

JIS

JAPANESE INDUSTRIAL STANDARD

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

JIS B 0419—1991
(ISO 2768—2 : 1989)

Translated and Published

by

Japanese Standards Association

Printed in Japan

In the event of any doubt arising,
the original Standard in Japanese is to be final authority.

JAPANESE INDUSTRIAL STANDARD**J I S****General tolerances — Part 2: Geometrical
tolerances for features without
individual tolerance indications****B 0419-1991
(ISO 2768-2: 1989)****Foreword as the Japanese Industrial Standard**

This Standard is the Japanese Industrial Standard drawn up without changing the technical contents and the form of copy of Standard, translating the ISO 2768-2 (General tolerances-Part 2: Geometrical tolerances for features without individual tolerance indications) published on 1989 as the first edition.

Furthermore, "Informative References" being attached with side lines (dotted lines) in this Standard are the matters not included in the original International Standard.

Introduction

All features on component parts always have a size and a geometrical shape. For the deviation of size and for the deviations of the geometrical characteristics (form, orientation and location) the function of the part requires limitations which, when exceeded, impair this function.

The tolerancing on the drawing should be complete to ensure that the elements of size and geometry of all features are controlled, i.e. nothing shall be implied or left to judgement in the workshop or in the inspection department.

The use of general tolerances for size and geometry simplifies the task of ensuring that this prerequisite is met.

1. Scope

This Japanese Industrial Standard 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 Standard 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 Standard.

2. General

When selecting the tolerance class, the respective customary workshop accuracies have 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 A.2 of Annex).

Informative reference: The contents of specification of ISO 1101 are equal to JIS B 0021-1984 (Indication of geometrical tolerances on drawings).

General geometrical tolerances in accordance with this Standard apply when drawings or associated specifications refer to this Standard in accordance with clause 6. These general geometrical tolerances 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 Standard should be used when the fundamental tolerancing principle in accordance with JIS B 0024 is used and indicated on the drawing (see B.1 of Annex).

3. Normative references

The following International Standards contain provisions which, through reference in this text, constitute provisions of this Standard. At the time of publication, the editions indicated were valid. All standards are subject to revision, and parties to agreements based on this Standard 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

Remarks: JIS B 0405-1991 (General tolerances-Part 1: Tolerances for linear and angular dimensions without individual tolerance indications) conforms to this International Standard.

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

ISO 8015: 1985 Technical drawings-Fundamental tolerancing principle

Remarks: JIS B 0024-1988 (Technical drawings-Fundamental tolerancing principle) conforms to this International Standard.

4. Definitions

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

Informative reference: The contents of the provisions of ISO 1101 and ISO 5459 are respectively equal to JIS B 0021-1984 and JIS B 0022-1984 (Datums for geometrical tolerances).

5. General geometrical tolerances (see B.1 of Annex)

5.1 General 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 this Table, 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.

Table 1. General tolerances on straightness and flatness

Unit: mm

Tolerance class	Straightness and flatness tolerances for ranges of nominal lengths					
	Up to 10	Over 10 up to 30	Over 30 up to 100	Over 100 up to 300	Over 300 up to 1000	Over 1000 up to 3000
	General tolerances on straightness and flatness					
H	0.02	0.05	0.1	0.2	0.3	0.4
K	0.05	0.1	0.2	0.4	0.6	0.8
L	0.1	0.2	0.4	0.8	1.2	1.6

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 value of circular radial run-out tolerance of Table 4 (see example of B.2 of Annex).

5.1.3 Cylindricity The general tolerances on cylindricity are not specified.

- Remarks 1. The cylindricity 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 cylindricity deviation has to be smaller than the combined effect (see B.3 of Annex) of the general tolerances of circularity, straightness and parallelism, an individual cylindricity 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 requirements ® is appropriate.

5.2 General 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 B.4 of Annex).

5.2.3 Perpendicularity The general tolerances on perpendicularity shall be as 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.