

1) For the purposes of this part of ISO 2768, a single feature of size comprises a cylindrical surface or two parallel plane surfaces.

Unless otherwise stated, workpieces exceeding the general geometrical tolerance shall not lead to automatic rejection provided that the ability of the workpiece to function is not impaired (see clause A.4).

7 Rejection

NOTE — The envelope requirement **(E)** cannot apply to features with individually indicated straightness tolerances which are greater than their size tolerances, e.g. stock material.

ISO 2768-mk-E

EXAMPLE

6.3 In cases where the envelope requirement **(E)** also applies to all single features of size **1**, the designation **"E"** shall be added to the general designation specified in 6.1:

ISO 2768-K

EXAMPLE

6.2 If the general dimensional tolerances (tolerance class m) shall not apply, the respective letter shall be omitted from the designation to be indicated on the drawing:

In this case the general tolerances for angular dimensions in accordance with ISO 2768-1 do not apply to right angles (90°), which are implied but not indicated, because this part of ISO 2768 specifies general tolerances on perpendicularity.

ISO 2768-2 : 1989 (E)

ISO 2768-mk

EXAMPLE

- c) the tolerance class in accordance with this part of ISO 2768.
- b) the tolerance class in accordance with ISO 2768-1;
- a) **"ISO 2768"**;

6.1 If general tolerances in accordance with this part of ISO 2768 shall apply in conjunction with the general tolerances in accordance with ISO 2768-1, the following information shall be indicated in or near the title block:

6 Indications on drawings

Tolerance class	H	K	L
Circular run-out tolerances	0.1	0.2	0.5

Values in millimetres

Table 4 — General tolerances on circular run-out

For general tolerances on circular run-out, the bearing surfaces shall be taken as the datum if they are designated as such. Otherwise, for circular radial run-out, the longer of the two features shall be taken as the datum; if the features are of equal nominal length, either may be taken as the datum.

Table 1 — General tolerances on straightness and flatness

Tolerance class	Straightness and flatness tolerances for ranges of nominal lengths					
	up to 10	10 to 30	30 to 100	100 to 300	300 to 1 000	over 1 000
H	0,02	0,05	0,1	0,2	0,3	0,4
K	0,06	0,1	0,2	0,4	0,6	0,8
L	0,1	0,2	0,4	0,8	1,2	1,6

Values in millimetres

5.1.2 Circularity

The general tolerance on circularity is equal to the numerical value of the diameter tolerance, but in no case shall it be greater than the respective tolerance value for circular run-out given in table 4 (see examples in clause B.2).

NOTES

1 The circularity deviation comprises three components: circularity deviation, straightness deviation and parallelism deviation of opposite generator lines. Each of these components is controlled by its individually indicated or its general tolerance.

2 If, for functional reasons, the circularity deviation has to be smaller than the combined effect (see clause B.3) of the general tolerances on circularity, straightness and parallelism, an individual circularity tolerance in accordance with ISO 1101 should be indicated for the feature concerned.

Sometimes, e.g. in the case of a fit, the indication of the envelope requirement **E** is appropriate.

5.2 Tolerances for related features

5.2.1 General

The tolerances specified in 5.2.2 to 5.2.6 apply to all features which are in relation to one another and which have no respective individual indication.

5.2.2 Parallelism

The general tolerance on parallelism is equal to the numerical value of the size tolerance or the flatness/straightness tolerance, whichever is the greater. The longer of the two features shall be taken as the datum; if the features are of equal nominal length, either may be taken as the datum (see clause B.4).

5.2.3 Perpendicularity

The general tolerances on perpendicularity are given in table 2. The longer of the two sides forming the right angle shall be taken as the datum; if the sides are of equal nominal length, either may be taken as the datum.

Table 2 — General tolerances on perpendicularity

Tolerance class	Perpendicularity tolerances for ranges of nominal lengths of the shorter side					
	up to 100	over 100	up to 300	over 300	up to 1 000	over 1 000
H	0,2	0,3	0,4	0,5	0,6	0,8
K	0,4	0,6	0,8	1	1,5	2
L	0,6	1	1,5	2	3	4

Values in millimetres

5.2.4 Symmetry

The general tolerances on symmetry are given in table 3. The longer of the two features shall be taken as the datum; if the features are of equal nominal length, either may be taken as the datum.

NOTE — The general tolerances on symmetry apply where

- at least one of the two features has a median plane, or
- the axes of the two features are perpendicular to each other.

See examples in clause B.5.

Table 3 — General tolerances on symmetry

Tolerance class	Symmetry tolerances for ranges of nominal lengths					
	up to 100	over 100	up to 300	over 300	up to 1 000	over 1 000
H	0,5	0,6	0,8	1	1,5	2
K	0,6	0,8	1	1,5	2	3
L	0,8	1	1,5	2	3	4

Values in millimetres

5.2.5 Coaxiality

General tolerances on coaxiality are not specified.

NOTE — The deviation in coaxiality may, in an extreme case, be as great as the tolerance value for circular radial run-out given in table 4, since the deviation in radial run-out comprises the deviation in coaxiality and the deviation in circularity.

5.2.6 Circular run-out

The general tolerances on circular run-out (radial, axial and any surface of revolution) are given in table 4.

General tolerances — Part 2: Geometrical tolerances for features without individual tolerance indications

1 Scope

This part of ISO 2768 is intended to simplify drawing indications and specifies general geometrical tolerances to control those features on the drawing which have no respective individual indication. It specifies general geometrical tolerances in three tolerance classes.

This part of ISO 2768 mainly applies to features which are produced by removal of material. Its application to features manufactured by other processes is possible; however, special examination is required to ascertain whether the customary workshop accuracy lies within the general geometrical tolerances specified in this part of ISO 2768.

2 General

When selecting the tolerance class, the respective customary workshop accuracy has to be taken into consideration. If smaller geometrical tolerances are required or larger geometrical tolerances are permissible and more economical for any individual feature, such tolerances should be indicated directly in accordance with ISO 1101 (see clause A.2).

General geometrical tolerances in accordance with this part of ISO 2768 apply when drawings or associated specifications refer to this part of ISO 2768 in accordance with clause 6. They apply to features which do not have respective individual geometrical tolerance indications.

General geometrical tolerances apply to all geometrical tolerance characteristics, excluding cylindricity, profile of any line, profile of any surface, angularity, coaxiality, positional tolerances and total run-out.

In any event, general geometrical tolerances in accordance with this part of ISO 2768 should be used when the fundamental tolerancing principle in accordance with ISO 8015 is used and indicated on the drawing (see clause B.1).

3 Normative references

The following standards contain provisions which, through reference in this text, constitute provisions of this part of

ISO 2768. At the time of publication, the editions indicated were valid. All standards are subject to revision, and parties to agreements based on this part of ISO 2768 are encouraged to investigate the possibility of applying the most recent editions of the standards indicated below. Members of IEC and ISO maintain registers of currently valid International Standards.

ISO 1101 : 1983, *Technical drawings — Geometrical tolerancing — Tolerancing of form, orientation, location and run-out — Generalities, definitions, symbols, indications on drawings*.

ISO 2768-1 : 1989, *General tolerances — Part 1: Tolerances for linear and angular dimensions without individual tolerance indications*.

ISO 5459 : 1981, *Technical drawings — Geometrical tolerancing — Datums and datum-systems for geometrical tolerances*.

ISO 8015 : 1985, *Technical drawings — Fundamental tolerancing principle*.

4 Definitions

For the purposes of this part of ISO 2768, the definitions for geometrical tolerances given in ISO 1101 and ISO 5459 apply.

5 General geometrical tolerances

(see also clause B.1)

5.1 Tolerances for single features

5.1.1 Straightness and flatness

The general tolerances on straightness and flatness are given in table 1. When a tolerance is selected from table 1, it shall be based, in the case of straightness, on the length of the corresponding line and, in the case of flatness, on the longer lateral length of the surface, or the diameter of the circular surface.

4.2 Angular dimensions

General tolerances 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 contacting line of ideal geometrical form. The maximum distance between the contacting line and the actual line shall be the least possible value (see ISO 8015).

If general tolerances in accordance with this part of ISO 2768 shall apply, the following information shall be indicated in or near the title block:

5 Indications on drawings

6 Rejection

ISO 2768-m

EXAMPLE

- a) "ISO 2768";
- b) the tolerance class in accordance with this part of ISO 2768.

Unless otherwise stated, workpieces exceeding the general tolerance shall not lead to automatic rejection provided that the ability of the workpiece to function is not impaired (see clause A.4).

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

Tolerance class		Permissible deviations for basic size range									
Designation	Description	0,5 ¹⁾	up to 3	3	6	up to 30	30	up to 120	120	up to 400	400
		over	over	over	over	over	over	over	over	over	over
f	fine	±0,05	±0,05	±0,1	±0,15	±0,2	±0,3	±0,5	±0,8	±1,2	±2
m	medium	±0,1	±0,1	±0,2	±0,3	±0,5	±0,8	±1,2	±2	±4	±8
c	coarse	±0,2	±0,3	±0,5	±0,8	±1,5	±2,5	±4	±6	±10	±20
v	very coarse	—	—	—	—	—	—	—	—	—	—

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)

Tolerance class		Permissible deviations for basic size range		
Designation	Description	0,5 ¹⁾	up to 3	over 3
		up to 6	over 6	over 10
f	fine	±0,2	±0,5	±1
m	medium	±0,2	±0,5	±1
c	coarse	±0,4	±1	±2
v	very coarse	±0,4	±1	±2

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'

General tolerances —

Part 1 :

Tolerances for linear and angular dimensions without individual tolerance indications

1 Scope

This part of ISO 2768 is intended to simplify drawing indications and it specifies general tolerances for linear and angular dimensions without individual tolerance indications in four tolerance classes.

NOTE 1 — The concepts behind the general tolerancing of linear and angular dimensions are described in annex A.

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

NOTES

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

3 Parallel International Standards exist or are planned, e.g. see ISO 8062¹⁾ for castings.

This part of ISO 2768 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 ISO 2768-2 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.

¹⁾ ISO 8062 : 1984, *Castings — System of dimensional tolerances*.

2 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 adjacent to the relevant nominal dimension(s).

General tolerances for linear and angular dimensions apply when drawings or associated specifications refer to this part of ISO 2768 in accordance with clauses 4 and 5. If there are general tolerances for other processes, as specified in 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 8062¹⁾.

3 Normative references

The following standards contain provisions which, through reference in this text, constitute provisions of this part of ISO 2768. At the time of publication, the editions indicated were valid. All standards are subject to revision, and parties to agreements based on this part of ISO 2768 are encouraged to investigate the possibility of applying the most recent editions of the standards indicated below. Members of IEC and ISO maintain registers of currently valid International Standards.

ISO 2768-2 : 1989, *General tolerances — Part 2: Geometrical tolerances for features without individual tolerance indications*.

ISO 8015 : 1985, *Technical drawings — Fundamental tolerancing principle*.

4 General tolerances

4.1 Linear dimensions

General tolerances for linear dimensions are given in tables 1 and 2.