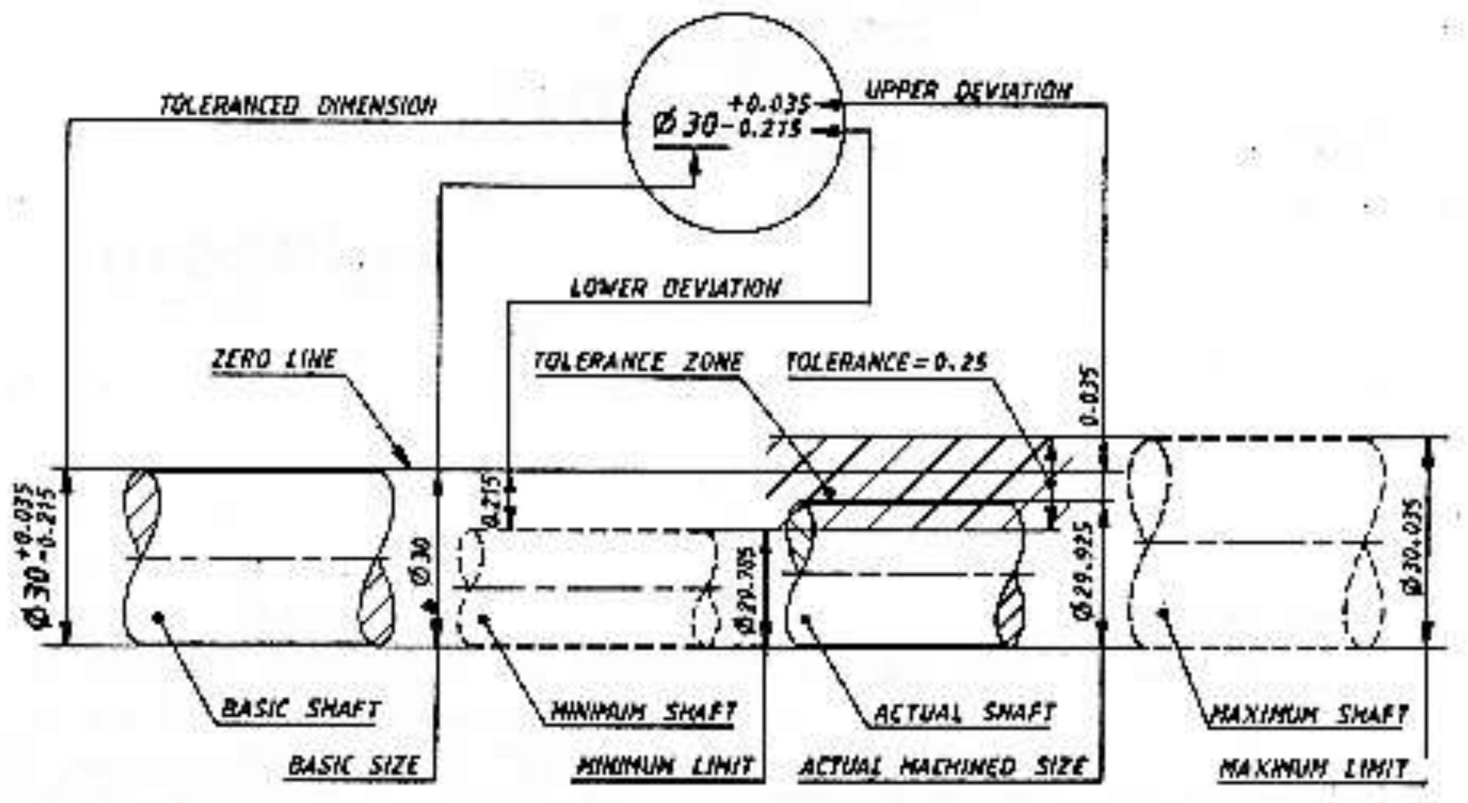
Case Study- Limits, Fits and Tolerances

Introduction

A machine is an assembly of innumerable number of parts of which some such as bolts, nuts, screws, Bearings, gear, chains, sprockets, couplings, etc., are the standard parts and the remaining are specially. Designed for a particular machine.

Definitions



All Dimensions in mm

Illustration of Basic Size Deviations, Limits and Tolerances

Definitions

Basic Size – It is defined as the theoretical size of a part, derived from the design after rounding off to the nearest whole millimeter.

Actual Size –It is defined as the size actually obtained by machining. It is found by actual measurement using measuring instruments.

Limits – The two extreme permissible sizes between which the actual size lies are called limits.

Definitions

Maximum Limit - It is defined as the maximum permissible size for a given basic size.

Minimum Limit – It is defined as the minimum permissible size for a given basic size.

Tolerance – It is defined as the amount of variation permitted of size. The difference between the Maximum and minimum limits of a basic size is called tolerance.

Definitions

Deviation – It is defined as the difference between the actual size or limit sizes, either maximum or minimum, and the corresponding basic sizes.

Actual Deviation – It is the algebraic difference between the actual measured size.

Upper Deviation – It is defined as the algebraic difference between the maximum limit of size and the corresponding basic size.

Lower Deviation – It is defined as the algebraic difference between the minimum limit of size and the corresponding basic size.

Definitions

Zero Line – Since the deviation are measured from the basic size, to indicate the deviations graphically, the basic shaft, the minimum shaft, the actual shaft and the maximum shaft are aligned at eh bottom and a straight line. Called *zero line* is drawn through the top generator of the basic shaft.

Tolerance Zone – In the graphic representation of a tolerance, the zone bounded by the upper and lower limits of the basic size called tolerance zone.

Tolerance

Tolerance on a basic size is obtained based on its performance. To achieve the required tolerance, the manufacturing process which is capable of accomplishing the specified to tolerance economically has to be selected.

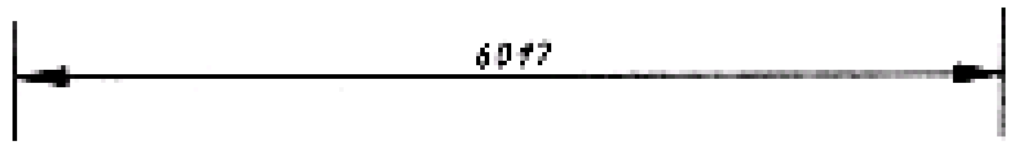
Methods of Indicating Tolerance on Drawing

The difference methods of indicating the tolerance is given below:

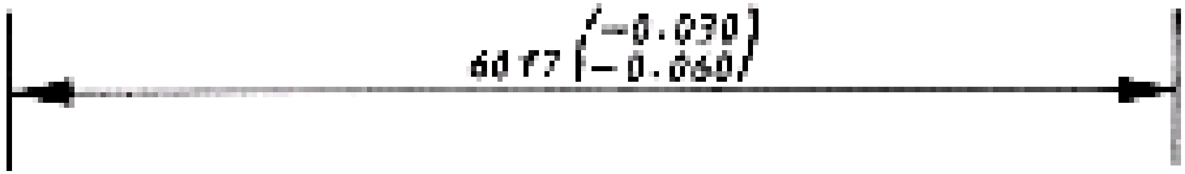
1. Basic size and the tolerance symbols
2. Basic size and the values of deviations
3. Limits of sizes

Methods of Indicating Tolerance on Drawing

- The toleranced dimension may be indicating using the toleranced symbol

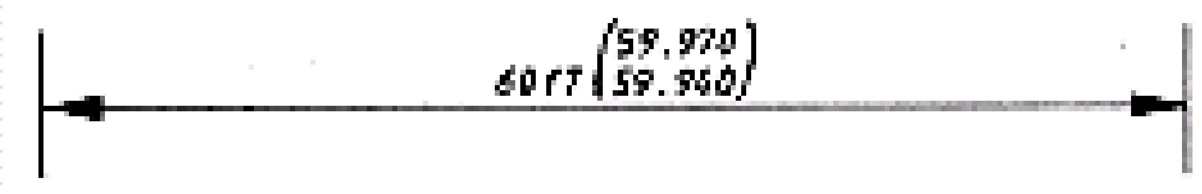


- If addition to the symbols, it is necessary to express the values of deviations.

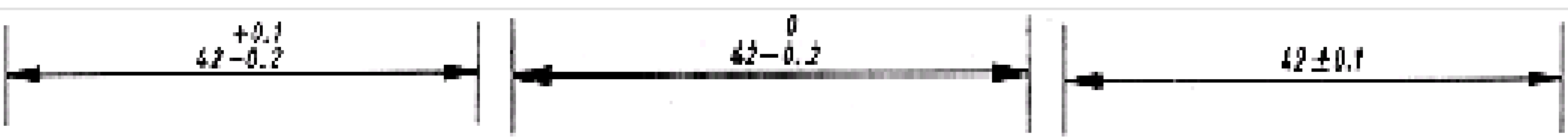


Methods of Indicating Tolerance on Drawing

- If addition to the symbols, it is necessary to express the limits of sizes.



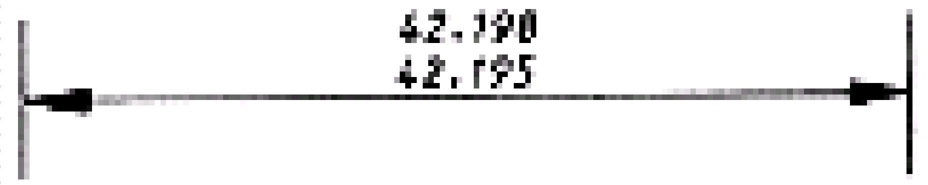
- When the toleranced dimension is to be indicated using the deviations, it may e shown any one of the methods



(a. Bilateral) (b. Unilateral) (c. Symmetrical)

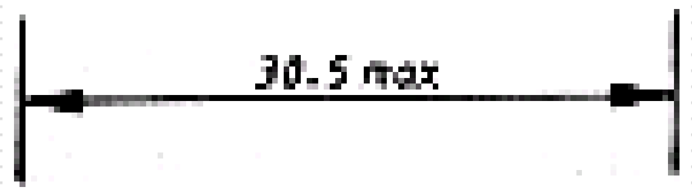
Methods of Indicating Tolerance on Drawing

- When the dimension is to be toleranced using the limit dimensions, they may be indicated directly.



- If a dimension is to be limited in one directio only, it must be indicating by adding "min" o "max" to the dimension.

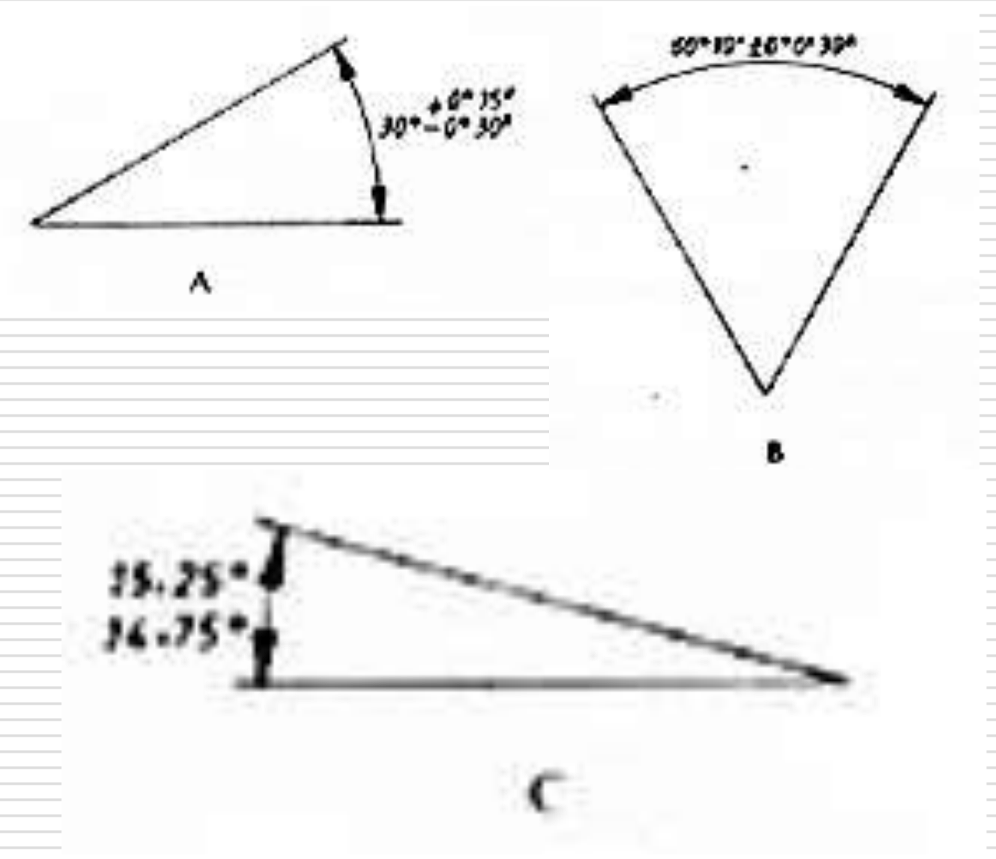
(a)



(b)

Indication of Tolerances on Angular Dimensions

The rules given for the indication of tolerances on liner dimensions are equally applicable. To Angular dimension except that the units of the basic angle and fraction there of, as well as the Deviations should be indicated as shown.



Machining Variations For Dimensions Without Tolerance Indication

All dimensions inscribed on a drawing of a machined part should in principle, be associated with the tolerance data which are normally indicated on the dimensioning line after the nominal Dimension

Deviations for Linear Dimensions

Class of Deviation		Range of Nominal Dimensions											
	Above	0.5	3	6	30	120	315	1000	2000	4000	8000	12000	16000
	Upto and Inculding	3	6	30	120	315	1000	2000	4000	8000	12000	16000	20000
Fine		± 0.05	± 0.05	± 0.1	± 0.15	± 0.2	± 0.3	± 0.5	± 0.8	-	-	-	-
Medium		± 0.1	± 0.1	± 0.2	± 0.3	± 0.5	± 0.8	± 1.2	± 2	± 3	± 4	± 5	± 6
Coarse		-	± 0.2	± 0.5	± 0.8	± 1.2	± 2	± 3	± 4	± 5	± 6	± 7	± 8
Extra coarse		-	± 0.5	± 1	± 1.5	± 2	± 3	± 4	± 6	± 8	± 10	± 12	± 12

All Dimension in mm

Machining Variations For Dimensions Without Tolerance Indication

Deviations for Radii and Chamfers

Class of Deviation	Range of Nominal Dimensions					
	Above	0.5	3	6	30	120
	Upto and including	3	6	30	120	315
Fine and medium		± 0.2	± 0.5	± 1	± 2	± 4
Coarse and extra coarse		± 0.2	± 1	± 2	± 4	± 8

Deviations for Angular Dimensions

Class of Deviation	Permissible Variations on Length of Shorter Side of Angle							
	Upto 10		Over 10 to 50		Over 50 to 120		Over 120	
	Degree	mm per 100 mm	Degree	mm per 100 mm	Degree	mm per 100 mm	Degree	mm per 100 mm
Fine and medium	± 1°	± 1.8	± 30′	± 0.9	± 20	± 0.6	± 10′	± 0.3
Coarse	± 1°30′	± 2.6	± 50′	± 1.5	± 25	± 0.7	± 15′	± 0.4
Extra coarse	± 3°	± 5.1	± 2°	± 3.5	± 1°	± 1.8	± 30′	± 0.9

All Dimension in mm

Fits

A machine is built by assembling all its constituting parts. During assembling sometimes a part may be required to be fitted into another part.

Shaft and Hole – Terminology

In mechanical engineering practice, generally a rod of circular cross section and a circular hole are termed as shaft and hole respectively.

Clearance

It is defined as the difference between the dimensions of the hole and the shaft assigned intentionally to obtain a particular type of a fit.

Classification of Fits:

Fit is established when one part is inserted into the other. The type of fit obtained between The two parts is governed by the dimensional deviations assigned for the basic size of the shaft and the hole

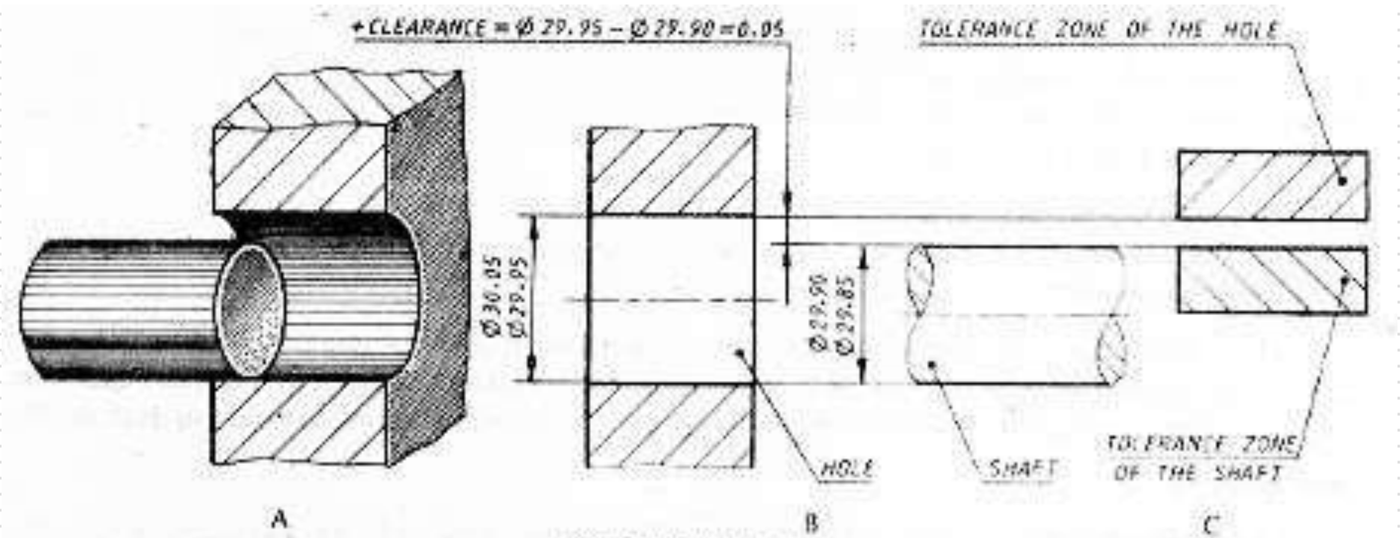
Types of Fit

1. Clearance Fit
2. Interference Fit
3. Transitions Fit

Types of Fit

1. Clearance Fit:

It is defined as the fit established, when a positive clearance exists between the hole and the shaft.



All Dimensions in mm

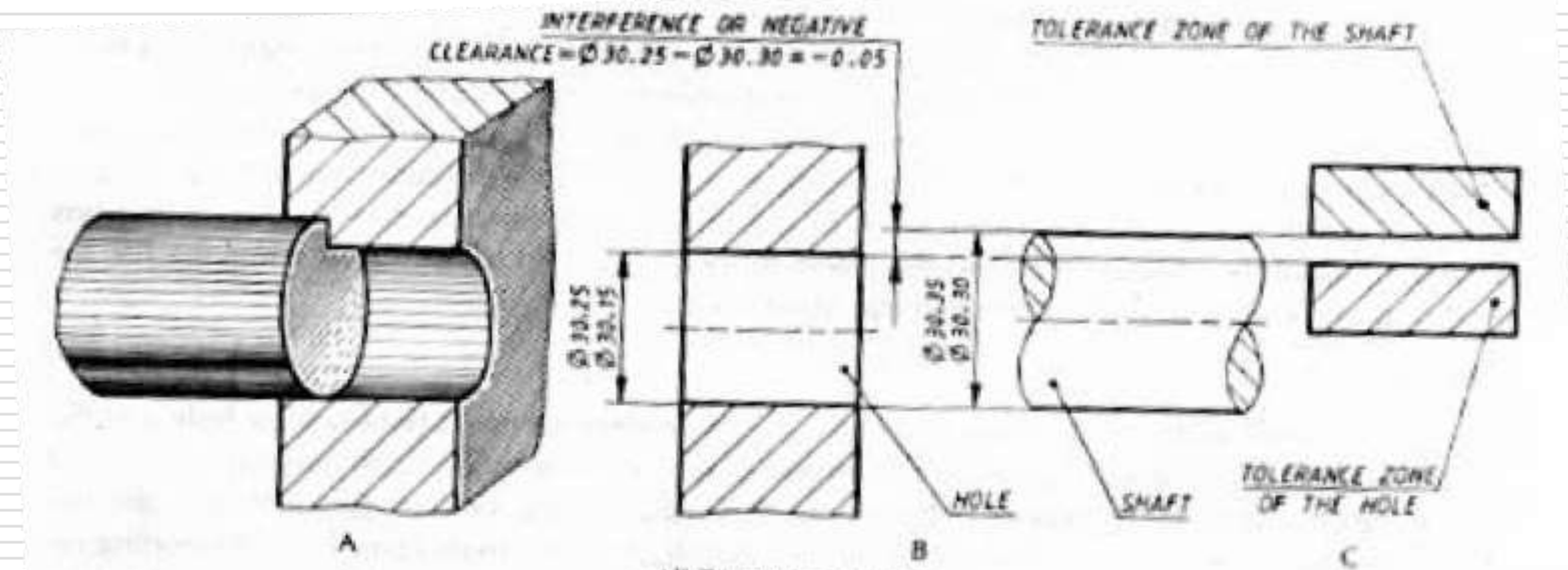
Clearance Fit

Fig.7.12

Types of Fit

2. Interference Fit:

It is defined as the fit established when a negative clearance exist between the sizes of the hole and the shaft.



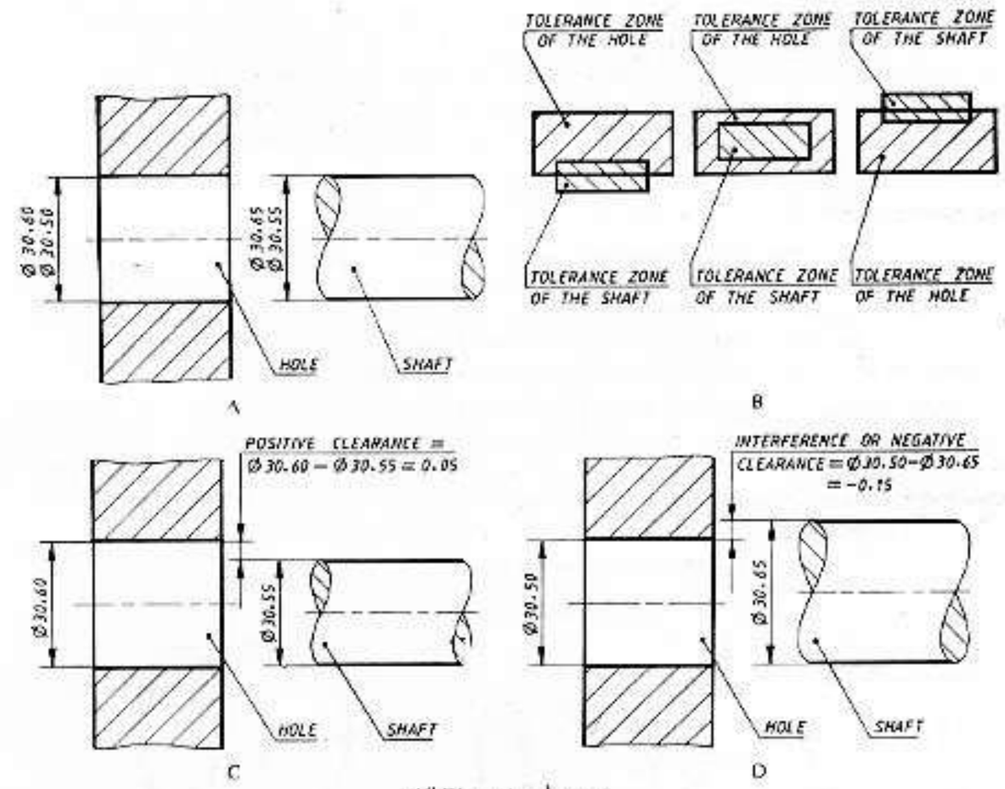
All Dimensions in mm

Interference Fit

Types of Fit

3. Transition Fit:

It is defined as the fit established when the dimensions of the hole and the shaft are such that there exists a *positive clearance* or a *negative clearance* when the shaft is finished into the hole.



All Dimensions in mm
Transition Fit
Fig.7.14