GENERAL TOLERANCES – TOLERANCES FOR LINEAR AND ANGULAR DIMENSIONS WITHOUT INDIVIDUAL TOLERANCE INDICATIONS

0.0 GENERAL

When selecting the tolerance class, the respective customary workshop accuracy has to be taken into consideration. If smaller tolerances are required or larger tolerances are permissible and more economical for any individual feature, such tolerances should be indicated to the relevant nominal dimension(s).

General tolerance for linear and angular dimensions apply when drawings or associated specifications refer to this standard in accordance with clauses 3 and 4. If there are general tolerances for other International standards, reference shall be made to them on the drawings or associated specifications. For a dimension between an unfinished and a finished surface, e.g. of cast or forged parts, for which no individual tolerance is directly indicated, the larger of the two general tolerances in question applies, e.g. for castings, see ISO 8062, Castings - system of Dimensional Tolerances.

1.0 SCOPE

The standard is intended to simplify drawing indications and it specifies general tolerances for linear and angular dimensions without individual tolerance indications in four tolerance classes.

It applies to the dimensions of parts that are produced by metal removal or parts that are formed from sheet metal.

NOTE:

1) The concepts behind the general tolerancing of linear and angular dimensions are described in Annex-A.

2) These tolerances may be suitable for use with materials other than metals.

This standard only applies for the following dimensions which do not have an individual tolerance indication:

a) Linear dimensions (e.g. external sizes, internal sizes, step sizes, diameters, radii, distances, external radii and chamfer heights for broken edges).

b) Angular dimensions, including angular dimensions usually not indicated, e.g. right angles (90°), unless reference to IS: 2102 (Part 2) is made, or angles of uniform polygons.

c) Linear and angular dimensions produced by machining assembled parts.

It does not apply for the following dimensions:

a) Linear and angular dimensions which are covered by reference to other standards on general tolerances.

b) Auxiliary dimensions indicated in brackets.

c) Theoretically exact dimensions indicated in rectangular frames.

2.0 COMPLIANCE WITH STANDARDS

This standard is based on IS: 2102 (Part 1)-1993 (ISO 2768-1).

3.0 GENERAL TOLERANCES

3.1 Linear dimensions are given in Table 1 and 2.

3.2 Angular dimensions: General tolerance specified in angular units control only the general orientation of lines or line elements of surfaces, but not their form deviations.

The general orientation of the line derived from the actual surface is the orientation of the contracting line of ideal geometrical form. The maximum distance between the contacting line and the actual line shall be the least possible value (see IS: 12160).

The permissible deviations of angular dimensions are given in Table 3.

4.0 INDICATIONS ON DRAWINGS:

If general tolerances in accordance with this standard shall apply, the following information shall be indicated.

Example: AA0230208 m

5.0 REJECTION

Unless otherwise stated, work pieces exceeding the general tolerance shall not lead to automatic rejection provided that the ability of the work piece to function is not impaired (see clause A4).

6.0 NOTE:

6.1 For "permissible deviations for Un-toleranced dimensions of castings" refer AA0230402.

6.2 For "Tolerances and machining allowances for flame cutting" refer AA0621101.

6.3 For "General tolerances for welding construction for length and angles" refer AA0621104.

6.4 For "General tolerances for welded structures form and position" refer AA0621105.

Table 1 – Permissible deviations for linear dimensions except for broken edges (external radii and chamfer heights, see table 2)

Values in millimetres

Tolerance class		Permissible deviations for basic size range							
Designation	Description	0.5[1] Up to 3	Over 3 Up to 6	Over 6 Up to 30	Over 30 Up to 120	Over 120 Up to 400	Over 400 Up to 1000	Over 1000 Up to 2000	Over 2000 Up to 4000
f	Fine	±0.05	±0.05	±0.1	±0.15	±0.2	±0.3	±0.5	-
m	Medium	±0.1	±0.1	±0.2	±0.3	±0.5	±0.8	±1.2	±2
c	Coarse	±0.2	±0.3	±0.5	±0.8	±1.2	±2	±3	±4
v	Very coarse	-	±0.5	±1	±1.5	±2.5	±4	±6	±8
1) For nominal sizes below 0.5 mm, the deviations shall be indicated adjacent to the relevant nominal size(s).									

Table 2 – Permissible deviations for broken edges (external radii and chamfer heights)

Values in millimetres

Tolerance class		Permissible deviations for basic size range		
Designation	Description	0.5[1] up to 3	Over 3 up to 6	Over 6
f	fine	±0.2	±0.5	±1
m	medium			
c	coarse	±0.4	±1	±2
v	very coarse			
1) For nominal sizes below 0.5 mm, the deviations shall be indicated adjacent to the relevant nominal size(s)				

Table 3 – Permissible deviations of angular dimensions

Tolerance class		Permissible deviations for ranges of lengths, in millimetres, of the shorter side of the angle concerned				
Designation	Description	Up to 10	Over 10 Up to 50	Over 50 Up to 120	Over 120 Up to 400	Over 400
f	fine	±1°	±0°30′	±0°20′	±0°10′	±0.5′
m	medium					
c	coarse	±1°30′	±1°	±0°30′	±0°15′	±0°10′
v	very coarse	±3°	±2°	±1°	±0°30′	±0°20′

Annex A
(informative)

Concepts behind general tolerancing of linear and angular dimensions

A.1 General tolerances should be indicated on the drawing by reference to this standard in accordance with clause 4.

The values of general tolerances correspond to tolerance classes of customary workshop accuracy, the appropriate tolerance class being selected and indicated on the drawing according to the requirement of the components.

A.2 Above certain tolerance values, there is usually no gain in manufacturing economy by enlarging the tolerance. For example, a feature having a 35 mm diameter could be manufactured to a high level of conformance in a workshop with "customary medium accuracy". Specifying a tolerance of ± 1 mm would be of not benefit in this particular workshop, as the general tolerance values of ± 0.3 mm would be quite adequate.

However, if, for functional reasons, a feature requires a smaller tolerance value than the general tolerance values, these should not be indicated adjacent to the dimension but should be stated on the drawing as described in clause 4. This type of tolerance allows full use of the concept of general tolerancing.

There will be "exceptions to the rule" where the function of the feature allows a larger tolerance than the general tolerances, and the larger tolerance will provide manufacturing economy. In these special cases, the larger tolerance should be indicated individually adjacent to the dimension for the particular feature, e.g. the depth of blind holes drilled at assembly.

A.3 Using general tolerances leads to the following advantages:

a) drawings are easier to read and thus communication is made more effective to the user of the drawing;

b) The design draughtsman saves time by avoiding detailed tolerance calculations as it is sufficient to know that the function allows a tolerance greater than or equal to the general tolerance;

c) The drawing readily indicates which feature can be produced by normal process capability, which also assists quality engineering by reducing inspection levels;

d) Those dimensions remaining, which have individually indicated tolerances, will, for the most part, be those controlling features for which the function requires relatively small tolerances and which therefore may require special effort in the production – this will be helpful for production planning and will assist quality control services in their analysis of inspection requirements;

e) Purchase and sub-contract supply engineers can negotiate orders more readily since the "customary workshop accuracy" is known before the contract is placed; this also avoids arguments on delivery between the buyer and supplier, since in this respect the drawing is complete.

These advantages are fully obtained only when there is sufficient reliability that the general tolerances will not be exceeded, i.e. when the customary workshop accuracy of the particular workshop is equal to or fine than the general tolerances indicated in the drawing.

The workshop should therefore

- Find out by measurements what is customary workshop accuracy is;
- Accept only those drawings having general tolerances equal to or greater than its customary workshop accuracy;
- Check by sampling that its customary workshop accuracy does not deteriorate.

Relying on underlined "good workmanship" with all its uncertainties and misunderstandings is no longer necessary with the concept of general geometrical tolerances. The general geometrical tolerances defines the required accuracy of "good workmanship".

A.4 The tolerance the function allows is often greater than the general tolerances. The function of the part is, therefore, not always impaired when the general tolerance is (occasionally) exceeded at any feature of the work piece. Exceeding the general tolerance should lead to a rejection of the work piece only if the function is impaired.