Limits, FITS AND TOLERANCES

Need of Limits, FITS AND TOLERANCES

- two parts can not be produced with identical measurements by any manufacturing process.
- In any production process, regardless of how well it is designed or how carefully it is maintained, a certain amount of natural variability will always exist.
- These natural variations are random in nature and are the cumulative effect of many small, essentially uncontrollable causes.
- Usually, variability arises from improperly adjusted machines, operator error, tool wear, and/or defective raw materials.
- No component can be manufactured precisely to a given dimension; it can only be made to lie between two limits, upper (maximum) and lower (minimum).
- The designer has to suggest these tolerance limits, which are acceptable for each of the dimensions used to define shape and form, and ensure satisfactory operation in service.

- 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.

Limits
The two extreme permissible sizes on the actual size are called limits.

Max. and mini. dimension of product .

Tolerance

The total variation permitted on the given dimension is called Tolerance. Or It is definedas the difference between a Higher limit and a Lower limit.
Tolerance = Higher Limit - Lower Limit.

Tolerance

Suppose we have to make a 10 mm nut for a 10 mm bolt. But due to human and machining error, the diameter inside it became 9.98mm. Therefore the nut will not fit in the bolt and our joint will be spoiled. So tolerance zones are used to avoid this error.

Classification of Tolerance

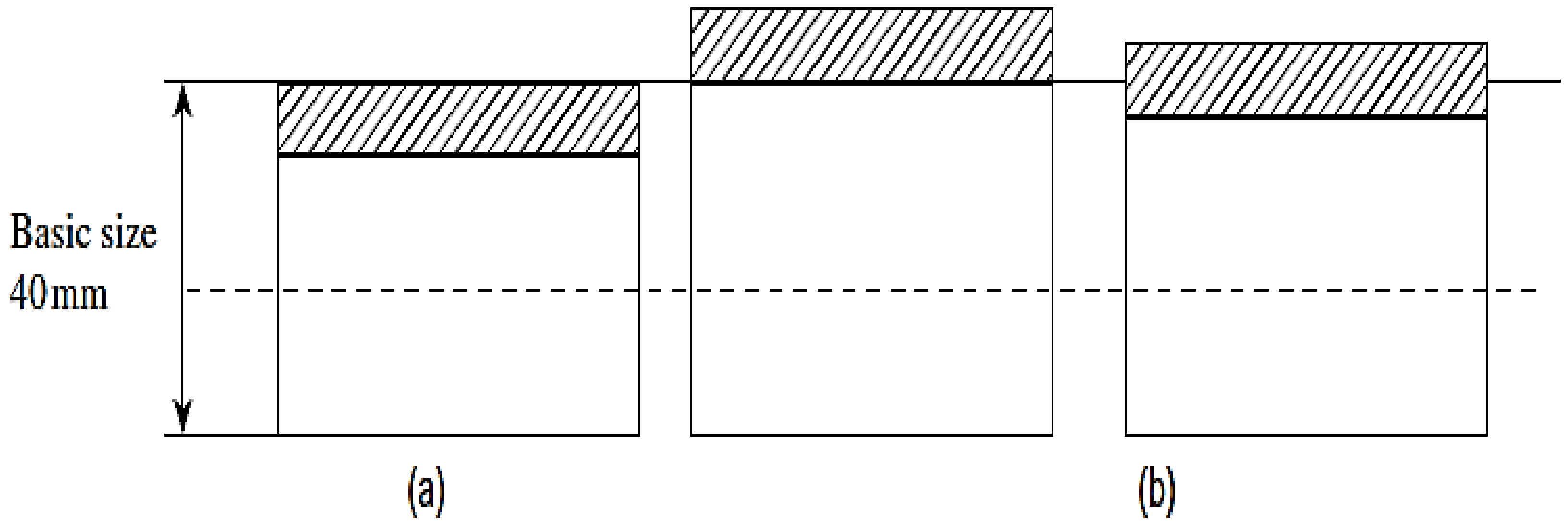
1. Unilateral tolerance
2. Bilateral tolerance

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.

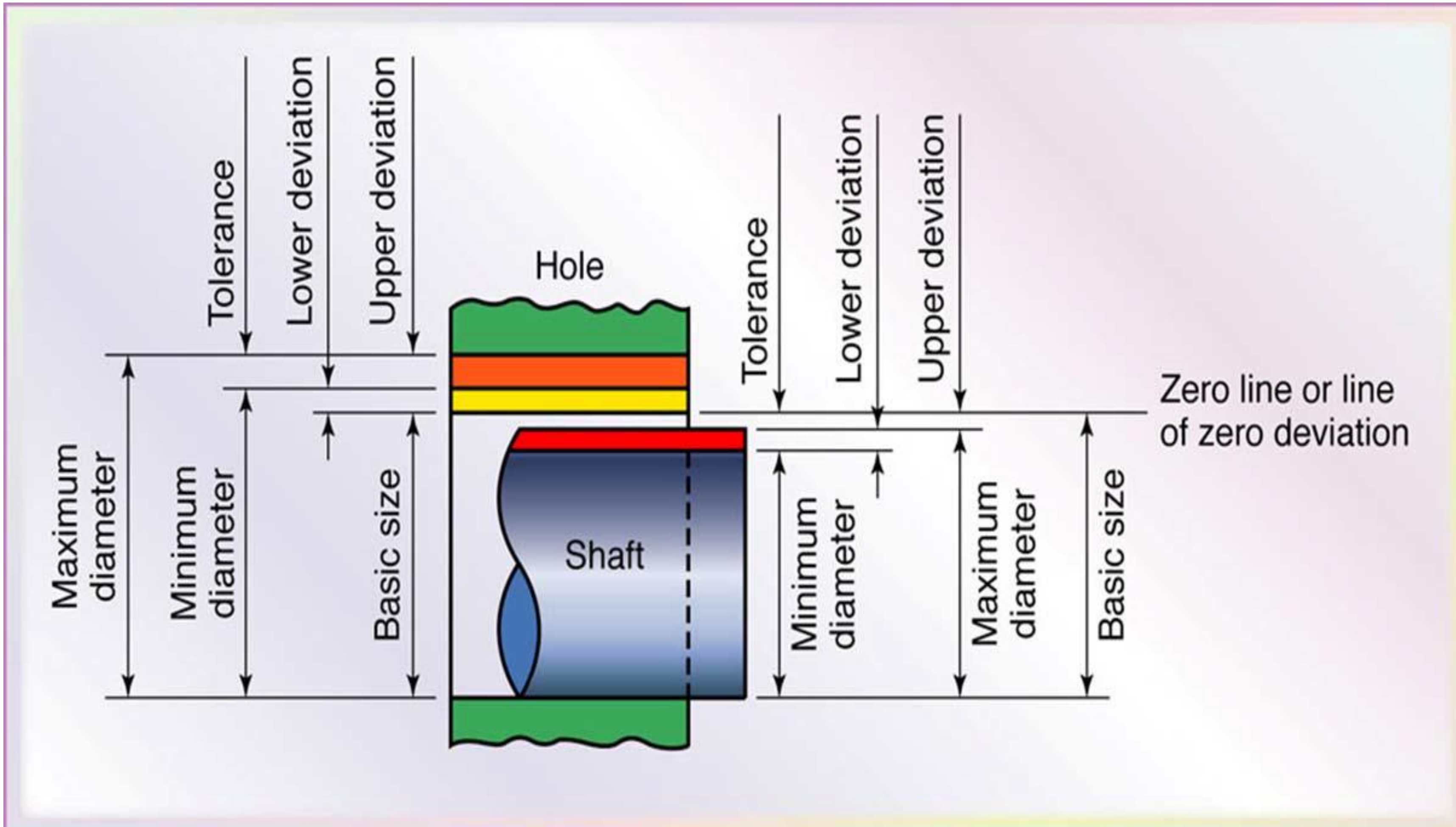
Bilateral Tolerance

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.



Tolerances


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

Basic Size:

It is the size based on which limits and tolerances of the component can be specified conveniently. If the nominal size itself is taken as round off value, the basic size and nominal size will be taken as the same.

Deviation:

The amount by which, the limit of a component is deviating from the basic size called a deviation.

The deviation can be an Upper(Higher) deviation and a Lower deviation.

Upper Deviation or Higher Deviation:

The difference between the upper limit and the basic size of a component is called the upper deviation.

Lower Deviation:

The difference between the lower limit and the basic size of a component is called the lower deviation.

Fundamental Deviation:

It is the deviation, either upper or lower deviation which is a fixed and conveniently chosen deviation.

Material limit

Least Material Condition denotes:

- Lower limit of the Shaft
- Upper limit of the Hole

Maximum Material Condition denotes:

- Lower limit of the Hole
- Upper limit of the Shaft

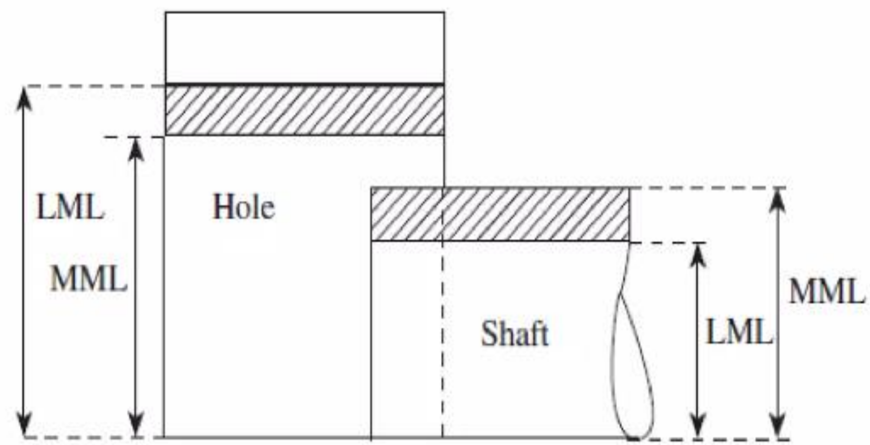
MAXIMUM AND MINIMUM METAL CONDITIONS

Consider a shaft having a dimension of 40 ± 0.05 mm and Hole having a dimension of 45 ± 0.05 mm.

For Shaft

Maximum metal limit (MML) = 40.05 mm

Least metal limit (LML) = 39.95 mm



For Hole

Maximum metal limit (MML) = 44.95 mm

Least metal limit (LML) = 45.05 mm

HOLE

Max Hole size – Basic Size = Upper Deviation

Min Hole size – Basic Size = Lower Deviation

SHAFT

Max shaft size – Basic Size = Upper Deviation

Min shaft size – Basic Size = Lower Deviation

FIT

FIT –

Fits are assembly conditions between Hole and Shaft.

Condition of looseness or tightness between two mating parts being assembled together.

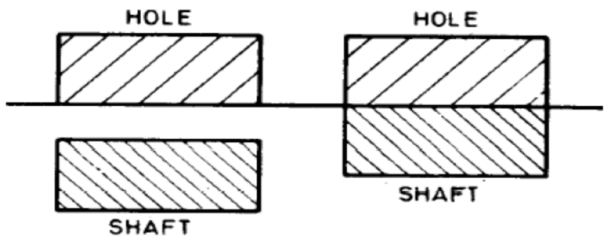
<u>Types of fits</u>

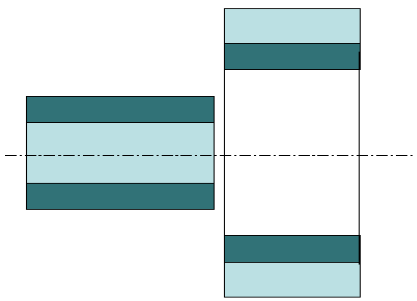
Clearance fit

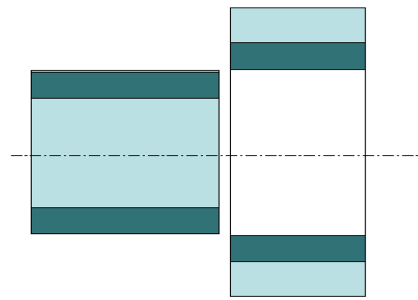
Interference fit

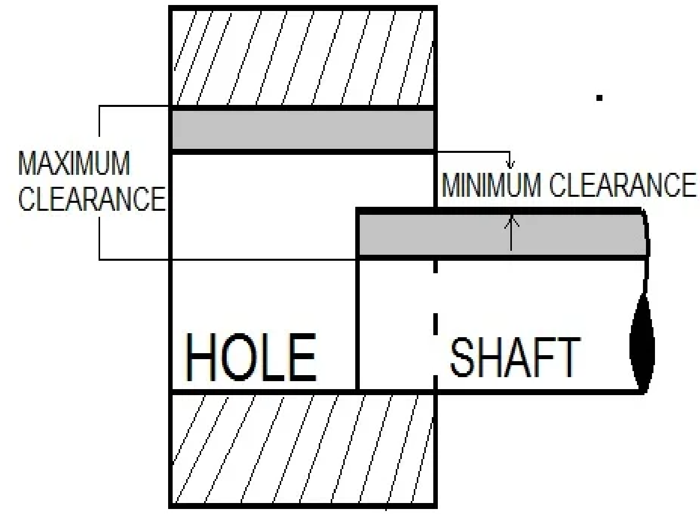
Transition fit

CLEARANCE FIT









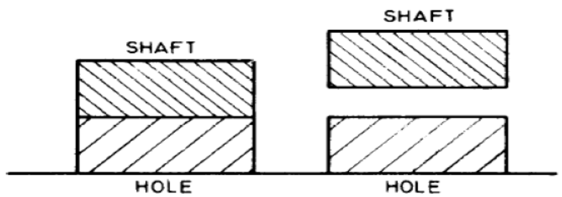
Maximum size of shaft < Minimum size of hole

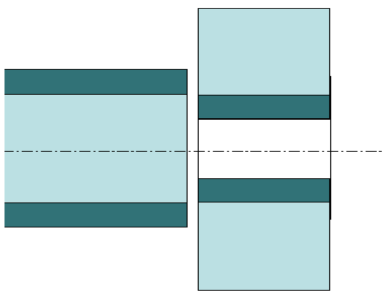
The clearance fit has a hole and a shaft in which there are upper and lower limits. Inside the hole along with these both come in the tolerance zone.

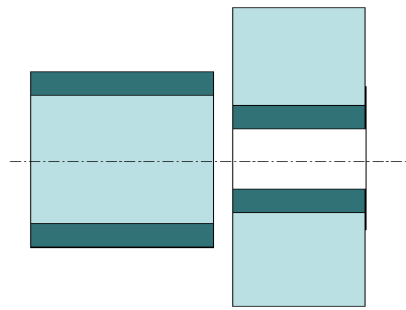
In this, the tolerance zone of the hole and the tolerance zone of the shaft, there is a lot of gap between them.

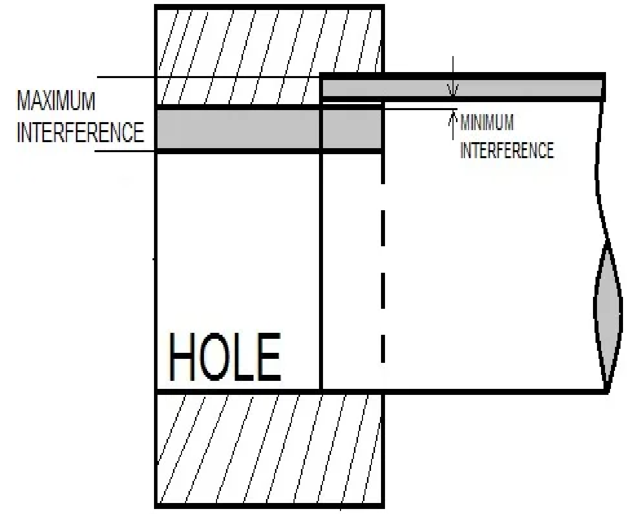
So if the hole is larger than the shaft and to allow the two mating parts to rotate or slide over each other, we call it clearance fit

INTERFERANCE FIT







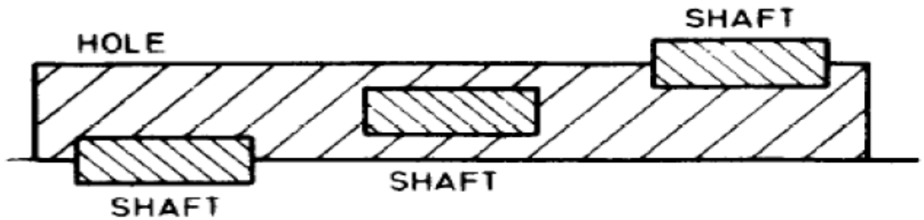


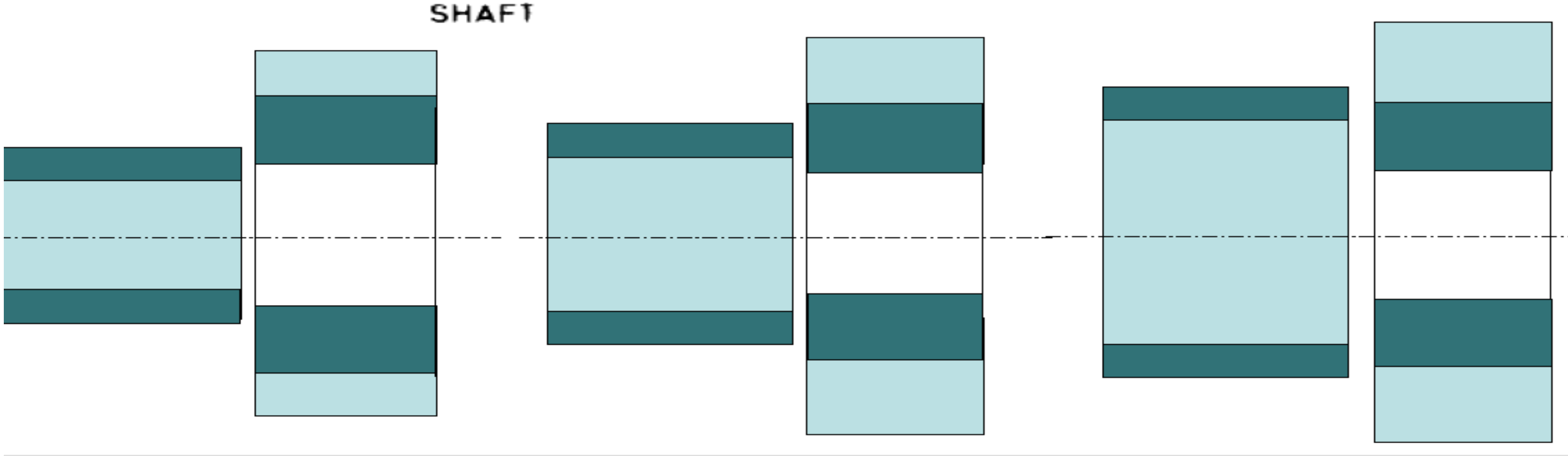
Minimum size of Shaft > Maximum size of Hole

In Interference Fit, the tolerance zone of the shaft goes above the tolerance zone of the hole. This means the shaft here is of large size, and the hole is of small size.

To make the fits between these two, we required high force to assemble and disassemble so generally we use a hammer. Another way is by using hydraulic press we can fit the shaft into the hole.
Another way is to heat the hole and put it in the shaft and let it cool, then it will shrink together and it will join each other.

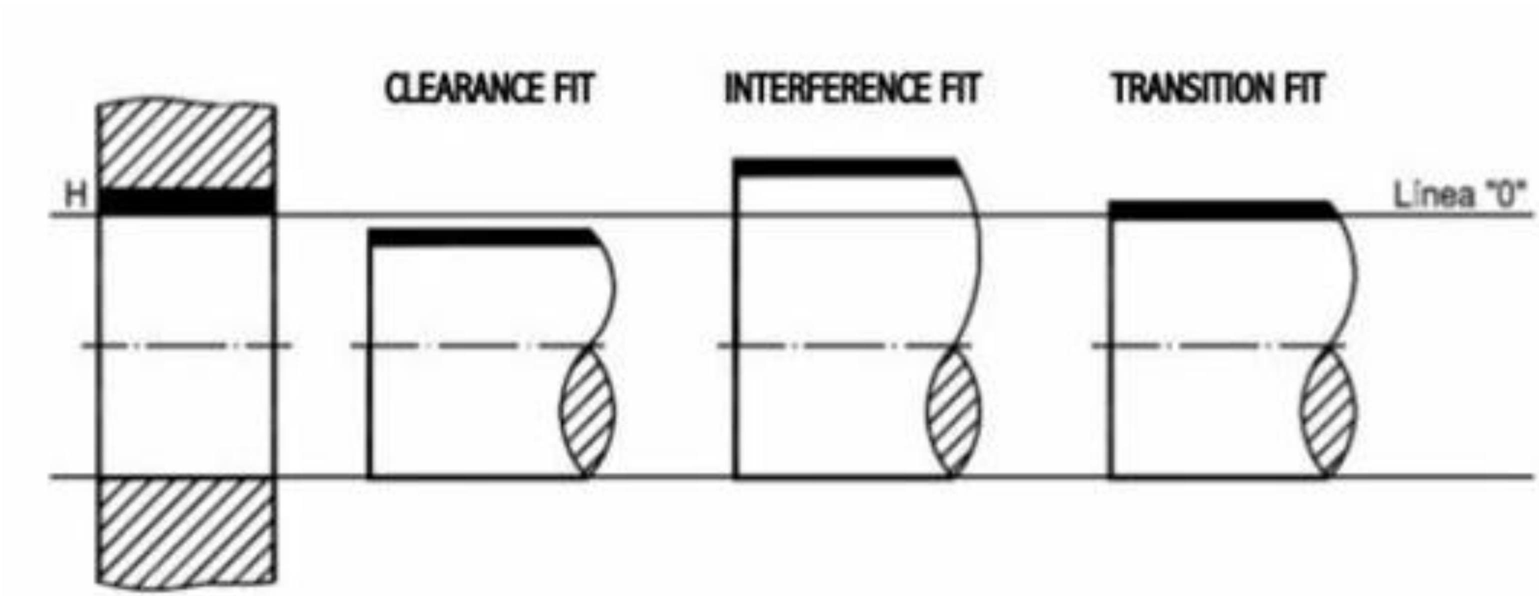
TRANSITION FIT





In transition fit, the tolerance zone of the shaft lies between the lower to the middle of the tolerance zone of the hole means here hole is smaller than the shaft.

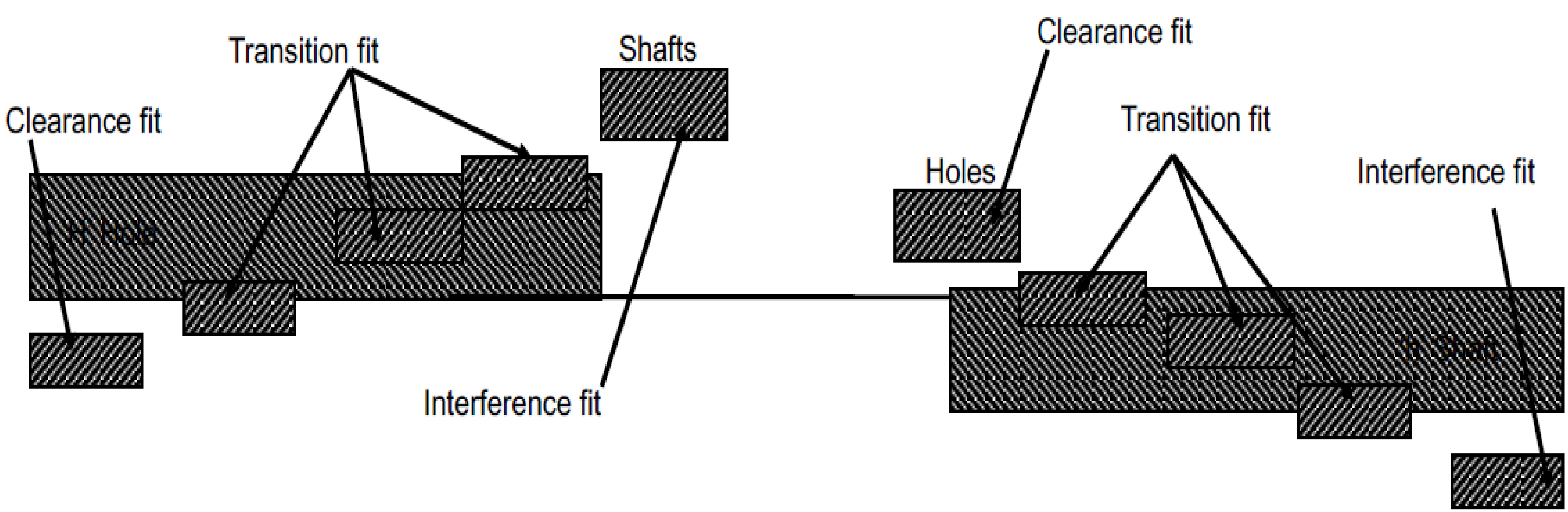
So to make this fit we have to make slight pressure on the shaft to go inside the hole. We also call it Push Fits. Transition fit has great precision and accurate alignment between two mating parts. Example: Shaft key.


CLEARANCE FIT
INTERFERENCE FIT
TRANSITION FIT
H
Línea "0"

Selection of Fits

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

1. Hole basis system
2. Shaft basis system



a. Hole Basis system

b. Shaft Basis system

Upper Deviation: The algebraic difference between the maximum limit of size (of either hole or shaft) and the corresponding basic size,

Lower Deviation: The algebraic difference between the minimum limit of size (of either hole or shaft) and the corresponding basic size,

Fundamental Deviation: It is one of the two deviations which is chosen to define the position of the tolerance zone.

Tolerance: The algebraic difference between upper and lower deviations. It is an absolute value.

Limits of Size: There are two permissible sizes for any particular dimension between which the actual size lies, maximum and minimum

Basic Shaft and Basic hole: The shafts and holes that have zero fundamental deviations. The basic hole has zero lower deviation whereas, the basic shaft has zero upper deviation.

Hole Based System:

In the make-to-suit assembly, if the hole is made first approximately near to the required dimension and the shaft is made slowly such that it can be assembled into the hole according to the required assembly conditions is called Hole Based System. According to the latest definition, if 'H' is used in the Assembly called Hole Based System.

Shaft Based System:

In the make suit assembly, If the shaft is made first approximately near to the required dimensions and the hole is enlarged slowly such that it can be assembled onto the shaft according to the required assembly conditions is called Shaft Based System.

Hole Based System, Hole 'H' = $50H_7d_8$

Shaft Based System, Shaft 'h' = $50F_5h_7$

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

Grades of Tolerance

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

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

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.

Diameter steps in I.S.I are: (a-b, where a is above and b is up to, Refer Table in the following sheet)

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

Grades of Tolerance

It is understood that the tolerances have parabolic relationship with the size of the products. As the size increases, the tolerance within which a part can be manufactured also increases.

IT01 – 0.3 + 0.008D

IT0 – 0.5 + 0.012 D

IT1 – 0.8 + 0.020D

IT2 to IT4 – the values of tolerance grades are placed geometrically between the tolerance grades of IT1 and IT5.

IT6 – 10 i; IT7 – 16i; IT8 – 25i; IT9 – 40i; IT10 – 64i; IT11 – 100i; IT12 – 160i; IT13 – 250i; IT14 – 400i; IT15 – 640i; IT16 – 1000i.