UNIT-III

LIMITS , FITS & TOLERANCE

INTRODUCTION

- No two things in nature can be identical, but they may found to be closely similar.
- Every production process is a combination of these elements i,e MAN, MATERIAL & MACHINE.
- A change in any one of these will constitute a change in the process.

- All these three elements are subjected to inherent and characteristics variations.
- These variables result in the variation of size of the components.
- It is the impossible to produce a part to be an exact size and some allowance known as TOLERANCE has to be allowed.
- The tolerance allowed also depends on the functional requirements which can’t be sacrified.

- Generally in engineering, any component manufactured is required to fit or match with some other component.
- The correct and prologed functioning of the two components in match depends upon the correct size relationships between the two, i.e., the parts must fit with each other in a desired way

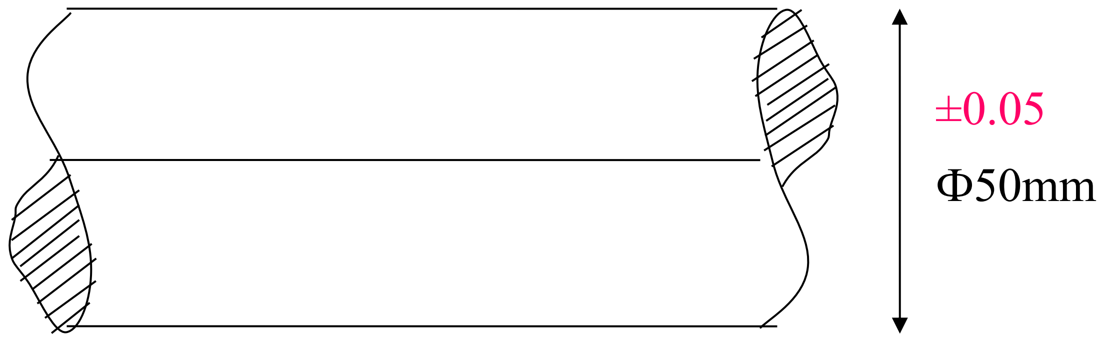
Eg.

If a shaft is to rotate in a hole, there must be enough clearance between the shaft and hole to allow the oil film to be maintained for lubrication. If the clearance is too small, excessive force would be required in rotation of shaft. If clearance is too wide, there would be vibrations and rapid wear and ultimate failure. Thus desired clearance to meet requirement has to be provided.

TOLERANCES

- Tolerance can be defined as the magnitude of permissible variation of a dimension or other measured or control criterion from the specified value.
- Tolerances have to be allowed because of the inevitable human failings and machine limitations which prevent ideal achievements during fabrication.
- In order to maintain economic production and facilitate the assembly of components it is necessary to allow a limited deviation from the designed size.

Example



Consider the dimensioning as shown in the figure.

When making the parts, a diameter of 50.00mm has to be acheived.
This dimension is called the BASIC or NOMINAL Diameter

The shaft will be satisfactory if its diameter lies between

50.00 + 0.05 = **50.05mm** and

50.0 - 0.05mm = **49.95mm**

The dimension 50.05mm is called the **UPPER LIMIT** and the dimension 49.95mm is called the **LOWER LIMIT**

The differnce between the upper limit and the lower limit is called **TOLERANCE**

In this example, 50.05 - 49.95 = **0.10mm** is the tolerance

Why tolerances are specified ?

(i) Variations in the properties of the material being machined introduce errors.

(ii) The production machines themselves have some inherent inaccuracies built into them and have the limitations to produce perfect parts.

(iii) It is impossible for an operator to make perfect settings. In setting up the machine, i.e. in adjusting the tools and workpiece on the machine, some errors are likely to creep in.

- An attempt to entirely overcome these factors with a view to obtain ideal conditions would result into enormous costs.
- The parts should, therefore, be made as inaccurate as tolerable to satisfy the functional requirements.
- Thus tolerances are specified to the dimensions of all manufactured parts and these should be just enough to do the intended job and no better.
- The tolerance is a compromise between accuracy required for proper functioning and the ability to economically produce this accuracy.

BASIC DIMENSION

- A basic dimension is the dimension, as worked out by purely design considerations.

- Since the ideal conditions of producing basic dimension, do not exist, the basic dimension can be treated as theoretical or nominal size, and it has only to be approximated.

- It is a general practice to specify a basic dimension and then indicate by tolerances as to how much variation in the basic dimension can be tolerated without affecting the functioning of the assembly into which this part will be used.

TOLERANCED DIMENSIONS

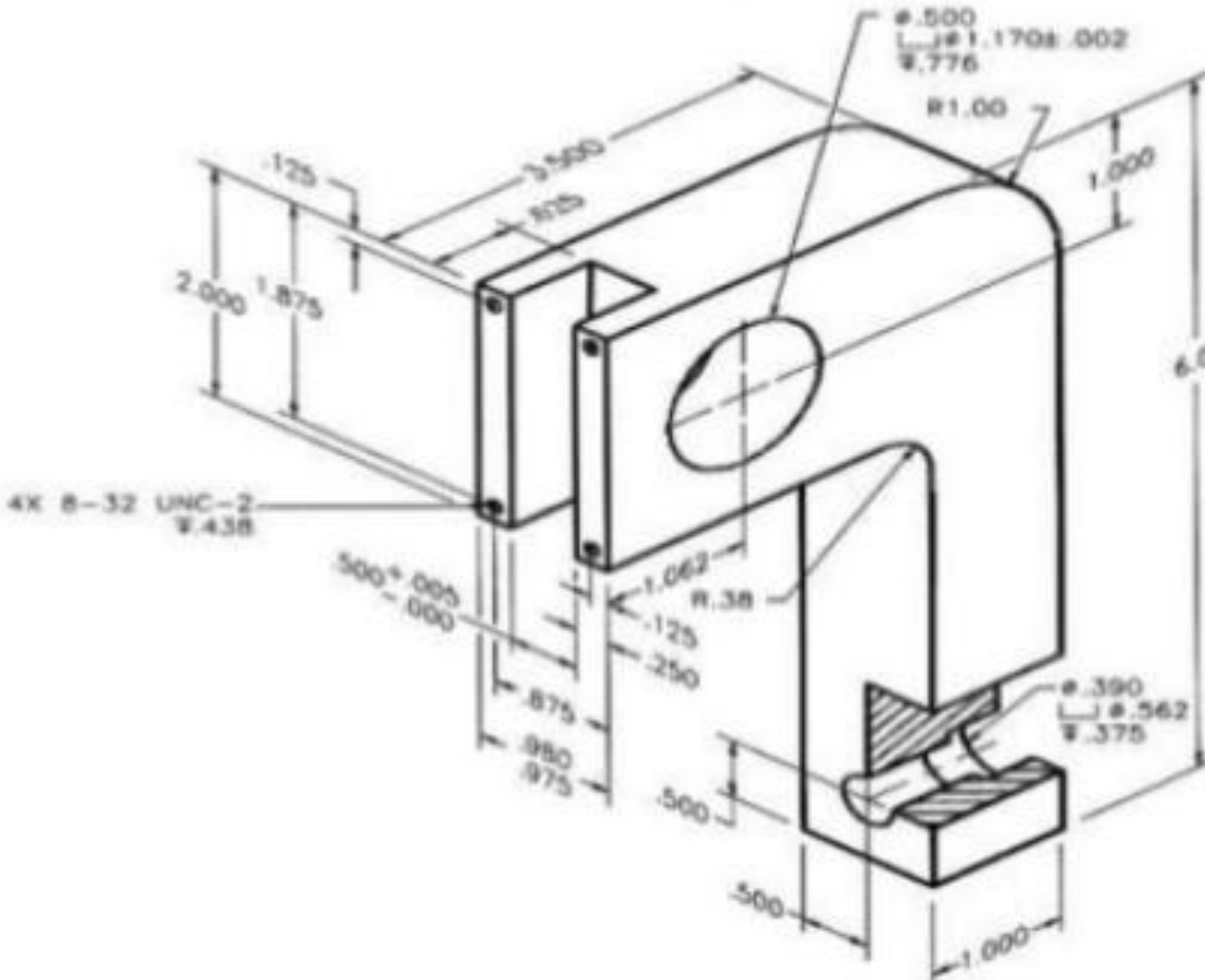
- Tolerances are basically specified in two forms, i.e. unilateral and bilateral. In unilateral tolerances, the total tolerance as related to a basic dimension is in one direction only.

- This form of tolerance is usually indicated when the machining of mating parts is called for, as this greatly assists the operator.

- The operator machines to the upper limit of shaft (lower limit for a hole) knowing fully well that he still has the whole tolerance left for machining before the parts are rejected.

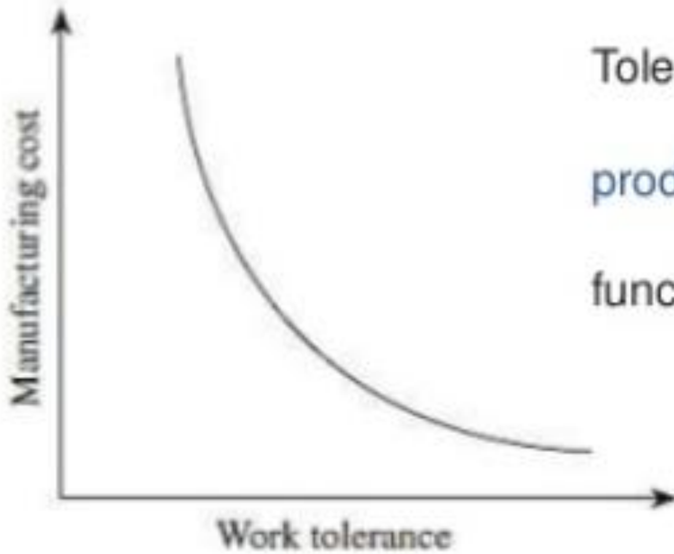
- Bilateral tolerances, the total tolerance is specified on both sides (plus and minus) of the basic dimension.
- Bilateral tolerances usually have plus and minus tolerance of equal amount, but not necessarily always.

Tolerances



⌀.500
⌴⌀1.170±.002
↧.776
R1.00
3.500
.125
.625
1.000
2.000
1.875
6.00
4X 8-32 UNC-2
↧.438
.500 +.005 -.000
1.062
R.38
.125
.250
.875
.980
.975
⌀.390
⌴⌀.562
↧.375
.500
.500
1.000

Manufacturing Cost and Work Tolerance



Tolerance is a trade-off between the economical production and the accuracy required for proper functioning of the product.

Relationship between work tolerance and manufacturing cost

UNILATERAL TOLERANCE

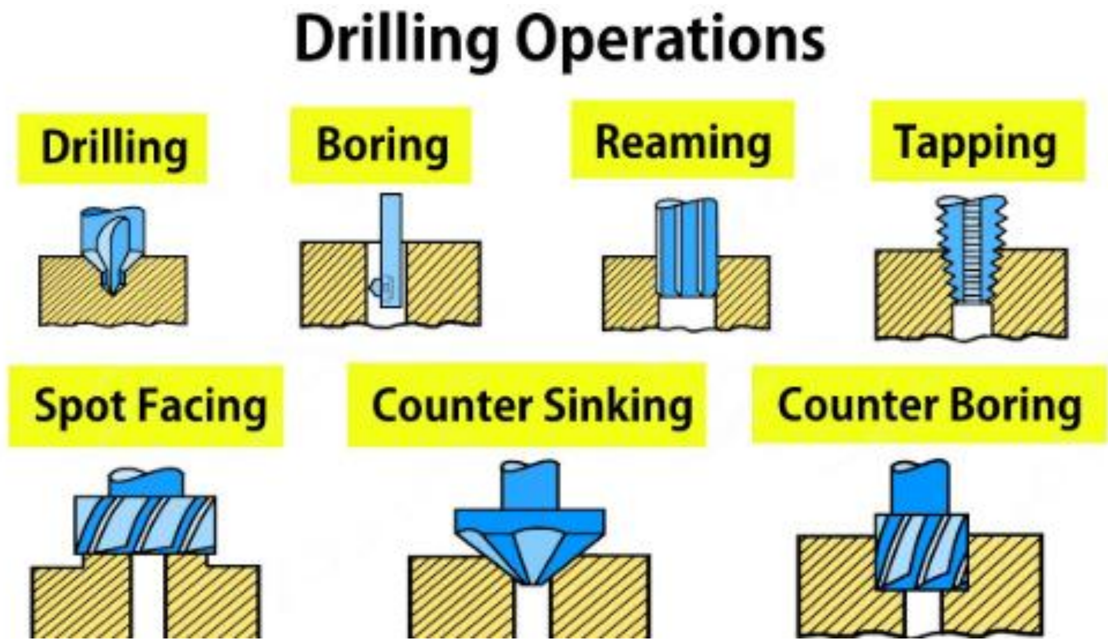
A **unilateral tolerance** is one where the dimension is allowed to vary **only in one direction** from the basic size — either **positive** or **negative**, but not both.

Form

- Basic size + tolerance (only positive)
  e.g., 50 mm +0.02 / +0.00 - size can increase up to 0.02 mm but cannot decrease
- Basic size – tolerance (only negative)
  e.g., 50 mm 0.00 / –0.03 - size can decrease by 0.03 mm but cannot increase

UNILATERAL TOLERANCE

- In the case of unilateral tolerances, the dimension is allowed to vary only in one direction.
- This system is used in machining processes like drilling in which case the dimensions are most likely to deviate in one direction only (in drilling, hole is always of over size rather than undersize).



BILATERAL TOLERANCE

A tolerance in which the size is allowed to vary **on both sides** of the nominal dimension (±).

Examples

- 50 ± 0.1
- 80 +0.2 / –0.1 (unequal bilateral)

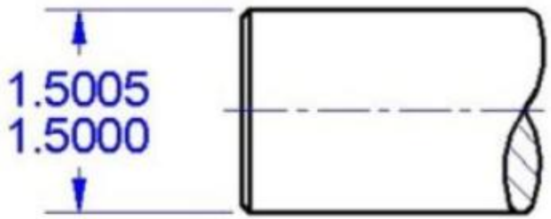
Example

A plate thickness can be: **10 ± 0.2 mm -** Both 9.8 mm and 10.2 mm will still function.

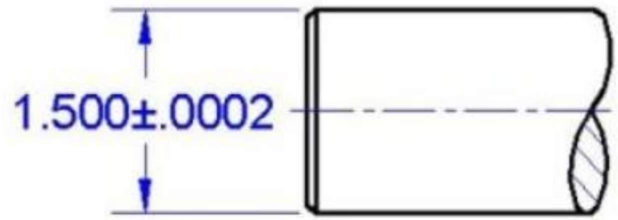
A bracket width: **50 ± 0.05 mm -** Either side is OK as long as it stays within the limit.

BILATERAL TOLERANCE

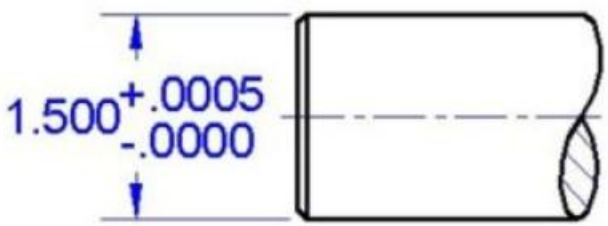
Specification of Tolerances



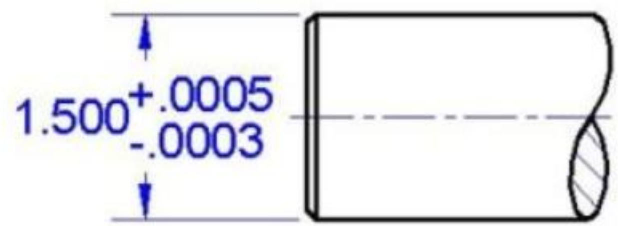
Limit Dimension



Bilateral-Equal

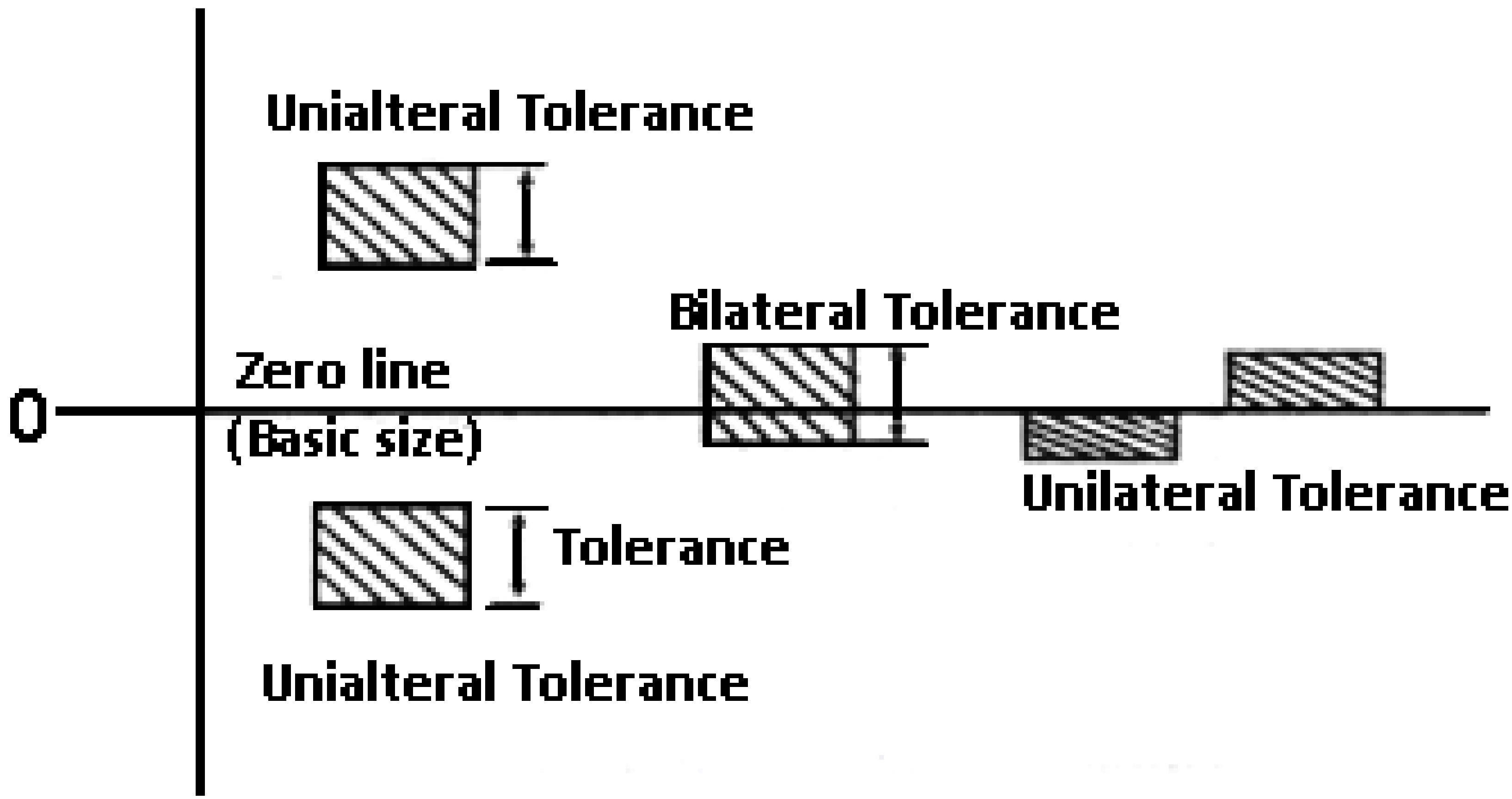


Unilateral



Bilateral-Unequal

Schematic representation of unilateral and bilateral tolerance



COMPOUND TOLERANCE

- A **compound tolerance** is the *resultant* or *combined* tolerance when two or more dimensions with their own tolerances are added, subtracted, or chained together in an assembly or mechanism.

- In simple terms: When multiple tolerances stack up to affect a final dimension → that final allowed variation is called compound tolerance.

Assembly Length Example

A machine assembly consists of:

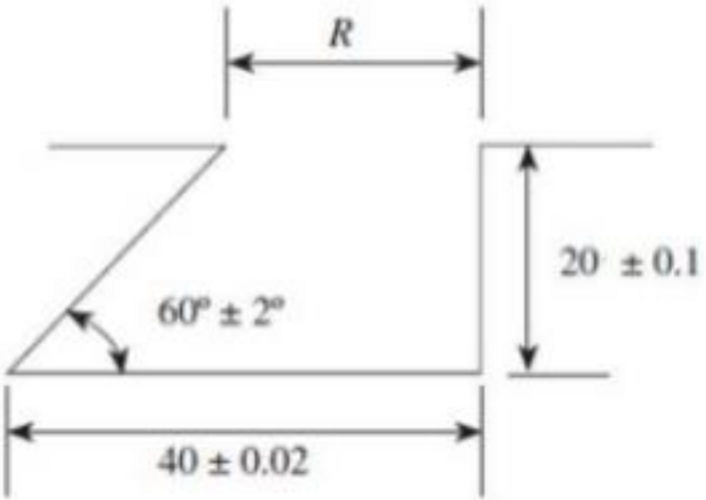
- Part A = 40 ± 0.1 mm
- Part B = 30 ± 0.05 mm
- Part C = 20 ± 0.1 mm

Total length = A + B + C

Tolerance = sum of tolerances

Compound tolerance =
0.1 + 0.05 + 0.1 = **±0.25 mm**
So total length = **90 ± 0.25 mm**

COMPOUND TOLERANCE

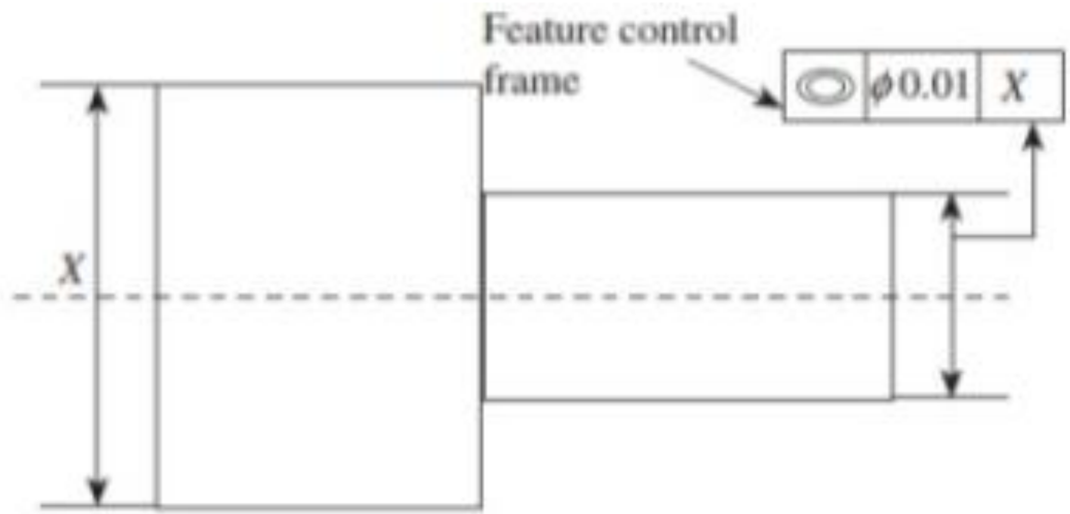


Tolerance for the dimension *R* is determined by the combined effects of tolerance on 40 mm dimension, on 60°, and on 20 mm dimension

GEOMETRIC TOLERANCE

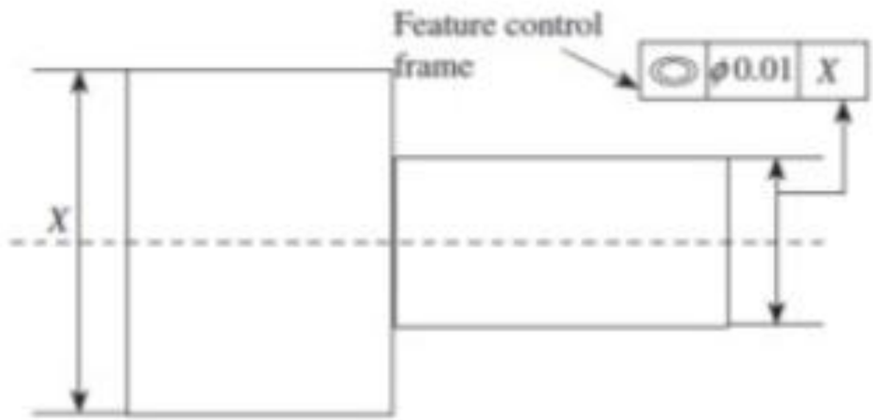
- **Geometric tolerance** controls the *shape*, *orientation*, *location*, and *runout* of features on a component.
- It specifies **how much a feature can deviate from its ideal geometry**.
- In simple terms:
  - Size tolerance = controls dimensions
  - Geometric tolerance = controls shape & accuracy of geometry
- Geometric tolerances follow the **GD&T system** (Geometric Dimensioning and Tolerancing).

4. Geometric tolerance



- Diameters of the cylinders need be concentric with each other.
- For proper fit between the two cylinders, both the centres to be in line.
- This information is represented in the feature control frame.
- Feature control frame comprises three boxes.

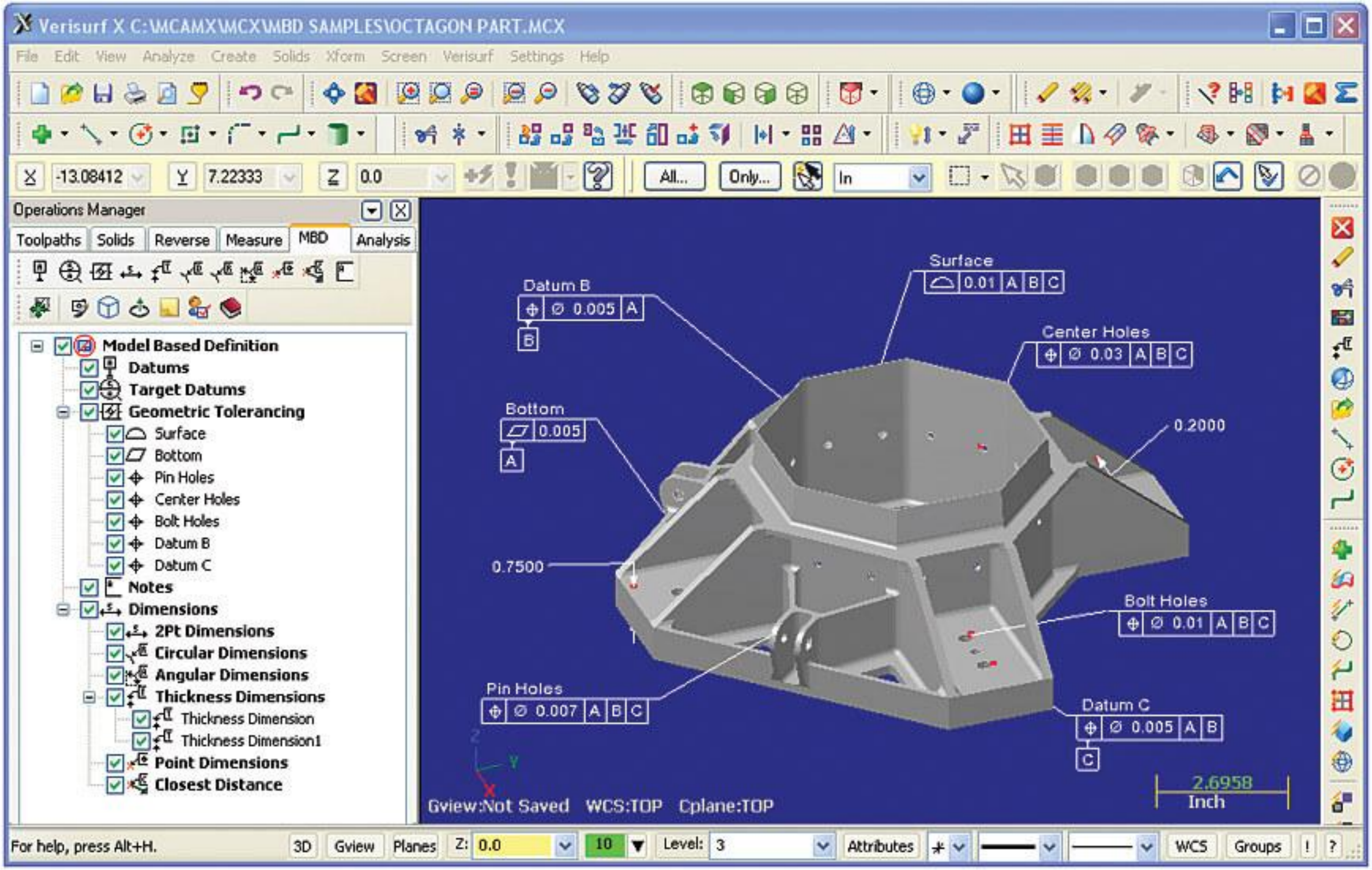
4. Geometric tolerance



- **First box:** On the left indicates the feature to be controlled, represented symbolically (example: concentricity).
- **Centre box:** indicates distance between the two cylinders, centres cannot be apart by more than 0.01 mm (Tolerance).
- **Third box:** Indicates that the datum is with X.

Geometric Category	Tolerance Type	Description	Symbol	ASME Section
For Individual Features	**Form**	**Straightness**	⏤	**6.4.1**
		Flatness	⏥	**6.4.2**
		Circularity	○	**6.4.3**
Indvidual or Related Features	**Profile**	**Line Profile**	⌒	**6.5.2(b)**
		Surface Profile	⌓	**6.5.2(a)**
For Related Features	**Orientation**	**Angularity**	∠	**6.6.2**
		Perpendicularity	⊥	**6.6.4**
		Parallelism	//	**6.6.3**
	Location	**Position**	⌖	**5.2**
		Concentricity	◎	**5.11.3**
		Symmetry	⌯	**5.13**
	Runout	**Circular Runout ***	↗	**6.7.1.2.1**
		Total Runout *	⌰	**6.7.1.2.2**

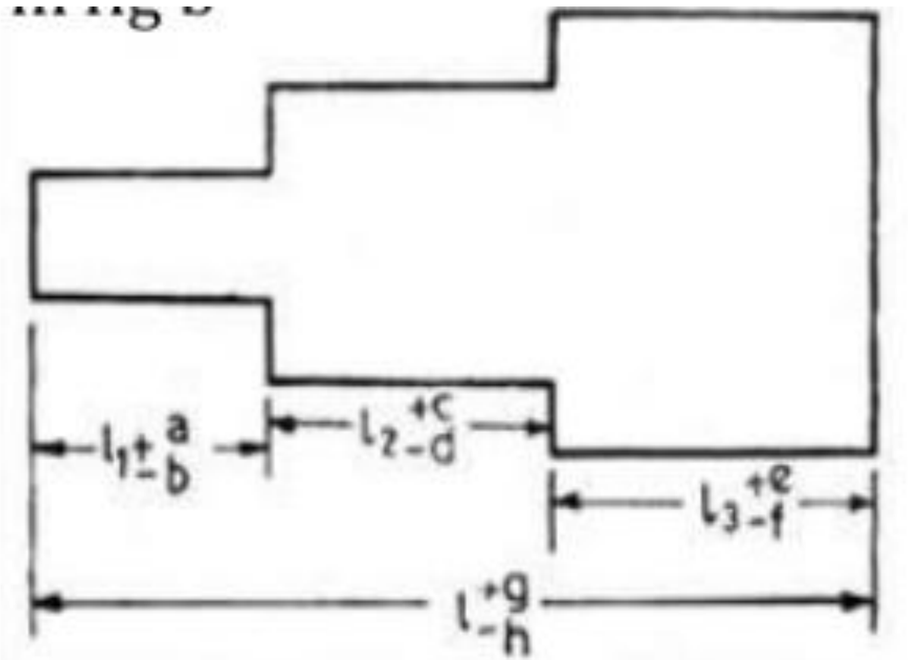
* Arrows may be filled or not filled



Verisurf X C:\MCAMX\MCX\MBD SAMPLES\OCTAGON PART.MCX
File Edit View Analyze Create Solids Xform Screen Verisurf Settings Help
Operations Manager
Toolpaths Solids Reverse Measure MBD Analysis
Model Based Definition
Datums
Target Datums
Geometric Tolerancing
Surface
Bottom
Pin Holes
Center Holes
Bolt Holes
Datum B
Datum C
Notes
Dimensions
2Pt Dimensions
Circular Dimensions
Angular Dimensions
Thickness Dimensions
Thickness Dimension
Thickness Dimension1
Point Dimensions
Closest Distance
Surface
0.01 A B C
Datum B
Ø 0.005 A
B
Center Holes
Ø 0.03 A B C
Bottom
0.005
A
0.2000
0.7500
Bolt Holes
Ø 0.01 A B C
Pin Holes
Ø 0.007 A B C
Datum C
Ø 0.005 A B
C
2.6958
Inch
Gview:Not Saved WCS:TOP Cplane:TOP
For help, press Alt+H.

Tolerance Accumulation.

If a part comprises of several steps, each step having some tolerance over its length, then overall tolerance on complete length will be sum of the tolerances on individual length.



In Fig,

g = a+c+e
h - b+d+f

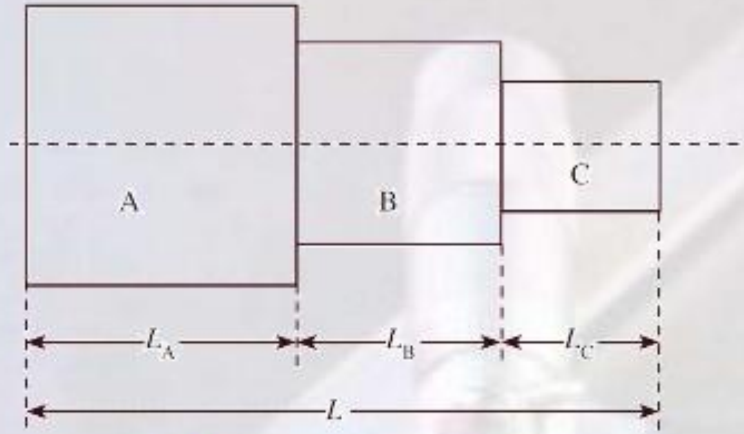


Fig. 3.6 Accumulation of tolerances

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

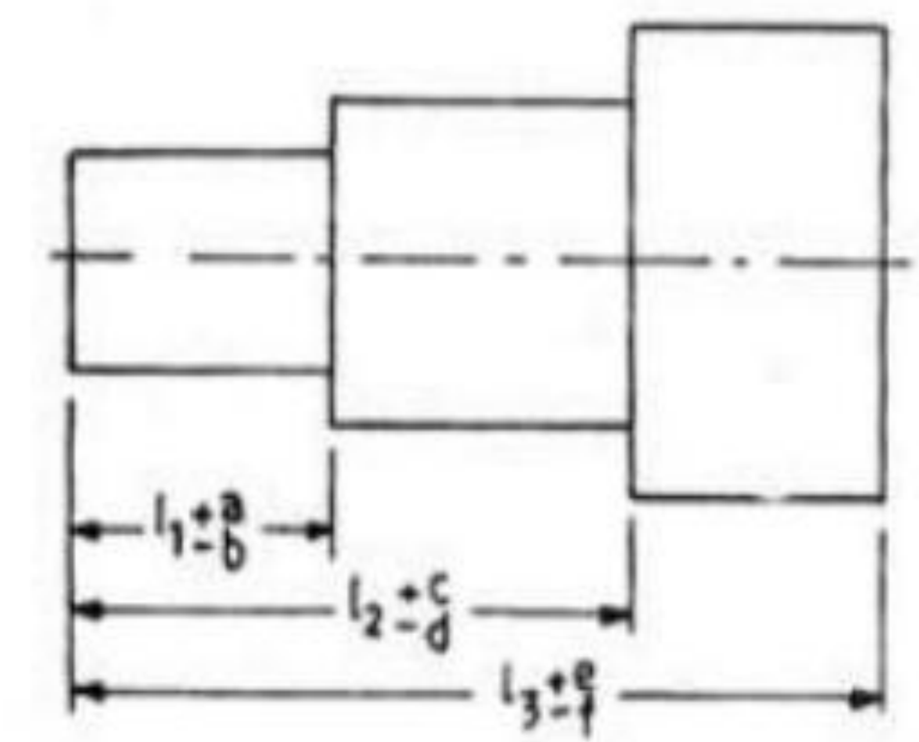
$L = L_A + L_B + L_C$
$L = 30 + 20 + 10 = 60$ mm

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.

The effect of accumulation of tolerances can be minimised by adopting **progressive dimensioning** from a datum



Interchangeability

Definition

Interchangeability is the ability of **manufactured parts to be replaced by identical parts** without any custom fitting, adjustment, or modification.

In simple words:
Any part should fit correctly into any assembly of the same type.

Why Interchangeability Is Important

1. **Mass Production**
   Components made at different times/places must fit together.
2. **Ease of Assembly**
   Saves time—no filing, grinding, or adjustments needed.
3. **Maintenance & Replacement**
   A damaged part can be replaced instantly with a standard spare.
4. **Cost Reduction**
   Uniform parts reduce machining, inspection, and repair costs.
5. **Quality Assurance**
   Ensures consistent performance of products.

How Interchangeability is Achieved

1. **Standardization**

   Using standard dimensions, fits, gauges, threads, tolerances (ISO, BIS standards).

2. **Limits, Fits, and Tolerances**

   Proper tolerance specification ensures every part stays within acceptable size variation.

3. **Selective Assembly (when needed)**

   Parts are grouped by size and matched.

4. **Precision manufacturing**

   CNC, jigs/fixtures, accurate tools ensure consistency.

Types of Interchangeability

1. Universal Interchangeability

- Parts produced by **different manufacturers**, at **different locations**, or under **different conditions**, can still work together.

Example:

- Nuts made in one factory fitting bolts made in another factory.
- Bearings from SKF fitting shafts from any machine manufacturer.

Requirement:

- Strict adherence to **international standards (ISO, BIS, ANSI, ASME, DIN etc.)**
- Because only then will **dimensions, tolerances, and fits** match exactly.

2. Local Interchangeability

- All mating parts are produced **within the same manufacturing plant.**
- Internal (local) standards may be used.

Example:

- Parts for a gearbox produced entirely within a single factory.

Limitation:

- If spare parts are needed later from another manufacturer, local standards must still be traceable to international standards.

Real-Life Examples:

1. **AA / AAA Batteries**
   Any battery from **Duracell, Eveready, Panasonic** will fit any torch, TV remote, or toy. - All follow **international size standards**.

2. **Light Bulbs (E27 / B22 Holders)**
   Bulbs from Philips, Wipro, Syska, or local brands fit the same holder.
   Because the thread size and cap dimensions are standardized.

3. **USB Cables & Chargers**
   A USB-C cable from any manufacturer fits any phone/laptop with USB-C - Achieved through **USB-IF international standards.**

4. **SIM Cards**
   Any SIM card fits any mobile phone slot (Nano-SIM, Micro-SIM) - Follows **GSM standardization.**

5. **Domestic LPG Cylinders**
   Any standard LPG regulator fits any LPG cylinder valve - Follows **BIS standards.**

- Example we have **100 parts** each with a hole and **100 shafts** which have to fit into these holes.
- If we have interchangeability then we can make any one of the **100 shaft** & fit it into any hole & be sure that the required fit can be obtained.

- **Any M6 bolt will fit to any M6 nut randomly selected.**

The advantages of interchangeability

1. The assembly of mating parts is easier. Since any component picked up from its lot will assemble with any other mating part from another lot without additional fitting and machining.

2. It enhances the production rate.

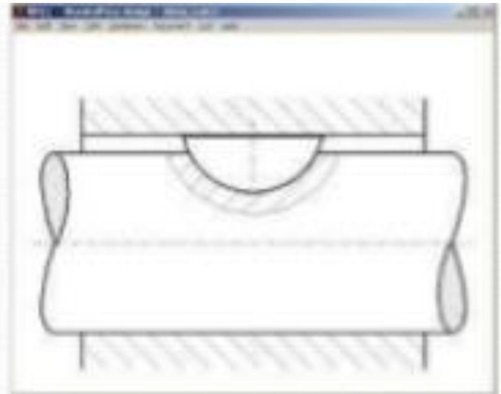
3. It brings down the assembling cost drastically.

4. Repairing of existing machines or products is simplified because component parts can be easily replaced.
5. Replacement of worn out parts is easy.

6. Without interchangeability mass production is not possible.

Examples

- **Keys**
- **Couplings**
- **Pin Joints**
- **Screwed Fasteners**
- **Gears**
- **Clutches**




- The required fit in an assembly can be obtained in two ways, namely

  (i) universal or full interchangeability, and

  (ii) selective assembly.

- **Full interchangeability** means that any component will mate with any other mating component without classifying manufactured components in subgroup or without carrying out any minor alterations for mating purposes.

Selective Assembly

- Today the consumer not only wants quality, precision and trouble-free products but also, he wants them at **attractive prices.**
- In selective assembly the components produced by a machine are classified into **several groups according to size.**
- Instead of making all parts the same, we **select** pairs that fit well together.

- If some parts (shaft and holes) to be assembled are manufactured to normal tolerances of 0.01 mm , an automatic gauge can segregate them into ten different groups with a 0.001 mm limit for selective assembly of the individual parts.

- In selective assembly, if the components are divided into different categories, then the groups will be organized according to the sizes and dimensions. We have to make sure that all the parts are grouped together and all are ready for mating so that every component will match with the corresponding sized component to form an assembled part.

- If a small shaft is mated with small holes, then large holes will be mated with large shafts.

- for example if a part at its low limit is assembled with the mating part a high limit, the fit so obtained may not fully satisfy the functional requirements of the assembly.
- also machine capabilities are sometimes not compatible with the requirements of interchangeable assembly.

Imagine a company is producing bearings and shafts that must fit together with a tight tolerance to ensure smooth rotation without excessive friction.

Manufacturing Process:

- Bearings are produced with an inner diameter (ID) tolerance of 10.00 mm ± 0.02 mm.
- Shafts are produced with an outer diameter (OD) tolerance of 10.00 mm ± 0.02 mm.

Measurement and Grouping:
After manufacturing, each bearing and shaft is measured.

- Bearings are grouped based on ID into three categories:

  Group 1: 9.98 mm – 9.99 mm
  Group 2: 10.00 mm – 10.01 mm
  Group 3: 10.02 mm – 10.03 mm

- Shafts are also grouped based on OD:

  Group A: 9.98 mm – 9.99 mm
  Group B: 10.00 mm – 10.01 mm
  Group C: 10.02 mm – 10.03 mm

Ball bearings must fit snugly into their inner race to avoid excess movement and ensure smooth rotation.

Manufacturing:

- Ball bearings are produced with an outer diameter tolerance of ±0.005 mm.
- Inner races are produced with an inner diameter tolerance of ±0.005 mm.

Grouping:

- Bearings are grouped as follows:
  - Group 1: 9.995 mm – 10.000 mm
  - Group 2: 10.001 mm – 10.005 mm
- Inner races are also grouped:
  - Group A: 10.001 mm – 10.005 mm
  - Group B: 9.995 mm – 10.000 mm

Pistons are measured and sorted into **five subgroups** based on their diameters:

Group 1: 49.96 mm – 49.97 mm
Group 2: 49.98 mm – 49.99 mm
Group 3: 50.00 mm – 50.01 mm
Group 4: 50.02 mm – 50.03 mm
Group 5: 50.04 mm – 50.05 mm

Cylinders are also measured and sorted into **five corresponding subgroups**:

Group A: 50.04 mm – 50.05 mm
Group B: 50.02 mm – 50.03 mm
Group C: 50.00 mm – 50.01 mm
Group D: 49.98 mm – 49.99 mm
Group E: 49.96 mm – 49.97 mm

Advantages

- There is a larger number of acceptable parts as original tolerances are greater
- This in turn allows the manufacture of cheaper parts as less will be consigned to the waste bin.
- Selective Assembly assures better and more accurate assembly of parts by insuring closer tolerances between the mating parts.
- Rise the quality and lower manufacturing costs by **avoiding tight tolerances**.
- Reduces the rejection rate (scrap rate)

- During usage of the assembly if one component fails, first we need manual of assembly and identify the group to which failure component belongs to and search the component in spare parts.
- By focusing on the fit between mating parts, rather than the absolute size of each component so there will small deviation in size of component.

Limits of Size

There are three considerations in deciding the limits necessary for a particular dimension:

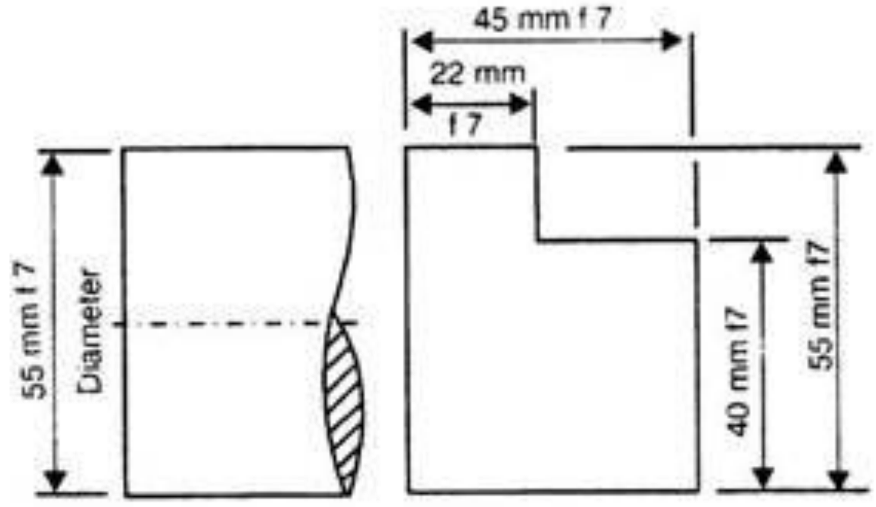
- **Functional requirement** (function of the component, what it is required to do),
- **Interchangeability** (ease of replacement in the event of failure), and
- **Economics** (minimisation of production time and cost).

Hence the degree of tolerance calls for the compromise.

Basic Definitions

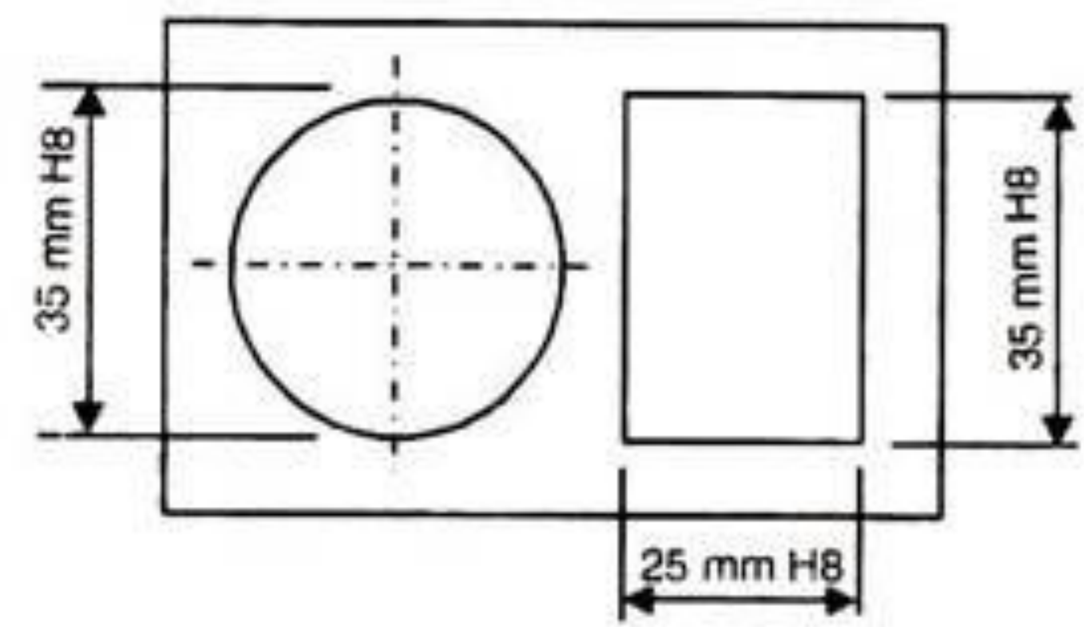
1. Shaft

The term shaft refers not only to the diameter of a circular shaft but to any external dimension on a component.



2. Hole

The term hole refers not only to the diameter of a circular hole but to any internal dimension of a component.



3. Size:

The term size refers to the numerical value of a linear dimension in a particular unit.

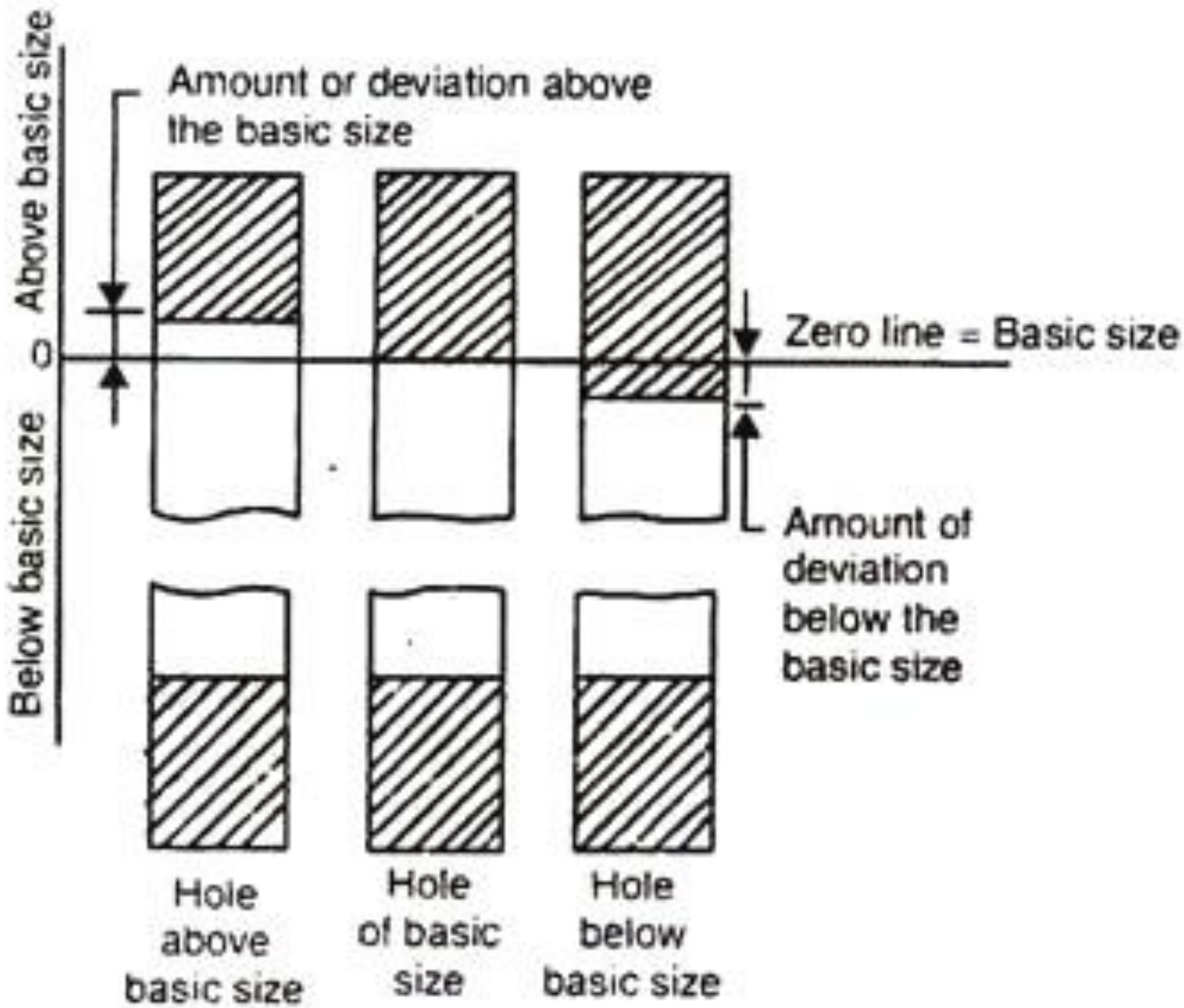
4. Basic Size:

The term basic size refers to the size from which the limits of size are derived by the application of tolerance (i.e. upper and lower deviation). The basic size or nominal size of a part is often the same and it is termed as zero line.

For example, if a hole and shaft are intended to mate with a basic size of 50 mm, then the dimensions and tolerances for both parts will be centered around 50 mm.

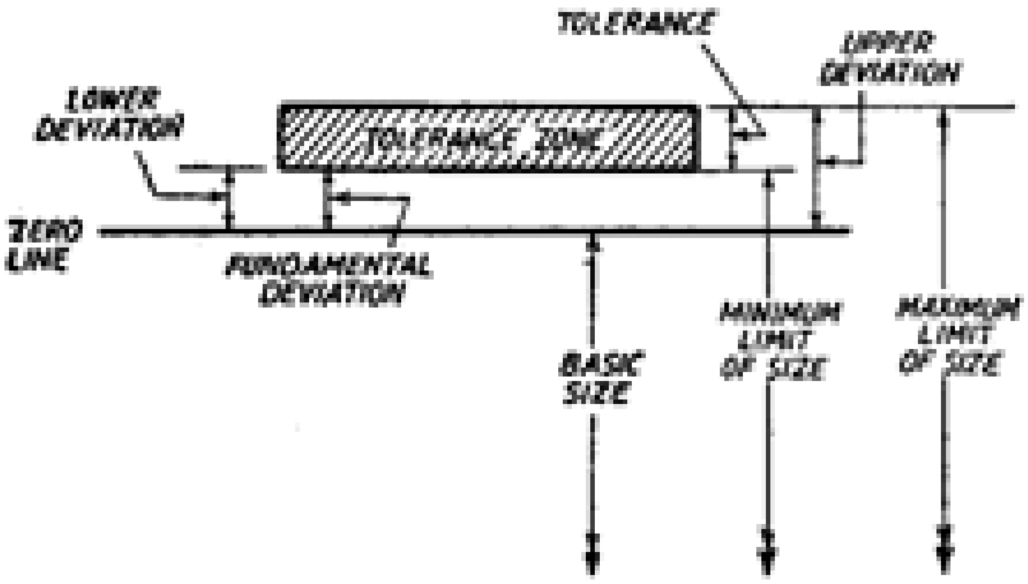
6. Actual Size:

- The term actual size referred to the actual measured dimension of a part.
- The difference between the basic size and the actual size should not exceed a certain limit, if so; it will disturb the interchangeability of assembly parts.
- It may differ slightly from the basic size due to manufacturing variations, but it must fall within the specified tolerance limits.


Above basic size
Amount or deviation above
the basic size
0
Zero line = Basic size
Below basic size
Amount of
deviation
below the
basic size
Hole
above
basic size
Hole
of basic
size
Hole
below
basic size

7. Zero Line:

It is a straight line corresponding to the basic size. The deviations are measured from this line. The positive and negative deviations are shown above and below the zero line respectively.



Limits of Size:

It is referred to the two extreme permissible sizes for a dimension of a part, between which the actual size should lie including the maximum and minimum sizes of the component.

The largest permissible size for a dimension is called upper or high or **maximum limit**, whereas the smallest size is called lower or **minimum limit.**

Minimum Limit of Size

The term minimum limit of size referred to the minimum or smallest permissible size of a component

Maximum Limit of Size

The term maximum limit of size referred to the maximum or greatest permissible size of a component

For example, if a shaft is specified with a diameter of 50mm ± 0.05mm, the limits of size would be:
Upper Limit: 50.05mm
Lower Limit: 49.95mm

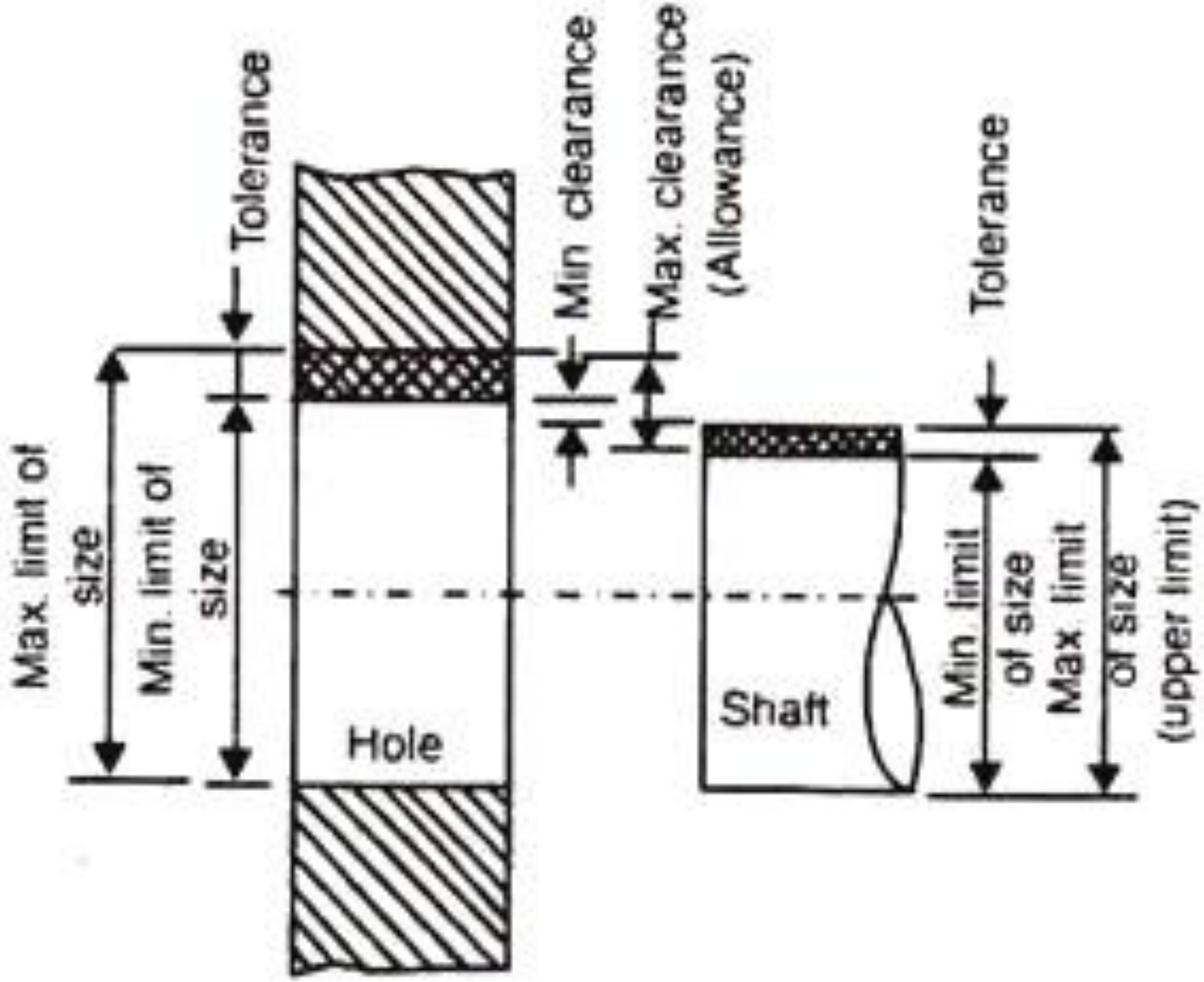
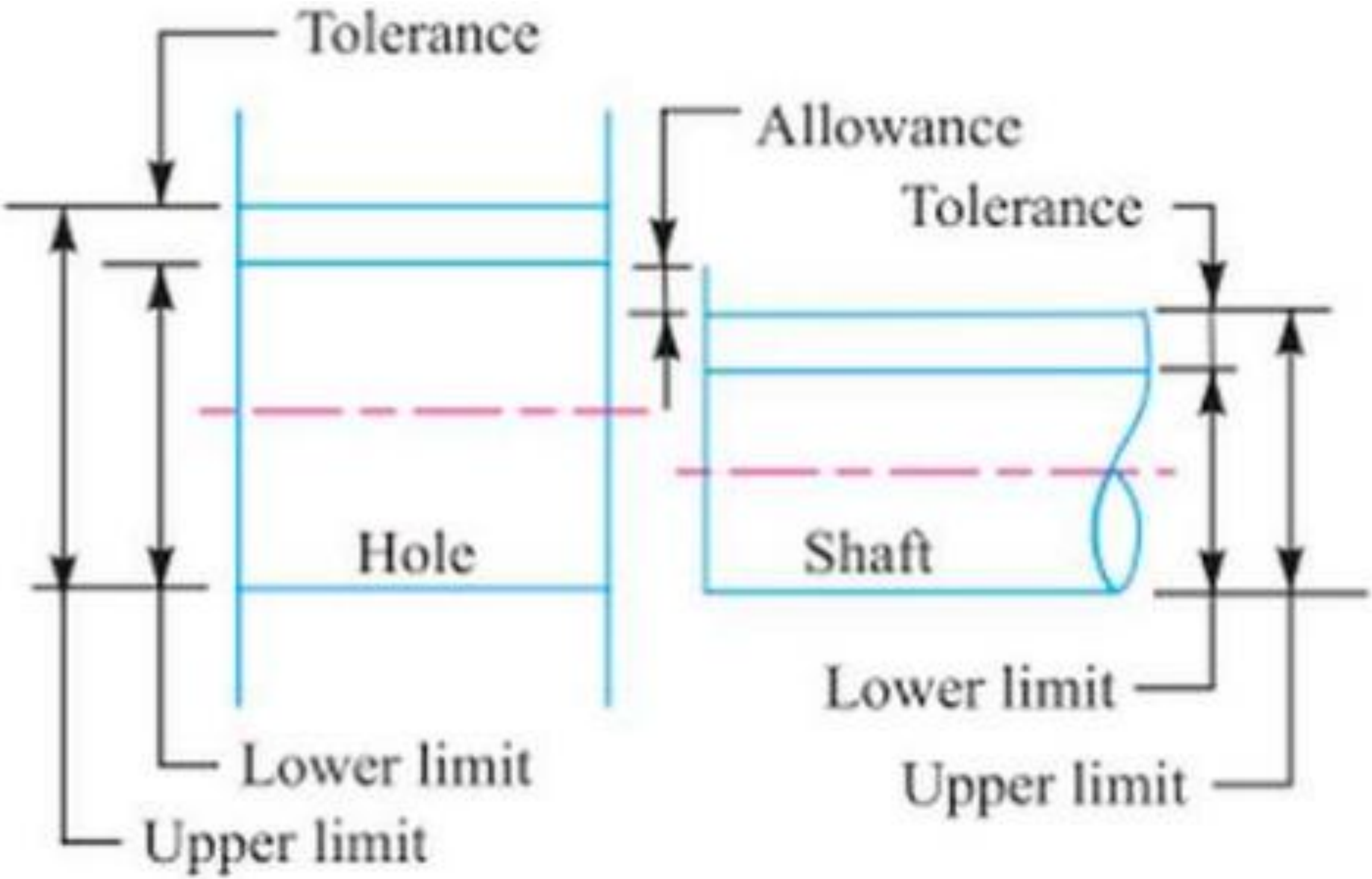


Fig. 1.52. Definition of Limits of size.

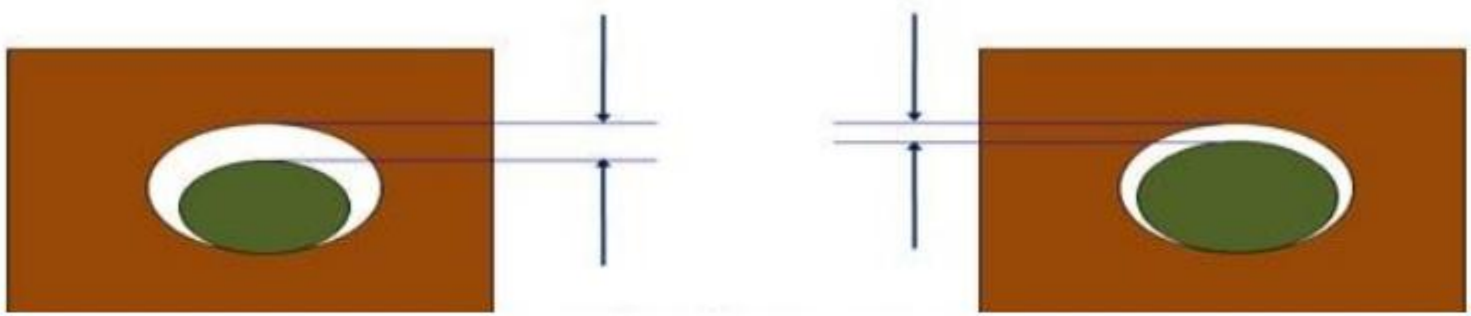
10. Allowance:

- The term allowance refers to the difference between the basic dimensions of mating parts.
- The allowance may be Positive or negative.
- In positive allowance the shaft size is less than the hole size, and in negative allowance the shaft size is greater than the hole size.


Tolerance
Allowance
Tolerance
Hole
Shaft
Lower limit
Lower limit
Upper limit
Upper limit

ALLOWANCE

Maximum Allowance **Minimum Allowance**



Clearance

The term clearance refers to the difference between the sizes of the hole and the shaft before assembly. Clearance must be positive.

Minimum clearance

The difference between the minimum size of hole and the maximum size of the shaft.

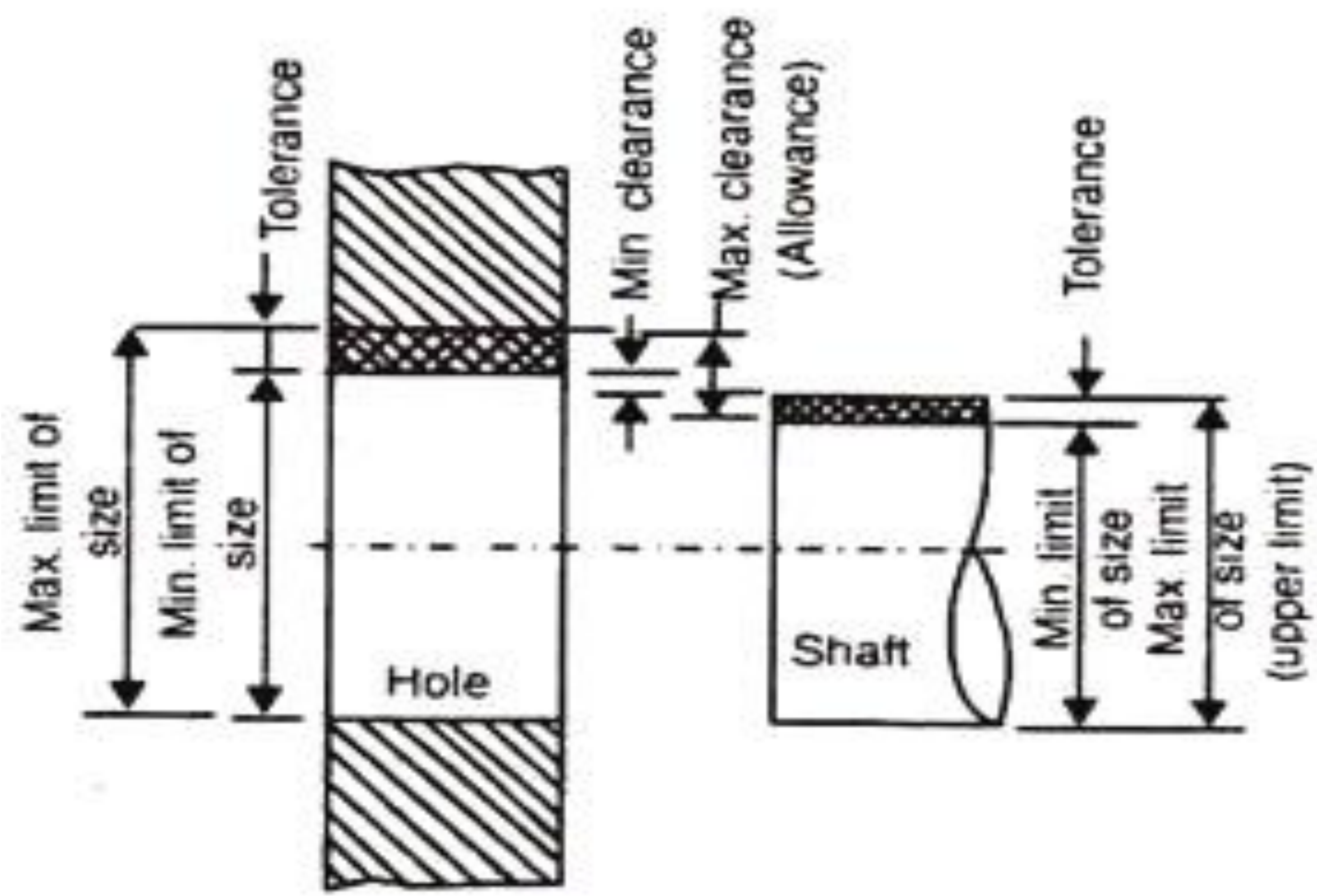
Maximum clearance

The difference bewteen the maximum size of hole and the minimum size of the shaft.

Key Differences

Aspect	Allowance	Clearance
Definition	Intended design gap (minimum tightness /looseness)	Actual gap between parts when assembled
Use	Controls desired type of fit (clearance or interference)	Allows for free movement or assembly without friction
Measurement	Based on design intent at maximum material condition	Varies based on actual dimensions when assembled
Types	Clearance Allowance, Interference Allowance	Minimum Clearance, Maximum Clearance

Concept of limits of size and tolerances

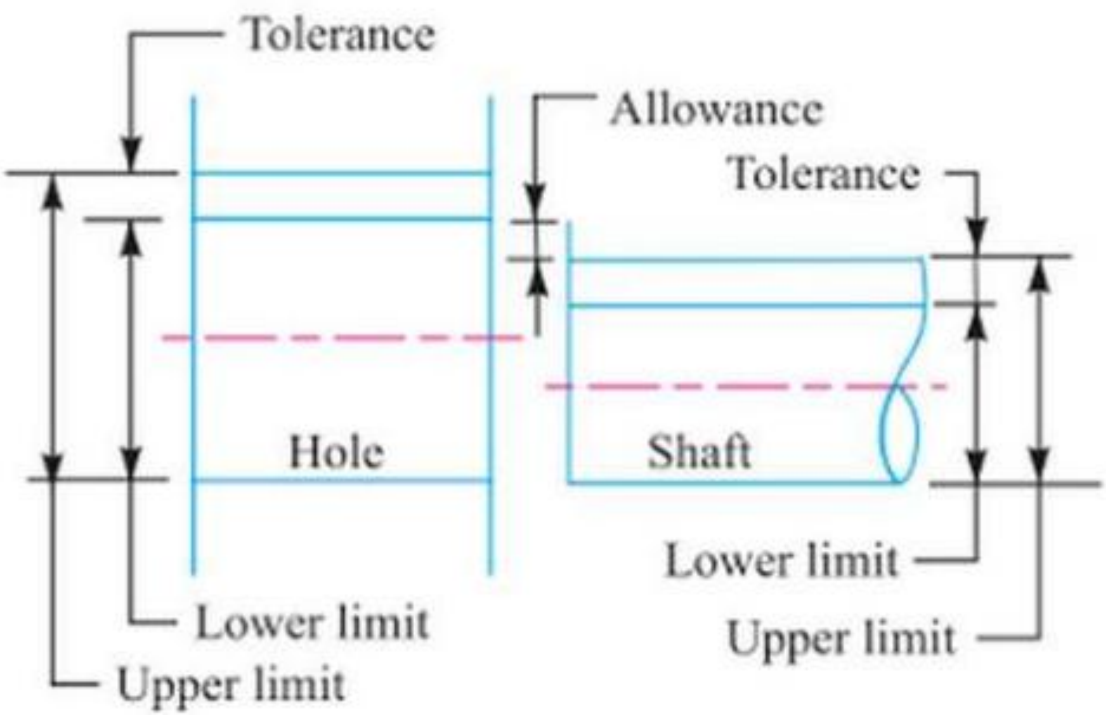


11. Tolerance:

- The term tolerance refers to the difference between the upper (maximum) limit and lower (minimum) limit of a dimension.
- In other words, tolerance is the maximum permissible variation in a dimension.
- The tolerance may be of two types i.e. unilateral or bilateral.

12. Tolerance Zone:

The term tolerance zone refers to the zone between the maximum and minimum limit size.



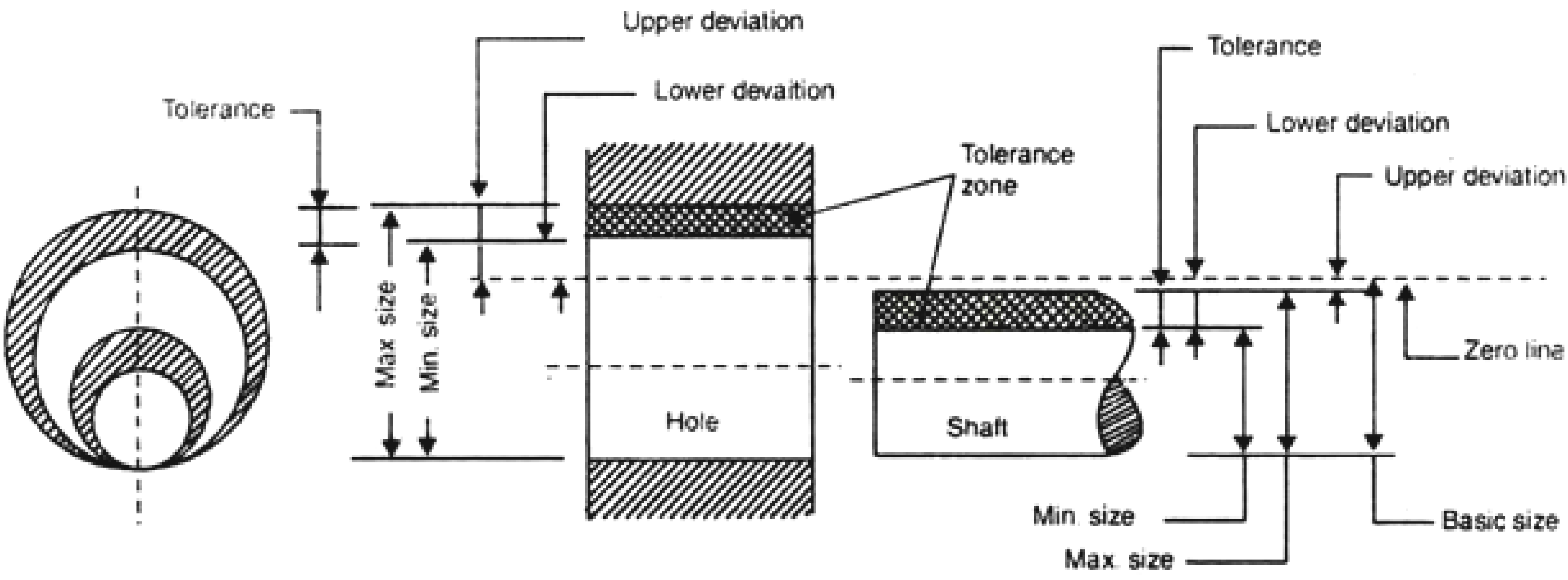


Fig. 1.54. Definition of Tolerance zone.

13. Zone Line:

- The term zero line refers to the straight line corresponding to the basic size, to which deviations and tolerances are referred.
- According to convention, the positive and negative deviations are shown above and below the zero line respectively.

14. Deviation:

The term deviation referred to the algebraic difference between a size (actual size limits of size, etc.) and the corresponding basic size.

15. Upper-Deviation:

- The term upper deviation refers to the algebraic difference between the maximum limit and the basic size.
- The upper deviation of a hole is demoted by a symbol 'ES' and of a shaft is denoted by a symbol 'es'.

16. Lower Deviation:

- The term lower deviation refers to the algebraic difference between the minimum limit and basic size.
- The lower deviation of a hole is denoted by a symbol ‘EI’ and of a shaft it is denoted by a symbol ‘ei’.

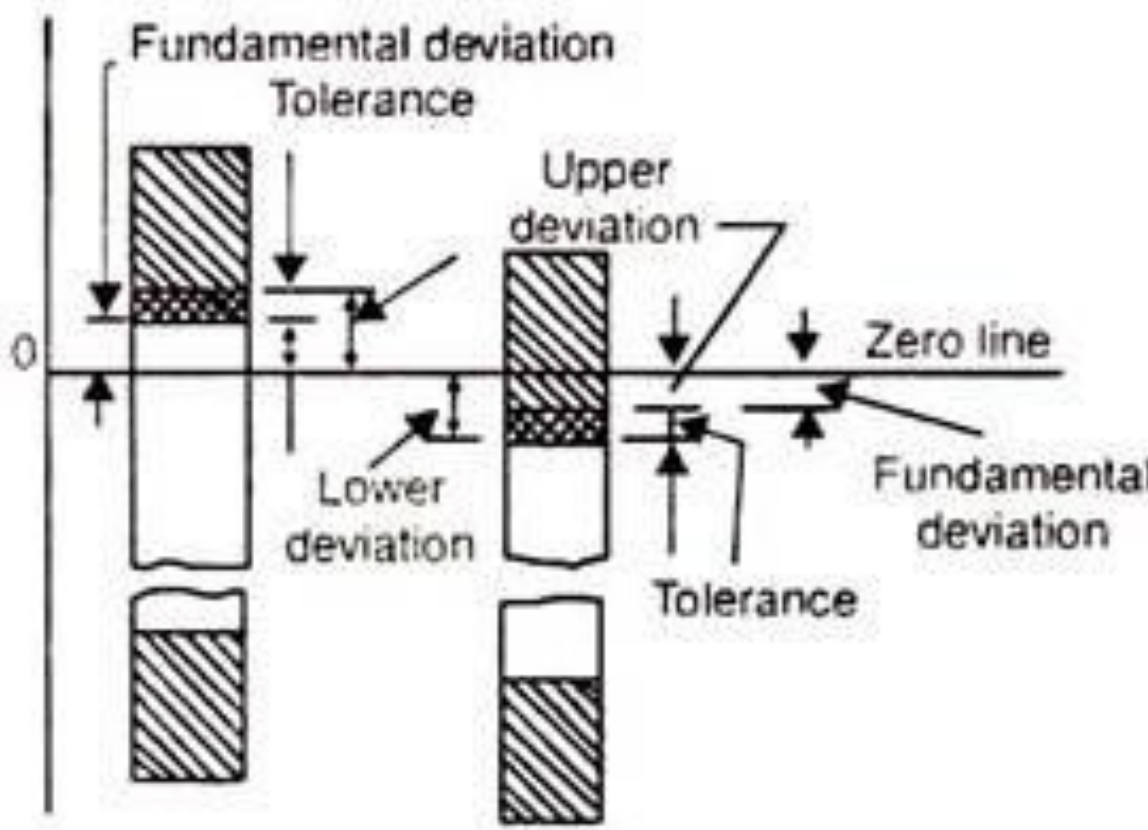
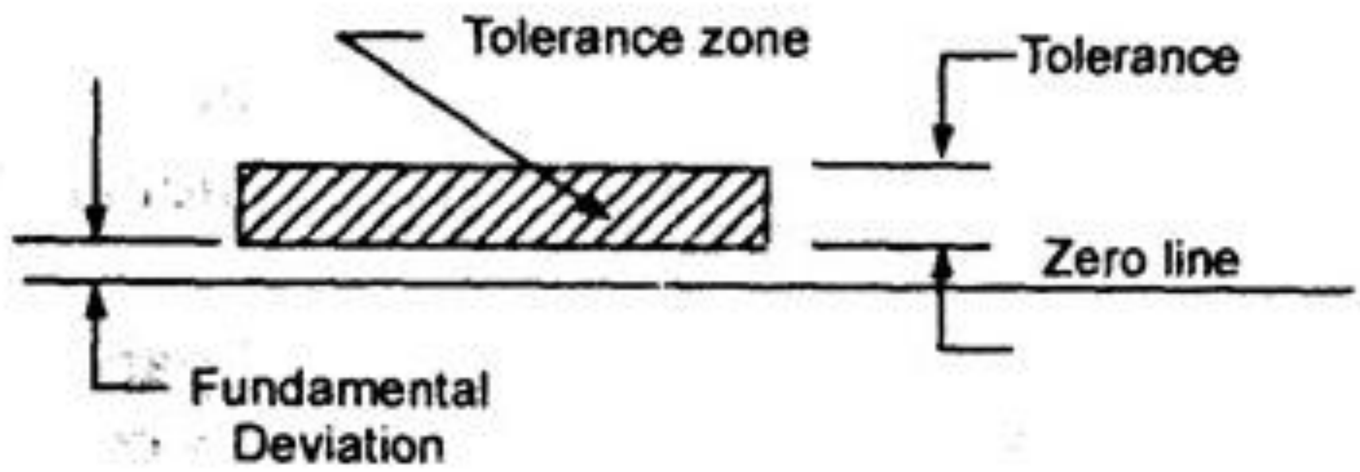


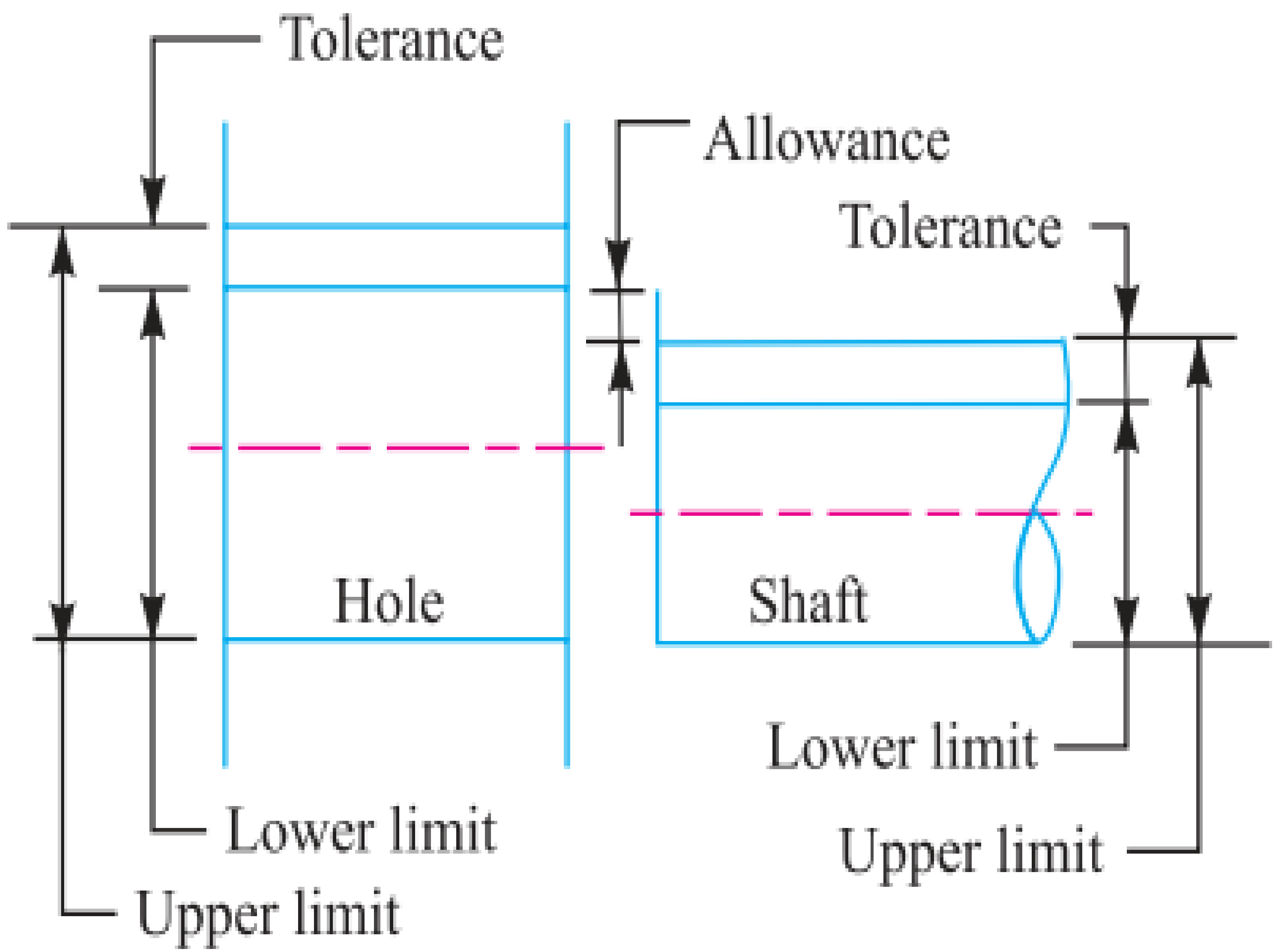
Fig. 1.55. Upper deviation, lower deviation and fundamental deviation.

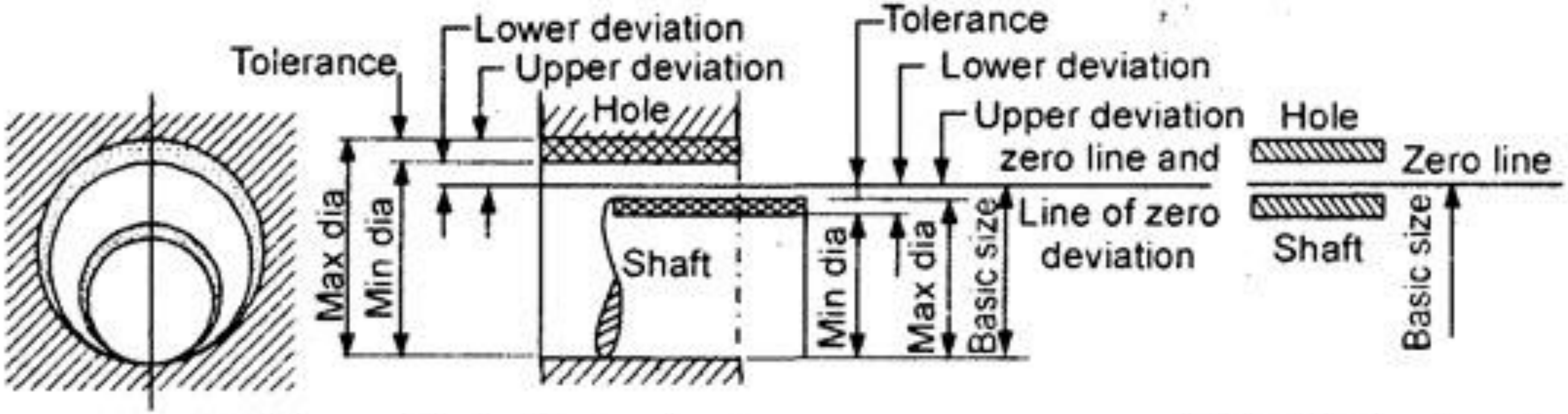
17. Fundamental Deviation:

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

- Fundamental deviation provides the position of the tolerance zone with respect to the zero line.

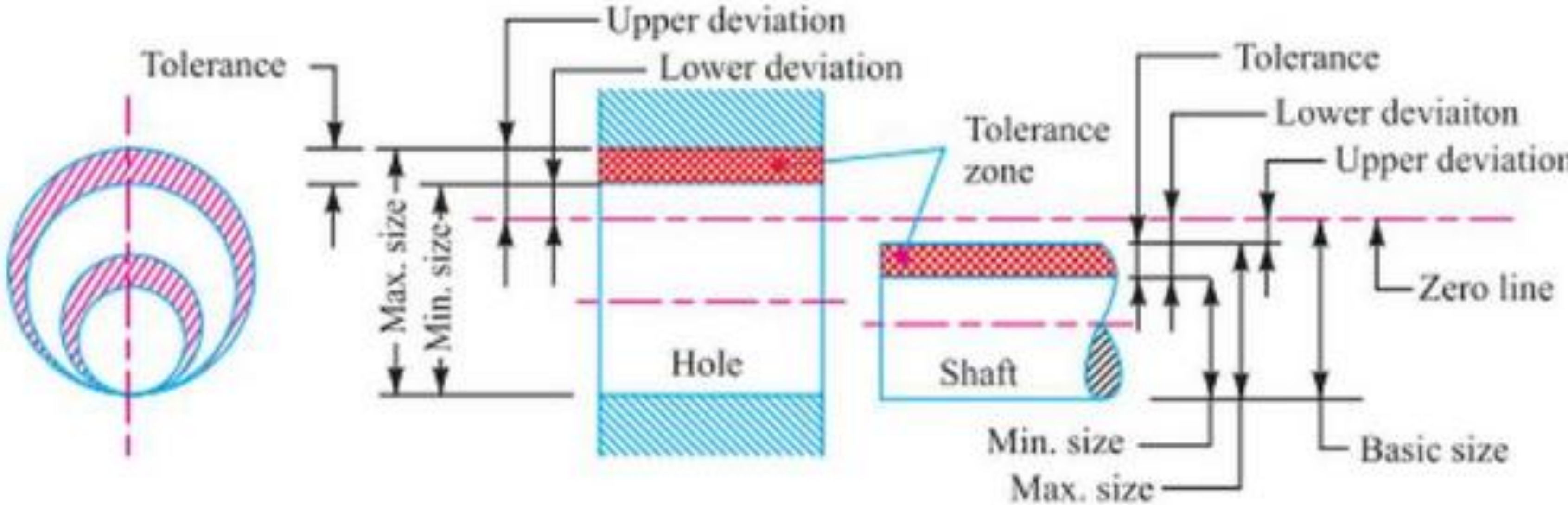


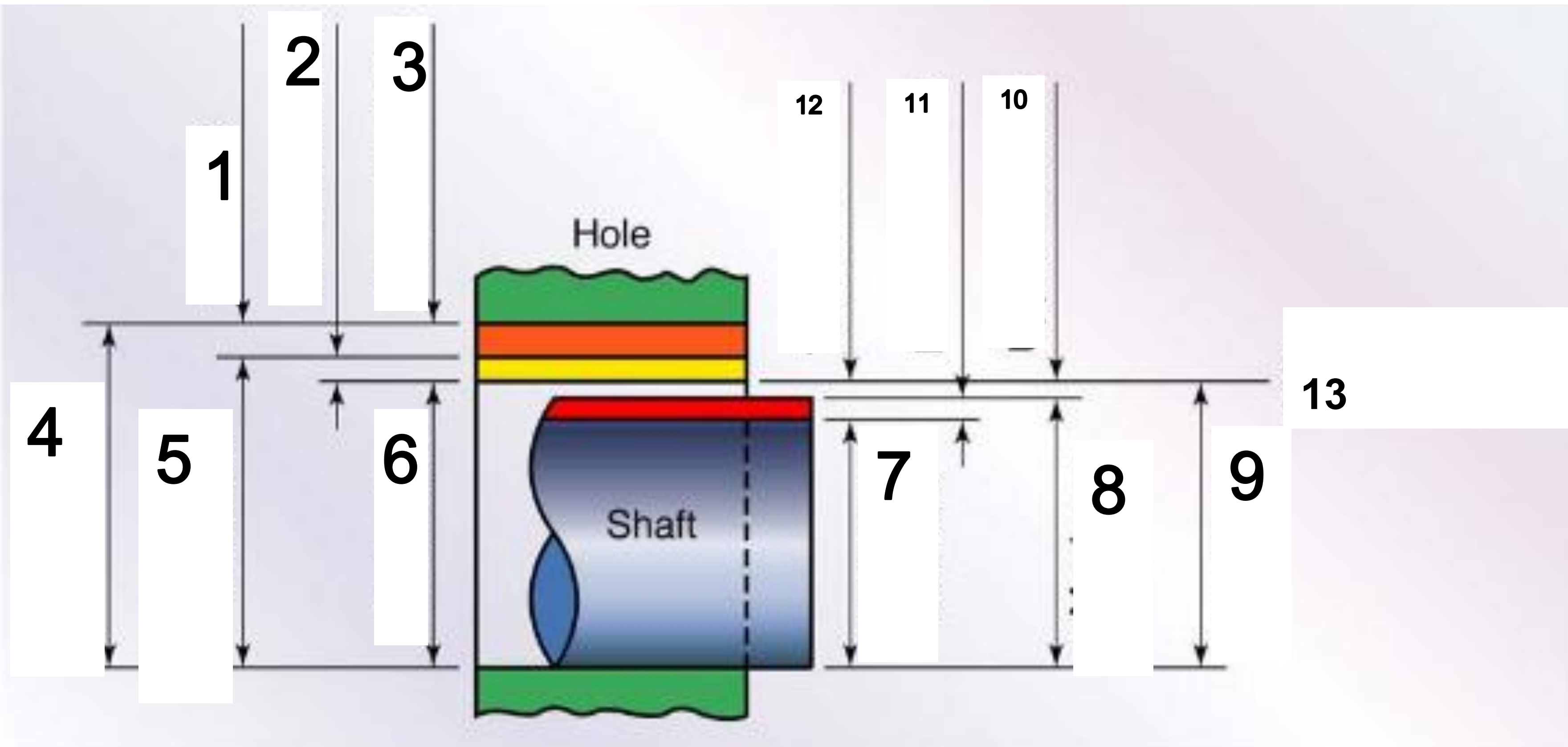

Tolerance
Allowance
Tolerance
Hole
Shaft
Lower limit
Lower limit
Upper limit
Upper limit



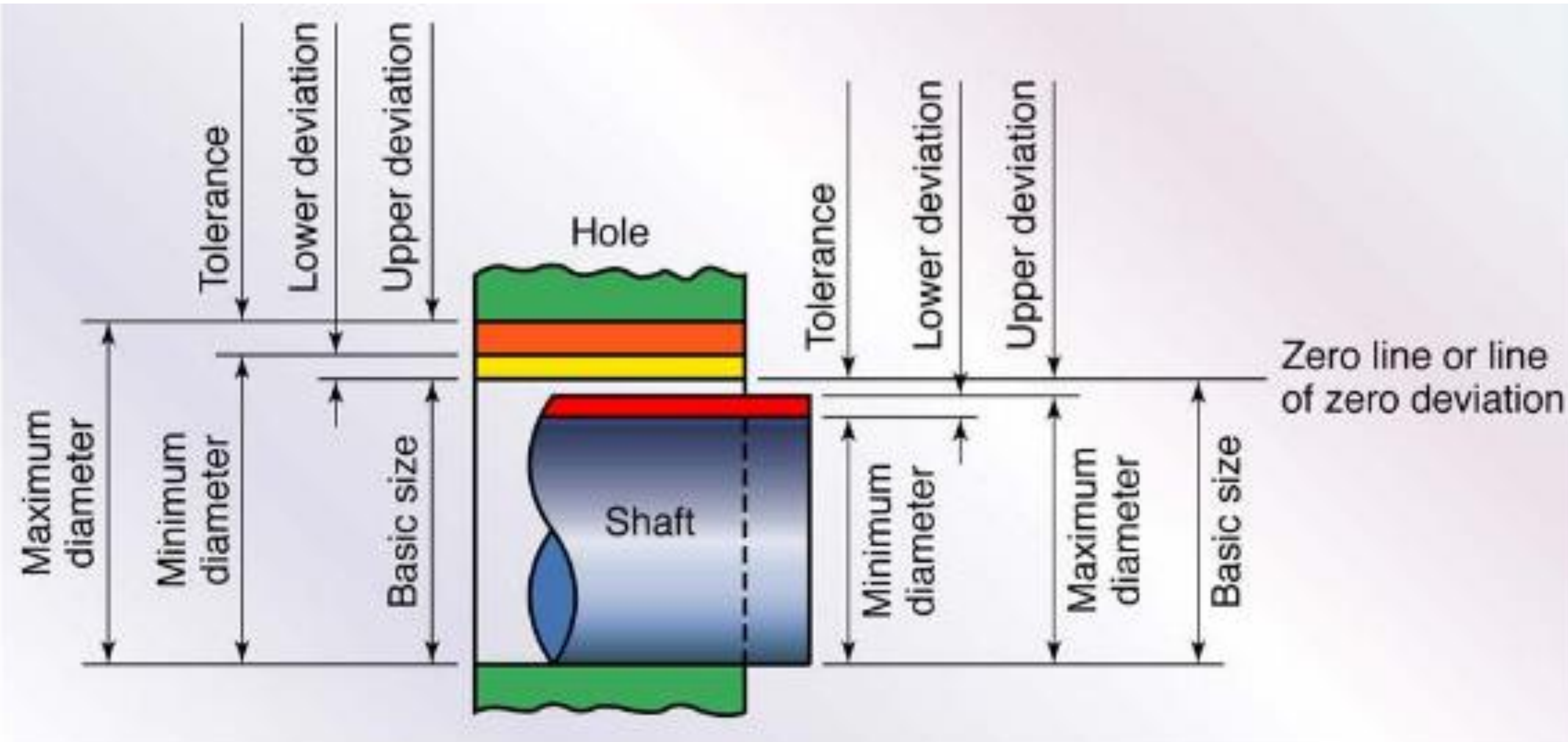
(a) Diagram illustrating basic size deviations and tolerances.

(b) Simlified schematic diagram of clearance fit.


Upper deviation
Tolerance
Lower deviation
Tolerance
Lower deviaiton
Tolerance
zone
Upper deviation
Max. size
Min. size
Zero line
Hole
Shaft
Min. size
Max. size
Basic size



1
2
3
12
11
10
Hole
4
5
6
Shaft
7
8
9
13

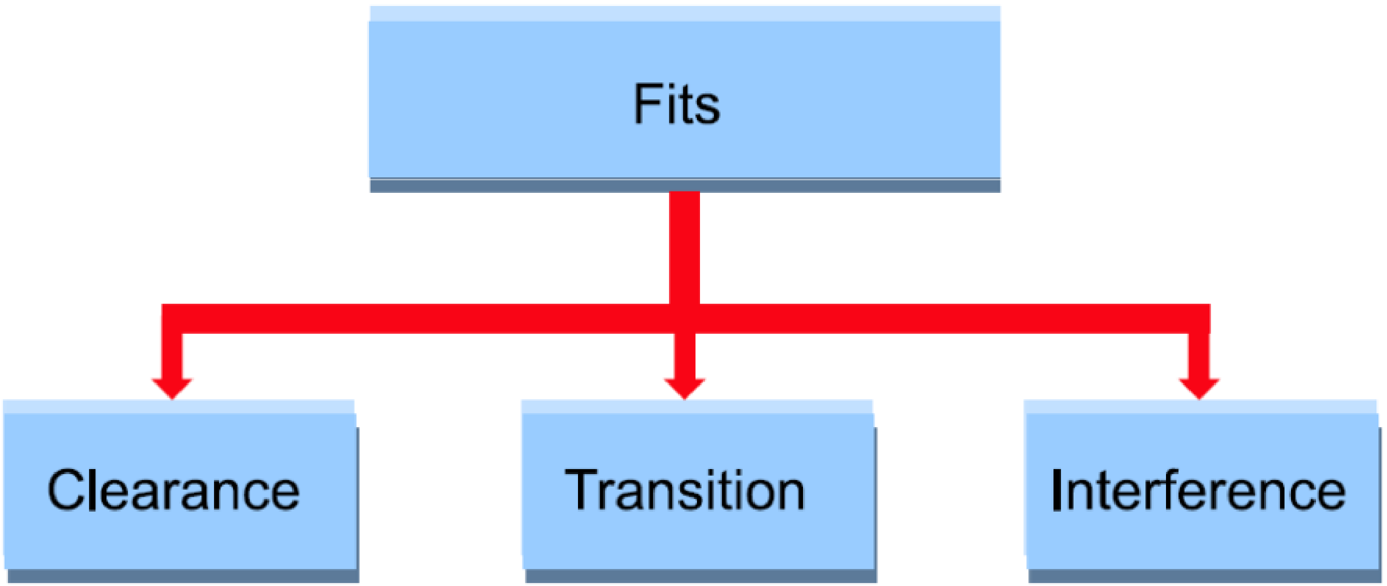


Hole
Shaft
Tolerance
Lower deviation
Upper deviation
Maximum diameter
Minimum diameter
Basic size
Tolerance
Lower deviation
Upper deviation
Zero line or line of zero deviation
Minimum diameter
Maximum diameter
Basic size

Fits:

The term fits refers to the **degree of tightness or looseness between two mating parts.** Depending upon the actual limits of the hole and shaft.

Fits may be classified into the following three types:



Clearance Fit:

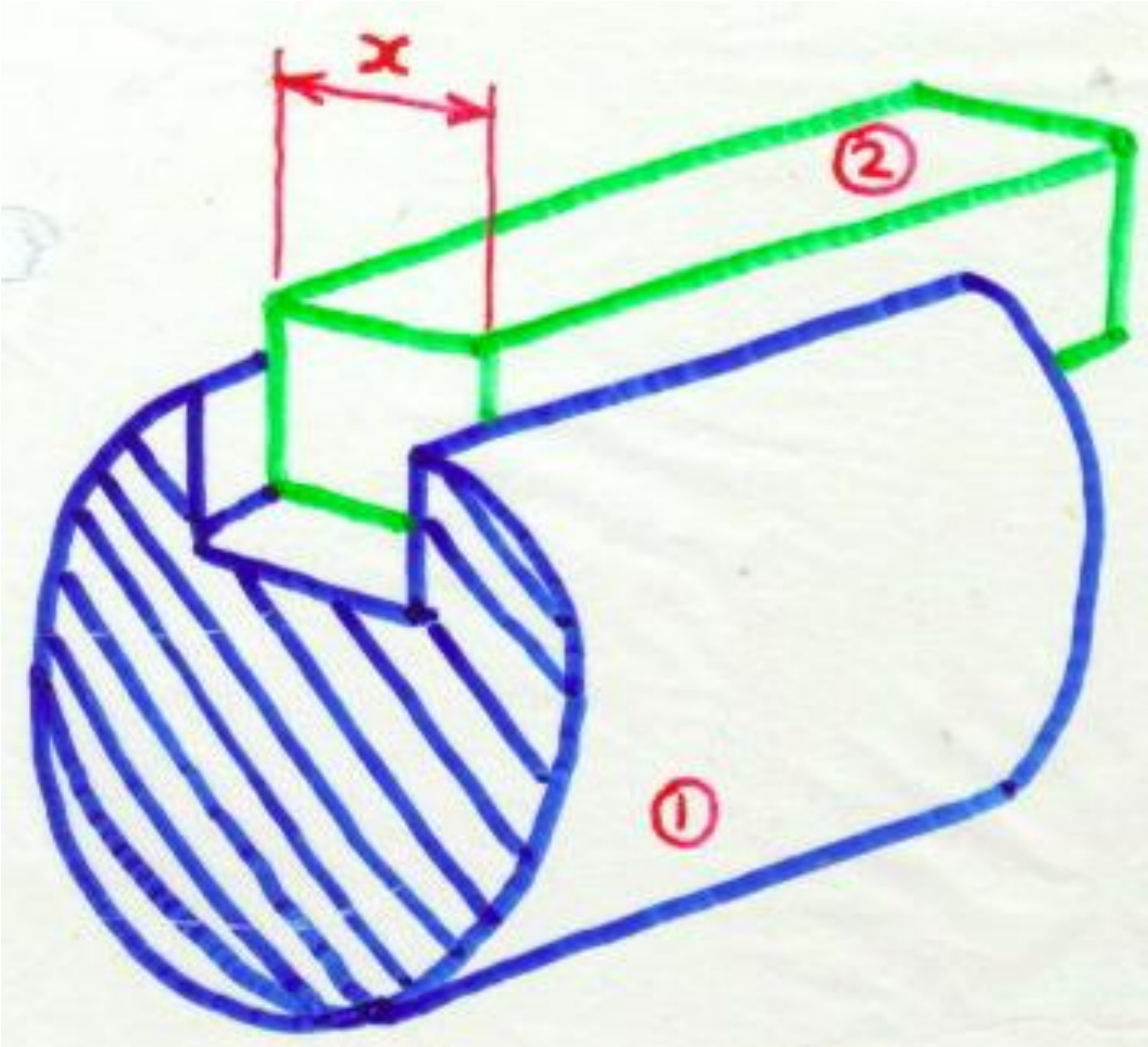
A fit that always provides a clearance (gap) between the hole and shaft when assembled is known as clearance fit.

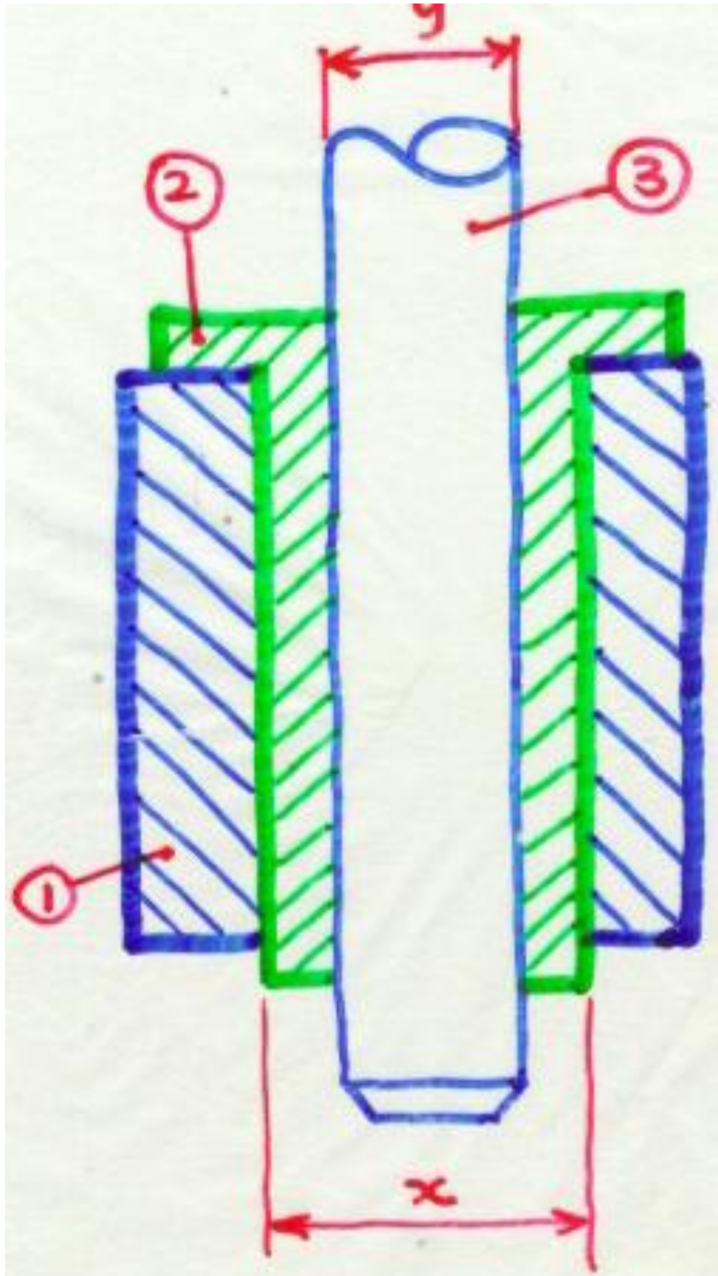
In clearance fit, there is positive clearance between the shaft and hole. *Such fits give loose joint*

In clearance fit, the minimum size of the hole is either greater than or, equal to (in extreme case) the maximum size of the shaft, so that the shaft can rotate or slide as per the purpose of the assembled members.

In clearance fit, the difference between the maximum size of the hole and minimum size of the shaft is called maximum clearance, whereas the minimum size of the hole and maximum size of the shaft is known as minimum clearance.

The clearance fit may be of different types, e.g., slide fit, easy sliding fit, running fit, slack running fit and loose running fit, etc.

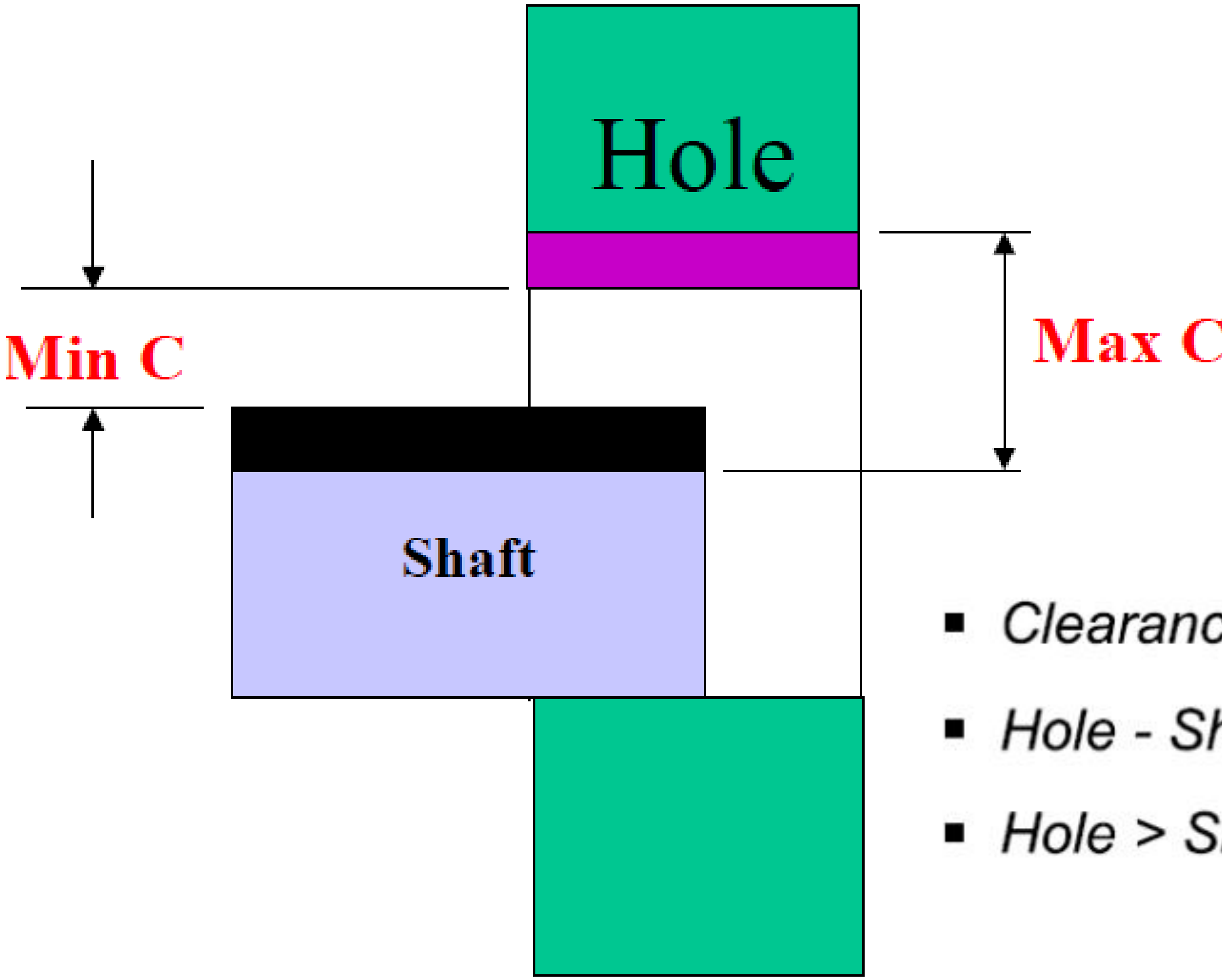

x
2
1


y
2
3
1
x

CLEARANCE FIT

Max. C = UL of hole - LL of shaft

Min. C = LL of hole - UL of shaft



- *Clearance = Hole - Shaft*
- *Hole - Shaft > 0*
- *Hole > Shaft*

Types of Clearance Fit

- 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.
- 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.

Types of Clearance Fit

1. Loose Fit

- Used where **high precision is not required**.
- There is **maximum clearance** (big gap) between hole and shaft.
- Parts can move freely without restriction.

Examples:

- Loose pulleys
- Agricultural machinery parts

2. Running Fit

- The shaft is slightly smaller than the hole so that it can **run or rotate smoothly**.
- A **thin film of oil** can form between the surfaces for lubrication.
- Ensures smooth, continuous motion.

Example:

- Bearing and shaft pair

3. Slide Fit / Medium Fit

- Used where **more accuracy (precision)** is needed than running fit.
- Provides **medium clearance**.
- Allows sliding motion, but without too much looseness.

Examples:

- Tool slides
- Slide valves
- Automobile components

Interference Fit:

An interference fit is a type of fit where the shaft is always larger than the hole.

Because of this, there is no clearance—instead, the parts must be forced together.

- There is tight contact between hole and shaft.
- Assembly requires force, pressing, or thermal methods (heating the hole or cooling the shaft).
- Provides very strong joint.
- Does not allow any relative motion between parts.

It may be noted from the figure that in interference fit, the tolerance zone of the hole is entirely below the tolerance zone of the shaft.

In interference fit, the difference between the minimum size of the hole and the maximum size of the shaft is called maximum interference.

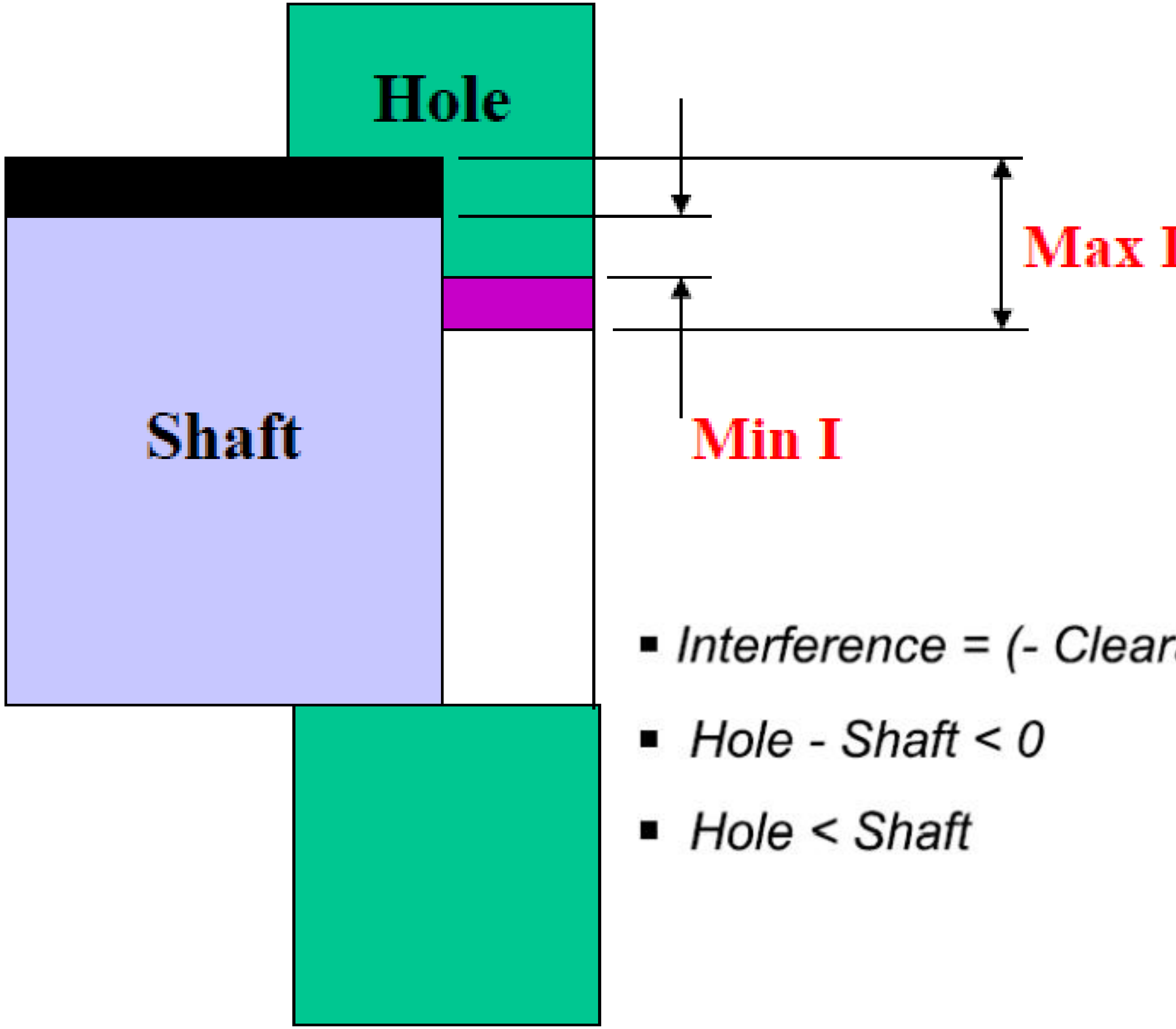
Whereas difference between the maximum size of the hole and the minimum size of the shaft is known as minimum interference.

The interference fit may be of different types, e.g., shrink fit, light drive fit, heavy drive fit.

INTERFERENCE FIT

Max. I = LL of hole - UL of shaft

Min. I = UL of hole - LL of shaft



- *Interference = (- Clearance)*
- *Hole - Shaft < 0*
- *Hole < Shaft*

Example

Hole size = 20.00 mm
Shaft size = 20.05 mm

$$\text{Interference} = 20.05 - 20.00 = 0.05\ mm$$

This means the shaft is **bigger**, so it will not assemble freely.

Types of Interference Fit

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 Force 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.

Types of Interference Fit

1. Shrink Fit / Heavy Force Fit

- Has **maximum negative allowance** (largest interference).
- For assembly, the **hole is heated** → it expands → shaft is inserted → hole cools and tightens strongly.
- Gives a **very strong and permanent joint**.

Used in:

- Fitting rims
- Railway wheels
- Heavy machinery parts

2. Medium Force Fit

- Has **medium interference**.
- Requires **considerable pressure** using hydraulic presses or mechanical presses.
- Not as tight as shrink fit but still very firm.

Used in:

- Car wheels
- Armature of dynamos
- Medium-duty hubs

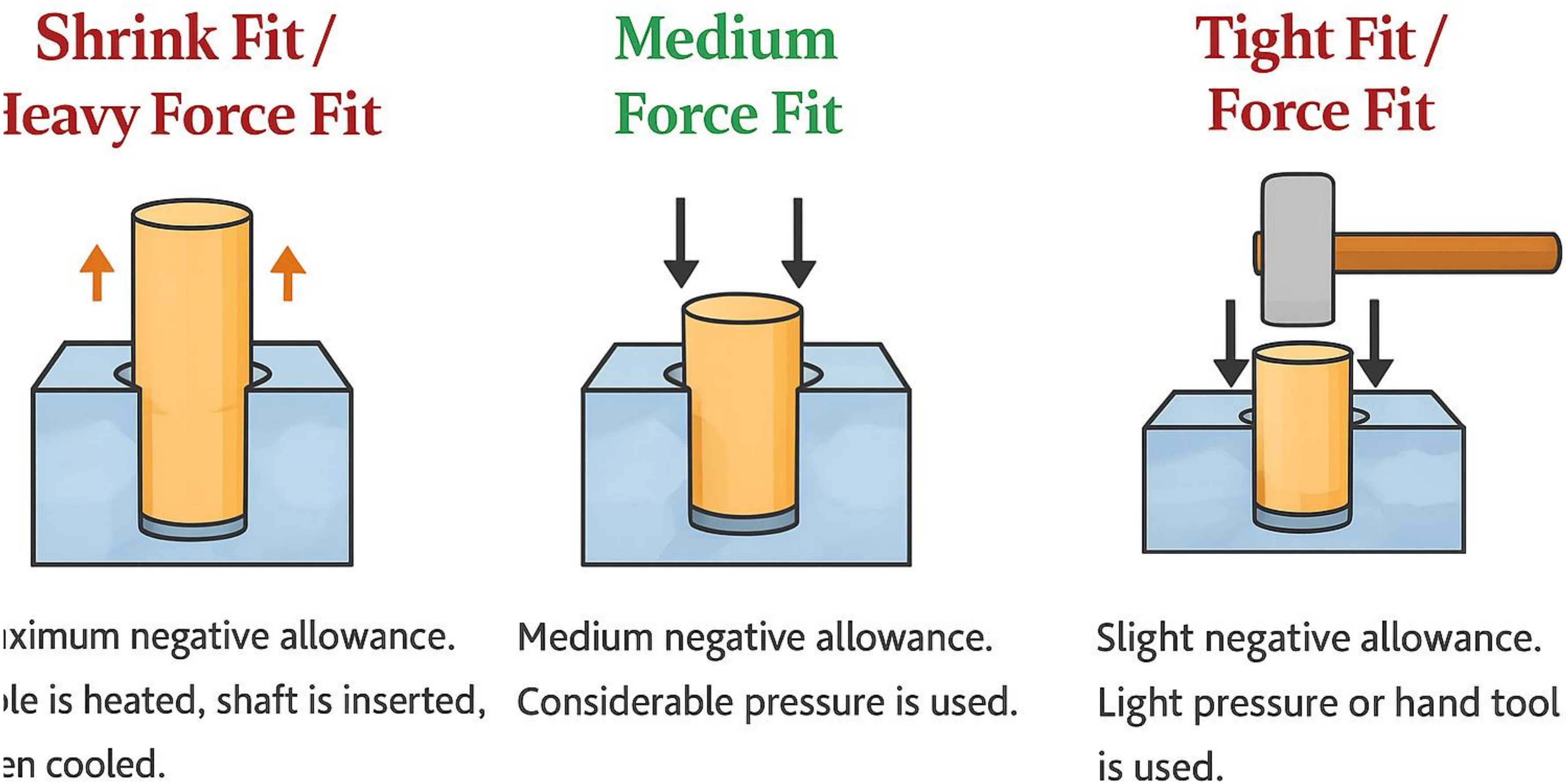
3. Tight Fit / Force Fit

- Has **small (slight) negative allowance**.
- Can be assembled using a **hand hammer or light pressure**.
- Gives a **semi-permanent joint**—can be removed but with effort.

Used in:

- Keyed pulley and shaft
- Rocker arm
- Light mechanical assemblies

Types of Interference Fit



Transition Fit:

A fit which may provide either a clearance or interference between the shaft and hole when assembled, depending on the actual sizes of the shaft and hole, is known as Transition fit.

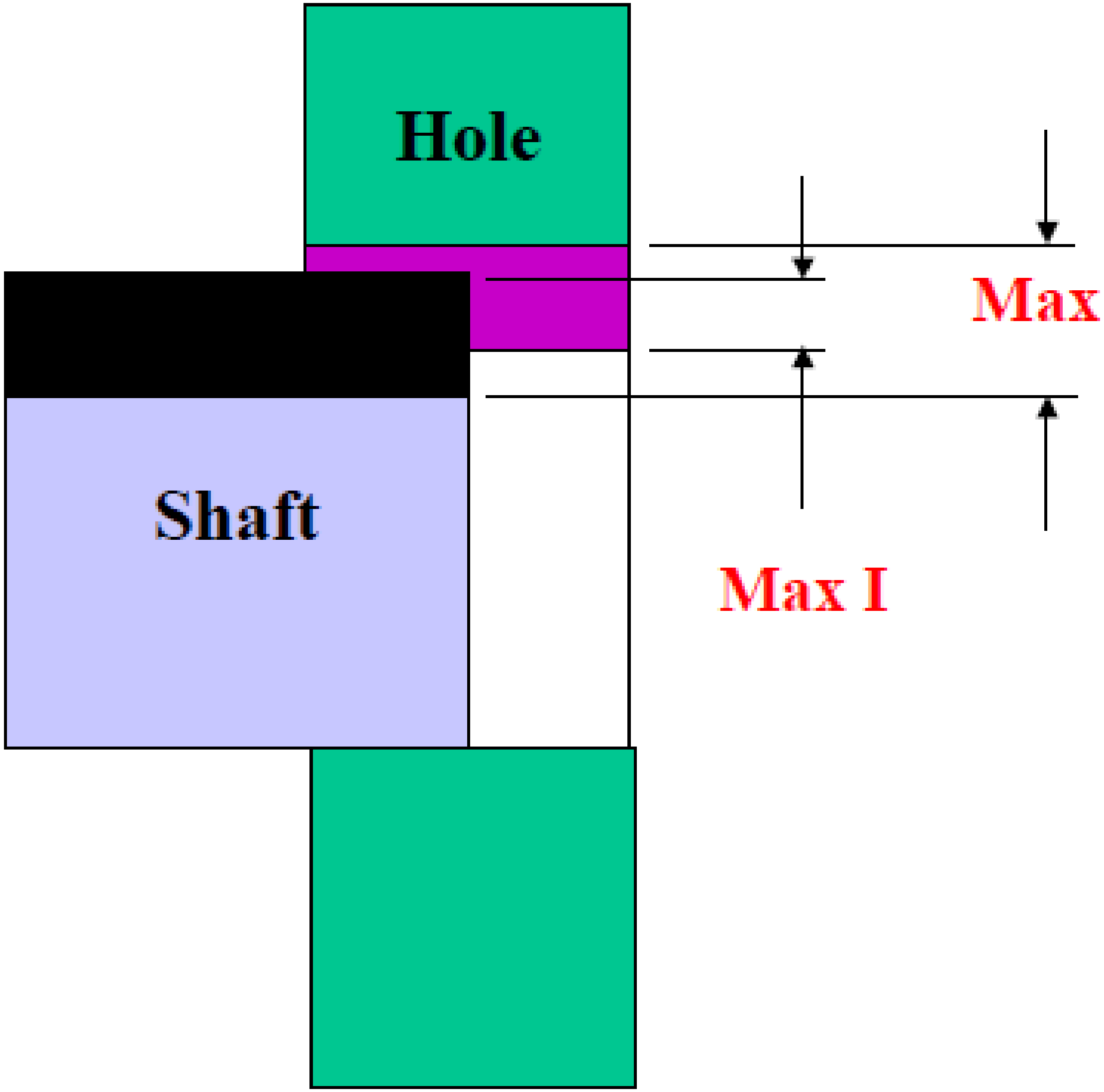
It may be noted that in a transition fit, the tolerance zone of shaft and hole overlap completely or partially.

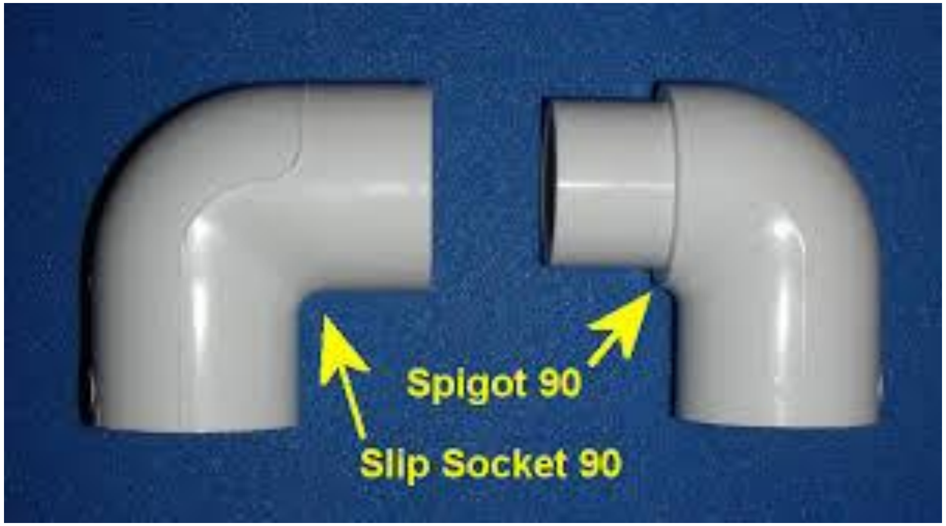
The transition fit may be of different types, e.g. Push fit, force fit, tight fit etc.

TRANSITION FIT

Max. C = UL of hole - LL of shaft

Max. I = LL of hole - UL of shaft




Spigot 90
Slip Socket 90


PTSMAKE.COM

Types of Transition Fit

Push Fit or Snug Fit

It refers to zero allowance and a light pressure is required in assembling the hole and the shaft. The moving parts show least vibration with this type of 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.

Types of Transition Fit

Push Fit / Snug Fit

A type of **light interference fit** where the shaft can be pushed into the hole with **hand pressure or slight force**.

Parts that need to stay in place but **may be removed later if needed**.

Example: Small gear on a shaft, dowel pins.

Key: Easy assembly, slight holding force, no permanent bonding.

Force Fit / Shrink Fit

A type of **tight interference fit** where **significant force** is required for assembly.

Sometimes, the **hole is heated or shaft is cooled** to make assembly easier (thermal expansion/contraction).

Permanent joints, where slipping is not allowed.

Example: Large pulleys, flywheels on shafts.

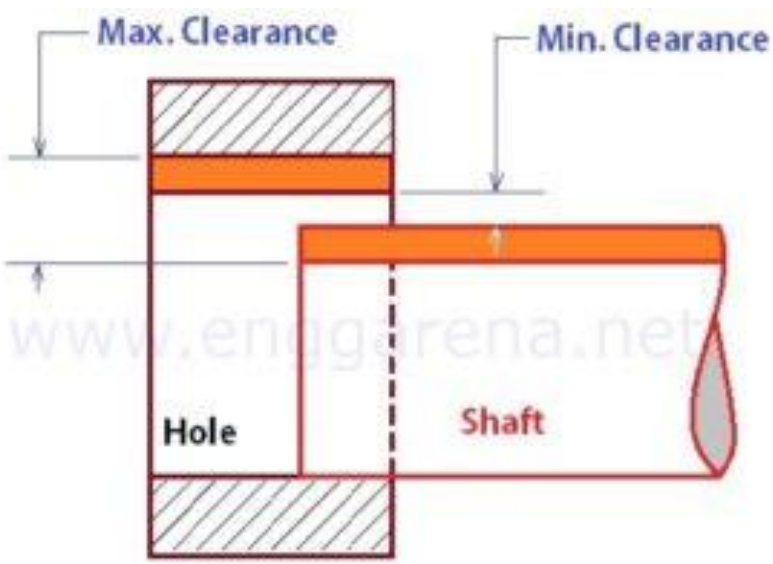
Key: Permanent, requires tools or heating/cooling for assembly.

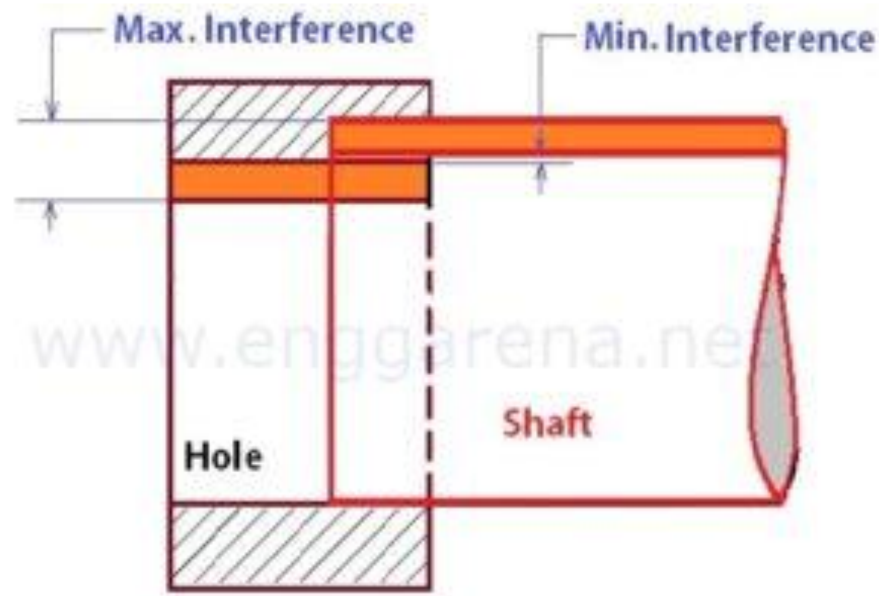
Wringing Fit

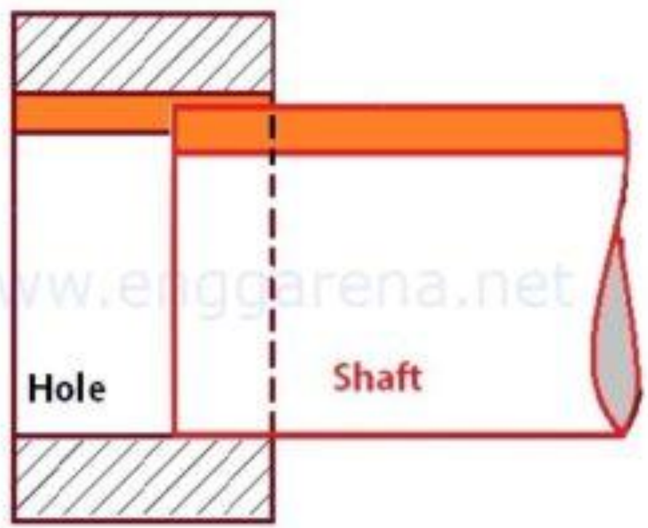
A **very precise fit** where the shaft and hole surfaces are so smooth and close that they **stick together due to friction and surface contact**. Also called a **“zero clearance” fit**.

Precision instruments, gauge blocks, micrometer anvils.

Key: Extremely accurate location, no play, held by surface adhesion/friction.


Max. Clearance
Min. Clearance
Hole
Shaft


Max. Interference
Min. Interference
Hole
Shaft


Hole
Shaft

CLEARANCE FIT: Hole is always bigger than shaft

Hole: 10.05 mm

Shaft: 9.95 mm

Result: ***Always loose***

INTERFERENCE FIT: Shaft is always bigger than hole

Hole: 10.00 mm

Shaft: 10.02 mm

Result: ***Always tight***

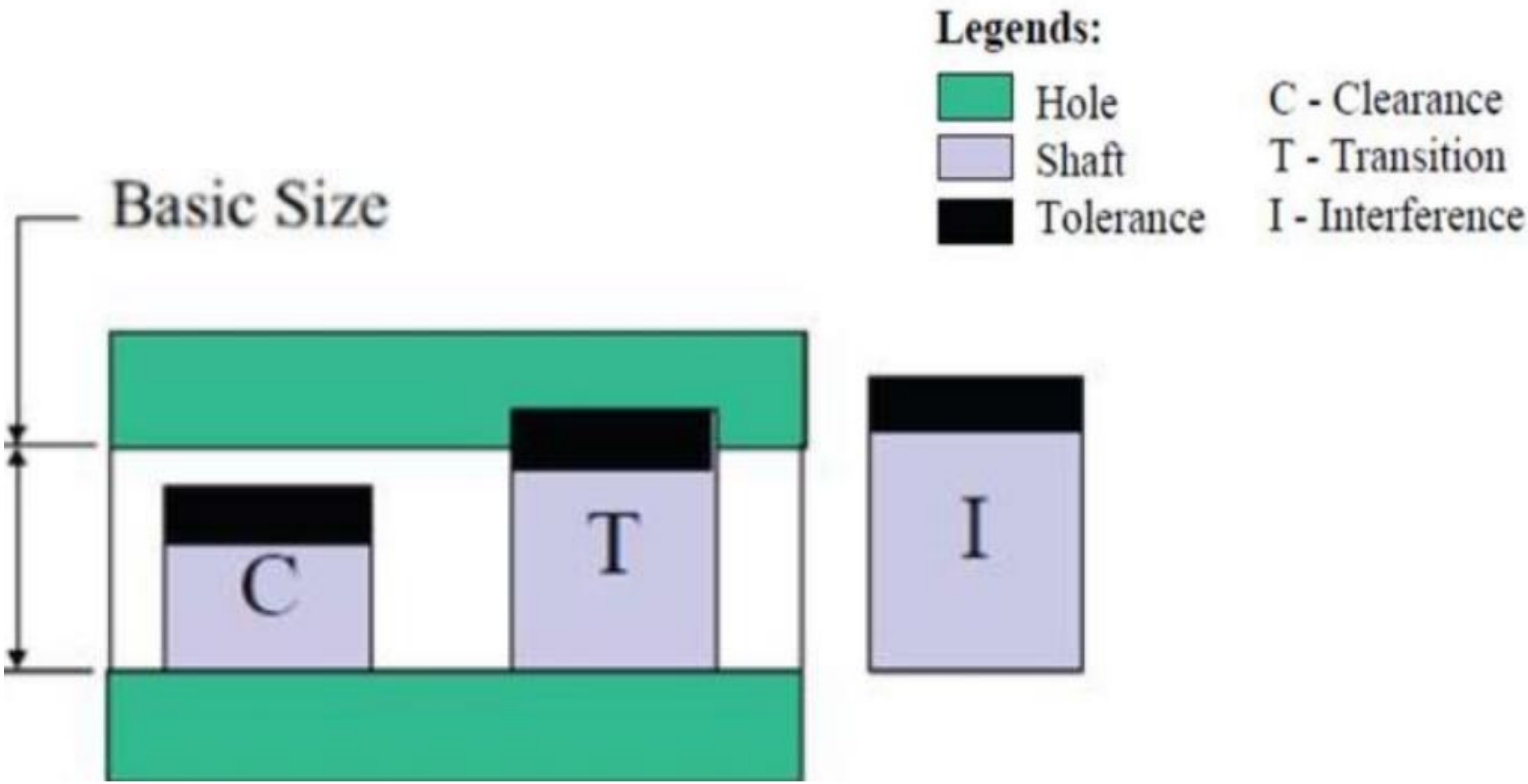
TRANSITION FIT: Hole can be slightly bigger OR slightly smaller

Hole: 10.02 mm to 10.00 mm

Shaft: 10.01 mm to 9.99 mm

Result: ***Sometimes loose, sometimes tight***

Hole basis system:



In a Hole Basis System, the hole's size is taken as the standard or basic size (nominal size), and the shaft's size is adjusted to achieve the desired fit — clearance, transition, or interference.

Why it is used:

1. Holes are **easier to standardize and manufacture** than shafts.
2. Shafts can be machined or ground to slightly smaller or larger sizes as needed.
3. Most industrial assemblies use this system for **cost efficiency and interchangeability**.

Example:

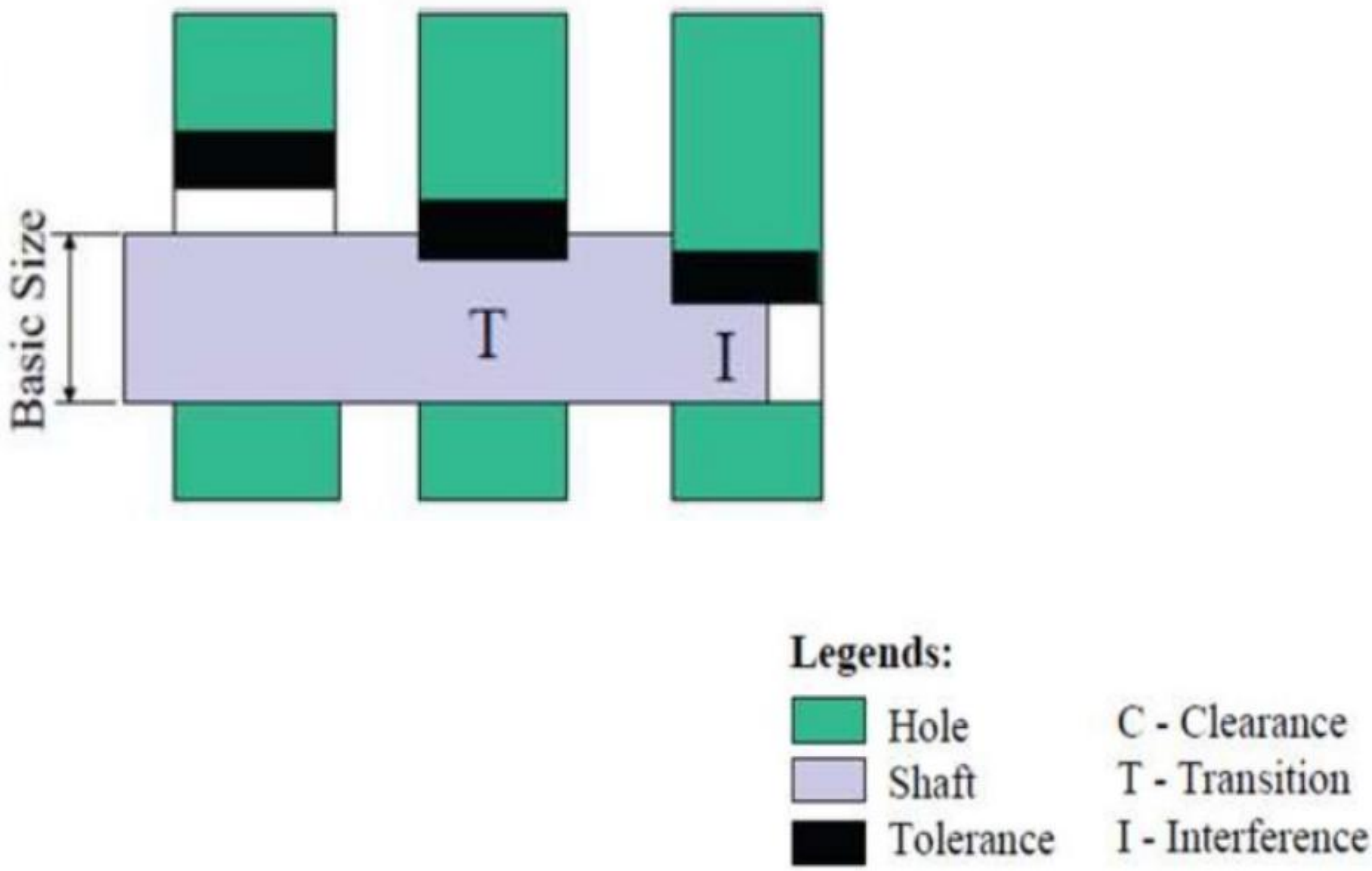
Nominal diameter = 20 mm

Hole = 20.00 mm (basic)

- Shaft = 19.95 mm → clearance fit
- Shaft = 20.00 mm → snug/transition fit
- Shaft = 20.05 mm → interference fit

✅ **Key:** Hole is fixed; shaft varies.

Shaft basis system



In a Shaft Basis System, the shaft is taken as the standard or basic size (nominal size), and the hole size is adjusted to achieve the desired fit.

Why it is used:

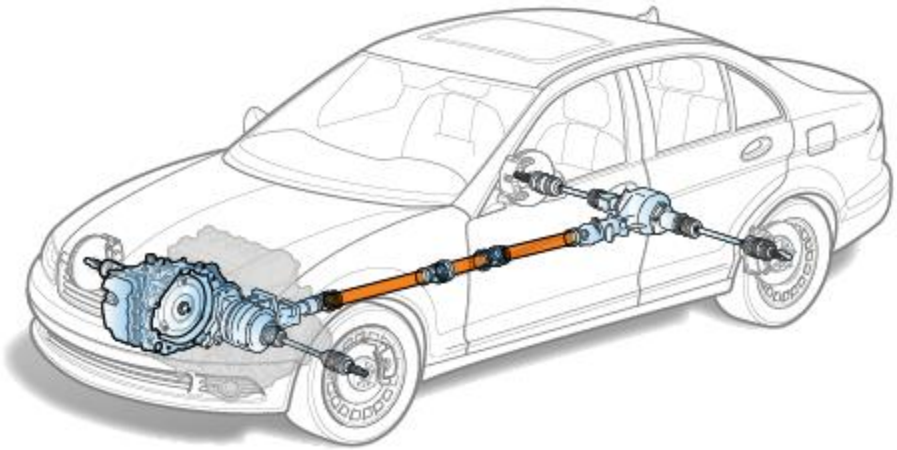
1. Shafts may be **standardized for interchangeability**, e.g., in motors or machinery.
2. Holes can be **machined or bored to slightly different sizes** for different fits.

Nominal diameter = 20 mm

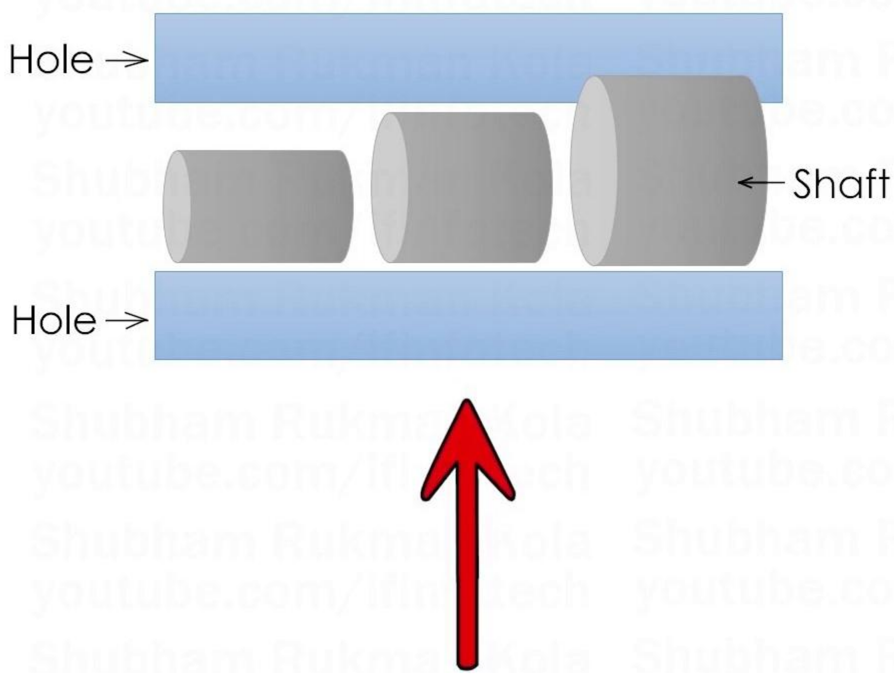
Shaft = 20.00 mm (basic)

- Hole = 20.05 mm → clearance fit
- Hole = 20.00 mm → snug/transition fit
- Hole = 19.95 mm → interference fit

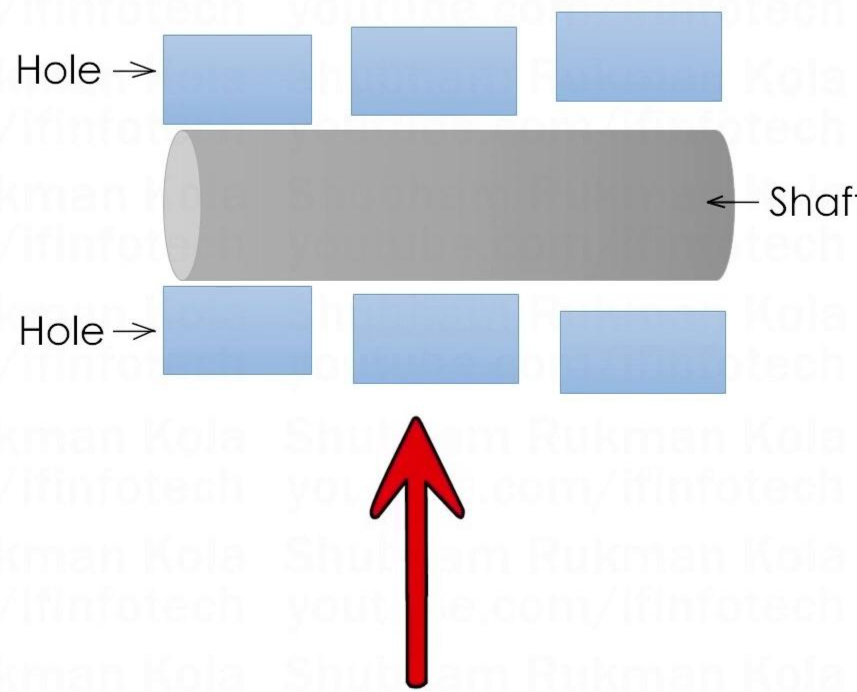
✅ **Key:** Shaft is fixed; hole varies.



Hole Basis System



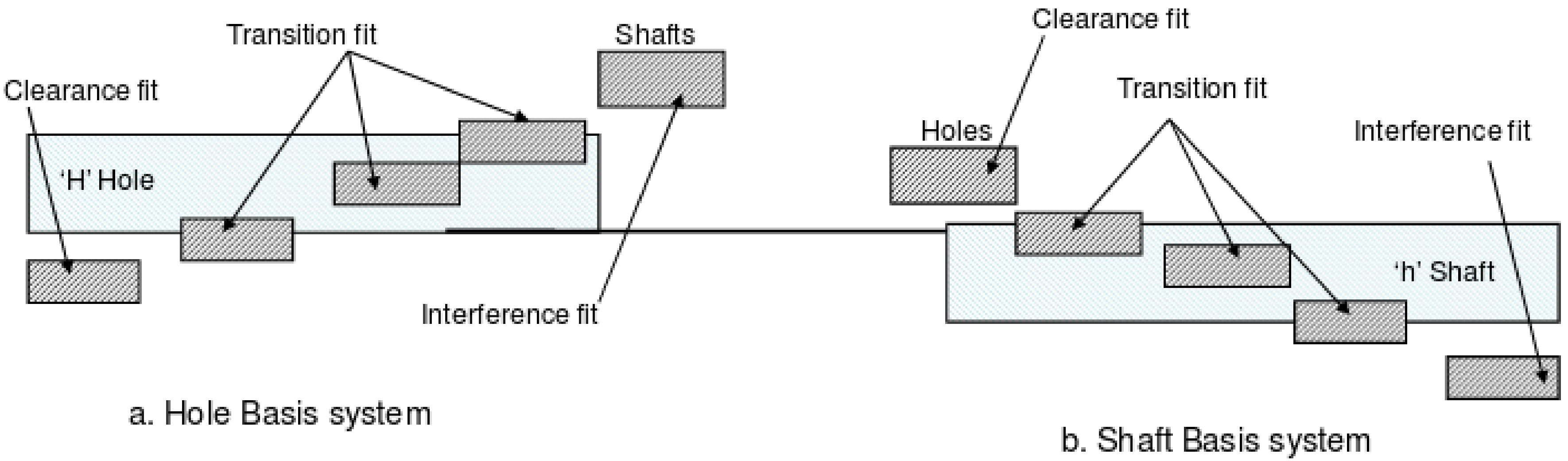
Shaft Basis System

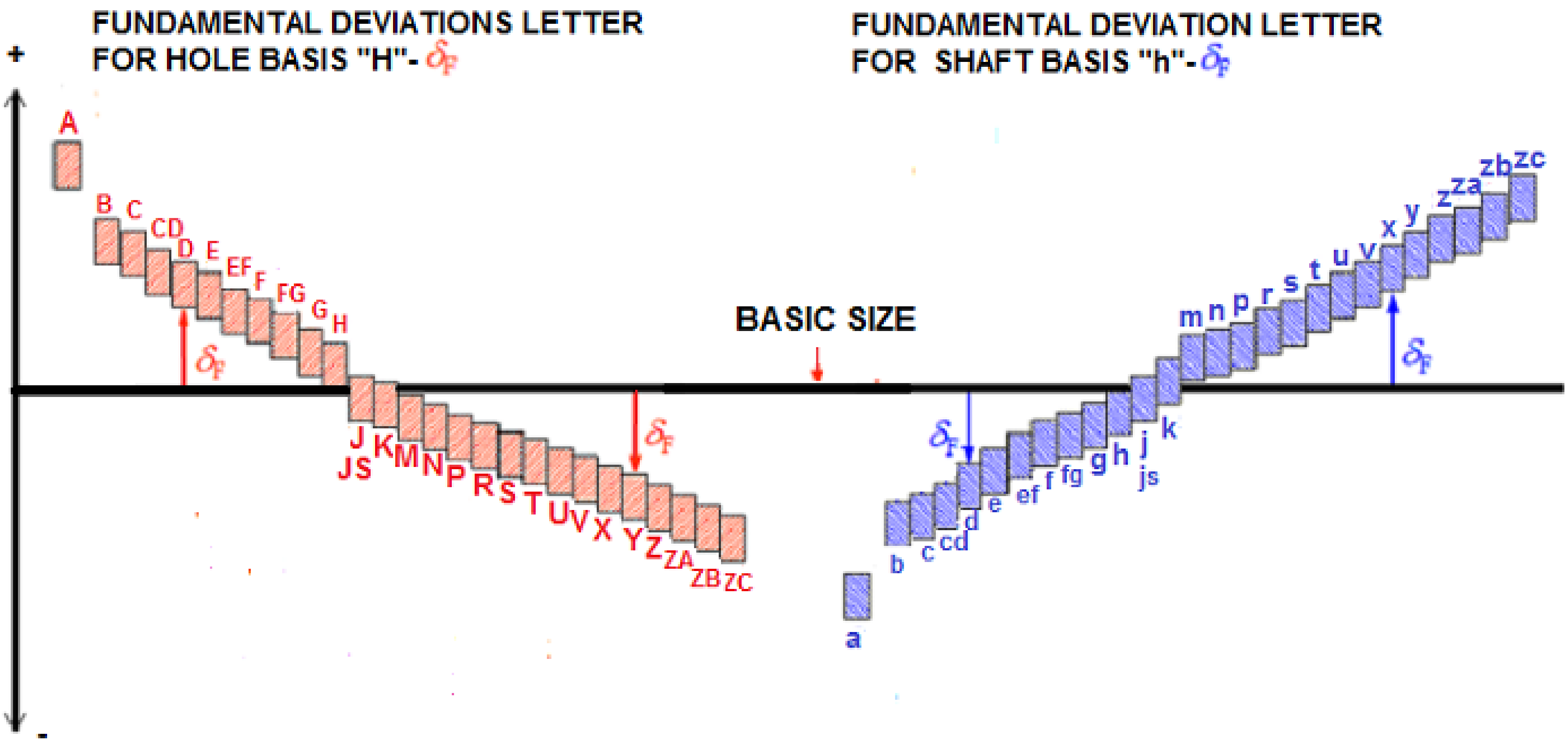


Standard Limit & Fit System

Systems of Fit: There are two systems by which a fits can be accomplished –

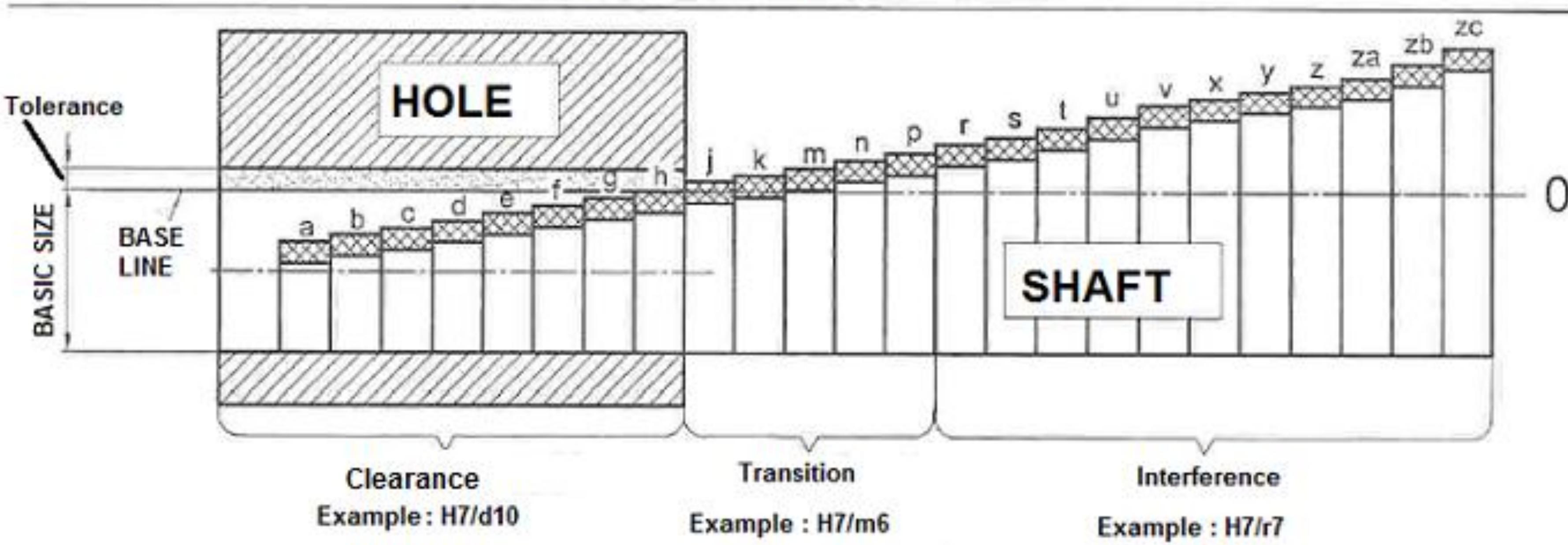
1. Hole basis system
2. Shaft basis system



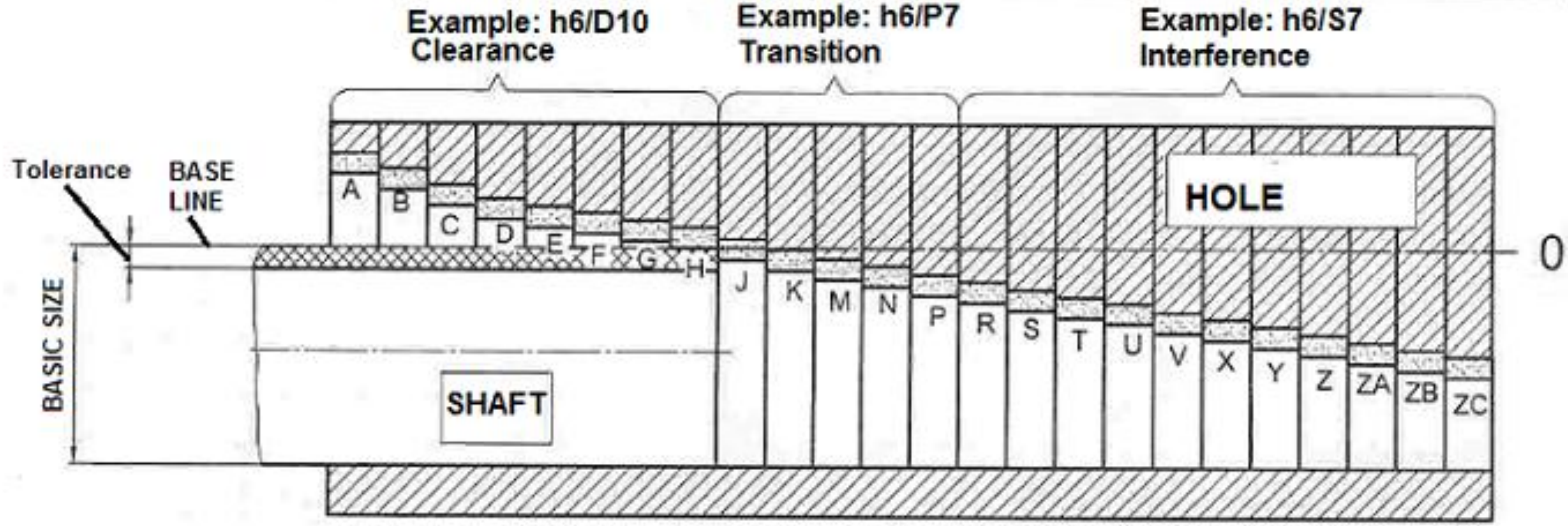


FUNDAMENTAL DEVIATIONS LETTER
FOR HOLE BASIS "H"- δ_F
FUNDAMENTAL DEVIATION LETTER
FOR SHAFT BASIS "h"- δ_F
+
-
BASIC SIZE
A B C CD D E EF F FG G H J JS K M N P R S T U V X Y Z ZA ZB ZC
a b c cd d e ef f fg g h j js k m n p r s t u v x y z za zb zc
δ_F

HOLE BASIS SYSTEM SHOWING WITH LETTER

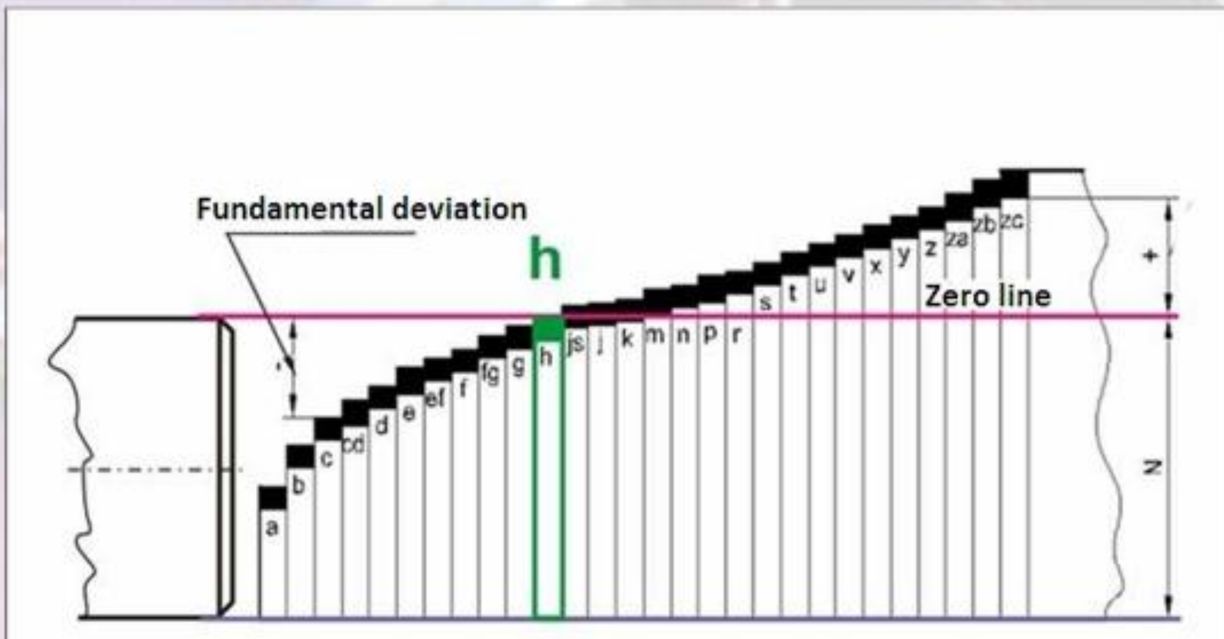


SHAFT BASIS SYSTEM SHOWING WITH LETTER



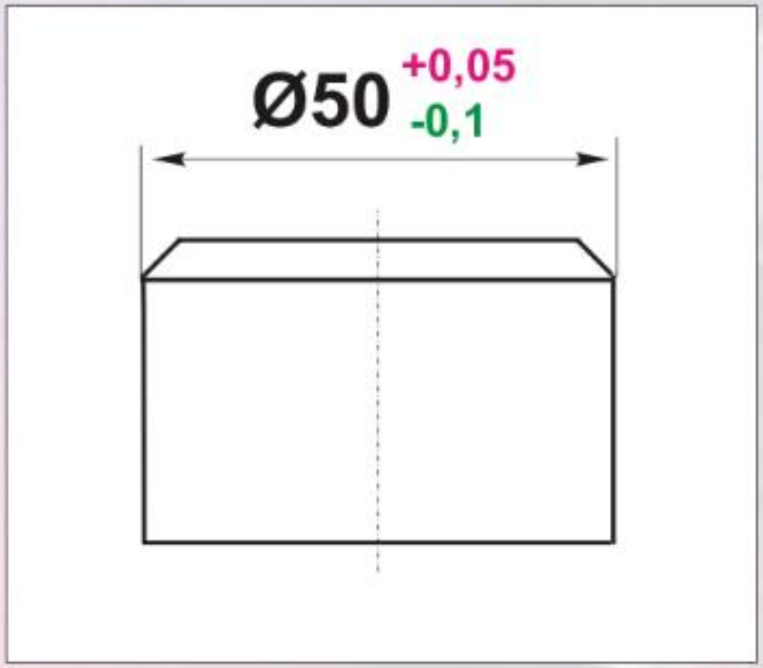
General Terminology in Fits

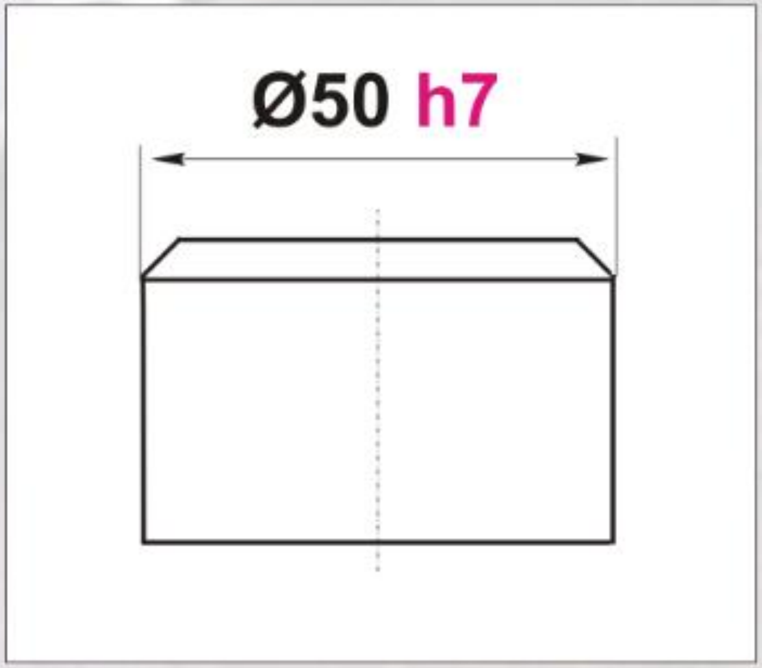
- ✓ <u>Tolerance symbols:</u> These are used to specify the tolerance and fits for mating components. For example, in 40 H8f7, the number 40 indicates the basic size in millimeters; capital letter H indicates the fundamental deviation for the hole; and lower-case letter f indicates the shaft. The numbers following the letters indicate corresponding IT grades.



Two ways of indicating tolerances on technical drawings

Limits of a dimension or the tolerance values are specified directly with the dimension.

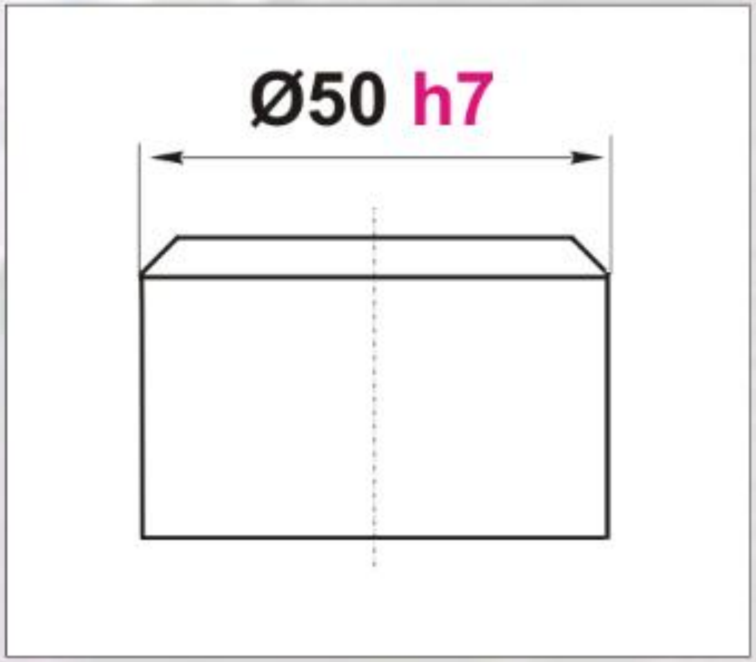




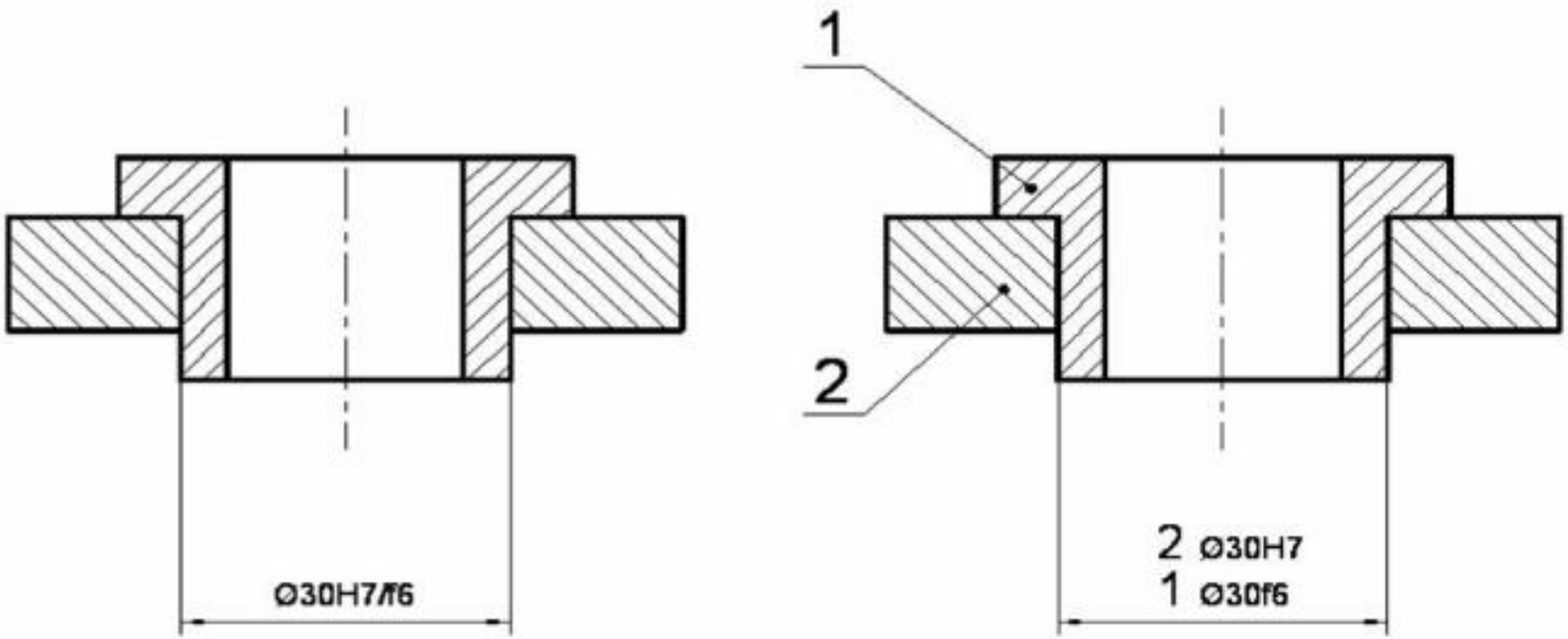
Indicating tolerances

The dimension is given by:

- a shape symbol,
- nominal size,
- a letter indicating the position of the tolerance zone in relation to zero line,
- a number indicating the width of the tolerance zone.



Specifying Fits in technical Drawing



Grades of Fundamental Deviation

For shafts 'a' to 'g' the Deviation is below the zero line and for shafts 'j' to 'zc' it is above the zero line

For shafts 'A' to 'G' the Deviation is above the zero line and for shafts 'J' to 'ZC' it is Below the zero line

For Shafts,

Upper deviation = 'es'

Lower deviation = 'ei'

For Holes,

Upper deviation = 'ES'

Lower deviation = 'EI'

Shafts			Holes			Formulae for deviation in µ
Type	**FD**	**Sign**	**Type**	**FD**	**Sign**	**For D in mm**
d	es	-	D	EI	+	16 D^0.44
e	es	-	E	EI	+	11 D^0.41
f	es	-	F	EI	+	5.5 D^0.41
g	es	-	G	EI	+	2.5 D^0.34
h	es	-	H	EI	+	0
js	ei	-	JS	ES	+	0.5 D^0.31
k	ei	+	K	ES	-	0
m	ei	+	M	ES	-	0.024D+12.6
n	ei	+	N	ES	-	0.04D+21
p	ei	+	P	ES	-	0.072D+37.8
r	ei	+	R	ES	-	GM bet p & s or P & S
s	ei	+	S	ES	-	IT7 + 0.4D
t	ei	+	T	ES	-	IT7 + 0.63D
u	ei	+	U	ES	-	IT7 + D

Grades of Tolerance

Grade of Tolerance: It is an indication of the level of accuracy. There are 18 grades of tolerances – IT01, IT0, IT1 to IT16

IT01 to IT4 - For 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

Tol Grades	IT5	IT6	IT7	IT8	IT9	IT10	IT11	IT12	IT13	IT14	IT15	IT16
Values	7i	10i	16i	25i	40i	64i	100i	160i	250i	400i	640i	1000i

Tolerance Grade	Class of work
01, 0, 1	Gauge Block
2	High quality Gauges, Plug Gauges
3	Good quality Gauges, Gap Gauges
4	Gauges, Precise Fit produced byLapping
5	Ball Bearings, Machine Lapping, Fine boring and grinding
6	Fine boring and grinding
7	High Quality Turning, Broaching, Boring
8	Centre Lathe Turning and Boring, Reaming, Capstan lathes in good condition

Tolerance Grade	Class of work
9	Capstan or automatic lathes, Boring machines
10	Milling, Slooting, Planing, Rolling, Extrusion
11	Drilling, Rough Turning and boring, Precision tube drawing
12	Light press work, tube drawing
13	Press work, Tube Rolling
14	Die Casting or Molding, rubber molding
15	Stamping
16	Sand Casting, Flame Cutting

Standard Tolerance: Various grades of tolerances are defined using the 'standard tolerance unit', (i) in µm, which is a function of basic size [3].

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

where, D *(mm)* is the geometric mean of the lower and upper diameters of a particular diameter step within which the chosen the diameter D lies.

1-3, 3-6, 6-10, 10-18, 18-30, 30-50, 50-80, 80-120, 120-180, 180-250, 250-315, 315-400 and 400-500 mm

Selection of Fits

Clearance Fits (Hole Basis System):

Shafts	Grades	Description of fit	Application
a, b, c	11	Very large clearance	Generally not used
d	8, 9, 10	Loose running	Loose pulleys
e	7, 8, 9	Loose clearance	Electric motor bearings, heavily loaded bearing
f	6, 7, 8	Normal running	Lubricated bearings (with oil or grease), pumps and smaller motors, gear boxes
g	5, 6	Precision running	Lightly loaded shafts, sliding spools, accurate bearings
h	5 to 11	Extreme clearance (preferably for non-running parts)	Sockets and spigots of joints

Preferred Clearance fits (in practice) [1] : H11/c11, H9/d9, H8/f7, H7/g6 (Guide Fit), H7/h6, C11/h11, D9/h9, F8/h7, G7/h6

Selection of Fits

Transition Fits (Hole Basis System):

Shafts	Grades	Description of fit	Application
js	5, 6, 7	Slight clearance to slight interference	Very accurate location, couplings, spigots, gears,
k	5, 6, 7	No clearance to little clearance	Precision joints likely to be subjected to vibrations
m	5, 6, 7	Slight interference (on average)	Forced assembly is required
n	5, 6, 7	Slight interference and very little clearance	Semi-permanent or tight fit assemblies

Preferred Transition fits [1]: H7/k6, H7/n6, K7/h6, N7/h6

Selection of Fits

Interference Fits (Hole Basis System):

Shafts	Grades	Description of fit	Application
p	6, 7, 8,	True interference (light)	Fixing bushes, standard press fit
r	5, 6, 7	Interference (but can be dismantled)	Tight press fit. Keys in key ways
s	5, 6, 7	Semi permanent/ permanent fit	Valve seating, collars on shafts
t, u	----	High degree of interference	Permanent assemblies

Preferred Interference fits [1]: H7/p6 (Press fit), H7/s6, H7/u6, P7/h6, S7/h6, U7/h6