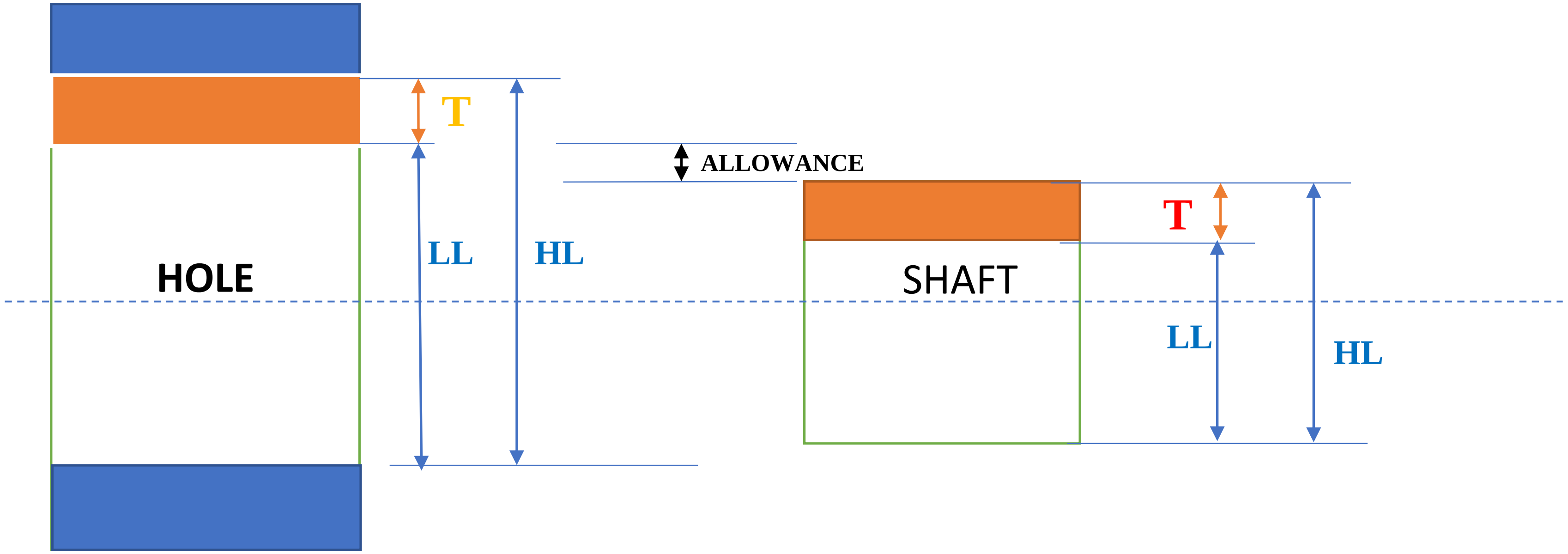
Dia. of the largest permissible hole is greater than the dia. of the smallest shaft.

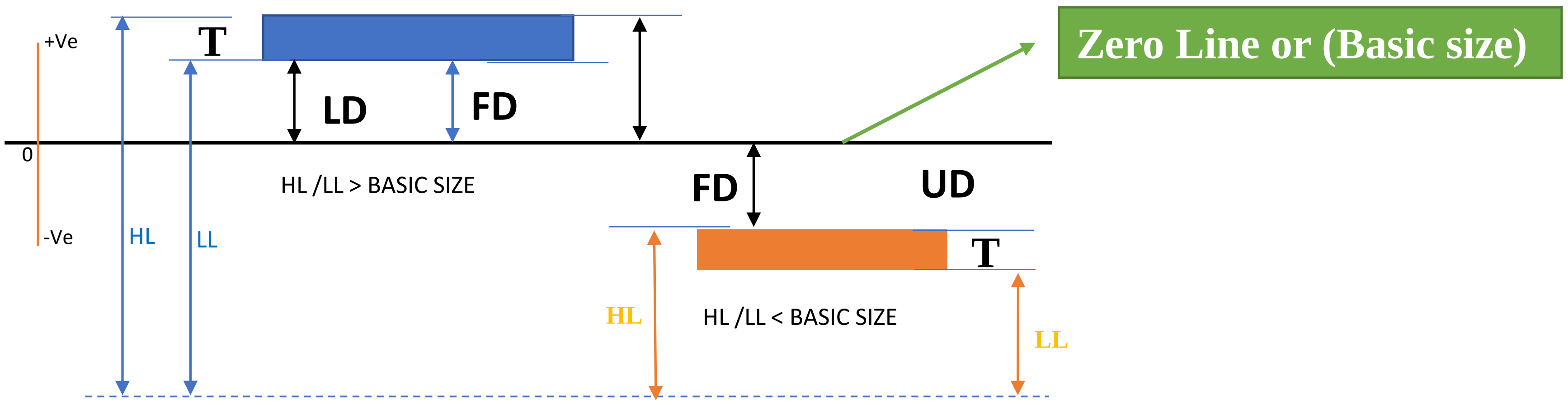






- Neither loose nor tight like clearance fit and interference fit.
- Tolerance zones of the shaft and the hole will be overlapped between the interference and clearance fits.


T
ALLOWANCE
T
LL
HL
HOLE
SHAFT
LL
HL

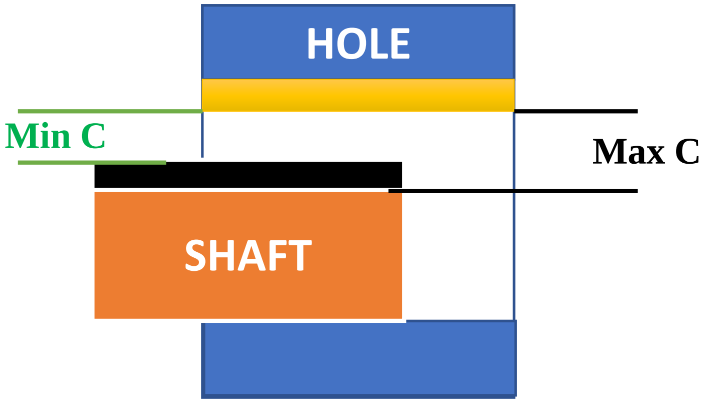


FD = Fundamental deviation

LD = Lower deviation from size

UD = Upper deviation from size

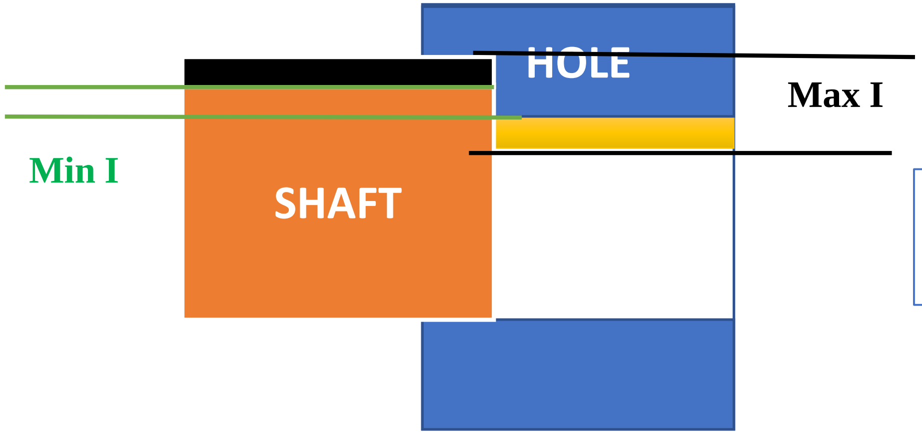
Clearance Fit



Max clearance = UL of Hole – LL of Shaft
Min Clearance = LL of Hole – UL of Shaft

Tolerance Zone Never meet

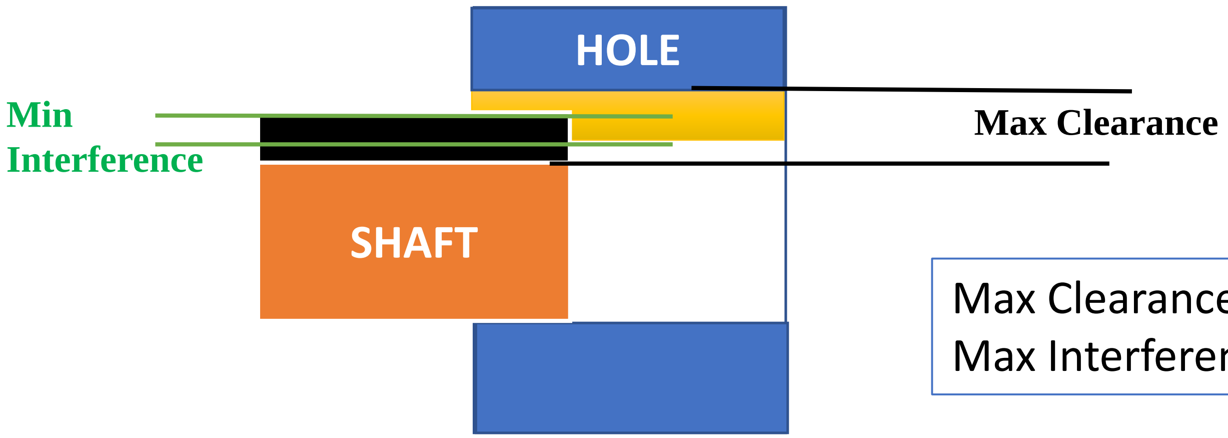
Interference Fit



Max Interference = UL of Shaft – LL of Hole
Min Interference = LL of Shaft – UL of Hole

Tolerance Zone never meet but crosses each other

Transition Fit

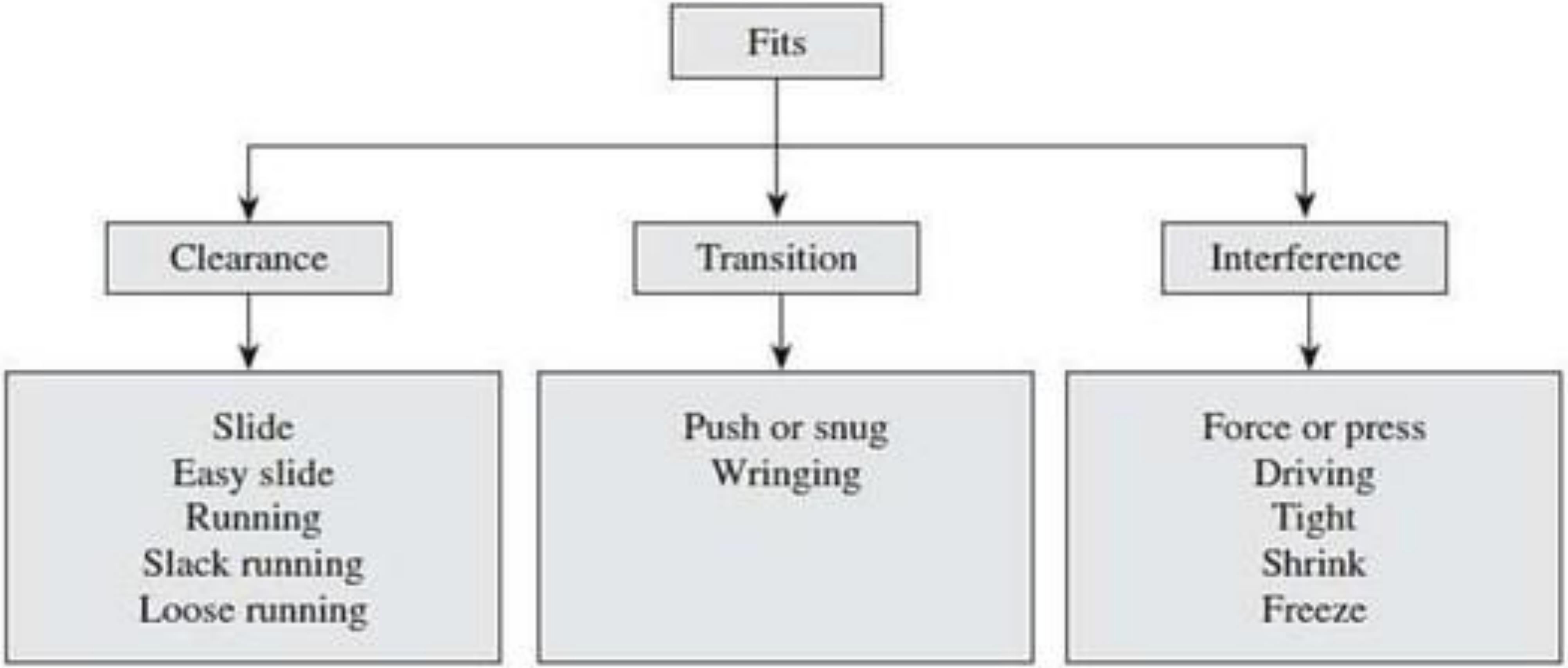


Max Clearance = UL of Hole – LL of Shaft
Max Interference = LL of Hole – UL of Shaft

Tolerance Zone is always Overlap

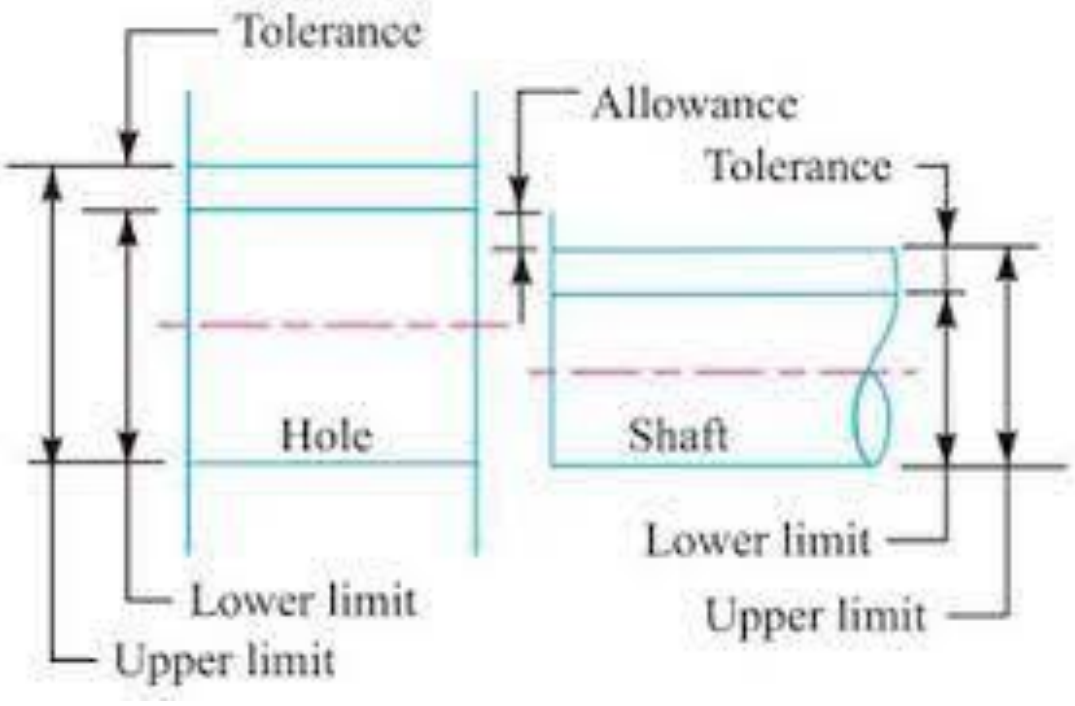
FITS

Detailed classification of Fits



Allowance :

- **Allowance:** An allowance is the intentional difference between the maximum material limits, that is, LLH and HLS (minimum clearance or maximum interference) of the two mating parts. It is the prescribed difference between the dimensions of the mating parts to obtain the desired type of fit.



- Allowance may be positive or negative. Positive allowance indicates a clearance fit, and an interference fit is indicated by a negative allowance.

Allowance = LLH − HLS

General Terminology in Fits :

Basic size: This is the size in relation to which all limits of size are derived. Basic or nominal size is defined as the size based on which the dimensional deviations are given. This is, in general, the same for both components.

Limits of size: These are the maximum and minimum permissible sizes acceptable for a specific dimension. The operator is expected to manufacture the component within these limits. The maximum limit of size is the greater of the two limits of size, whereas the minimum limit of size is the smaller of the two.

Tolerance: This is the total permissible variation in the size of a dimension, that is, the difference between the maximum and minimum limits of size. It is always positive.

General Terminology in Fits :

Allowance: It is the intentional difference between the LLH and HLS. An allowance may be either positive or negative.

Allowance = LLH − HLS

Grade: This is an indication of the tolerance magnitude; the lower the grade, the finer the tolerance.

Deviation: It is the algebraic difference between a size and its corresponding basic size. It may be positive, negative, or zero.

Upper deviation: It is the algebraic difference between the maximum limit of size and its corresponding basic size. This is designated as ‘ES’ for a hole and as ‘es’ for a shaft.

General Terminology in Fits :

Lower deviation: It is the algebraic difference between the minimum limit of size and its corresponding basic size. This is designated as ‘EI’ for a hole and as ‘ei’ for a shaft.

Actual deviation: It is the algebraic difference between the actual size and its corresponding basic size.

Fundamental deviation: It is the minimum difference between the size of a component and its basic size. This is identical to the upper deviation for shafts and lower deviation for holes.

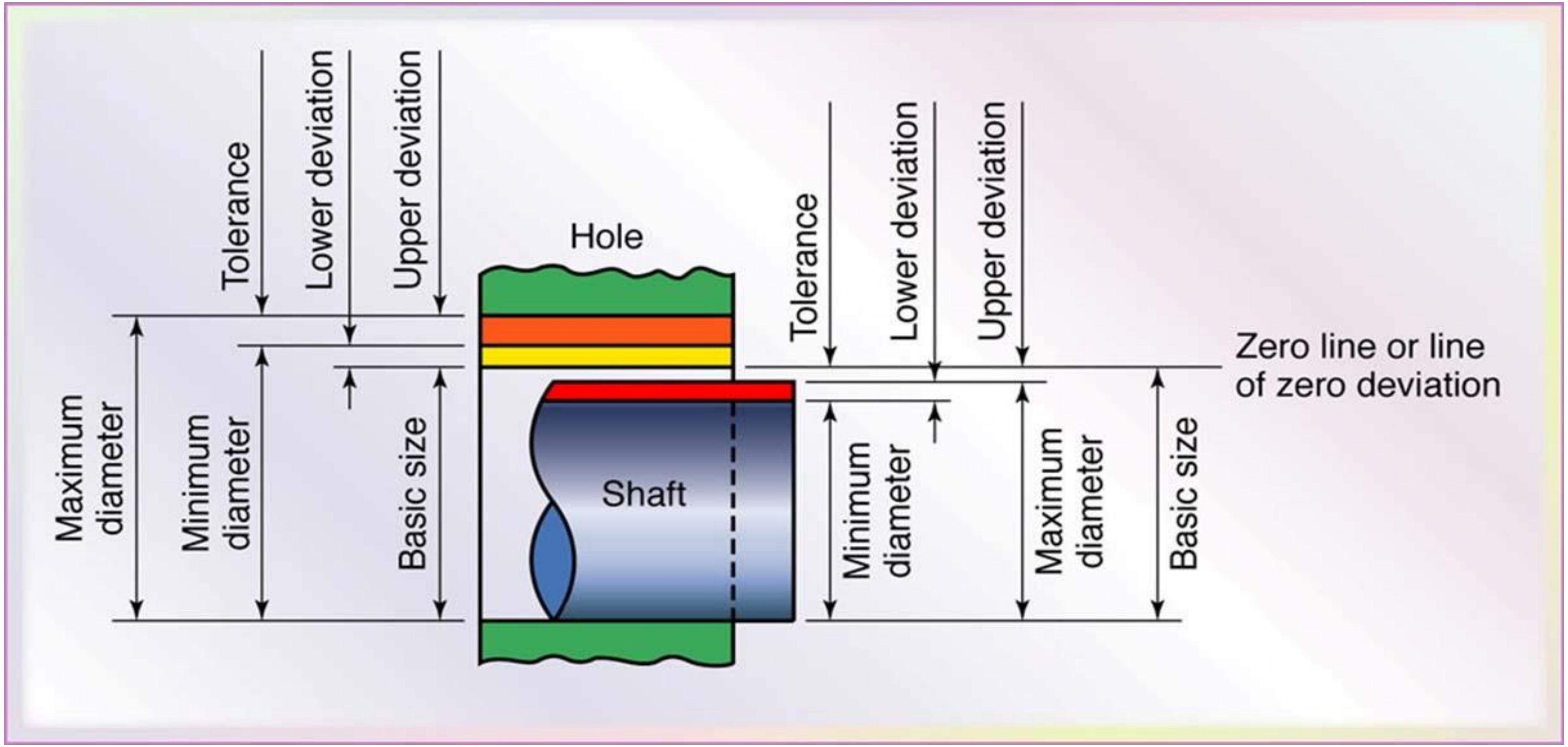
General Terminology in Fits :

Zero line: This line is also known as the line of zero deviation. The convention is to draw the zero line horizontally with positive deviations represented above and negative deviations indicated below. The zero line represents the basic size in the graphical representation.

Shaft and hole: These terms are used to designate all the external and internal features of any shape and not necessarily cylindrical.

Fit: It is the relationship that exists between two mating parts, a hole and a shaft, with respect to their dimensional difference before assembly.

General Terminology in Fits :

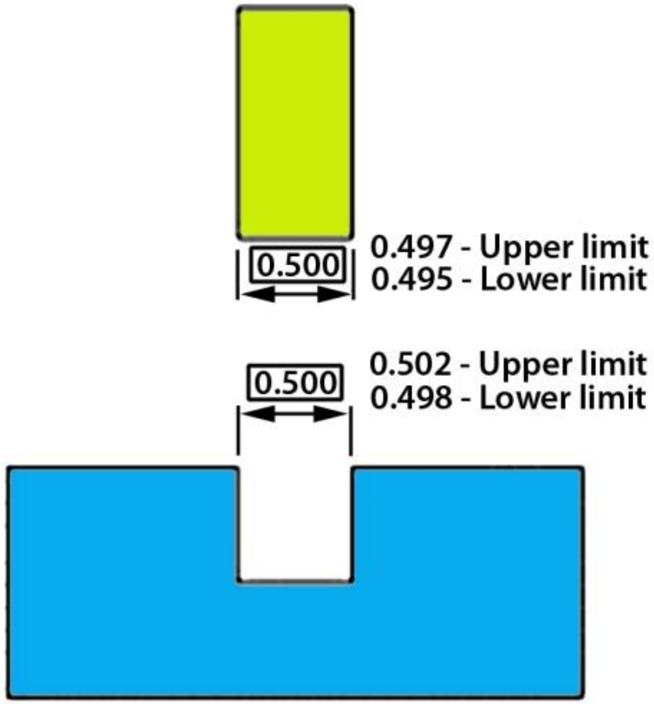


Types of Assemblies

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

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.

Types of Assemblies

(2) Interchangeable Assembly:

When a large number of components are to be produced then it will not be economical to produce both the mating components by the same operator.

- In addition to economy it is also essential to produce the components within the minimum possible time.

This is only possible by mass production system.

- In mass production system there is a division of labour. The components are produced in one or more batches by different operators on different machines. Under such conditions in order to assemble the mating components with a *desired fit, a strict control is exercised* and the parts are manufactured with specified tolerance limits.

(2) Interchangeable Assembly:

When a system of this kind is used any one component selected at random will assemble correctly with any other mating component that too, selected at random, the system is called **interchangeable assembly**.

- The manufacture of components under such conditions is called **interchangeable manufacture.**
- Production on an interchangeable basis results in increased output with a corresponding reduction in manufacturing cost.

Example : Suppose a clearance fit is required between the mating parts with hole, specified as

$$25^{+0.04}_{-0.00}\ \text{mm} \qquad \text{and} \qquad \text{shaft } 25^{-0.02}_{-0.04}\ \text{mm}$$

Interchangeable assembly requires precise machines or processes whose process capability is equal to or less than the manufacturing tolerance allowed for that part. Only then every component produced will be within **desired tolerance and capable** of mating with any other mating component to give the required fit

Advantages of Interchangeability

1. The operator is not required to waste his skill in fitting the mating components by trial and error and thus assembly time is reduced considerably
2. There is an increased output with reduced production cost

3. There is a division of labour, the operator has to perform same limited operations again and again thus he becomes specialized in that particular work, which helps to improve quality and reduce the time for operations.

4. It facilitates production of mating components at different places, by different operators.
5. The replacement of worn-out or defective parts and repair becomes very easy.

6. The cost of maintenance and shutdown period is also reduced to minimum

(3) Selective Assembly :

- It is sometimes found that it is not economical to manufacture parts to the required high degree of accuracy so as to make them interchangeable
- The consumer not only wants quality and precision trouble-free products but also he wants them at economical prices
- In selective assembly the components produced are classified into groups according to .their sizes by automatic gauging.
- This is done for both mating parts, holes and shafts, and only matched groups of mating parts are assembled.
- It results in complete protection against defective assemblies and reduces matching costs since the parts may be produced with wider tolerances.

A practical example of this system is the assembly of pistons with cylinder bores.

Let the bore size be **50 mm** and the clearance required for the assembly **is 0.12 mm** on the diameter. Let the tolerance on bore and the piston each **= 0.04 mm.** Then,

Dimension of bore diameter is **$50^{\pm 0.02}$**

Dimension of piston shirt is mm **$49.88^{\pm 0{\cdot}02}$**

By grading and making the bores and the pistons they may be selectively assembled to give the clearance of 0.12 mm as given below:

Cylinder bore	49.98	50.00	50.02
Piston	49.86	49.88	49.90

Hole Basis and Shaft Basis Systems

Hole Basis System :

In this system, the size of the hole is kept constant and the shaft size is varied to give various types of fits.

In a hole basis system, the fundamental deviation or lower deviation of the hole is zero, that is, the lower limit of the hole is the same as the basic size.

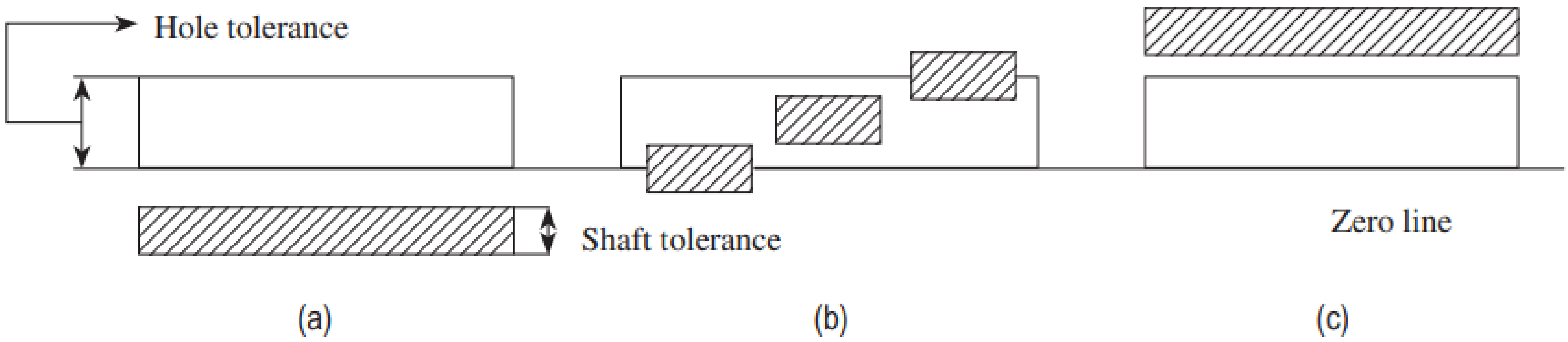


Fig. 3.13 Hole basis system (a) Clearance fit (b) Transition fit (c) Interference fit

Hole Basis System :

- This type of system is widely adopted in industries, as it is easier to manufacture shafts of varying sizes to the required tolerances.
- Standard size drills or reamers can be used to obtain a variety of fits by varying only the shaft limits, which leads to greater economy of production.
- The shaft can be accurately produced to the required size by standard manufacturing processes, and standard-size plug gauges are used to check hole sizes accurately and conveniently.

Shaft Basis System :

The system in which the dimension of the shaft is kept constant and the hole size is varied to obtain various types of fits is referred to as shaft basis system.

- In this system, the fundamental deviation or the upper deviation of the shaft is zero, that is, the HLH equals the basic size.
- The desired class of fits is obtained by varying the lower limit of the shaft and both limits of the hole

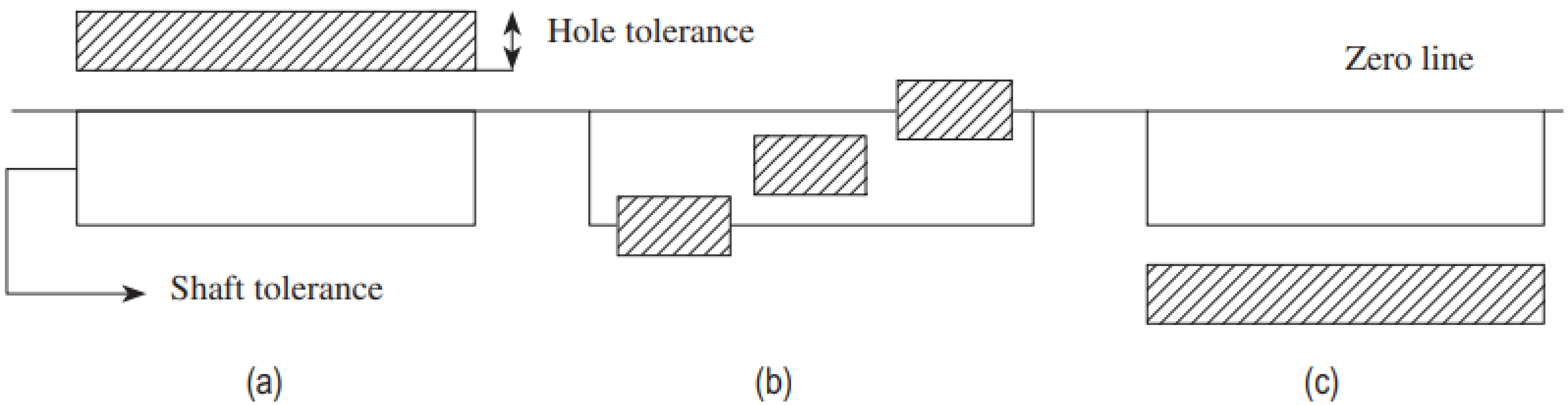
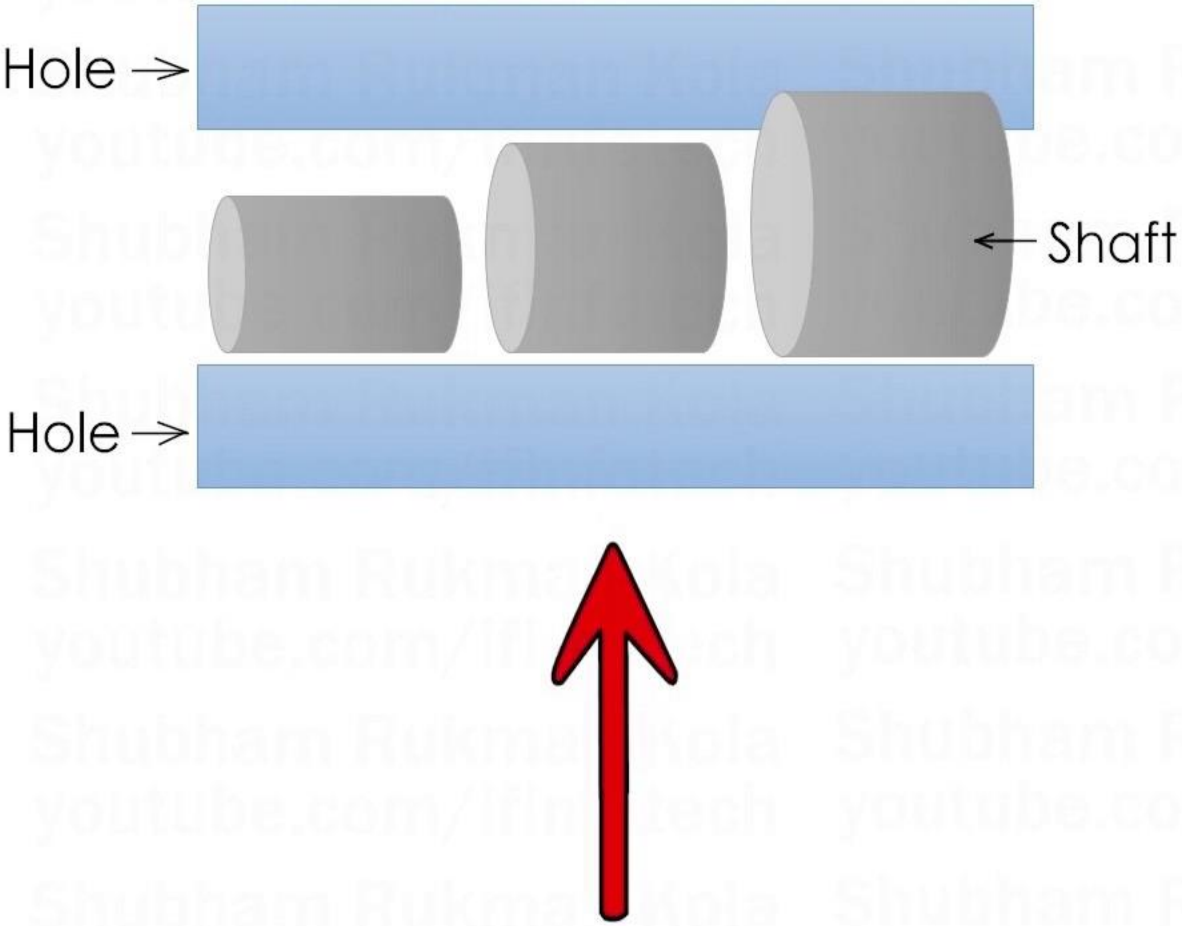


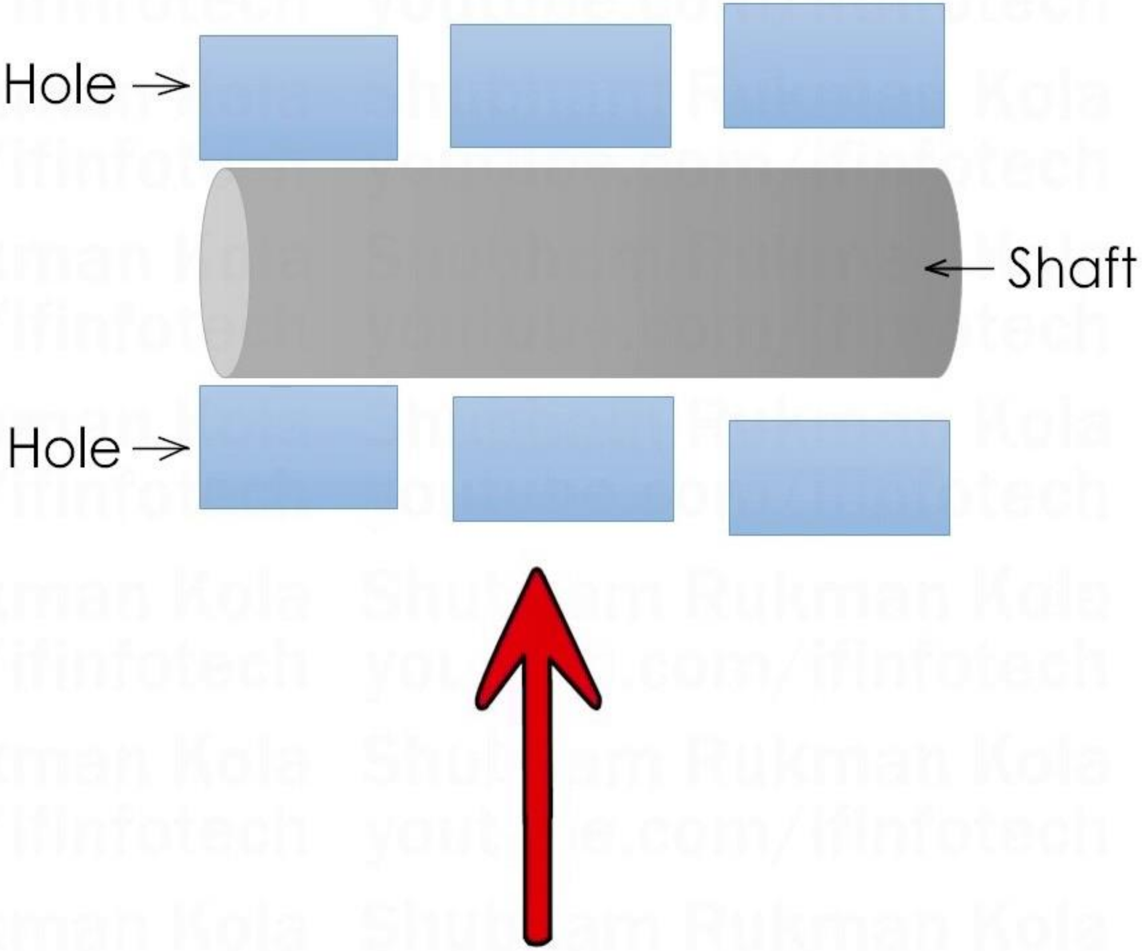
Fig. 3.14 Shaft basis system (a) Clearance fit (b) Transition fit (c) Interference fit

- This system is **not preferred in industries**, as it requires more number of standard-size tools such as reamers, broaches, and gauges, which increases manufacturing and inspection costs.
- It is normally preferred where the hole dimension is dependent on the shaft dimension and is used in situations where the standard shaft determines the dimensions of the mating parts such as couplings, bearings, collars, gears, and bushings.

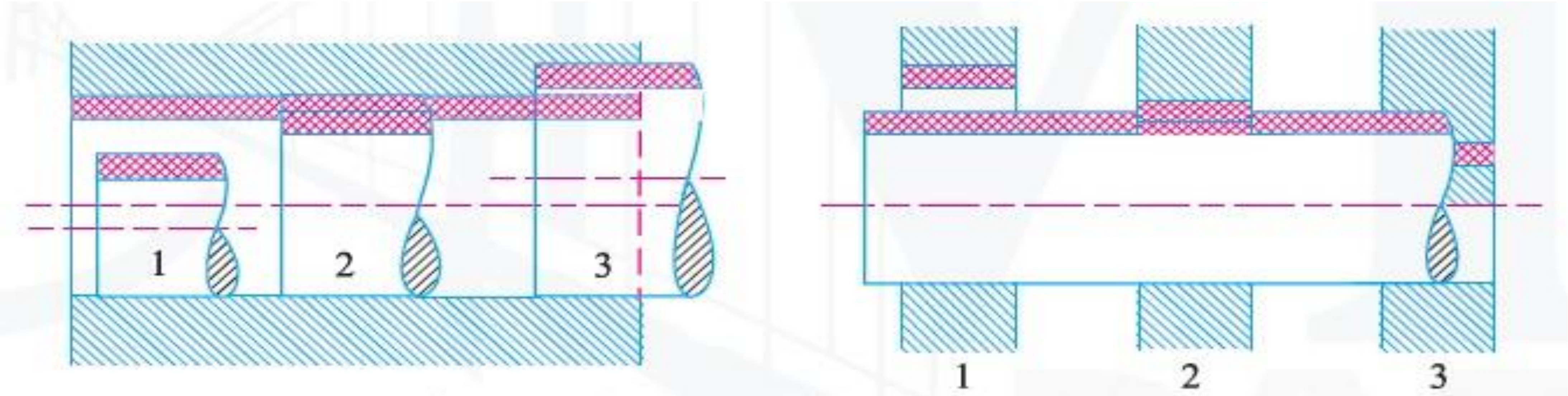
Hole Basis System



Shaft Basis System



HOLE BASIS SYSTEM	SHAFT BASIS SYSTEM



1. Clearance fit. 2. Transition fit. 3. Interference fit.

(*a*) Hole basis system. (*b*) Shaft basis system.

Fig. 3.6. Bases of limit system.

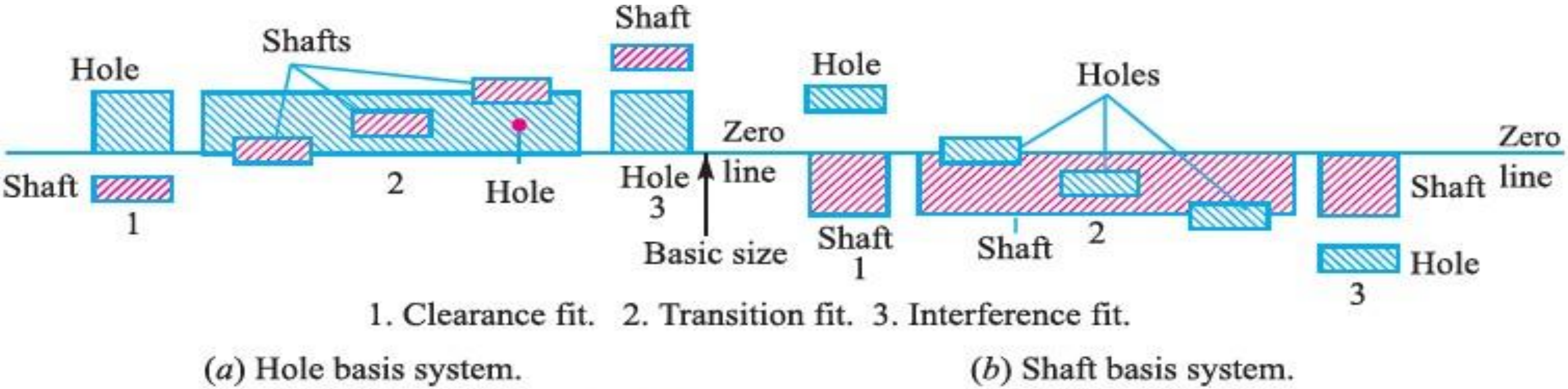
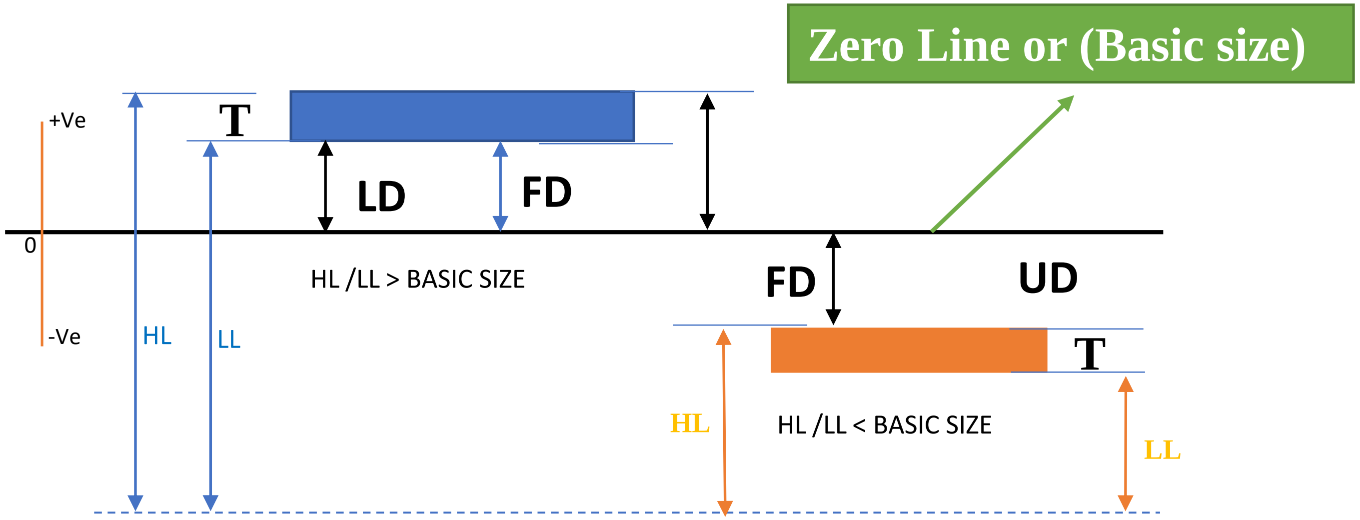


Fig. 3.7. Bases of limit system.

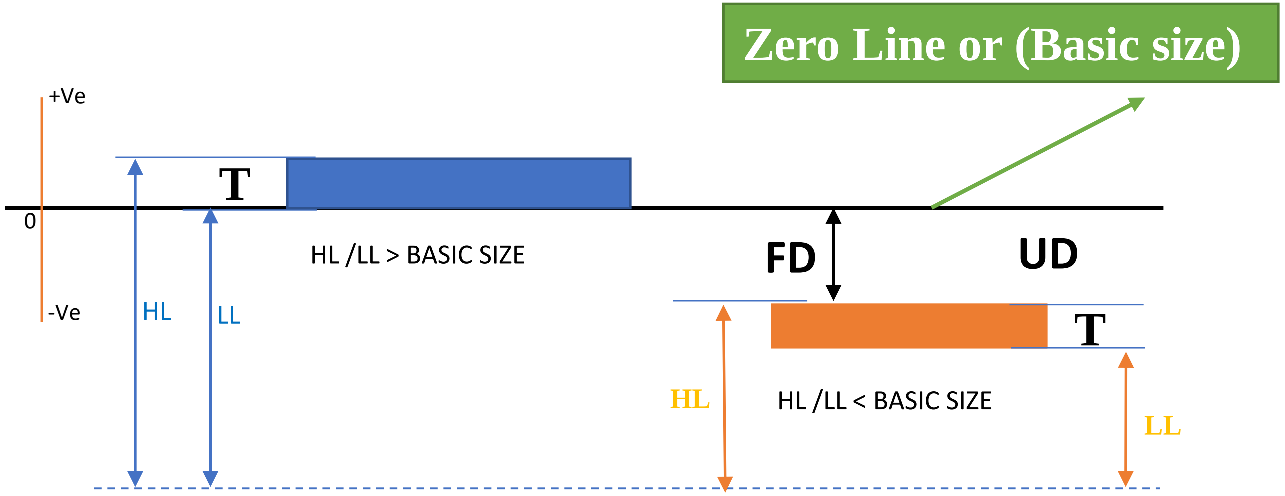


FD = Fundamental deviation

LD = Lower deviation from size

UD = Upper deviation from size

Hole Basis system



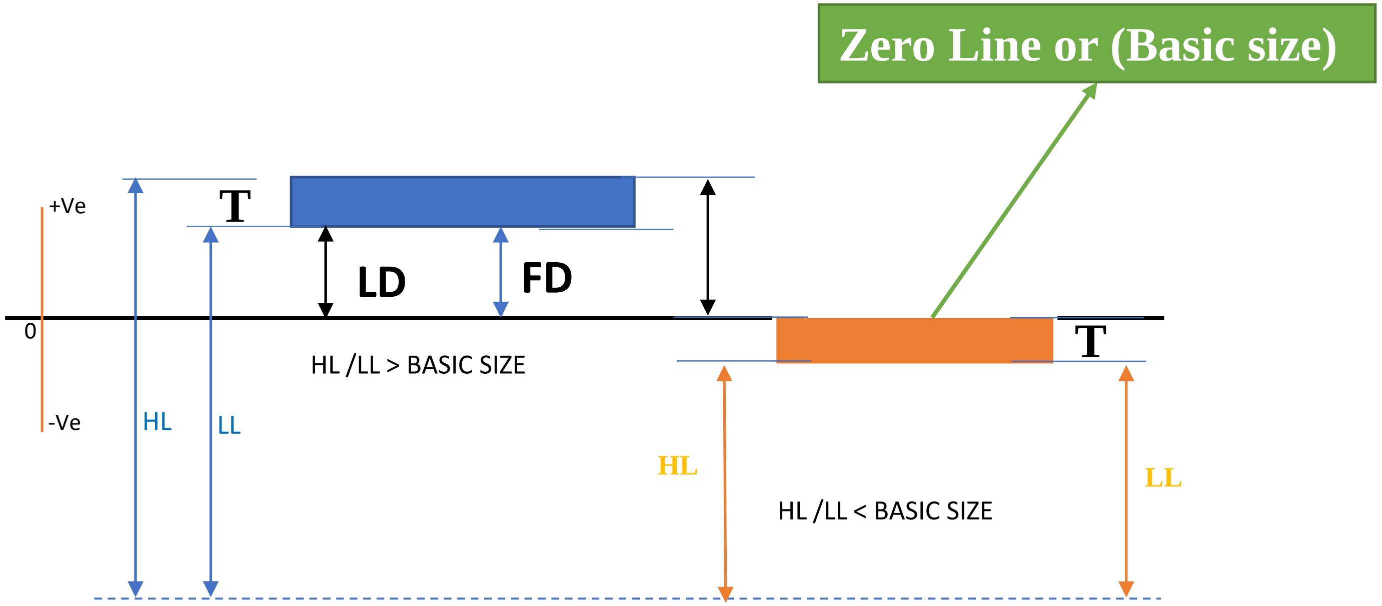
FD = Fundamental deviation

LD = Lower deviation from size

UD = Upper deviation from size

FD = LD = Zero value

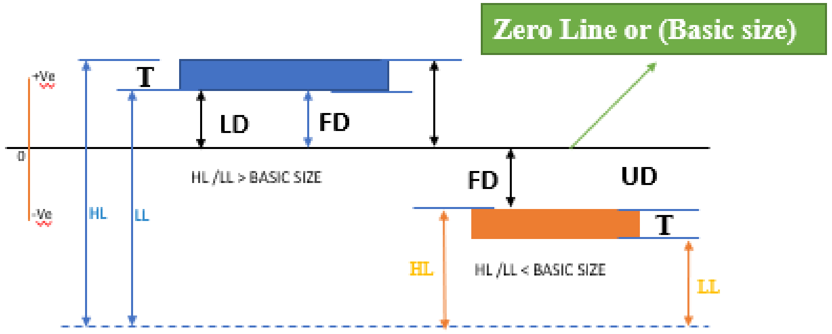
Shaft Basis system



FD = Fundamental deviation

LD = Lower deviation from size

UD = Upper deviation from size

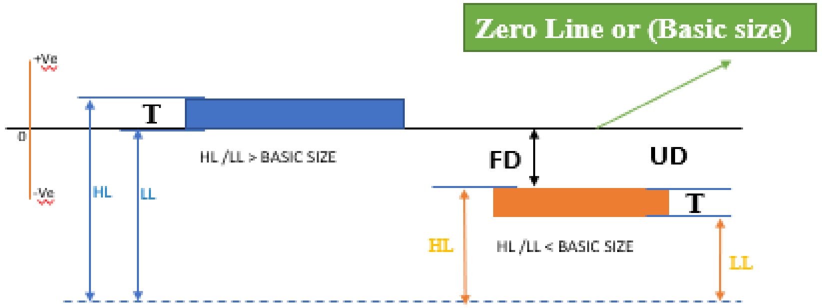


FD = Fundamental deviation

LD = Lower deviation from size

UD = Upper deviation from size

Hole Basis system



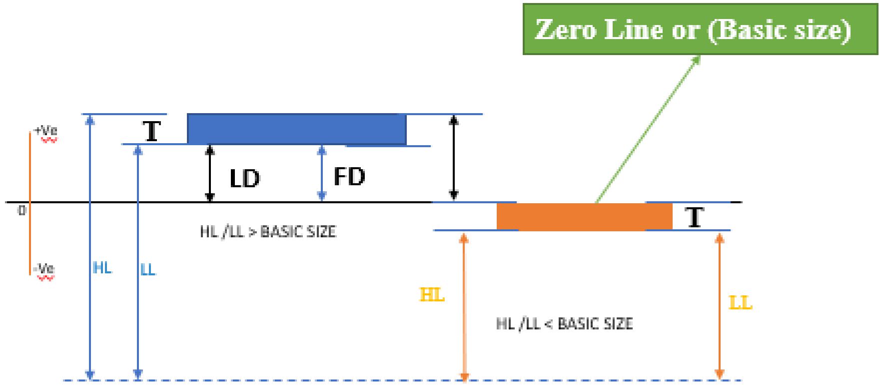
FD = Fundamental deviation

LD = Lower deviation from size

UD = Upper deviation from size

FD = LD = Zero value

Shaft Basis system



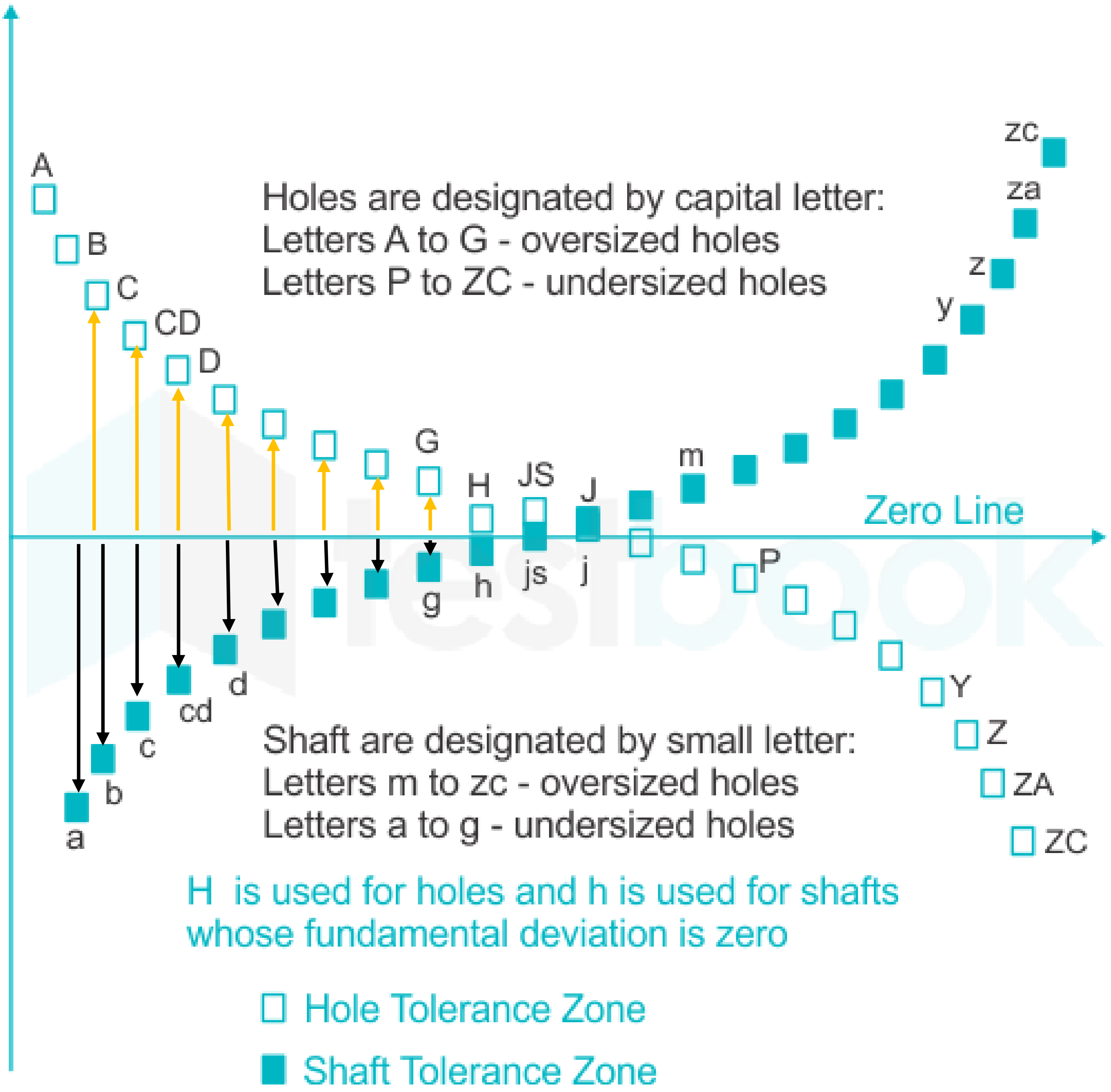
FD = Fundamental deviation

LD = Lower deviation from size

UD = Upper deviation from size

FD = LD

FD = UD



Holes are designated by capital letter:
Letters A to G - oversized holes
Letters P to ZC - undersized holes
A
B
C
CD
D
G
H
JS
J
P
Y
Z
ZA
ZC
zc
za
z
y
m
Zero Line
a
b
c
cd
d
g
h
js
j
Shaft are designated by small letter:
Letters m to zc - oversized holes
Letters a to g - undersized holes
H is used for holes and h is used for shafts
whose fundamental deviation is zero
Hole Tolerance Zone
Shaft Tolerance Zone

Indian Standard System of Limits and Fits

- According to Indian standard [IS : 919 (Part I)-1993], the system of limits and fits comprises **18 grades** of fundamental tolerances i.e. grades of accuracy of manufacture and 25 types of fundamental deviations indicated by letter symbols for both holes and shafts (**capital letter A to ZC for holes** and **small letters a to zc for shafts**) in diameter steps ranging from 1 to 500 mm.
- A unilateral hole basis system is recommended but if necessary a unilateral or bilateral shaft basis system may also be used.
- The 18 tolerance grades are designated as IT 01, IT 0 and IT 1 to IT 16. These are called standard tolerances.

Tolerance Designation(ISO)

Tolerance on a shaft or hole can also be calculated by using formulas provided by ISO

Tolerance (T) = K × i

Where **T** is the tolerance (in µm)

$$i\ (\text{microns}) = 0.45\sqrt[3]{D} + 0.001\,D$$

(Unit tolerance in µm

$$D = \sqrt{D_{max} \times D_{min}}$$

(D max and D min are the nominal sizes marking the beginning and the end of a range of sizes in mm

Indian Standard System of Limits and Fits

- The standard tolerances for grades IT 5 to IT 7 are determined in terms of standard tolerance unit (i) in microns, where

$$i\ (\text{microns}) = 0.45\ \sqrt[3]{D} + 0.001\ D$$

where D is the size or geometric mean diameter in mm.

- The following table shows the relative magnitude for grades between IT 5 and IT 16.

Table 3.2. Relative magnitude of tolerance grades.

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

Indian Standard System of Limits and Fits

The various steps specified for the diameter steps are as follows: 1–3, 3–6, 6–10, 10–18, 18–30, 30–50, 50–80, 80–120, 120–180, 180–250, 250–315, 315– 400, 400–500 500–630, 630–800, and 800–1000 mm

The values of standard tolerances corresponding to grades IT 01, IT 0 and IT 1 are as given below:

- For IT 01, i (microns) = 0.3 + 0.008 D,
- For IT 0, i (microns) = 0.5 + 0.012 D, and
- For IT 1, i (microns) = 0.8 + 0.020 D,
- where D is the size or geometric mean diameter in mm.
- The tolerance values of grades IT 2 to IT 4 are scaled approximately geometrically between IT 1 and IT 5.

Indian Standard System of Limits and Fits

$$i \text{ (microns)} = 0.45 \sqrt[3]{D} + 0.001 D$$

The various steps specified for the diameter steps are as follows:

1–3,

3–6,

6–10,

10–18,

18–30,

30–50,

50–80,

80–120,

120–180,

180–250,

250–315,

315– 400,

400–500

500–630,

630–800,

and

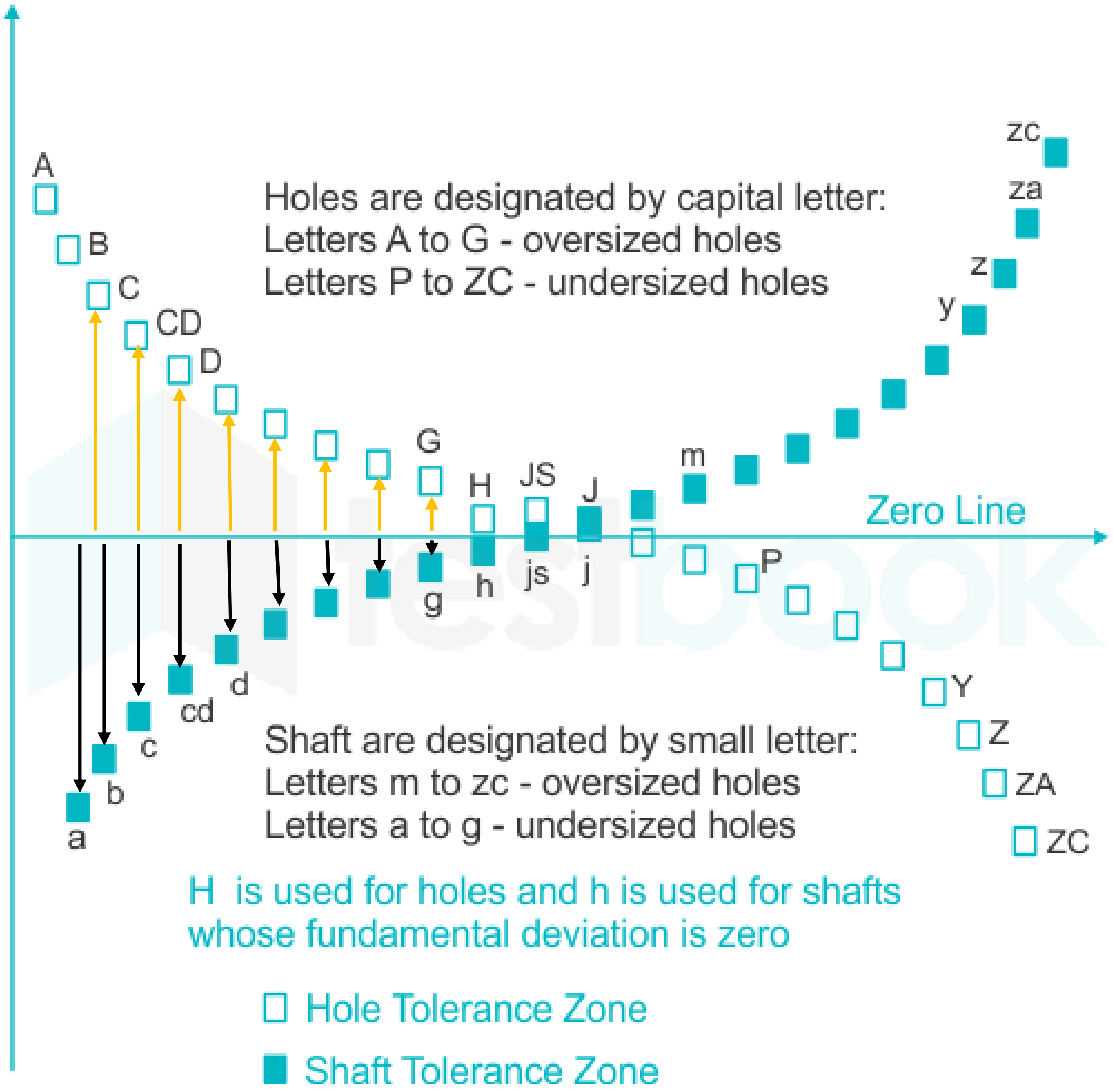
800–1000 mm

where D is the size or geometric mean diameter in mm.

$$\sqrt{D_{max} \times D_{min}}$$

FD = LD

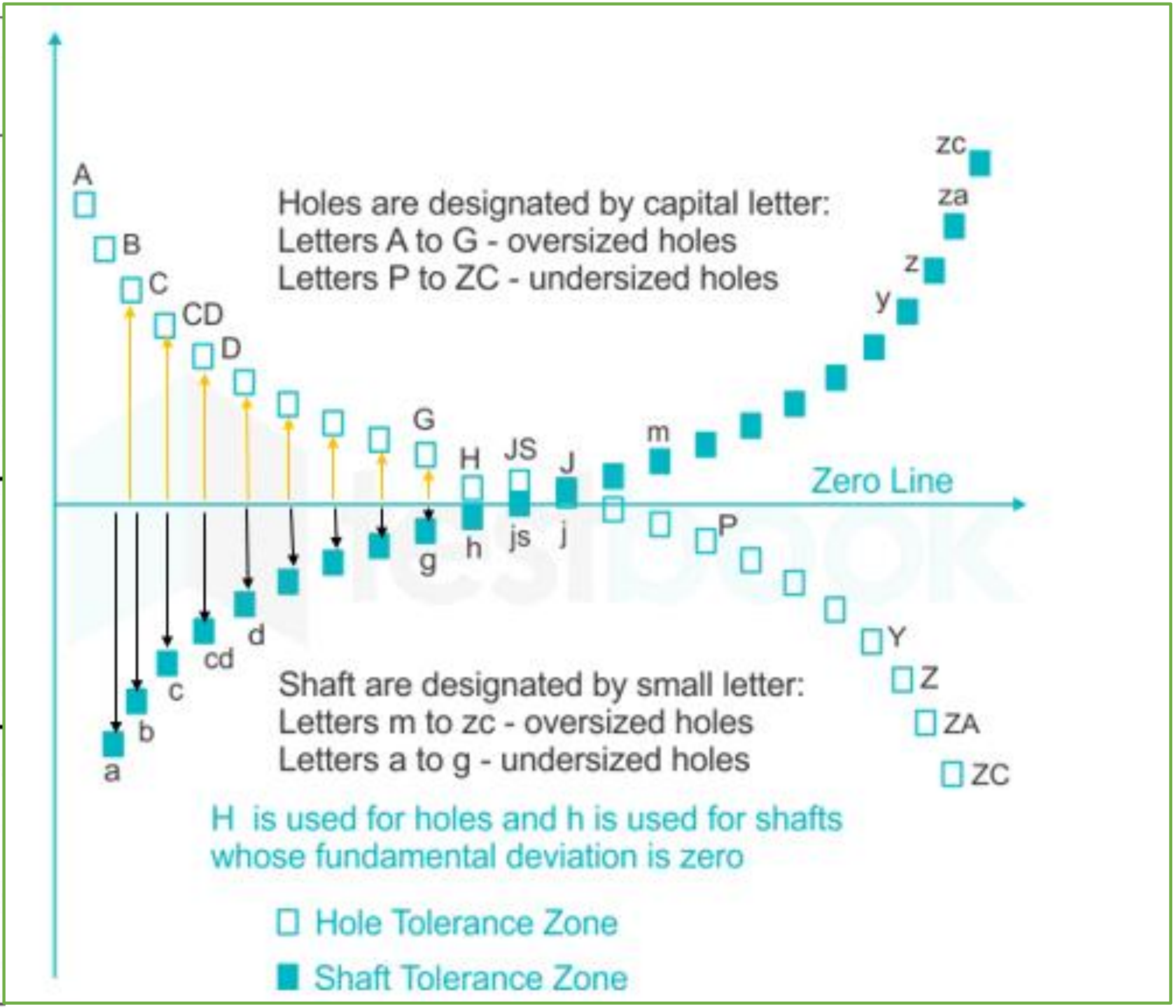
FD = UD


A
B
C
CD
D
G
H
JS
J
m
P
Y
Z
ZA
ZC
zc
za
z
y
Zero Line
a
b
c
cd
d
g
h
js
j
Holes are designated by capital letter:
Letters A to G - oversized holes
Letters P to ZC - undersized holes
Shaft are designated by small letter:
Letters m to zc - oversized holes
Letters a to g - undersized holes
H is used for holes and h is used for shafts whose fundamental deviation is zero
Hole Tolerance Zone
Shaft Tolerance Zone

	ISO SYMBOL		DESCRIPTION	
	Hole Basis	Shaft Basis		
Clearance Fits	H11/c11	C11/h11	Loose running fit for wide commercial tolerances or allowances on external members.	More Clearance ↑
	H9/d9	D9/h9	Free running fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures.	
	H8/f7	F8/h7	Close running fit for running on accurate machines and for accurate location at moderate speeds and journal pressures.	
	H7/g6	G7/h6	Sliding fit not intended to run freely, but to move and turn freely and locate accurately.	
Transition Fits	H7/h6	H7/h6	Locational clearance fit provides snug fit for locating stationary parts; but can be freely assembled and disassembled.	
	H7/k6	K7/h6	Locational transition fit for accurate location, a compromise between clearance and interference.	
	H7/n6	N7/h6	Locational transition fit for more accurate location where greater interference is permissible.	
Interference Fits	H7/p6[1]	P7/h6	Locational interference fit for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements.	More Interference ↓
	H7/s6	S7/h6	Medium drive fit for ordinary steel parts or shrink fits on light sections, the tightest fit usable with cast iron.	
	H7/u6	U7/h6	Force fit suitable for parts which can be highly stressed or for shrink fits where the heavy pressing forces required are impractical.	

[1]Transition fit for basic sizes in range from 0 through 3 mm.

	ISO SYMBOL	
	Hole Basis	**Shaft Basis**
Clearance Fits	H11/c11	C11/h11
	H9/d9	D9/h9
	H8/f7	F8/h7
	H7/g6	G7/h6
Transition Fits	H7/h6	H7/h6
	H7/k6	K7/h6
	H7/n6	N7/h6
Interference Fits	H7/p6[1]	P7/h6
	H7/s6	S7/h6
	H7/u6	U7/h6



Note :

Hole : (A – G) lower limit or Deviation
H Tolerance zone or /Zero line
(1 –ZC) Upper limit /Deviation (Fundamental Deviation below the zero line)

Shaft : (a – g) Upper deviation below the zero line
h on zero line
(j – z) lower deviation (Above the zero line)

Φ50 H8 f7

Basic size

IT #

Fundamental deviation

Example 1

1) **In bush and pin assembly, pin of 30 mm diameter rotates in a bush. The tolerance for pin is 0.025 mm while for bush is 0.04 mm. If allowance is 0.1 mm, determine dimensions of pin and bush considering hole-basis system.**

Given Data:-

Basic size = 30 mm,

Allowance = 0.1 mm,

Tolerance for pin = 0.025 mm,

Tolerance for bush = 0.04 mm

Hole basis system To find: Dimension of pin & bush.

Step: -1 Calculate the dimension of bush.

For hole basis system, fundamental deviation of bush is zero.

Lower limit of bush = Basic size + fundamental deviation = 30 + 0 = 30 mm.

Upper limit of bush = Lower limit of bush + Tolerance for bush = 30 + 0.04 = 30.04 mm.

Step: -2 Calculate the dimension of pin.

With positive allowance, it is a clearance fit.

Allowance = Lower limit of bush – Upper limit of pin = 30 – Upper limit of pin

Upper limit of pin = 29.9 mm.

Lower limit of pin = Upper limit of pin – Tolerance for pin = 29.9 - .025 = 29.875 mm.

Dimension of bush = 30 +0.04 +0.00 mm.

Dimension of pin = 30 -0.100 -0.125 mm.

Example 2

2) A journal of nominal diameter 79 mm rotates in a bearing. The upper and lower deviations in hole diameter are respectively +0.05 mm and 0.00 mm, while those for shaft are respectively -0.03 mm and -0.07 mm. Calculate: (i) Extreme diameters for hole and shaft, (ii) Tolerances for hole and shaft and (iii) maximum and minimum clearance.

Given Data

Nominal Diameter = Ø 79 mm.

Hole - Upper Deviation = + 0.05 mm.

Lower Deviation = 0.00 mm.

Shaft - Upper Deviation = - 0.03 mm.

Lower Deviation = - 0.07 mm.

Solution

Upper Deviation = Max limit – Basic size.

Max limit = Upper Deviation + Basic size.

For hole Max limit = + 0.05 + 79 = 79.05 mm.

For shaft Max limit = - 0.03 + 79 = 78.97 mm.

Lower Deviation = Min limit – Basic size.

Min limit = Lower Deviation + Basic size.

For hole Min limit = 0.00 + 79 = 79.00 mm.

For shaft Min limit = - 0.07 + 79 = 78.93 mm.

Tolerance = Max limit – Min limit.

Tolerance for hole = Max limit – Min limit = 79.05 – 79 = 0.05 mm.

Tolerance for shat = Max limit – Min limit = 78.97 – 78.93 = 0.04 mm. Min

Clearance = Min hole – Max shaft = 79 – 78.97 = 0.03 mm.

Max Clearance = Max hole – Min shaft = 79.05 – 78.93 = 0.12 mm.

Example 3

3) Find the tolerances, maximum interference and type of fit for the data for the following given data:

Hole ϕ50 +0.25 -0.10 and Shaft ϕ50 +0.20-0.20

Given Data

Hole Ø 50 + 0.25 -0.10,

Shaft Ø 50 + 0.20 -0.20.

Solution

Upper Deviation = Max limit – Basic size. Max limit = Upper

Deviation + Basic size. For hole Max limit = + 0.25 + 50 = 50.25 mm.

For shaft Max limit = + 0.20 + 50 = 50.20 mm.

Lower Deviation = Min limit – Basic size.

Lower Deviation = Min limit – Basic size.

Min limit = Lower Deviation + Basic size.

For hole Min limit = - 0.10 + 50 = 49.90 mm.

For shaft Min limit = - 0.20 + 50 = 49.80 mm.

Tolerance = Max limit – Min limit.

Tolerance for hole = Max limit – Min limit = 50.25 – 49.90 = 0.35 mm.

Tolerance for shat = Max limit – Min limit = 50.20 – 49.80 = 0.40 mm.

Max Interference = Max shaft – Min hole = 50.20 – 49.90 = 0.30 mm.

Type of fit is Interference.

LIMIT GAUGES

- Gauges are inspection tools which serve to check the dimensions of the manufactured pmts. Limit gauges ensure the size of the component lies within the specified limits.
- They are non-recording and do not determine the size of the part. Gauges are generally classified as:

1. **Standard gauges** are made to the nominal size of the part to be tested and have the measuring member equal in size to the mean permissible dimension of the part to be checked.

LIMIT GAUGES

2. Limit Gauges are also called 'GO' and 'NO GO' gauges.

- These are made to the limit sizes of the work to be measured.
- One of the sides or ends of the gauge is made to correspond to maximum and the other end to the minimum permissible size.
- The function of limit gauges is to determine whether the actual dimensions of the work are within outside the specified limits.

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

LIMIT GAUGES

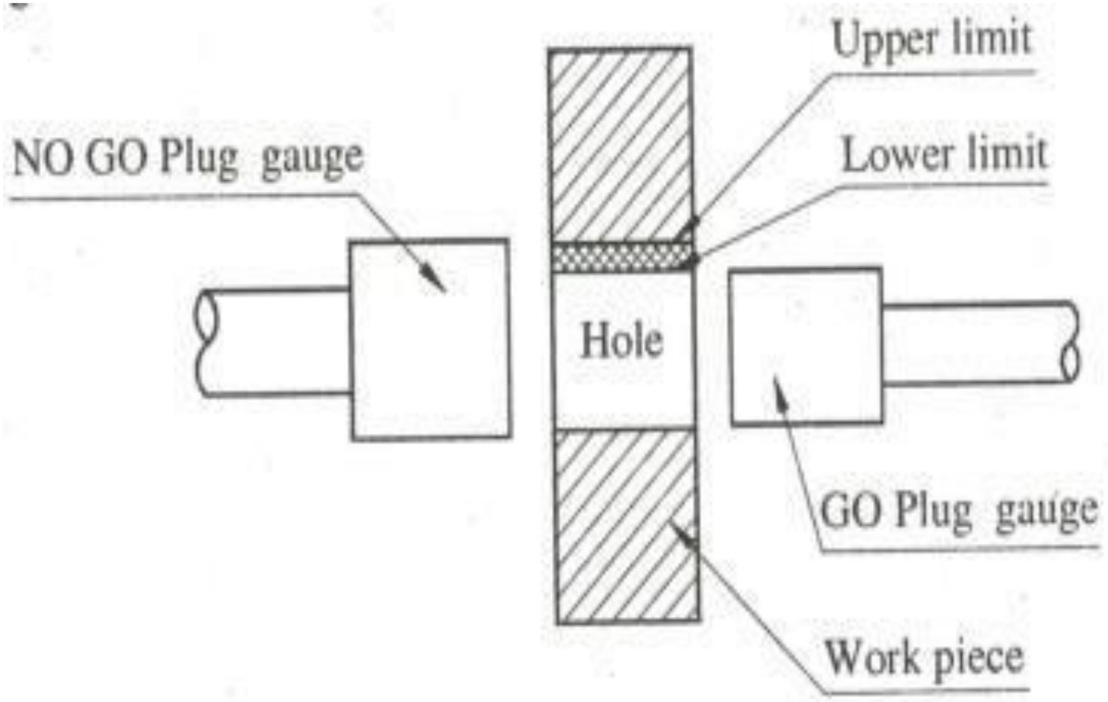
- They are well suited for use in the production area of the' factory as they require little skill or interpretation to use effectively and have few, if any, moving parts to be damaged in the often hostile production environment.

Why limit gauges necessary?

- In the manufacturing firm, the components are manufactured as per the specified tolerance limits, upper limit and lower limit. The dimension of each component should be within this upper and lower limit.
- If the dimensions are outside these limits, the components will be rejected. If we use any measuring instruments to check these dimensions, the process will consume more time.
- Also, in mass production, we are not interested in knowing the amount of error in dimensions. It is just enough whether the size of the component is within the prescribed limits or not. For this purpose, limit gauges are used.
- This procedure is mostly performed by the quality control department of each and every industry. Limit gauge are mainly used for checking for **cylindrical holes of identical** components with a large numbers in mass production.

Basic concept of Gauge Design (Taylor's Principle)

- According to Taylor, '**GO**' and '**NOGO**' gauges should be designed to check maximum and minimum material limits.
- The terms minimum metal condition, and maximum metal condition are used to describe the tolerance state of a work piece. The **GO** gauge is made near the maximum metal condition.
- The GO gauge must be able to slip inside/over the feature without obstruction. For example plug gauge (for checking hole size), as shown in Fig. having exactly the GO limit.



Basic concept of Gauge Design (Taylor's Principle)

- Diameter and a length equal to the engagement length of the fit to be made for checking the GO limit of the work piece and this gauge must perfectly assemble with the work piece to be inspected

- The NO GO gauge is made near the minimum metal condition. NO GO gauge which contacts the work piece surface only in two diametrically opposite points and at those points it should have exactly NO GO limit diameter.

- The NO GO gauge must not be able to slip inside/over the work piece in any consecutive position in various diametric directions on the work piece length.

TYPES OF LIMIT GAUGES

1. According to their purpose:

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

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

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

TYPES OF LIMIT GAUGES

2. According to form of tested surfaces:

a) **Plug gauges:** They check the dimensions of a hole.

b) **Ring gauges:** They check the dimensions of a shaft.

c) **Snap gauges:** They also check the dimensions of a shaft. Snap gauges can be used for both cylindrical as well as non-cylindrical work as compared toting gauges which are conveniently used only for cylindrical work.

3. According to their design:

a) Single limit & double limit gauges

b) Single ended and double ended gauges

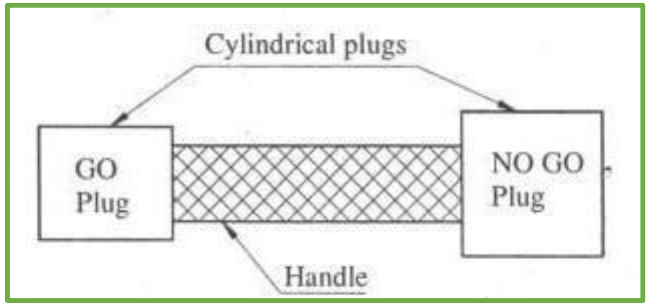
c) Fixed & adjustable gauges

TYPES OF LIMIT GAUGES

- Plug gauges are used for checking holes and consist of two cylindrical wear resistant plugs. The plug made to the lower limit of the hole is known as 'GO' end and this will enter any hole which is not smaller than the lower limit allowed.
- The plug made to the upper limit of the hole is known as 'NO GO' end and this will not enter any hole which is smaller than the upper limit allowed. The plugs are arranged on either ends of a common handle.
- The ends are hardened and accurately finished by grinding. One end is the **GO** end and the other end is NO GO end. If the size of the hole is within the limits, the GO end should go inside the hole and NO GO end should not go.

Plug gauges

- If the GO end does not go, the hole is under size and also if NO GO end goes, the hole is over size. Hence, the components are rejected in both the cases.
- The GO end and NO GO end are arranged on both the ends of the plug as shown in Figure This type has the advantage of easy handling and are called double ended plug gauges.



Double ended plug gauge

Plug gauges

- **In case of progressive type** of plug gauges as shown in the Figure both the GO end and NO GO end are arranged in the same side of the plug. We can use the plug gauge ends progressively one after the other while checking the hole. It saves time.
- Generally, the GO end is made longer than the NO GO end in plug gauges

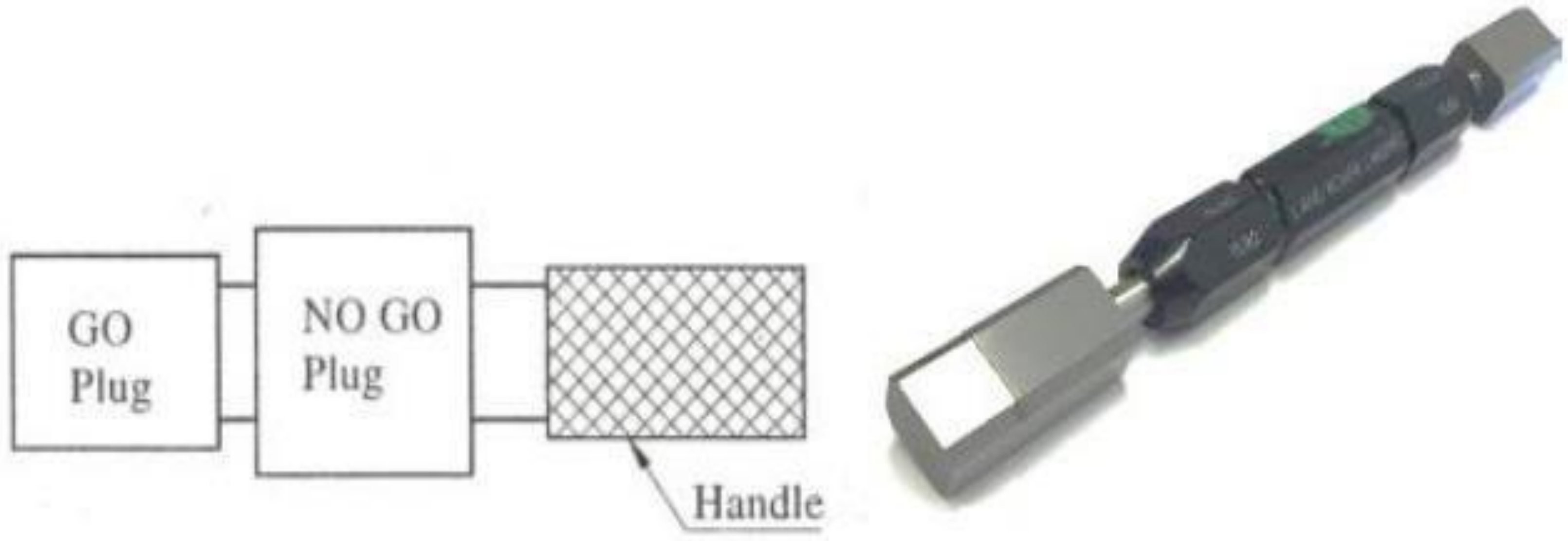


Figure 2.33 Progressive plug gauge

Plug gauges

- As shown in Figure taper plug gauges are used to check tapered holes. It has two check lines, one is a GO line and another is a NO GO line. During the checking of work, NO GO line remains outside the hole and GO line remains inside the hole. They are various types taper plug gauges are available as taper plug gauge- plain, taper plug gauge- tanged, taper ring gauge- plain and taper ring gauge- tanged.

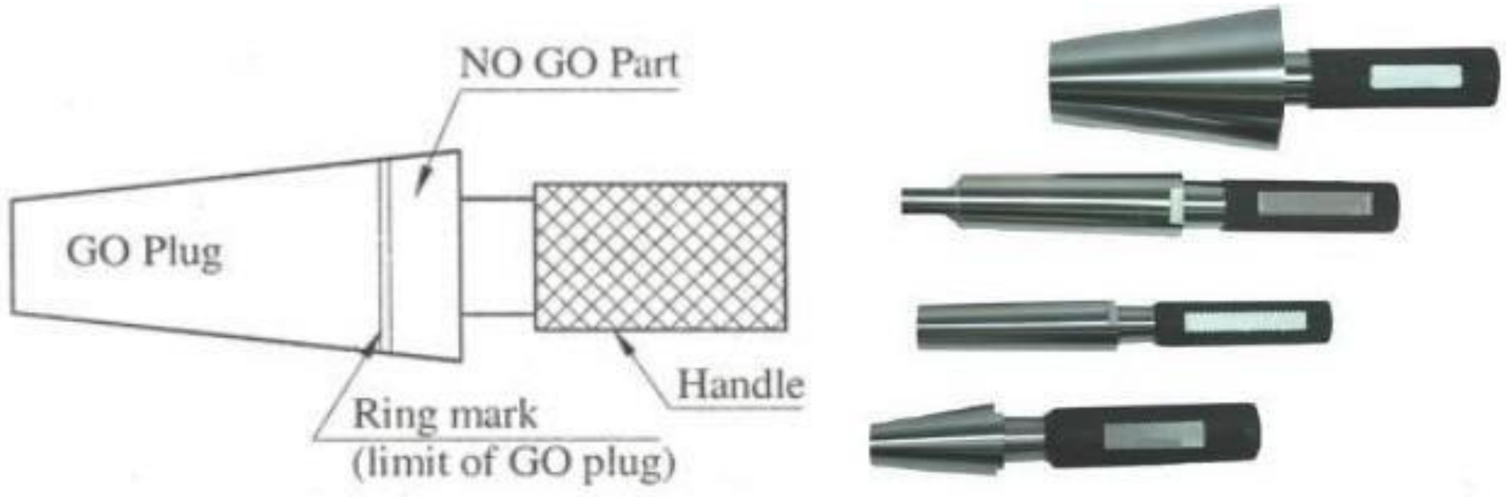


Figure 2.34 Taper plug gauge

Ring gauge

- Ring gauges are used for checking the diameter of shafts. They are used in a similar manner to that of **GO** and **NO GO plug gauges**. A ring gauge consists of a piece of metal in which a hole of required size is bored as shown in Figure.

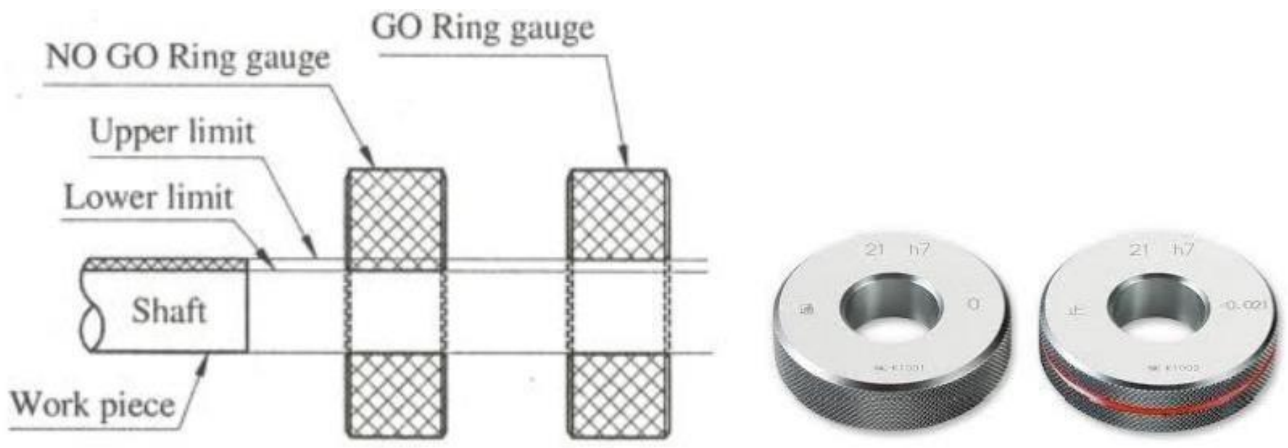


Figure 2.35 Ring gauges

Ring gauge

- The hole or bore is accurately finished by grinding and lapping after taking hardening process. The hole of GO ring gauge is made to the. Upper limit size of the shaft and NO GO for the lower limit.
- The periphery of the ring is knurled to give more grips while handling the gauges.
- Nominally, the GO ring gauge and NO GO ring gauge are separately used to check the shaft.
- Generally, NO GO made with a red mark or a small groove cut on its periphery in order to identify NO GO gauge.

Snap gauge

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

Figure 2.36 Double ended snap gauge

Snap gauge

- A double ended snap gauge as shown in Figure is having two ends in the form of anvils. The GO anvil is made to lower limit and NO GO anvil is made to upper limit of the shaft. This snap gauge is also known as solid snap gauges.

Figure 2.36 Double ended snap gauge

Snap gauge

- If both GO and NO GO anvils are form at the same end as shown in Figure then it is called progressive snap gauge. It is mainly used for checking large diameters up to 100 mm.
- The GO anvil should be at the front and NO GO anvil at the rear. So, the diameter of the shaft is checked progressively by these two ends. This type of gauge is made of horse shoe shaped frame with I section to reduce the weight of the snap gauges. This snap gauge is also called caliper gauge.

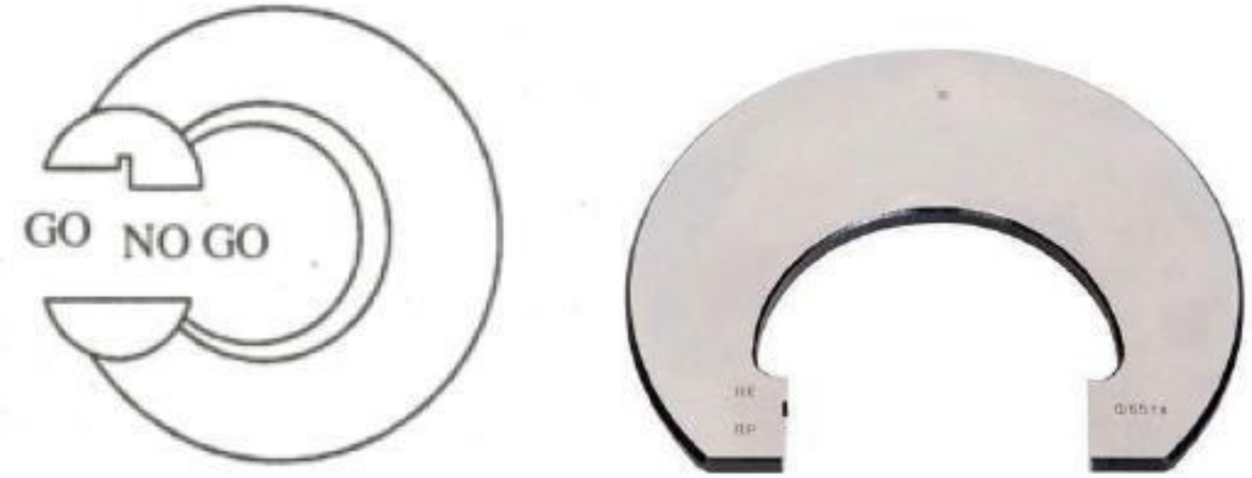


Figure 2.37 Progressive snap gauge

Snap gauge

- An adjustable snap gauges as shown in Figure consists of one fixed anvil and two small adjustable anvils. The distance between the two anvils is adjusted by adjusting the adjustable anvils by means of set screws. This adjustment can be made with the help of slip gauges for specified limits of size. They are used for checking large size shafts.

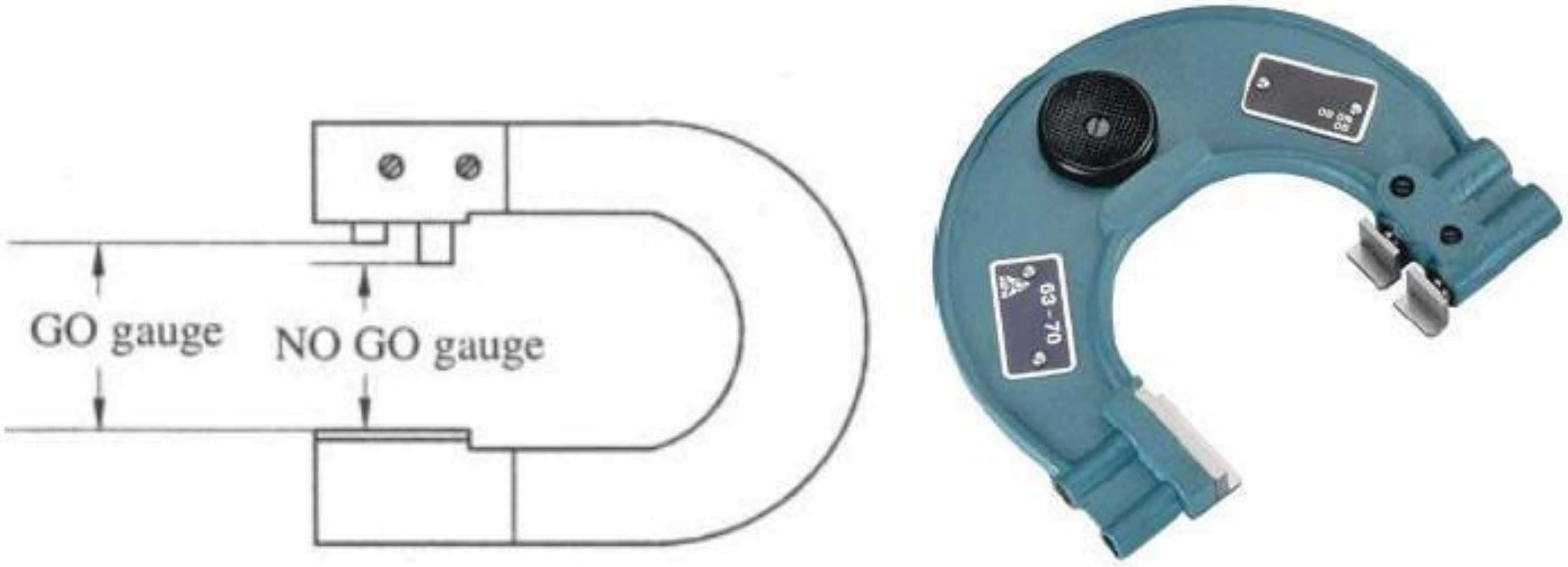


Figure 2.38 Adjustable snap gauges

Thread gauge

Thread gauges are used to check the pitch diameter of the thread

For checking internal threads (nut, bushes, etc.),plug thread gauges are used, while for checking external threads (screws, bolts, etc.), ring thread gauges are used.

Single-piece thread gauges serve for measuring small diameters.

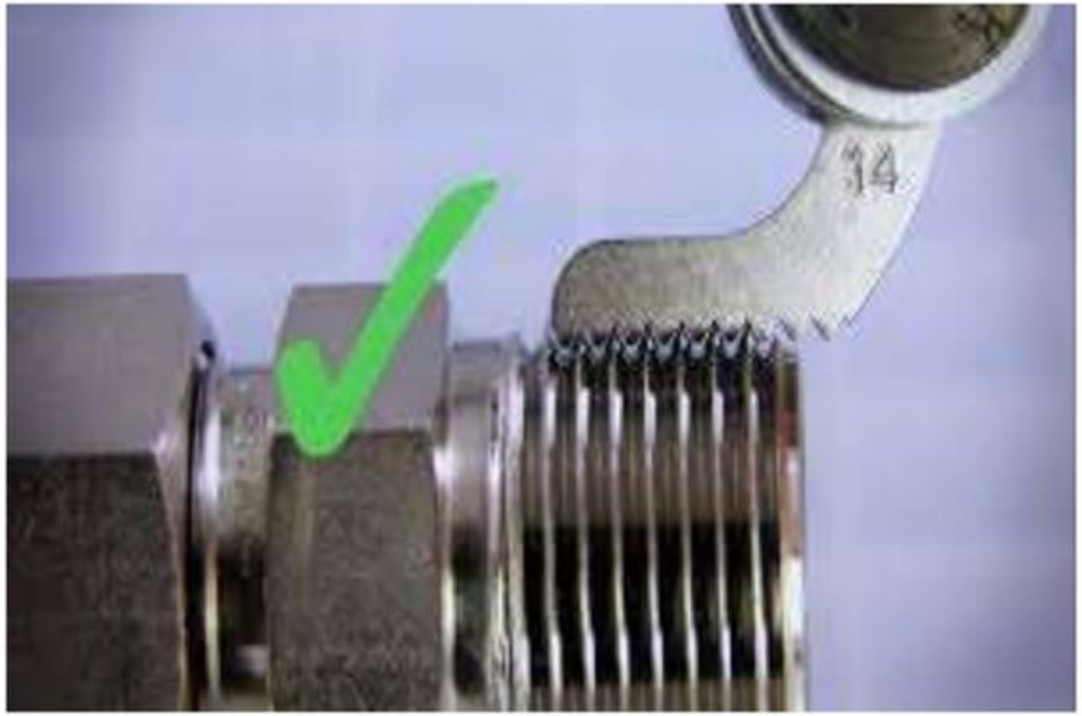
For large diameters the gauges are made with removable plugs machined with a tang.

Threaded

THREAD GAUGE

SCREW PITCH GAUGE

- Screw pitch gauges are used in picking out a required screw and for checking the pitch of the screw threads.



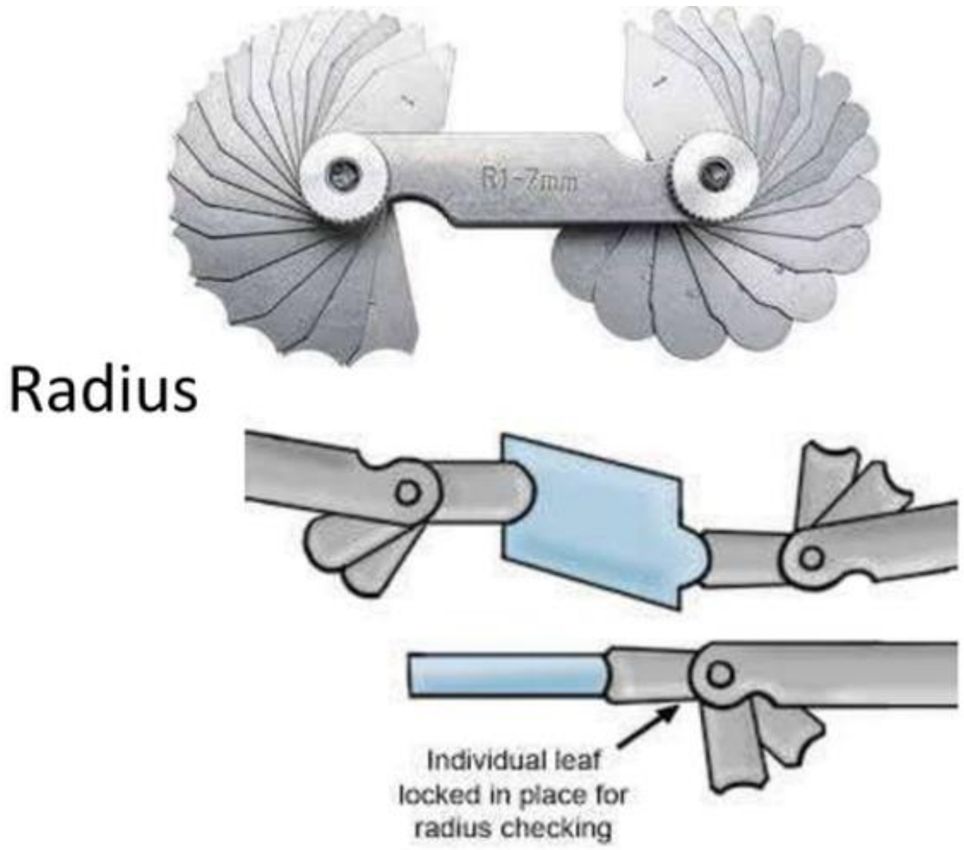
- They consist of a number of flat blades which are cut out to a given pitch and pivoted in a holder.
- Each blade is stamped with the pitch or number of thread per inch and the holder bears an identifying number designing the thread it is intended for.

RADIUS AND FILLET GAUGE

- The function of these gauges is to check the radius of curvature of convex and concave surfaces over a range from 1 to 25 mm.

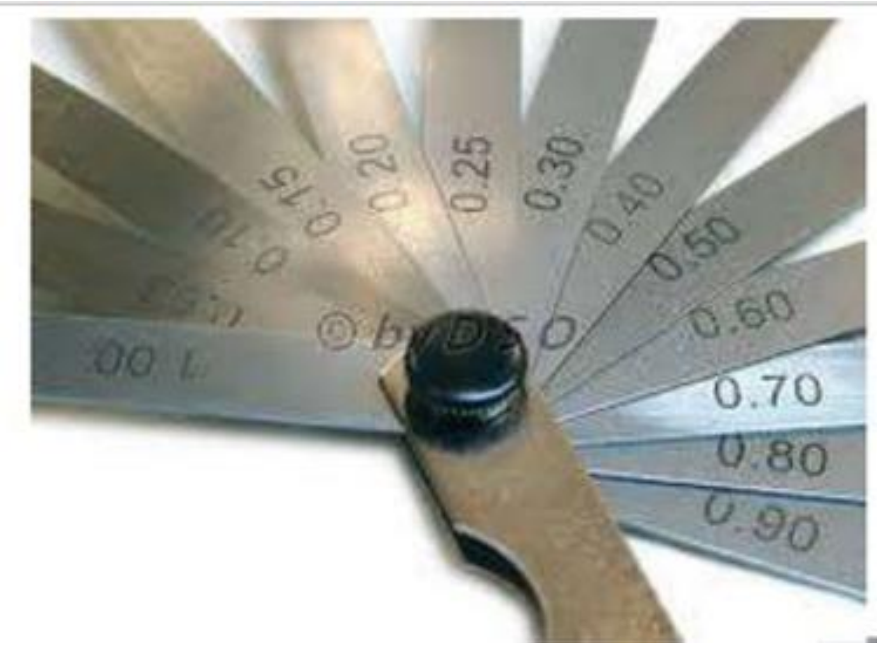
- The gauges are made in sets of thin plates curved to different radius at the ends.

- Each set consists of 16 convex and 16 concave blades.



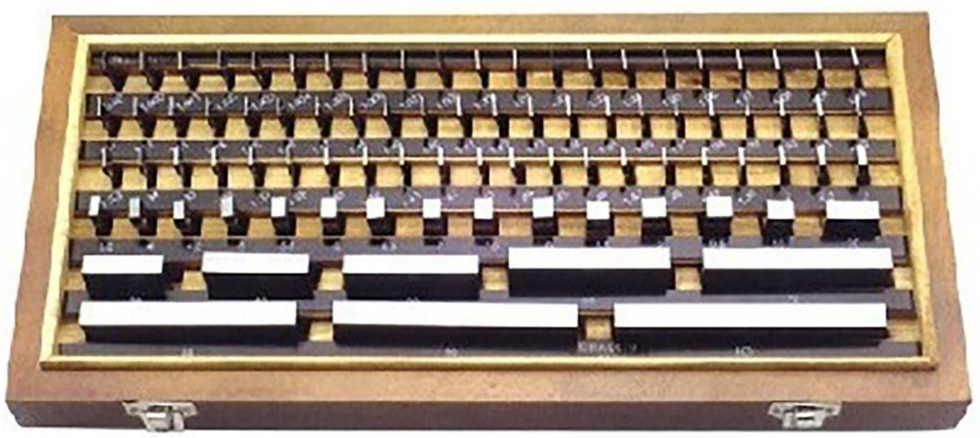
FEELER GAUGE

- Feeler gauges are used for checking clearances between mating surfaces.
- They are made in form of a set of steel, precision machined blade 0.03 to 1.0 mm thick and 100mm long.
- Each blade has an indication of its thickness.
- To find the size of the clearance, one or two blades are inserted and tried for a fit between the contacting surfaces until blades of suitable thickness are found.

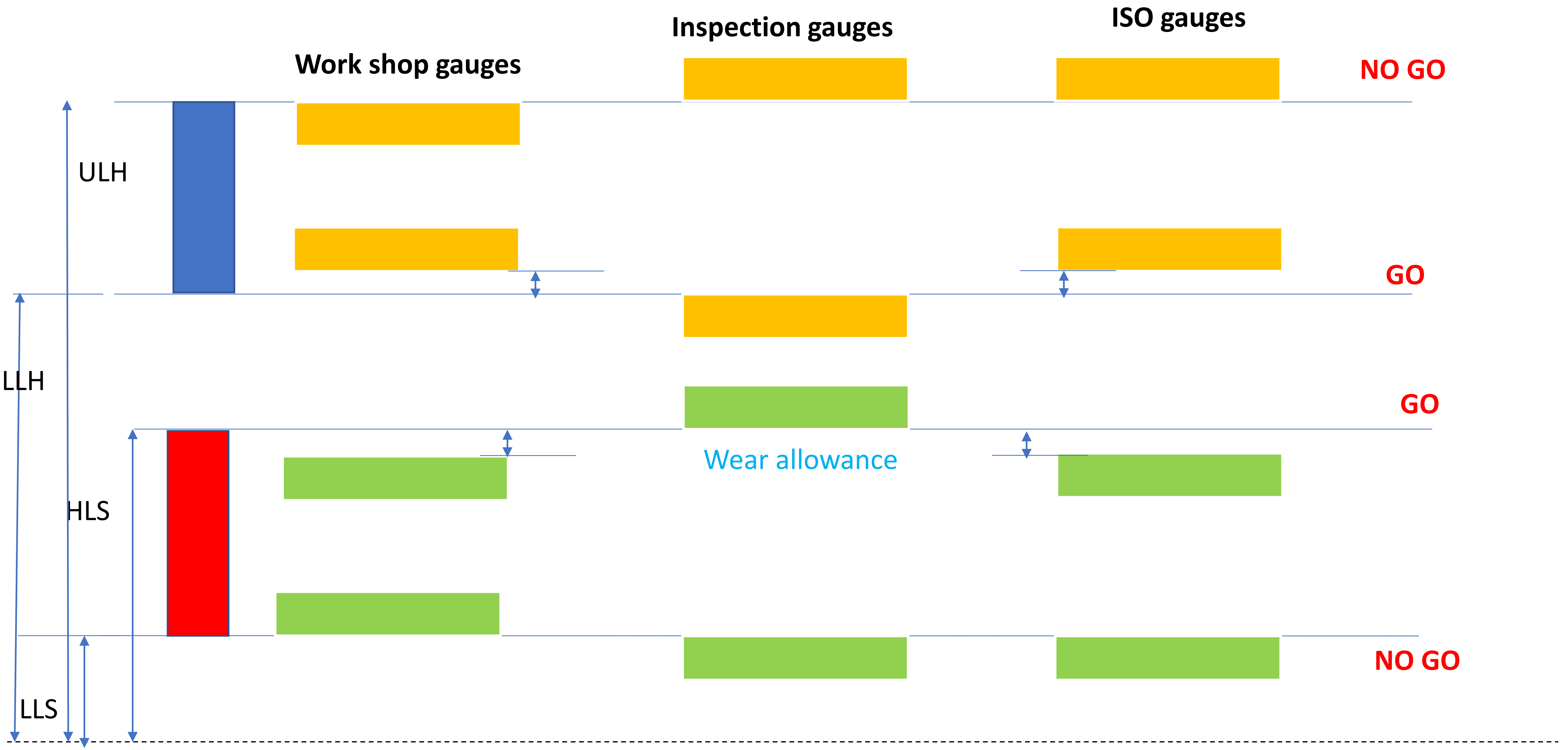


Slip Gauges or Gauge blocks

- Slip gauges are used in the manufacturing shops as length
- They are not to be used for regular and continuous measurement.
- Slip gauges are rectangular blocks with thickness representing the dimension of the block.
- The cross-section of the block is usually 32 mm x 9 mm.
- These are small blocks of alloy steel.
- The slip gauges are hardened and finished to size.
- The measuring surfaces of the gauge blocks are finished to a very high degree of finish, flatness and accuracy.



wear allowance on gauges



Allocation of Tolerance

Wear Allowance

- The surface of “Go” gauge is constantly rub against the surface of the part in inspection and loose their initial size.
- The size of plug gauge is reduced but size of snap gauge is increased.
- 10% wear allowance is provided only for the “GO "gauge if working tolerance is greater than 0.09 mm.