

JAPANESE
INDUSTRIAL
STANDARD

Translated and Published by
Japanese Standards Association

JIS B 0021 : 1998

(ISO/DIS 1101 : 1996)

**Geometrical product
specifications (GPS)—
Geometrical tolerancing—
Tolerancing of form, orientation,
location and run-out**

ICS 17.040.10

Descriptors : engineering drawings, form tolerances, tolerances of position, technical drawing, mechanical tolerances

Reference number : JIS B 0021 : 1998 (E)

B 0021 : 1998 (ISO/DIS 1101 : 1996)

Foreword

This translation has been made based on the original Japanese Industrial Standard revised by the Minister of International Trade and Industry through deliberations at Japanese Industrial Standards Committee in accordance with the Industrial Standardization Law. Consequently **JIS B 0021 : 1984** is replaced with **JIS B 0021 : 1998**.

This Standard which is entitled *Geometrical product specifications (GPS)—Geometrical tolerancing—Tolerancing of form, orientation, location and run-out* consists of the items given in the Contents.

In the revision of this time this Standard has resulted in an International conformance standard by ensuring the complete conformance with the International Standard **ISO/DIS 1101 : 1996** *Geometrical product specifications (GPS)—Geometrical tolerancing—Tolerancing of form, orientation, location and run-out*.

Date of Establishment: 1972-02-01

Date of Revision: 1998-01-20

Date of Public Notice in Official Gazette: 1998-01-20

Investigated by: Japanese Industrial Standards Committee
Divisional Council on Machine Elements

JIS B 0021 : 1998, First English edition published in 1999-11

Translated and published by: Japanese Standards Association
4-1-24, Akasaka, Minato-ku, Tokyo, 107-8440 JAPAN

In the event of any doubts arising as to the contents,
the original JIS is to be the final authority.

© JSA 1999

All rights reserved. Unless otherwise specified, no part of this publication may be reproduced or utilized in any form or by any means, electronic or mechanical, including photocopying and microfilm, without permission in writing from the publisher.

Printed in Japan

Contents

	Page
Introduction	1
1 Scope	2
2 Normative references	2
3 Definitions	3
4 Basic concepts	3
5 Symbols	4
6 Tolerance frame	5
7 Toleranced features	6
8 Tolerance zones	7
9 Datums	9
10 Supplementary indications	11
11 Theoretically exact dimensions	12
12 Restrictive specifications	12
13 Projected tolerance zone	13
14 Maximum material requirement	13
15 Least material requirement	14
16 Free state condition	14
17 Interrelationship of geometrical tolerances	14
18 Definitions of geometrical tolerances	14
18.1 Straightness tolerance	15
18.2 Flatness tolerance	16
18.3 Circularity tolerance	16
18.4 Cylindricity tolerance	17
18.5 Profile tolerance of a line not related to a datum (ISO 1660)	17
18.6 Profile tolerance of a line related to a datum (ISO 1660)	18
18.7 Profile tolerance of a surface not related to a datum (ISO 1660)	19
18.8 Profile tolerance of a surfaces related to a datum (ISO 1660)	19
18.9 Parallelism tolerance	20

B 0021 : 1998 (ISO/DIS 1101 : 1996)

18.10	Perpendicularity tolerance	24
18.11	Angularity tolerance.....	29
18.12	Position tolerance (ISO 5458).....	32
18.13	Concentricity and coaxiality tolerance.....	36
18.14	Symmetry tolerance.....	37
18.15	Circular run-out tolerance	38
18.16	Total run-out tolerance.....	42
Annex A (informative)	Former practices.....	43
Annex B (informative)	Definitions	45
Annex C (normative)	Assessment of geometrical deviations	46
Annex D (informative)	GPS matrix model.....	50

Geometrical product specifications (GPS)— Geometrical tolerancing— Tolerancing of form, orientation, location and run-out

Introduction This Japanese Industrial Standard has been prepared based on the second edition issued in 1996 of **ISO/DIS 1101** *Geometrical product specifications (GPS)—Geometrical tolerancing—Tolerancing of form, orientation, location and run-out* without changing the technical contents.

The “Informative reference” underlined with a dotted line in this Standard is not included in the original International Standard.

This Standard **JIS B 0021** is a Geometrical Product Specification (GPS) standard and is to be regarded as a General GPS standard (see **ISO/TR 14638**, *Geometrical Product Specifications (GPS)—Masterplan*). It influences the chain link 1 and 2 of the chain of standards **ISO/TR 14638** on form, orientation, location and run out and chain link 1 of the chain of standards on datums.

For more detailed information of the relation of this Standard to the GPS matrix model see Annex D.

This Standard contains all information necessary for the geometrical definition of workpieces. It represents the initial basis and describes the required fundamentals for geometrical tolerancing. Nevertheless, it is advisable to consult the separate standards referenced in 2 and the references described in Table 1 and Table 2 for more detailed information.

For the purposes of this Standard, all numbers and Remarks in all figures have been prepared in upright lettering. It should be understood that these indications could have been written in inclined (italic) lettering without altering the meaning of the indications.

For the presentation of lettering (proportions and dimensions), see **JIS Z 8313-1** *Technical drawings—Lettering—Part 1: Currently used characters*.

Remarks: This standard above is identical to **ISO 3098-1**, *Technical drawings—Lettering—Part 1: Currently used characters*.

In the interest of uniformity, all figures in this Standard have been drawn in third angle projection with dimensions and tolerances in millimetres. It should be understood that first angle projection and other units of measurement could have been used equally well without prejudice to the principles established.

The figures in this Standard illustrate the text and are not intended to reflect an actual application. Consequently, the figures are not fully dimensioned and tolerances showing only the relevant general principles.

For a definitive presentation (proportions and dimensions) of the symbols, symbolization for geometrical tolerancing, see **ISO 7083**.

Annex A (Former practices) of this Standard has been provided for information only. It presents previous drawing indications that have been omitted from this Standard and are no longer used.

1 Scope

1.1 This Standard contains basic information for the geometrical tolerancing of workpieces. It represents the initial basis and describes the fundamentals for geometrical tolerancing. There are separate standards referenced in 2 and in Table 2 that provide more detailed information on geometrical tolerancing.

1.2 Geometrical tolerances shall be specified in accordance with functional requirements. Manufacturing and inspection requirements may also influence geometrical tolerancing.

1.3 Indicating geometrical tolerances on a drawing does not necessarily imply the use of any particular method of production, measurement or gauging.

2 Normative references The following standards contain provisions which, through reference in this text, constitute provisions of this Standard. For dated references, out of these normative references, only the edition indicated year constitutes the provisions of this Standard, and subsequent revisions are not applied. For undated references, the latest edition of the publication referred to applies.

ISO 1660 : 1987 *Technical drawings — Dimensioning and tolerancing of profiles*

JIS B 0022 *Datums and datum-systems for geometrical tolerances*

Informative reference : This standard above is equivalent to ISO 5459 : 1981, Technical drawings — Geometrical tolerancing — Datums and datum-systems for geometrical tolerances.

JIS B 0023 : 1995 *Technical drawings — Geometrical tolerancing — Maximum material requirement and least material requirement*

Remarks: This standard above is identical to ISO 2692 : 1988, *Technical drawings — Geometrical tolerancing — Maximum material principles* ⁽¹⁾ and ISO 2692 Amd. 1 : 1992, *Technical drawings — Geometrical tolerancing — Maximum material principles — Amendment 1 : Least material requirement*.

JIS B 0024 *Technical drawings — Fundamental tolerancing principle*

Remarks: This standard above is identical to ISO 8015 : 1985, *Technical drawings — Fundamental tolerancing principle* ⁽²⁾.

JIS B 0025 *Technical drawings — Geometrical tolerancing — Positional tolerancing*

Remarks: This standard above is identical to ISO 5458 : 1987, *Technical drawings — Geometrical tolerancing — Positional tolerancing* ⁽³⁾.

JIS B 0614 *System of cone tolerances*

Informative reference : ISO 3040 : 1990, Technical drawings — Dimensioning and tolerancing — Cones is identical to the relevant provisions in the said standard.

Notes ⁽¹⁾ At present at ISO/CD stage

⁽²⁾ At present under revision

⁽³⁾ At present at ISO/DIS stage

JIS B 0026 *Technical drawings—Dimensioning and tolerancing—Non-rigid parts*

Remarks: This standard above is identical to **ISO 10579**:1993, *Technical drawings—Dimensioning and tolerancing—Non-rigid parts*.

ISO 7083:1983 *Technical drawings—Symbols for geometrical tolerancing—Proportions and dimensions*

ISO 10578:1992 *Technical drawings—Tolerancing of orientation and location—Projected tolerance zone*

JIS/TR B 0003 *Technical drawings—Geometrical tolerancing—Tolerancing of form, orientation, location and run-out—Verification principles and methods—Guidelines*

Remarks: This standard above is identical to **ISO/TR 5460**:1985, *Technical drawings—Geometrical tolerancing—Tolerancing of form, orientation, location and run-out—Verification principles and methods—Guidelines*.

3 Definitions Definitions related to features are under development and will be issued as **ISO 14660** *Geometrical product specification (GPS)—Geometric features*. This work will result in new terms that are different than those used in this Standard. These new terms are explained in the Annex B and are identified in the main body of this Standard by the term being added to the current term and enclosed in parentheses.

For the purposes of this Standard, the following definition also applies.

3.1 Tolerance zone Space limited by one or several geometrically perfect lines or surfaces, and for which the tolerances (unit of length) are indicated.

4 Basic concepts

4.1 A geometrical tolerance applied to a feature defines the tolerance zone within which that feature is to be contained.

4.2 A feature is a specific portion of the workpiece, such as a surface, a hole, a slot, a thread, a chamfered part or a profile; these features can be physically existing (e.g. the external surface of a cylinder) or derived (e.g. an axis or median surface).

4.3 According to the characteristic which is to be toleranced and the manner in which it is dimensioned, the tolerance zone is one of the following:

- the space within a circle;
- the space between two concentric circles;
- the space between two equidistant lines or two parallel straight lines;
- the space within a cylinder;
- the space between two coaxial cylinders;
- the space between two equidistant surfaces or two parallel planes;
- the space within a sphere.

4.4 Unless a more restrictive indication is required, for example by an explanatory note (see Fig. 7), the toleranced feature may be of any form or orientation within this tolerance zone.

4.5 The tolerance applies to the whole extent of the considered feature unless otherwise specified as in **11**, and **12**.

4.6 Geometrical tolerances which are assigned to features related to a datum do not limit the form deviations of the datum feature itself. It may be necessary to specify tolerances of form for the datum feature(s).

5 Symbols

Table 1 Symbols for geometrical characteristics

Tolerances	Characteristics	Symbols	Datum needed	References
Form	Straightness	—	no	18.1
	Flatness		no	18.2
	Circularity		no	18.3
	Cylindricity		no	18.4
	Profile of any line		no	18.5
	Profile of any surface		no	18.7
Orientation	Parallelism		yes	18.9
	Perpendicularity		yes	18.10
	Angularity		yes	18.11
	Profile of any line		yes	18.6
	Profile of any surface		yes	18.8
Location	Position		no or yes	18.12
	Concentricity (for centre points)		yes	18.13
	Coaxiality (for axes)		yes	18.13
	Symmetry		yes	18.14
	Profile of any line		yes	18.6
	Profile of any surface		yes	18.8
Run-out	Circular run-out		yes	18.15
	Total run-out		yes	18.16

Table 2 Additional Symbols

Descriptions	Symbols	References
Toleranced feature indication		7
Datum feature indication		9 and JIS B 0022
Datum target indication		JIS B 0022
Theoretically exact dimension		12
Projected tolerance zone		13 and ISO 10578
Maximum material requirement		14 and JIS B 0023
Least material requirement		15 and JIS B 0023
Free state condition (non-rigid parts)		16 and JIS B 0026
All around (profile)		10.1
Envelope requirement		JIS B 0024
Common zone	CZ	8.5

Informative reference: The symbol letters other than P, M, L, F, E and CZ show an example.

6 Tolerance frame

6.1 The requirements are shown in a rectangular frame which is divided into two or more compartments. These compartments contain, from left to right, in the following order (see Figs. 1, 2, 3 and 4):

- the symbol for the geometrical characteristic;
- the tolerance value in the unit used for linear dimensions (unit of length). This value is preceded by the sign “ ϕ ” if the tolerance zone is circular or cylindrical; or by “S ϕ ” if the tolerance zone is spherical.
- if appropriate, the letter or letters identifying the datum or datum system (see Figs. 2, 3 and 4).

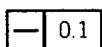


Fig. 1

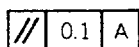


Fig. 2

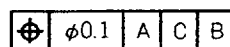


Fig. 3

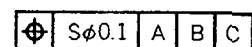


Fig. 4

6.2 When a tolerance applies to more than one feature this shall be indicated above the tolerance frame by the number of features followed by the sign “ $\times$ ” (see Figs. 5 and 6).

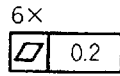


Fig. 5

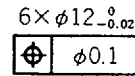
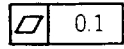


Fig. 6

6.3 If required, indications qualifying the form of the feature within the tolerance zone shall be written near the tolerance frame (see Fig. 7).



not convex

Fig. 7

6.4 It is necessary to specify more than one geometrical characteristic for a feature, the requirements may be given in tolerance frames one under the other for convenience (see Fig. 8).

Informative reference: No discrepancy shall be between plural tolerance indications.

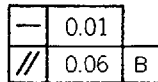


Fig. 8

7 Toleranced features The tolerance frame is connected to the toleranced feature by a leader line starting from either side of the frame and terminating with an arrow in the following way:

- on the outline of the feature or an extension of the outline (but clearly separated from the dimension line) when the tolerance refers to the line or surface itself (see Figs. 9 and 10). The arrow may be placed on a leader line pointing to the surface (see Fig. 11).

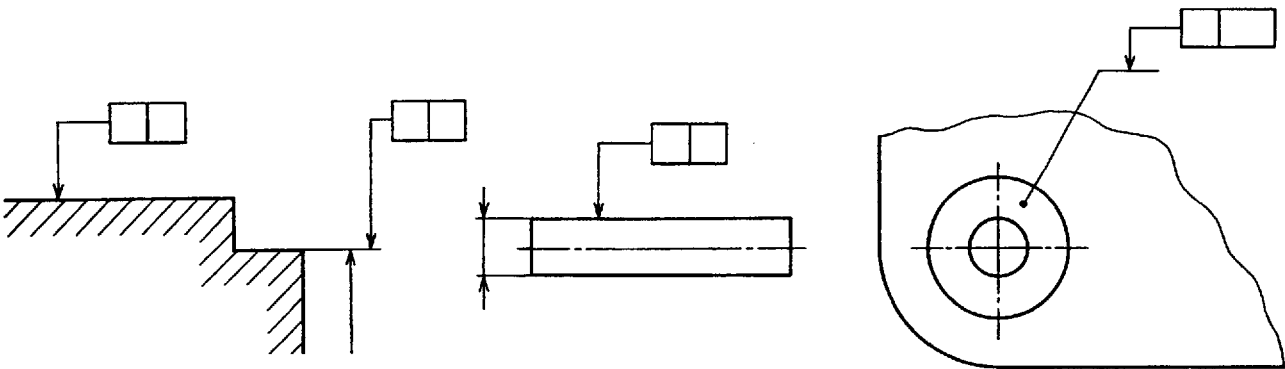


Fig. 9

Fig. 10

Fig. 11

- as an extension of the dimension line when the tolerance refers to the axis or median plane or a point defined by the feature so dimensioned (see Figs. 12, 13 and 14).

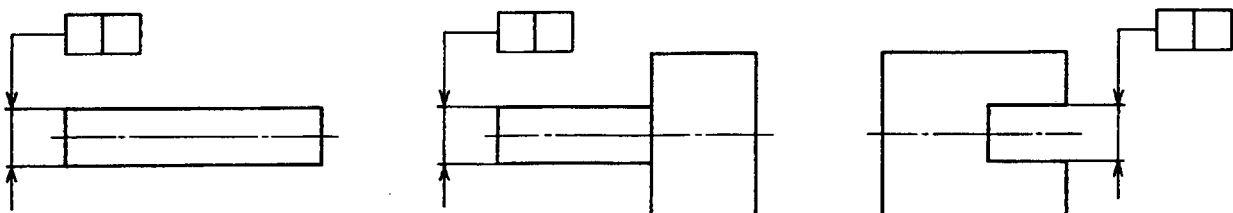


Fig. 12

Fig. 13

8 Tolerance zones

8.1 The width of the tolerance zone applies normal to the specified geometry (see Figs. 15, 16 and 17) unless otherwise indicated (see Figs. 18 and 19).

In the case of circularity, the width of the tolerance zone always applies in the direction where a straight line perpendicular to the tangent line intersects the nominal axis.

8.2 In the case of an axis or centre point toleranced in one direction, the orientation of the width of a positioning tolerance zone is based on the pattern of the theoretically exact dimensions and is at 0° degrees or 90° degrees as indicated by the direction of the arrow of the leader line unless otherwise indicated (see Fig. 15).

— the orientation of the width of the tolerance zone is at 0° degrees or 90° degrees relative to the datum as indicated by the direction of the arrow of the leader line unless otherwise indicated (see Figs. 18 and 19).

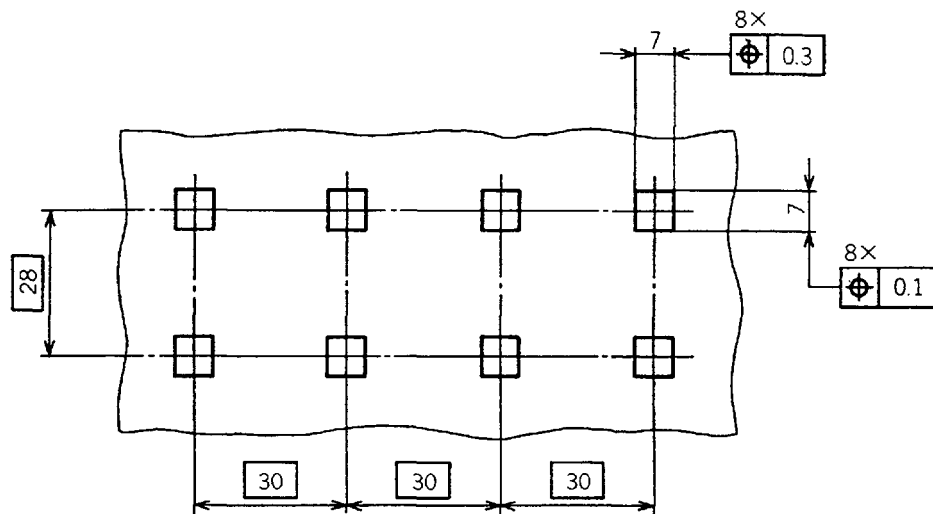


Fig. 15

— when two tolerances are stated, they are perpendicular to each other unless otherwise specified (see Figs. 16 and 17).

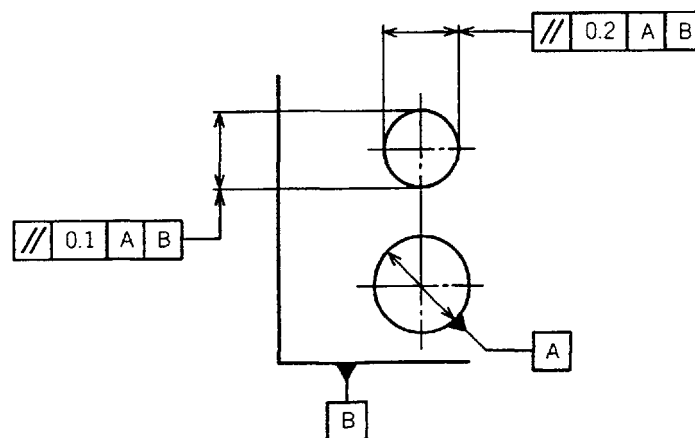


Fig. 16

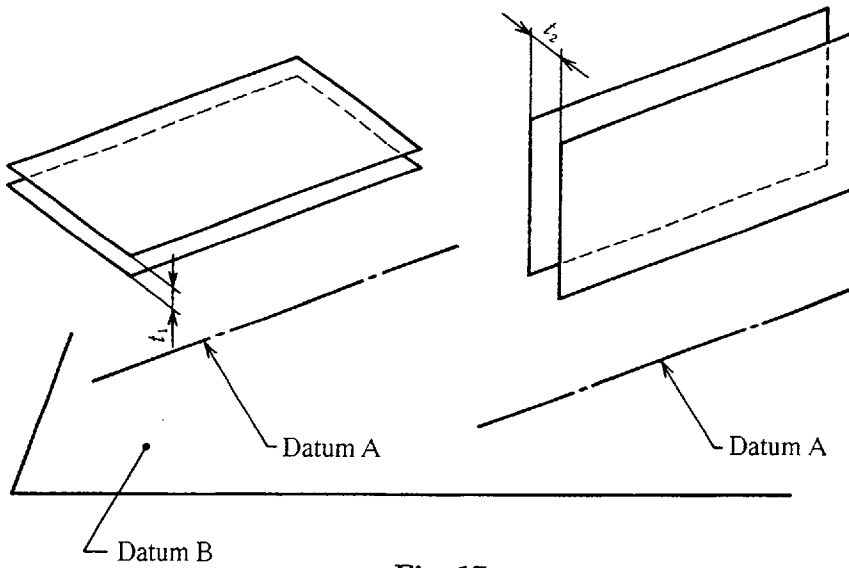


Fig. 17

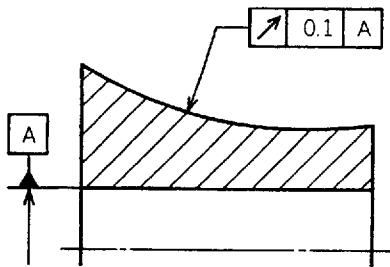


Fig. 18

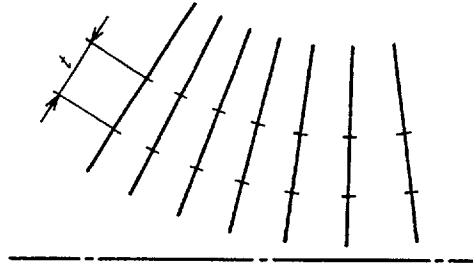


Fig. 19

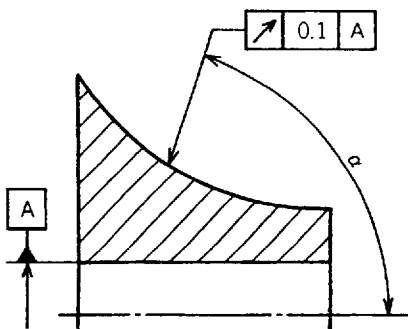


Fig. 20

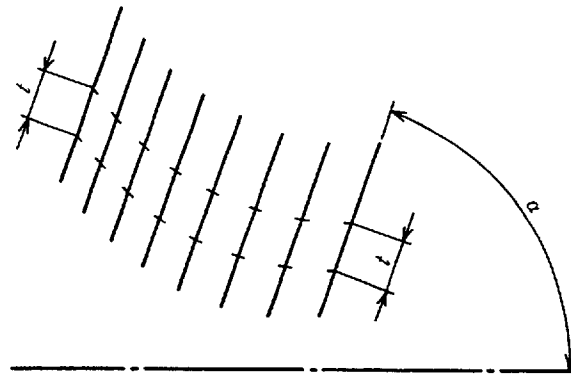


Fig. 21

Remarks: The angle α shown in Fig. 20 shall be indicated, even if it is equal to 90° .

8.3 The tolerance zone is cylindrical (see Figs. 22 and 23) or circular if the tolerance value is preceded by the symbol " ϕ " or spherical if it is preceded by the symbol " $S\phi$ ".

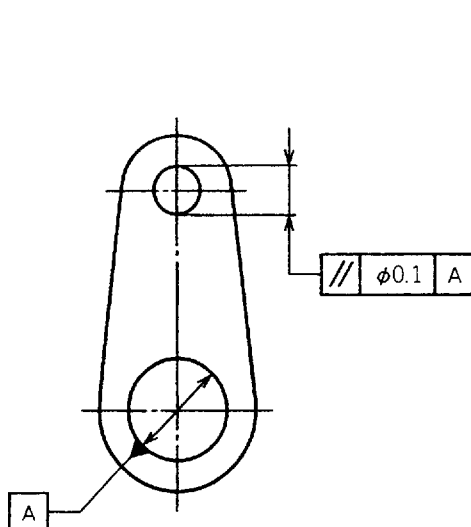


Fig. 22

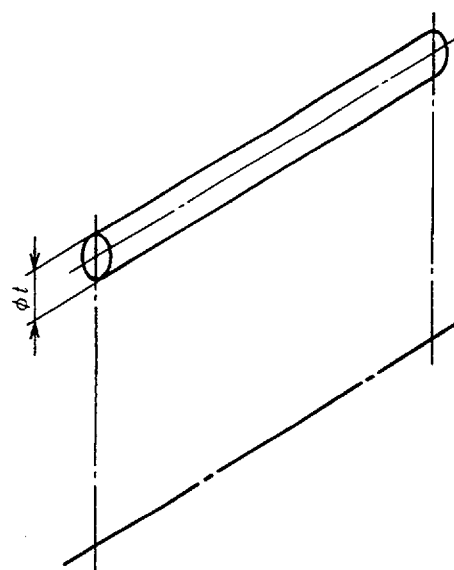


Fig. 23

8.4 Individual tolerance zones of the same value applied to several separate features can be specified as shown on Fig. 24.

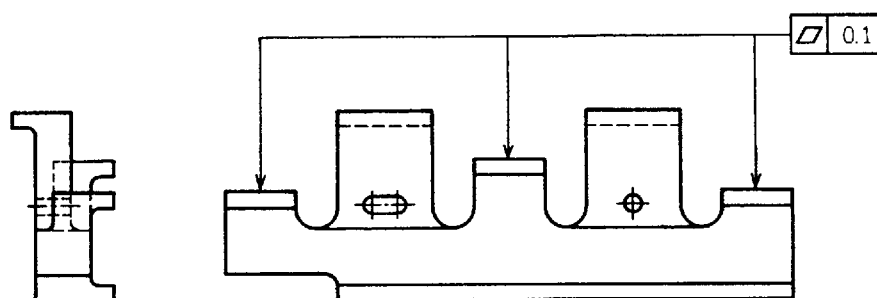


Fig. 24

8.5 Where a single tolerance zone is applied to several separate features, the requirement is indicated by the label "CZ" for common zone following the tolerance in the tolerance frame (see Fig. 25).

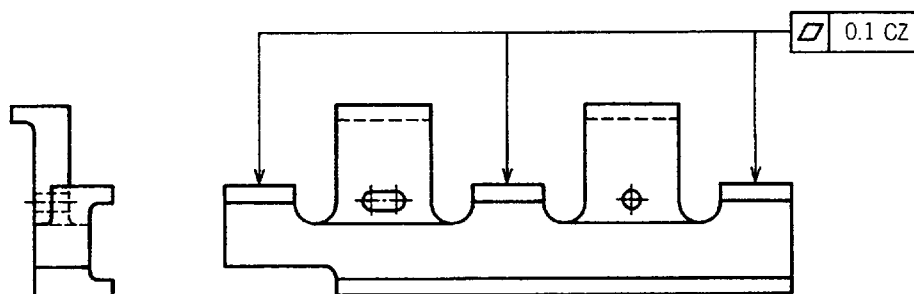


Fig. 25

9 Datums Datums are to be indicated as shown in the following sub-clauses. For additional information see JIS B 0022.

9.1 A datum that is related to a tolerated feature is designated by a datum letter. A capital letter enclosed in a frame is connected to a solid or blank datum triangle to identify the datum (see Figs. 26 and 27). The same letter which defines the datum is also indicated in the tolerance frame. There is no difference in the meaning between a solid and a blank datum triangle.

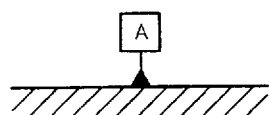


Fig. 26

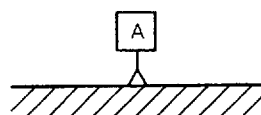


Fig. 27

9.2 The datum triangle with the datum letter is placed:

- on the outline of the feature or an extension of the outline (but clearly separated from the dimension line), when the datum is the line or surface shown (see Fig. 28). The datum triangle may be placed on a leader line pointing to the surface (see Fig. 29).

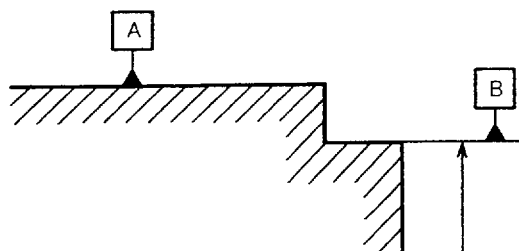


Fig. 28

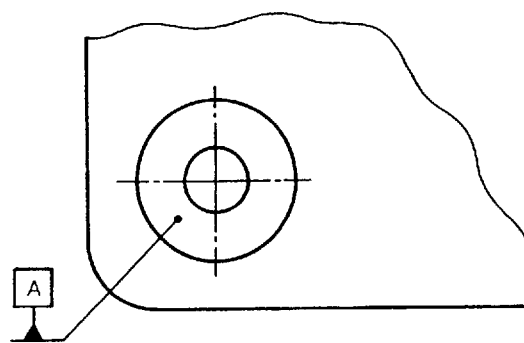


Fig. 29

- as an extension of the dimension line when the datum is the axis or median plane or a point defined by the feature so dimensioned (see Figs. 30 to 32). If there is insufficient space for two arrows, one of them may be replaced by the datum triangle (see Figs. 31 and 32).

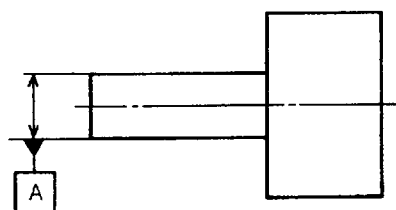


Fig. 30

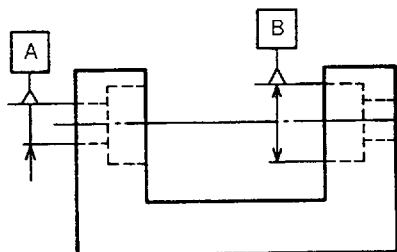


Fig. 31

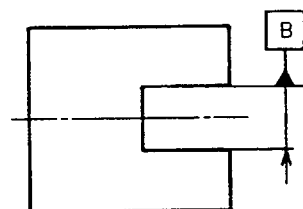


Fig. 32

9.3 If a datum is applied to a restricted part of a datum feature only, this restriction shall be shown as a thick chain line and dimensioned (see Fig. 33).

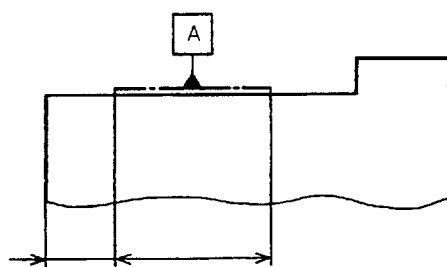


Fig. 33

9.4 A datum established by a single feature is identified by a capital letter (see Fig. 34).

A common datum established by two features is identified by two capital letters separated by a hyphen (see Fig. 35).

Where a datum system is established by two or three features, i.e. multiple datums, the capital letters for identifying the datums are indicated in an order of priority, from left to right, in separate compartments (see Fig. 36).



Fig. 34



Fig. 35

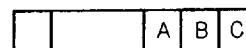


Fig. 36

10 Supplementary indications

10.1 If a profile characteristic is applied to the entire outline of the cross sections or if it is applied to the entire surface of the boundary it shall be expressed by using the symbol “all around” (see Figs. 37 and 38). The all around symbol does not involve the entire workpiece but only the surfaces identified by the tolerance indication.

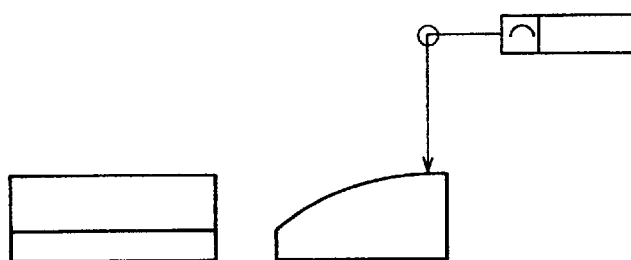


Fig. 37

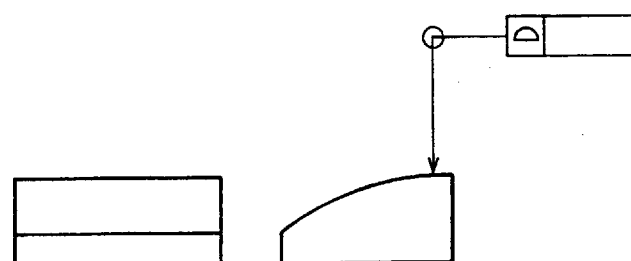
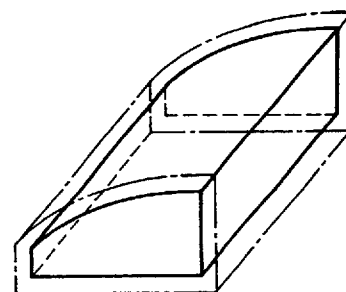
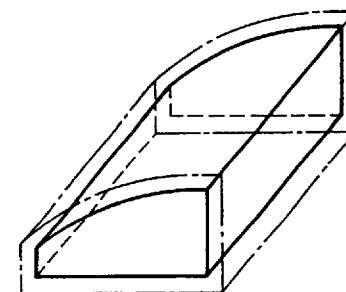


Fig. 38



10.2 Tolerances and datum references specified for screw threads apply to the axis derived from the pitch cylinder, unless otherwise specified, e.g. “MD” for major diameter (see Figs. 39 and 40). Tolerances and datum references specified for gears and splines shall designate the specific feature to which they apply, e.g. “PD” for pitch diameter, “MD” for major diameter or “LD” for minor diameter.

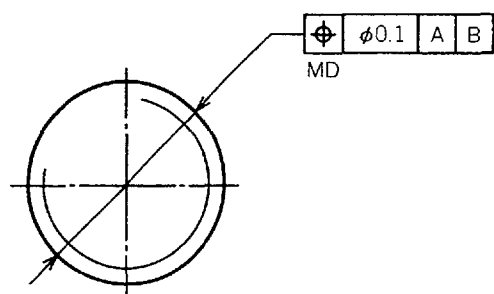


Fig. 39

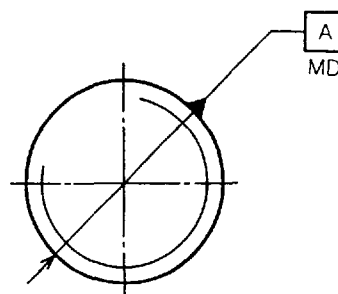


Fig. 40

11 Theoretically exact dimensions If tolerances of location, orientation or profile are prescribed for a feature or a group of features, the dimensions (including distance) determining the theoretically exact location, orientation or profile respectively are called “theoretically exact dimensions”. Theoretically exact dimensions also apply to the dimensions determining the relative orientation of the datums of a system.

Theoretically exact dimensions shall not be tolerated. They are enclosed in a frame (see Figs. 41 and 42).

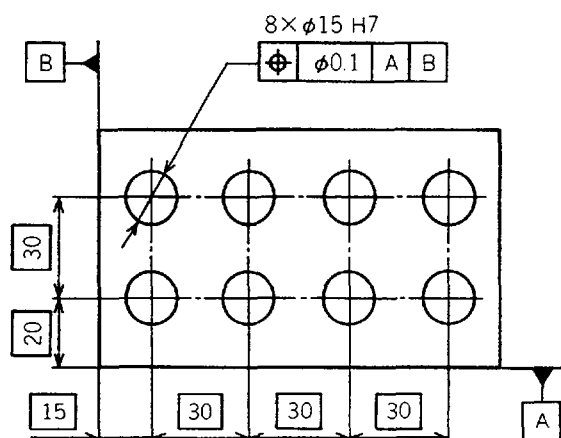


Fig. 41

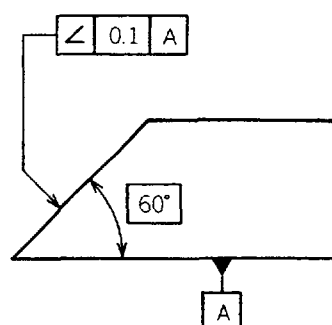


Fig. 42

12 Restrictive specifications

12.1 If a tolerance of the same characteristic is applied to a restricted length, lying anywhere within the total extent of the feature, the value of the restricted length shall be added after the tolerance value and be separated from it by an oblique stroke. Such indication may be placed in the lower compartment of the tolerance frame directly under the tolerance for the entire feature (see Fig. 43).

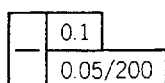


Fig. 43

12.2 If a tolerance is applied to a restricted part of a feature only, this restriction shall be shown as a thick chain line and dimensioned (see Figs. 44 and 45).

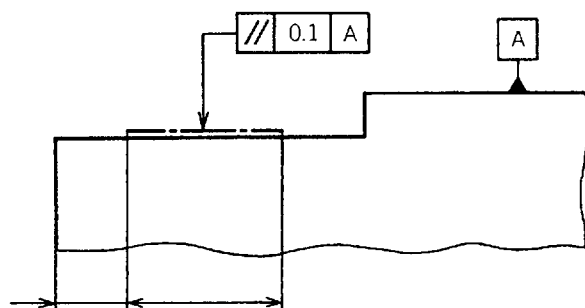


Fig. 44

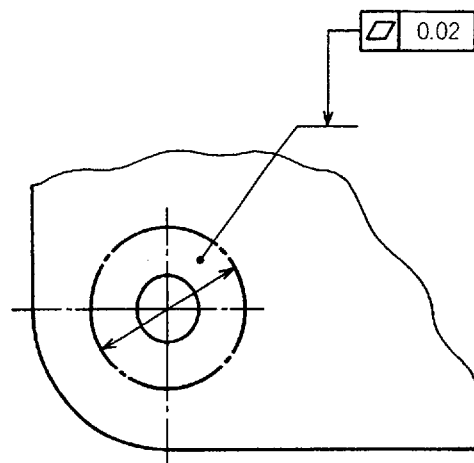


Fig. 45

12.3 Restricted part of a datum (see 9.3)

12.4 Restrictions to the form of a feature within the tolerance zone are given in 6.3.

13 Projected tolerance zone Projected tolerance zones are indicated by the symbol $\textcircled{P}$ (see Fig. 46). See ISO 10578 for additional information.

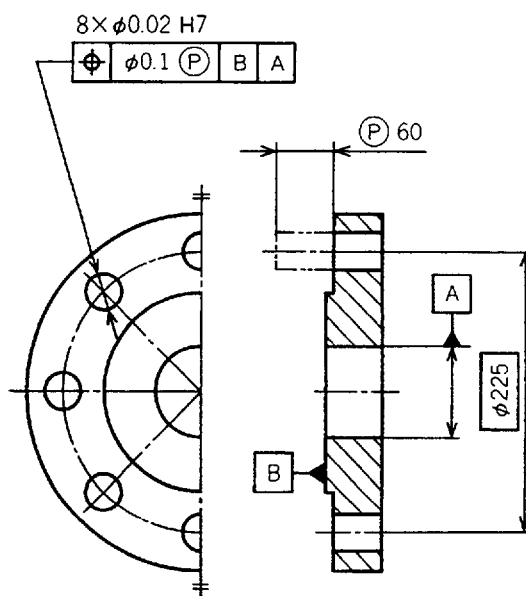


Fig. 46

14 Maximum material requirement The maximum material requirement is identical by the symbol $\textcircled{M}$. The symbol is placed after the specified tolerance value, datum letter or both as appropriate (see Figs. 47, 48 and 49). See JIS B 0023 for additional information.

Remarks: The provisions specified in this standard above-mentioned are identical to ISO 2692:1988, *Technical drawings—Geometrical tolerancing—Maximum material principle*.