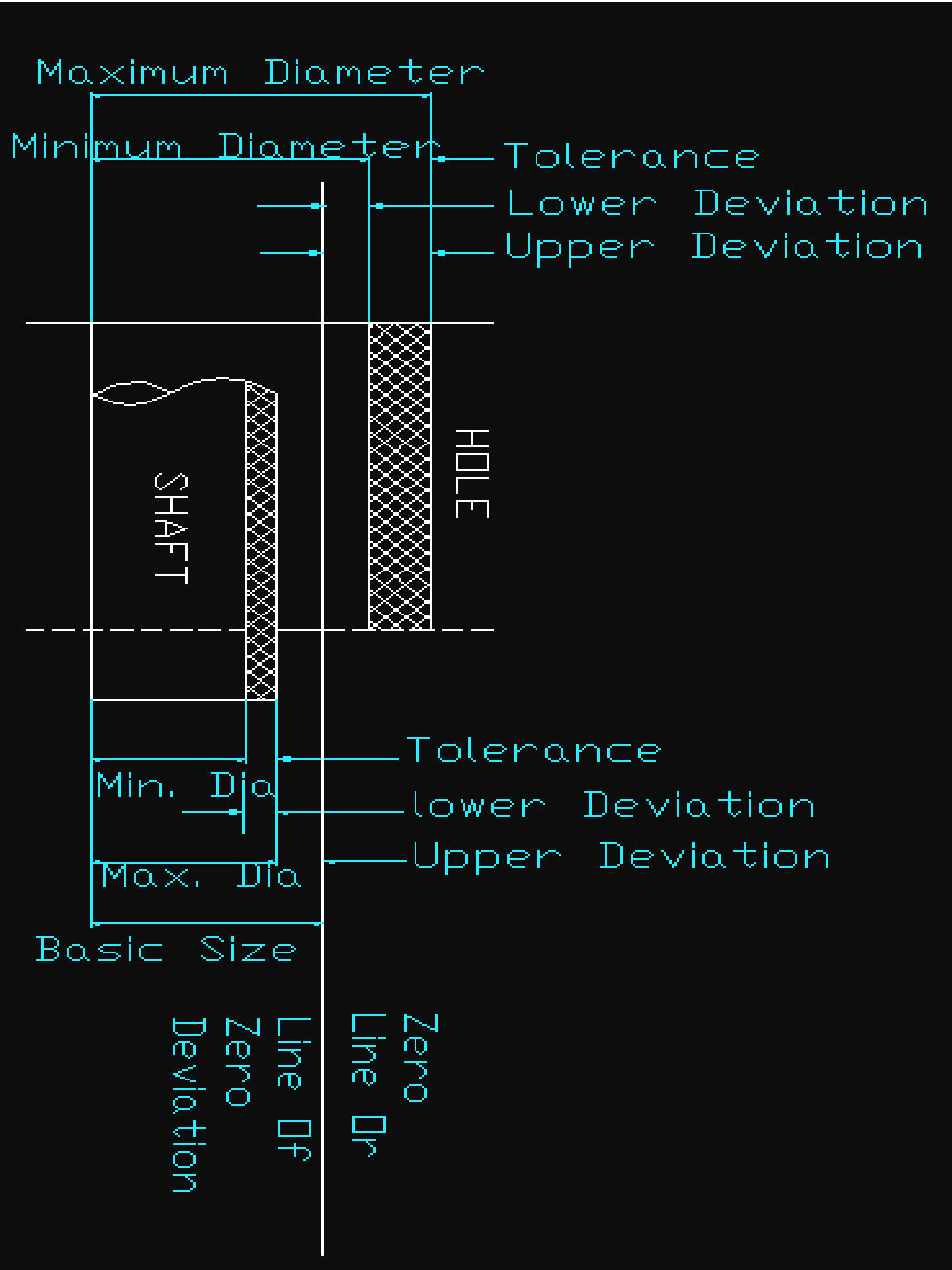
LIMITS FITS AND TOLERANCE

Prof. D.S.Shastri


Maximum Diameter
Minimum Diameter
Tolerance
Lower Deviation
Upper Deviation
SHAFT
HOLE
Tolerance
Min. Dia
lower Deviation
Upper Deviation
Max. Dia
Basic Size
Zero Line Or Deviation
Zero Line Or

LIMITS

- The concept of limits means essentially that a precisely defined basic condition (express by a numerical value or specification) is replaced by two limiting conditions.
- Any result lying between these two limits of size is acceptable.

- The difference between these two limits of size is called the permissible tolerance. One standard level is replaced by two limiting levels enclosing a zone of acceptance or tolerance.
- In this way, a workable scheme of interchangeable manufacture, to facilitate mass production method, has been established.

- Limits Definition: The two extreme permissible sizes between which the actual size is contained are called ***limits***.
- There are two limits dimensions, a maximum and a minimum in any basic or a design size.
- The largest permissible dimension is known as ***upper limit*** or ***upper tolerance limit***, while the lowest permissible dimension is called the ***lower limit*** or ***lower tolerance limit***.

UNILATERAL AND BILATERAL TOLERANCE

- When tolerance distribution is on only one side of basic size i.e. if one of the two tolerance limit is on zero line then it is referred to as **unilateral tolerance**.
- If permissible variation is both plus and minus it is referred to as a **bilateral tolerance**.

- ***Allowance:*** It is the dimensional difference between the maximum mating limits of mating parts, intentionally provided to obtain the desired degree or class of fit.
- If the allowance is positive, it will result in minimum clearance between the mating parts, and of the allowance is negative, it will result in maximum interference.

- ***Basic Size:*** The basic size of a part is determined solely from design calculations.
- If the strength and stiffness requirements demand a 50mm diameter shaft, then 50mm is the basic size. **OR**
- it is defined as the size of the component from which the limits of variations are determined.

- ***Design Size:*** The size from which the limits of a size are obtained by the application of tolerances in the design size. If there is no allowance, the design size then is the same as the basic size.

- ***Actual Size:*** This is the size of the finished part.

- ***Zero Line:*** A line drawn representing the basic size is called line of zero deviation or zero line.

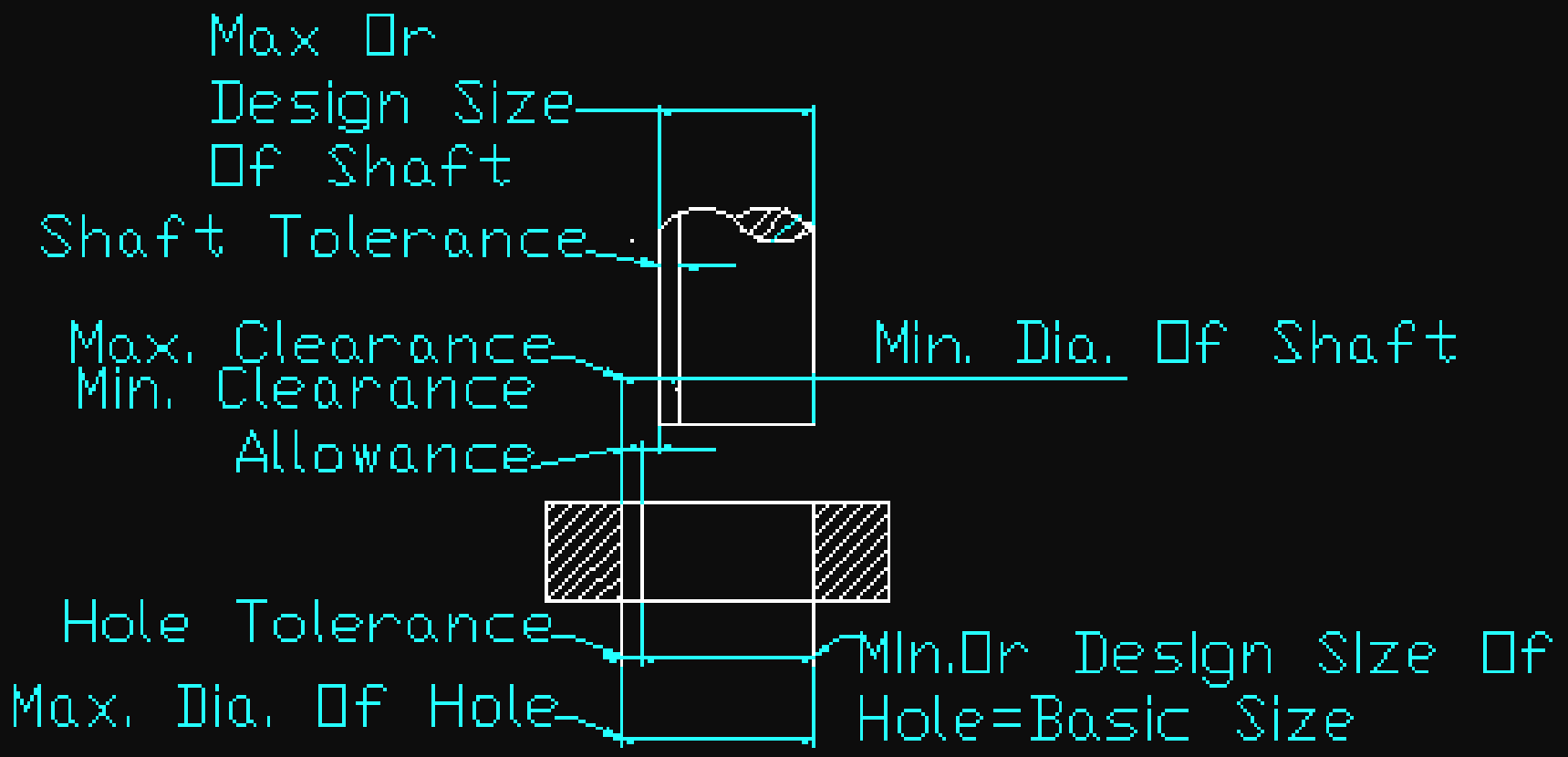
FITS

- Since even the simplest machine involves the fitting together of several parts for the purpose of design and production, it is necessary to know how the various parts fit together.
- A fit between two parts to be assembled can be defined as it is degree of tightness or looseness between two mating parts.

- ***FIT*** is the general term to signify the range of tightness or looseness resulting from the application of a specific combination of allowances and tolerances in the design of the mating parts.
- Fits are of three general types: **clearance**, ***interference***, *and* ***transition***, depending on the actual limits of the hole or shaft.

1) Clearance fits

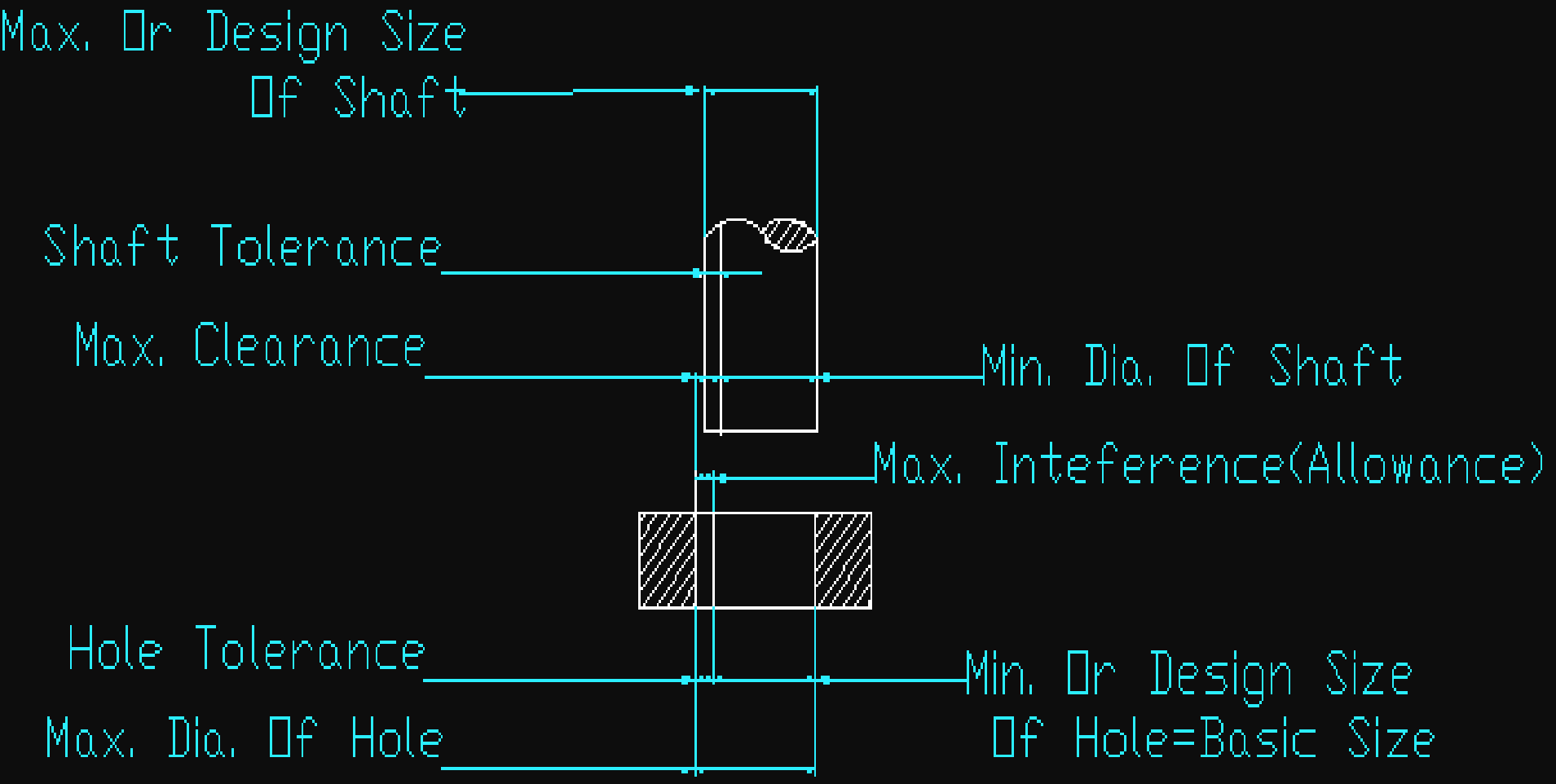
- In this, the difference between the hole and shaft sizes before assembly is positive.
- Clearance fits have limits of size prescribed such that a clearance always results when the mating parts are assembled.
- Clearance fits are intended for accurate assembly of parts and bearings .
- The parts can be assembled by hand because the hole is always larger than the shaft.



CLEARANCE FIT

2) Transition fits

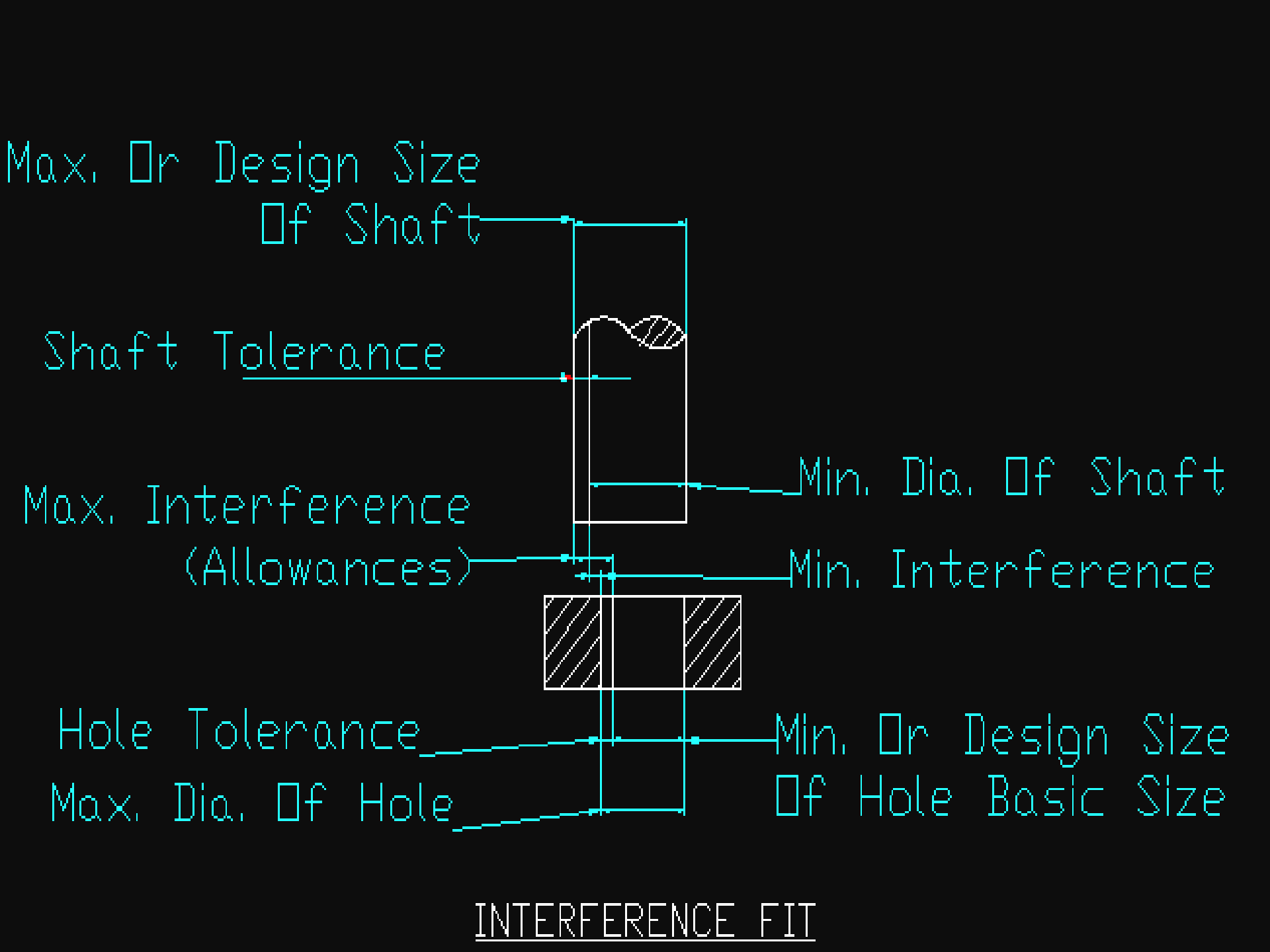
- This fit may provide either clearance or interference, depending on the actual value of the tolerance of individual parts .
- Transition fits are a compromise between the clearance and Interference fits .
- They are used for applications where accurate location is important, but either a small amount of clearance or interference is permissible.



TRANSITION FIT

3) Interference fit

- In this, the arithmetic difference between the hole and shaft sizes before assembly is negative.
- Interference fits have a limits of size prescribed that an interference always results when mating parts are assembled.
- The hole is always smaller than the shaft .
- Interference fits are for permanent assemblies of parts which require rigidity and alignment, such as dowel pins and bearings in casting.



INTERFERENCE FIT

Important definitions concerning to fits

1. **Clearance** In a fit, this is the difference between the sizes of the hole and the shaft, before assembly.

2. **Minimum clearance**: It is a clearance fit. It is the difference between the minimum size of the hole and the maximum size of the shaft.

3. **Maximum clearance**: In a clearance or transition fit, the difference between the maximum size of the hole and the minimum size of the shaft.

4. Minimum interference: In an interference fit, the arithmetical difference between the maximum size of the hole and the minimum size of the shaft.

5. Maximum interference: In an interference or transition fit the arithmetical difference between the minimum size of the hole and the maximum size of the shaft before assembly.

6. Hole basis system: It is a system of fits each of which has a basic hole.

7. Shaft basis system: It is a system of fits each of which has a basic shaft

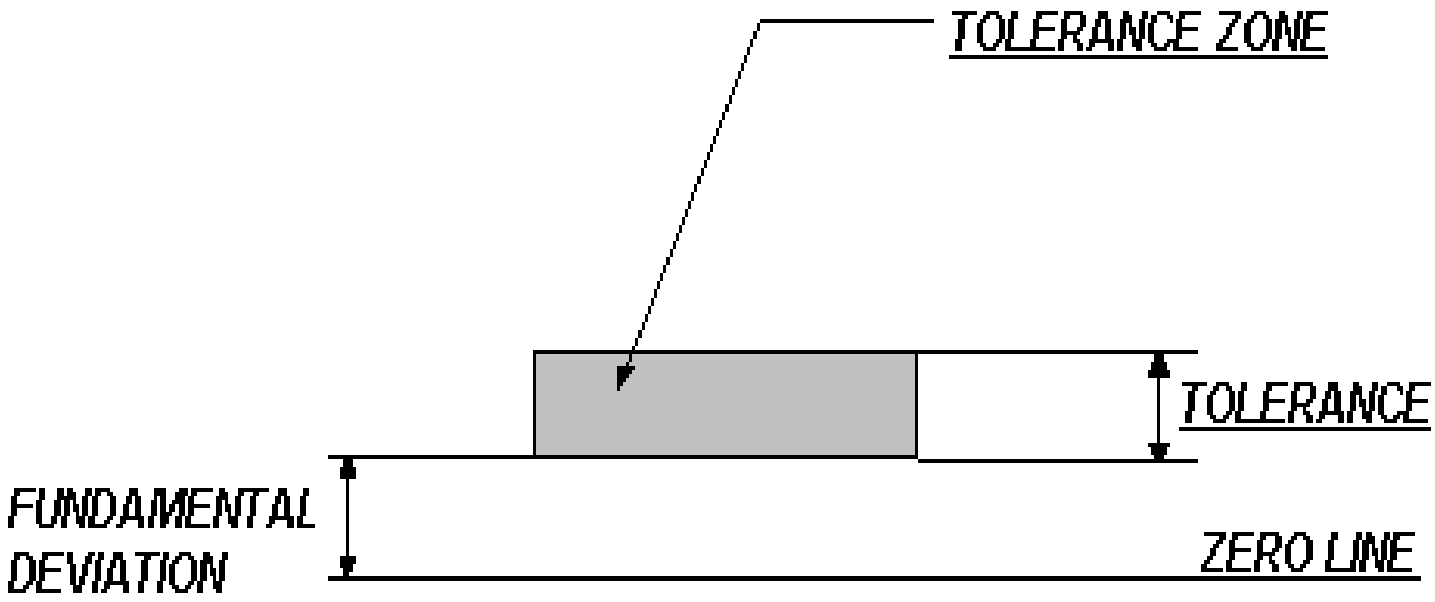
TOLERANCE

- The permissible variations of size is called the tolerance .
- It is the difference between the maximum and minimum permissible limits of the given size.
- Tolerance is denoted by two symbols – a number symbol called the grade and a letter symbol (a capital letter being used for holes and small letter for shafts).

TERMS RELATED TO TOLERANCE

- **Tolerance zone**: in a graphical representation of tolerance, it is the zone bounded by two limits of size of the part. It is defined by its magnitude and position in relation to the zero line.

- **Standard Tolerance**: it is the tolerance belonging to any one of the standard grades.


TOLERANCE ZONE
TOLERANCE
FUNDAMENTAL
DEVIATION
ZERO LINE

- Grade of tolerance In a standard system of limits and fits, groups of tolerances are considered as corresponding to the same level of accuracy for all basic sizes.
- It is the name given to one standard series of tolerances calculated according to certain law in terms of the basic size. *Tolerance grade* is an indication of the magnitude of tolerance. The lower the grade, finer will be the tolerance.

- A *unilateral tolerance* is one which applies in one direction from the specified size. So the permissible variation in the other direction is zero. When a unilateral tolerance is specified on the drawing, the plus or minus sign is not shown with the zero .A *bilateral tolerance* is a tolerance which is expressed as plus and minus values. These values need not be of the same size.