Brief overview of design and manufacturing

Instructional Objectives:

At the end of this lesson, the students should be able to understand:

- Concept of limits and fits
- Preferred numbers
- Various manufacturing processes

1.3.1 Design and Manufacturing

A machine element, after design, requires to be manufactured to give it a shape of a product. Therefore, in addition to standard design practices like, selection of proper material, ensuring proper strength and dimension to guard against failure, a designer should have knowledge of basic manufacturing aspects.

In this lesson, we will discuss briefly about some of the basic manufacturing requirements and processes.

First and foremost is assigning proper size to a machine element from manufacturing view point. As for example, a shaft may be designed to diameter of, say, 40 mm. This means, the nominal diameter of the shaft is 40 mm, but the actual size will be slightly different, because it is impossible to manufacture a shaft of exactly 40 mm diameter, no matter what machine is used. In case the machine element is a mating part with another one, then dimensions of both the parts become important, because they dictate the nature of assembly. The allowable variation in size for the mating parts is called limits and the nature of assembly due to such variation in size is known as fits.

1.3.2 Limits

Fig. 1.3.1 explains the terminologies used in defining tolerance and limit. The zero line, shown in the figure, is the basic size or the nominal size. The definition of the terminologies is given below. For the convenience, shaft and hole are chosen to be two mating components.

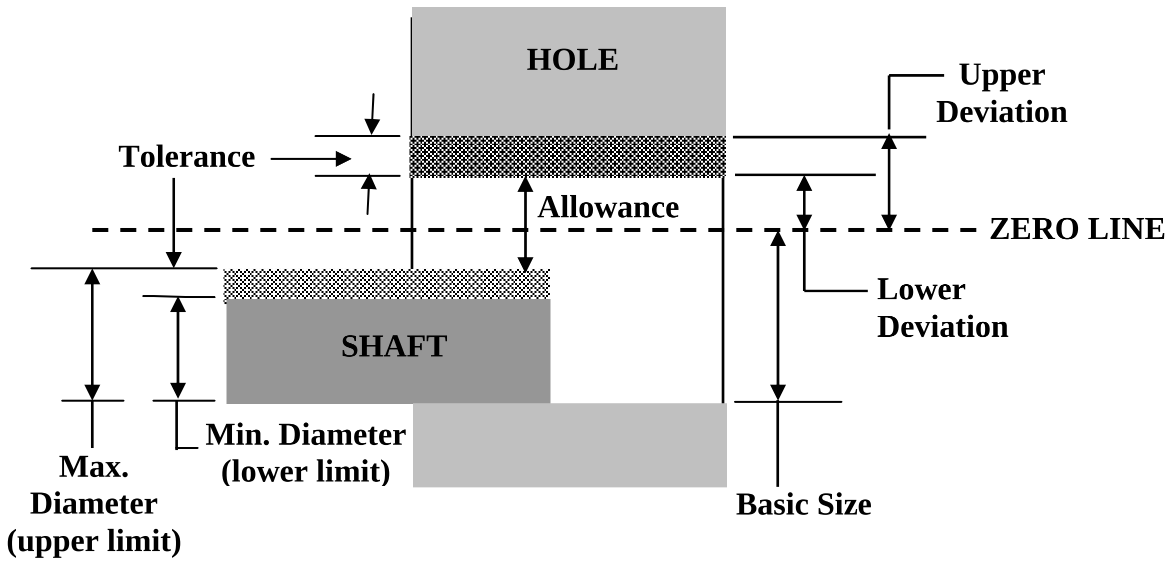


Fig. 1.3.1 Interrelationship between tolerances and limits

Tolerance

Tolerance is the difference between maximum and minimum dimensions of a component, ie, between upper limit and lower limit. Depending on the type of application, the permissible variation of dimension is set as per available standard grades.

Tolerance is of two types, bilateral and unilateral. When tolerance is present on both sides of nominal size, it is termed as bilateral; unilateral has tolerance only on one side. The Fig.1.3.2 shows the types of tolerance. 50^{0}_{-y}, 50^{+x}_{0} and 50^{+x}_{-y} is a typical example of specifying tolerance for a shaft of nominal diameter of 50mm. First two values denote unilateral tolerance and the third value denotes bilateral tolerance. Values of the tolerance are given as x and y respectively.

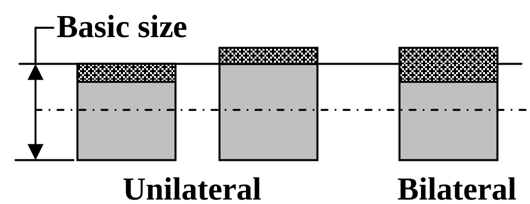


Fig. 1.3.2 Types of tolerance

Allowance

It is the difference of dimension between two mating parts.

Upper deviation

It is the difference of dimension between the maximum possible size of the component and its nominal size.

Lower deviation
Similarly, it is the difference of dimension between the minimum possible size of the component and its nominal size.

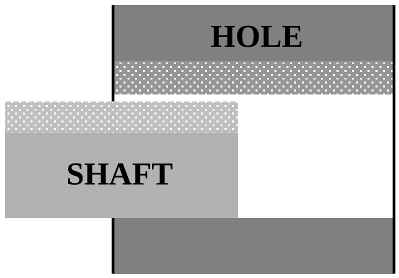
Fundamental deviation

It defines the location of the tolerance zone with respect to the nominal size. For that matter, either of the deviations may be considered.

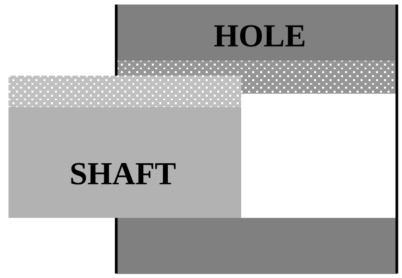
1.3.3 Fit System

We have learnt above that a machine part when manufactured has a specified tolerance. Therefore, when two mating parts fit with each other, the nature of fit is dependent on the limits of tolerances and fundamental deviations of the mating parts. The nature of assembly of two mating parts is defined by three types of fit system, Clearance Fit, Transition Fit and Interference Fit. The fit system is shown schematically in Fig.1.3.3.

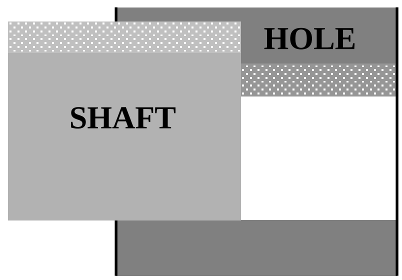
There are two ways of representing a system. One is the hole basis and the other is the shaft basis. In the hole basis system the dimension of the hole is considered to be the datum, whereas, in the shaft basis system dimension of the shaft is considered to be the datum. The holes are normally made by drilling, followed by reaming. Therefore, the dimension of a hole is fixed due to the nature of the tool used. On the contrary, the dimension of a shaft is easily controllable by standard manufacturing processes. For this reason, the hole basis system is much more popular than the shaft basis system. Here, we shall discuss fit system on hole basis.



Clearance fit



Transition fit



Interference fit

Fig. 1.3.3 Schematic view of Fit system

Clearance Fit

In this type of fit, the shaft of largest possible diameter can also be fitted easily even in the hole of smallest possible diameter.

Transition Fit

In this case, there will be a clearance between the minimum dimension of the shaft and the minimum dimension of the hole. If we look at the figure carefully, then it is observed that if the shaft dimension is maximum and the hole dimension is minimum then an overlap will result and this creates a certain amount of tightness in the fitting of the shaft inside the hole. Hence, transition fit may have either clearance or overlap in the fit.

Interference Fit

In this case, no matter whatever may be the tolerance level in shaft and the hole, there is always a overlapping of the matting parts. This is known as interference fit. Interference fit is a form of a tight fit.

1.3.4 Standard limit and fit system

Fig. 1.3.4 shows the schematic view of a standard limit and fit system. In this figure tolerance is denoted as IT and it has 18 grades; greater the number, more is the tolerance limit. The fundamental deviations for the hole are denoted by capital letters from A and ZC, having altogether 25 divisions. Similarly, the fundamental deviations for the shaft is denoted by small letters from a to zc.

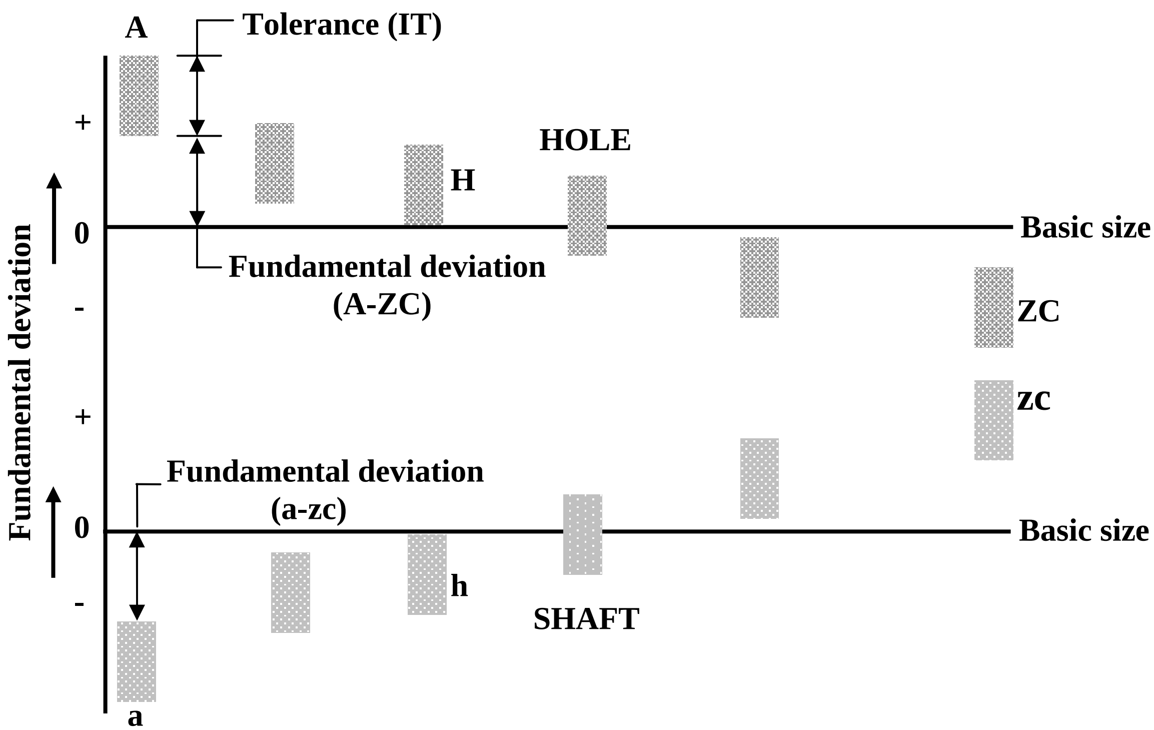


Fig. 1.3.4 Schematic view of standard limit and fit system

Here H or h is a typical case, where the fundamental deviation is zero having an unilateral tolerance of a specified IT grade.

Therefore in standard limits and fit system we find that,

<u>Standard tolerances</u>

18 grades: IT01 ,IT0 and IT1-1T16

Fundamental deviations

25 types: A- ZC (For holes)
a- zc (For shafts)

The values of standard tolerances and fundamental deviations can be obtained by consulting design hand book. It is to be noted that the choice of tolerance grade is related to the type of manufacturing process; for example, attainable tolerance grade for lapping process is lower compared to plain milling. Similarly, choice of fundamental deviation largely depends on the nature of fit, running fit or tight fit etc. The approximate zones for fit are shown in Fig. 1.3.5. Manufacturing processes involving lower tolerance grade are generally costly. Hence the designer has to keep in view the manufacturing processes to make the design effective and inexpensive.

Sample designation of limit and fit, 50H6/g5.

The designation means that the nominal size of the hole and the shaft is 50 mm. H is the nature of fit for the hole basis system and its fundamental deviation is zero. The tolerance grade for making the hole is IT6. Similarly, the shaft has the fit type g, for which the fundamental deviation is negative, that is, its dimension is lower than the nominal size, and tolerance grade is IT5.

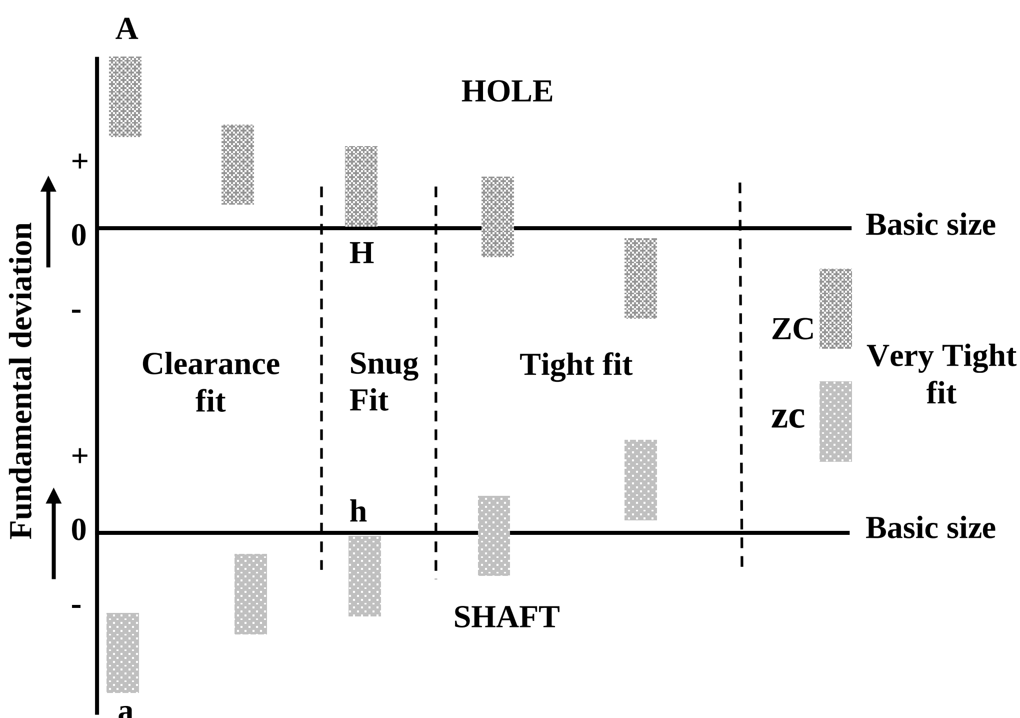


Fig. 1.3.5 Typical zones of fit

1.3.5 Preferred numbers

A designed product needs standardization. It means that some of its important specified parameter should be common in nature. For example, the sizes of the ingots available in the market have standard sizes. A manufacturer does not produce ingots of sizes of his wish, he follows a definite pattern and for that matter designer can choose the dimensions from those standard available sizes. Motor speed, engine power of a tractor, machine tool speed and feed, all follow a definite pattern or series. This also helps in interchangeability of products. It has been observed that if the sizes are put in the form of geometric progression, then wide ranges are covered with a definite sequence. These numbers are called preferred numbers having common ratios as,

$$\sqrt[5]{10} \approx 1.58,\ \sqrt[10]{10} \approx 1.26,\ \sqrt[20]{10} \approx 1.12 \ \text{ and } \ \sqrt[40]{10} \approx 1.06$$

Depending on the common ratio, four basic series are formed; these are R_5 , R_{10} , R_{20} and R_{40} . These are named as Renard series. Many other derived series are formed by multiplying or dividing the basic series by 10, 100 etc.

Typical values of the common ratio for four basic G.P. series are given below.

Preferred Numbers

R5:	$\sqrt[5]{10}$	***1.58*: 1.0, 1.6, 2.5, 4.0,…**
R10:	$\sqrt[10]{10}$	***1.26*: 1.0, 1.25, 1.6, 2.0,…**
R20:	$\sqrt[20]{10}$	***1.12*: 1.0, 1.12, 1.25, 1.4,…**
R40:	$\sqrt[40]{10}$	***1.06*: 1.0, 1.06, 1.12, 1.18,..**

Few examples

R_{10} , R_{20} and R_{40} : Thickness of sheet metals, wire diameter
R_5 , R_{10} , R_{20} : Speed layout in a machine tool (R_{10} : 1000, 1250,1600, 2000)
R_{20} or R_{40} : Machine tool feed
R_5 : Capacities of hydraulic cylinder

1.3.6 Common manufacturing processes

The types of common manufacturing processes are given below in the Fig.1.3.6.

Manufacturing processes

- **Shaping**
- **Joining**
- **Machining**
- **Surface finishing**
- **Non-conventional machining**
- **Heat treatment of the product**

Fig. 1.3.6 Common manufacturing processes

The types of shaping processes are given below in the Fig.1.3.7.

Shaping processes

- **Casting**
- **Forging**
- **Extruding**
- **Rolling**

Fig. 1.3.7 Shaping processes

Following are the type of machining processes, shown in Fig.1.3.8.

Machining

- **Turning**
- **Shaping**
- **Milling**
- **Drilling**

Fig. 1.3.8 Machining processes

Various joining processes are shown in Fig.1.3.9.

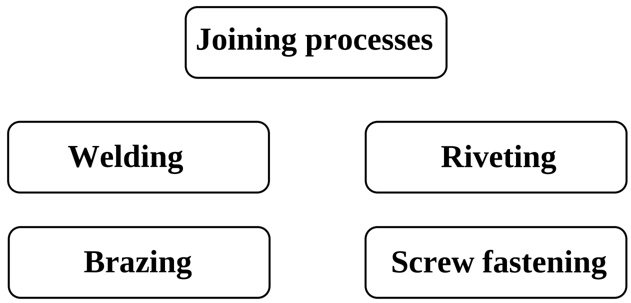


Fig. 1.3.9 Joining processes

The surface finishing processes are given below (Fig.1.3.10),

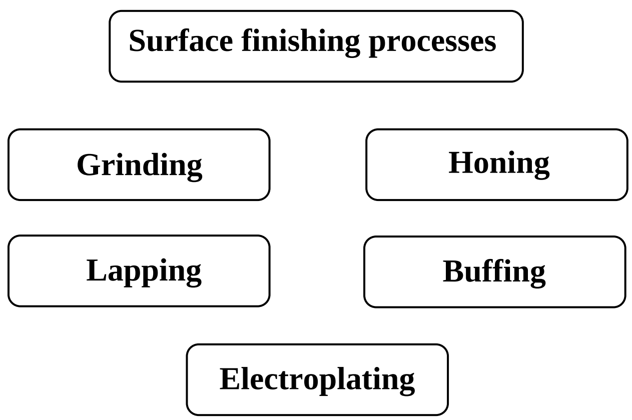


Fig. 1.3.10 Surface finishing processes

The non-conventional machining processes are as follows (Fig.1.3.11),

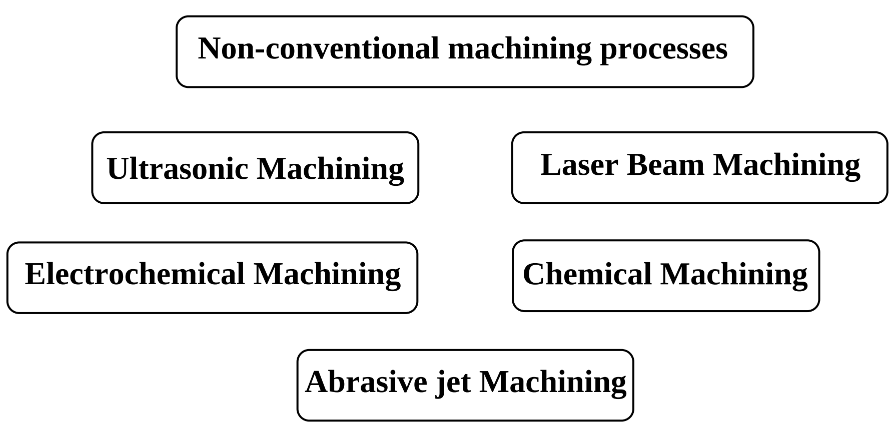


Fig. 1.3.11 Non conventional machining processes

Questions and answers

Q1. What is meant by tolerance? How many types of tolerance is there?

A1. Tolerance is the difference between maximum and minimum dimensions of a component, ie, between upper limit and lower limit. Depending on the type of application, the permissible variation of dimension is set as per available standard grades. Tolerance is of two types, bilateral and unilateral. When tolerance is present on both sides of nominal size, it is termed as bilateral; unilateral has tolerance only on one side.

Q2. What are the types fit? Describe the differences.

A2. The nature of assembly of two mating parts is defined by three types of fit system, Clearance Fit, Transition Fit and Interference Fit.

Clearance Fit: In this type of fit, the shaft of largest possible diameter can be fitted easily in the hole of smallest possible diameter.
Interference Fit : In this type of fit, irrespective of tolerance grade there is always a overlapping of the matting parts.
Transition Fit: In this case, a clearance is present between the minimum dimension of the shaft and the minimum dimension of the hole. However, the fit is tight, if the shaft dimension is maximum and the hole dimension is minimum. Hence, transition fit have both the characteristics of clearance fit and interference fit.

Q3. What are preferred numbers?

A3. Preferred numbers are the numbers belonging to four categories of geometric progression series, called basic series, having common ratio of,

$$\sqrt[5]{10} \approx 1.58,\ \sqrt[10]{10} \approx 1.26,\ \sqrt[20]{10} \approx 1.12 \text{ and } \sqrt[40]{10} \approx 1.06$$

Preferred numbers of derived series are formed by multiplying or dividing the basic series by 10, 100 etc. These numbers are used to build-up or manufacture a product range. The range of operational speeds of a machine or the range of powers of a typical machine may be also as per a series of preferred numbers.

References

1. J.E Shigley and C.R Mischke , Mechanical Engineering Design , McGraw Hill Publication, 5th Edition. 1989.
2. Khurmi, R.S. and Gupta J.K., Text book on Machine Design, Eurasia Publishing House, New Delhi.
3. Sharma, C.S. and Purohit Kamalesh, Design of Machine Elements, Prentice Hall of India, New Delhi, 2003.
4. Chapman, W.A.J., Workshop Technology (part 2), ELBS, 4th edition, 1975
5. Maitra, G.M., Handbook of Design, Tata McGraw Hill Publication, New Delhi, 1998.