

Mechanical Engineering 232

Limits and Fits, Tolerance Dimensioning

Definitions:

nominal size: The size designation used for general identification. The nominal size of a shaft and a hole are the same. This value is often expressed as a fraction.

basic size: The exact theoretical size of a part. This is the value from which limit dimensions are computed. Basic size is a four decimal place equivalent to the nominal size. The number of significant digits imply the accuracy of the dimension.

example: nominal size = 1 1/4
basic size = 1.2500

design size: The ideal size for each component (shaft and hole) based upon a selected fit. The difference between the design size of the shaft and the design size of the hole is equal to the allowance of the fit. The design size of a part corresponds to the Maximum Material Condition (MMC). That is, the largest shaft permitted by the limits and the smallest hole. Emphasis is placed upon the design size in the writing of the actual limit dimension, so the design size is placed in the top position of the pair.

tolerance: The total amount by which a dimension is allowed to vary. For fractional linear dimensions we have assumed a bilateral tolerance of 1/64 inch. For the fit of a shaft/hole combination, the tolerance is considered to be unilateral, that is, it is only applied in one direction from design size of the part. Standards for limits and fits state that tolerances are applied such that the hole size can only vary larger from design size and the shaft size smaller.

basic hole system: Most common system for limit dimensions. In this system the design size of the hole is taken to be equivalent to the basic size for the pair (see above). This means that the lower (in size) limit of the hole dimension is equal to design size. The basic hole system is more frequently used since most hole generating devices are of fixed size (for example, drills, reams, etc.) When designing using purchased components with fixed outer diameters (bearings, bushings, etc.) a basic shaft system may be used.

allowance: The allowance is the intended difference in the sizes of mating parts. This allowance may be: positive (indicated with a "+" symbol), which means there is intended clearance between parts; negative ("-"), for intentional interference; or "zero allowance" if the two parts are intended to be the "same size". This last case is common to selective assembly.

Example:

The figure attached (see separate sheet) shows an example of a limit dimension for the shaft in a shaft/hole assembly. The top number in the limit represents the design size of the shaft. The two values define a range of acceptable component sizes.

The drawing is for a shaft/hole assembly of 3/4 inch nominal diameter. A standard Class RC3 fit is being used. The specifications for the fit are found in the fit tables in the appendix of your text. By finding the appropriate range for the nominal size in the left hand column of the table and reading under the RC3 fit heading we obtain the following values.

<u>Hole</u>	<u>Shaft</u>
+ 0.8	- 0.8
+ 0.0	- 1.3

These values are applied algebraically to the basic size for the assembly in order to obtain the limit dimensions for each part. Note that the values are given in thousandths of an inch. Further note that one of the values for the hole is zero. This will make one limit for the hole the same as basic size, hence the basic hole system.

We now have the following:

nominal size =	3/4
basic size =	.7500
limit dimension for the hole =	.7500 .7508
limit dimension for the shaft =	.7492 .7487
design size for the hole =	.7500
design size for the shaft =	.7482

This means that for this standard fit, the allowance = +.0008. This is the difference between the design sizes of the parts, or, in other words, the largest shaft and the smallest hole. The allowance is positive since the shaft is always smaller than the hole and will assemble freely.

In most design situations the nominal size of the combination is chosen through engineering analysis of the problem. A standard fit is then selected based upon function. The fit tables include short descriptions of the applications for the specific fits. The limit dimensions for each part can be written using the previous procedure.

Unilateral tolerance:

This is an alternative form for the writing of fit dimensions. The information is the same as that for limit dimensions, just the format is different. In the unilateral tolerance format, the design size of the component is given and the tolerances on each part are listed in an indented position.

example:	<u>limits</u>	<u>unilateral tolerance</u>
	.7492	.7492
	.7487	+ .0000 - .0005

