

ISO

INTERNATIONAL ORGANIZATION FOR STANDARDIZATION

ISO RECOMMENDATION

R 1101 / E

TOLERANCES OF FORM AND OF POSITION

iTeh STANDARD PREVIEW

PART I

(standards.iteh.ai)

GENERALITIES, SYMBOLS, INDICATIONS ON DRAWINGS

ISO/R 1101-1:1969

<https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-9b98-9604e169d2c1/iso-r-1101-1-1969>

1st EDITION

July 1969

COPYRIGHT RESERVED

The copyright of ISO Recommendations and ISO Standards belongs to ISO Member Bodies. Reproduction of these documents, in any country, may be authorized therefore only by the national standards organization of that country, being a member of ISO.

For each individual country the only valid standard is the national standard of that country.

Printed in Switzerland

Also issued in French and Russian. Copies to be obtained through the national standards organizations.

iTeh STANDARD PREVIEW
(standards.iteh.ai)

ISO/R 1101-1:1969

<https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-9b98-9604e169d2c1/iso-r-1101-1-1969>

BRIEF HISTORY

The ISO Recommendation R 1101, *Tolerances of form and of position – Part I : Generalities, symbols, indications on drawings*, was drawn up by Technical Committee ISO/TC 10, *Drawings (General principles)*, the Secretariat of which is held by the Association Suisse de Normalisation (SNV).

Work on this question led to the adoption of a Draft ISO Recommendation.

In August 1966, this Draft ISO Recommendation (No. 1016) was circulated to all the ISO Member Bodies for enquiry. It was approved, subject to a few modifications of an editorial nature, by the following Member Bodies :

Argentina	Iran	Spain
Austria	Ireland	Sweden
Belgium	Israel	Switzerland
Canada	Italy	Turkey
Chile	Japan	U.A.R.
Denmark	Korea Rep. of	United Kingdom
Finland	Netherlands	U.S.A.
France	Norway	U.S.S.R.
Germany	Portugal	Yugoslavia
Greece	Romania	
India	South Africa, Rep. of	

One Member Body opposed the approval of the Draft :

Australia

The Draft ISO Recommendation was then submitted by correspondence to the ISO Council, which decided, in July 1969, to accept it as an ISO RECOMMENDATION.

CONTENTS

	Page
1. Scope	5
2. Definitions	6
3. Symbols	7
4. Indications on the drawings	8
5. Detailed definitions of tolerances	12
5.1 Straightness tolerance	13
5.2 Flatness tolerance	13
5.3 Circularity tolerance	14
5.4 Cylindricity tolerance	14
5.5 Profile tolerance of any line	14
5.6 Profile tolerance of any surface	14
5.7 Parallelism tolerance	15
5.8 Perpendicularity tolerance	17
5.9 Angularity tolerance	18
5.10 Positional tolerance	19
5.11 Concentricity, coaxiality and symmetry tolerance	21
5.12 Run-out tolerance	22
6. Synoptic table	24

iTeh STANDARD PREVIEW
(standards.iteh.ai)

[ISO/R 1101-1:1969](https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-9b98-9604e169d2c1/iso-r-1101-1-1969)

[https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-](https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-9b98-9604e169d2c1/iso-r-1101-1-1969)

[9b98-9604e169d2c1/iso-r-1101-1-1969](https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-9b98-9604e169d2c1/iso-r-1101-1-1969)

TOLERANCES OF FORM AND OF POSITION

PART I

GENERALITIES, SYMBOLS, INDICATIONS ON DRAWINGS

INTRODUCTORY NOTE

For uniformity all the linear dimensions given in this ISO Recommendation are in metric units only and the figures are in projection method E (called European or first angle projection). It should be understood that inch units and/or projection method A (called American or third angle projection) could equally well have been used without prejudice to the principles established.

(standards.iteh.ai)

1. SCOPE

[ISO/R 1101-1:1969](https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-9b98-9604e169d3e1/iso-r-1101-1-1969)

[https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-](https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-9b98-9604e169d3e1/iso-r-1101-1-1969)

- 1.1 This ISO Recommendation gives the principles of the symbolization and of the indication on technical drawings of tolerances of form and of position (form, orientation, position and run-out) and establishes the appropriate geometrical definitions. The primary object of this method of tolerancing is to ensure satisfactory functioning and interchangeability.
- 1.2 These tolerances of form and position should be specified only where they are essential to ensure the fitness of the part for its purpose.
- 1.3 When only a tolerance of size is specified, this tolerance also limits certain errors of form and of position (e.g. flatness and parallelism – see the table in section 3). The actual surfaces of the manufactured part may then deviate from the specified geometrical form on condition that they remain within the tolerances of size. If the errors of form must remain within other limits, a tolerance of form must be specified.
- 1.4 A tolerance of form or of position may be specified even if no tolerance of size is given.
- 1.5 The system of indicating tolerances of form and of position does not necessarily imply the use of any particular method of production, measurement or gauging.

2. DEFINITIONS

- 2.1 A tolerance of form or of position of a geometrical feature (point, line, surface or median plane) defines the zone within which this feature is to be contained. (See Note 1 below).
- 2.2 According to the characteristic which is to be tolerated and the manner in which it is dimensioned, the tolerance zone is one of the following :
 - the area within a circle;
 - the area between two concentric circles;
 - the area between two parallel lines or two parallel straight lines;
 - the space within a sphere;
 - the space within a cylinder or between two coaxial cylinders;
 - the space between two parallel surfaces or two parallel planes;
 - the space within a parallelepiped.
- 2.3 The feature may be of any form or orientation within this tolerance zone, unless a more restrictive indication is given by an explanatory note.
- 2.4 Unless otherwise specified in accordance with clauses 4.5 to 4.7, the tolerance applies to the whole length or surface of the considered feature.
- 2.5 The datum feature is the feature to which tolerances of orientation, position and run-out are related.
- 2.6 The form of a datum feature should be sufficiently accurate for its purpose and it may therefore be necessary, in some cases, to specify tolerances of form for the datum features. (See Note 2 below).

NOTES

- 1. The form of a single feature is deemed to be correct when the distance of its individual points from a superimposed surface of ideal geometrical form is equal to or less than the value of the specified tolerance. The orientation of the ideal surface should be chosen so that the maximum distance between it and the actual surface of the feature concerned is the least possible value.

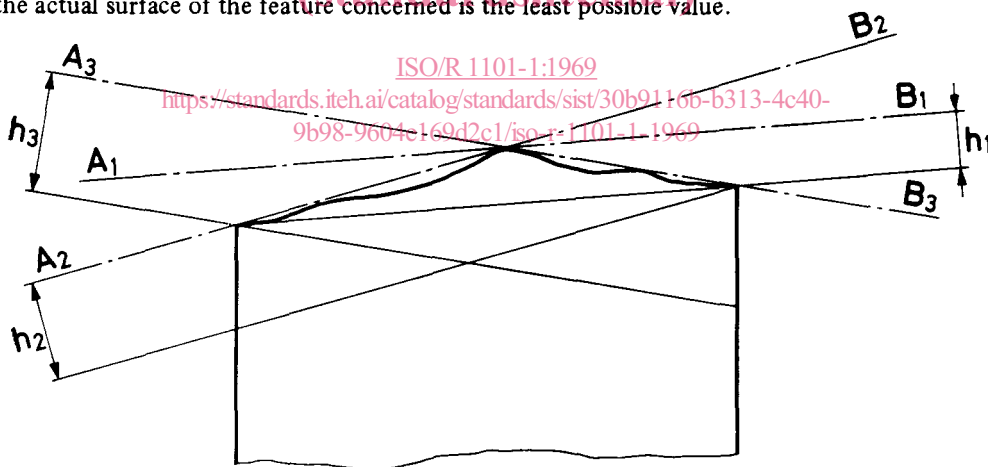


FIG. 1

Possible orientations of the ideal surface :	A_1-B_1	A_2-B_2	A_3-B_3
Corresponding maximum distances :	h_1	h_2	h_3
In the case of Figure 1 :	h_1	$< h_2$	$< h_3$

Therefore the orientation of the ideal surface is A_1-B_1 , and h_1 is to be equal to or less than the specified tolerance.

- 2. In some cases it may be desirable to indicate the position of certain points which will possibly form a temporary datum feature for the purposes of both manufacture and inspection of components.

3. SYMBOLS

The following symbols represent the types of characteristics to be controlled by the tolerance.

Characteristics to be tolerated		Symbols
Form of single features	Straightness	—
	Flatness	
	Circularity (Roundness)	○
	Cylindricity	
	Profile of any line	⌒
	Profile of any surface	⌒
Orientation of related features	Parallelism	//
	Perpendicularity (Squareness)	⊥
	Angularity	∠
Position of related features	Position	⊕
	Concentricity and coaxiality	◎
	Symmetry	≡
Run-out		

4. INDICATIONS ON THE DRAWINGS

4.1 The necessary indications are written in a rectangular frame which is divided into two, sometimes three, compartments. These compartments are filled in, from left to right, in the following order (see Fig. 2 and 3) :

- the symbol for the characteristic to be tolerated, as in section 3;
- tolerance value (total value) in the unit used for linear dimensions. This value is preceded by the sign ϕ if the tolerance zone is circular or cylindrical, or by the indication "sphere ϕ " if the tolerance zone is spherical;
- if appropriate, the letter or letters identifying the datum feature or features.



FIG. 2

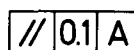


FIG. 3

4.2 The tolerance frame is connected to the tolerated feature by a leader line terminating with an arrow in the following way :

- on the outline of the feature or an extension of the outline (but not at a dimension line), when the tolerance refers to the line or surface itself (see Fig. 4);
- on the projection line at the dimension line when the tolerance refers to the axis or median plane of the part so dimensioned (see Fig. 5 and 7) or on the axis when the tolerance refers to the axis or median plane of all features common to that axis or median plane (see Fig. 6, 8 and 9).

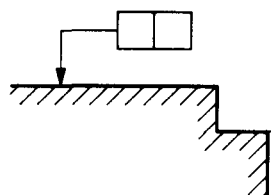


FIG. 4

ISO/R 1101-1:1969
<https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-9b98-9604e169d2c1/iso-r-1101-1-1969>

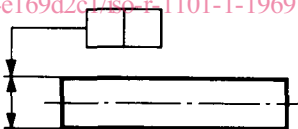


FIG. 5

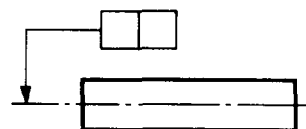


FIG. 6

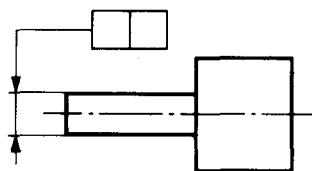


FIG. 7

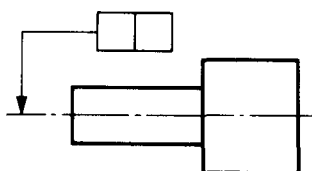


FIG. 8

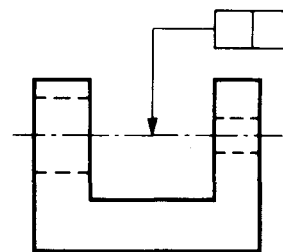


FIG. 9

If the tolerance zone is not circular, cylindrical or spherical, its width is in the direction of the arrow terminating the line joining the symbol frame to the feature which is tolerated.

4.3 The datum feature or datum features are indicated by a leader line terminating in a solid triangle, the base of which lies

- on the outline of the feature or an extension of the outline (but not at a dimension line), when the datum feature is the line or surface itself (see Fig. 10);
- on the projection line at the dimension line when the datum feature is the axis or median plane of the part so dimensioned (see Fig. 11, 13 and 16) or on the axis or median plane of all features common to that axis or median plane (see Fig. 12, 14 and 15) if such an axis can be determined with sufficient accuracy.

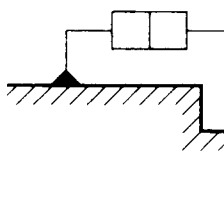


FIG. 10

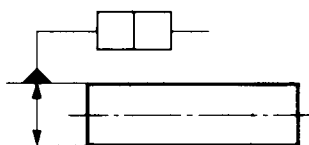


FIG. 11

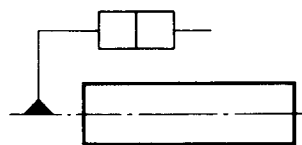


FIG. 12

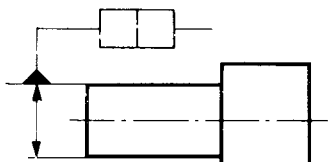


FIG. 13

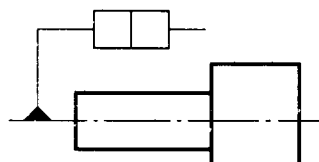


FIG. 14

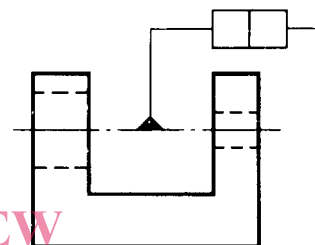


FIG. 15

ITeh STANDARD PREVIEW
(standards.iteh.ai)

If there is insufficient space for two arrows, one of them may be replaced by this triangle (see Fig. 16).

ISO/R 1101-1:1969

<https://standards.iteh.ai/catalog/standards/sist/5089116b-b313-4c40-9b78-9604e169d2c1/iso-r-1101-1-1969>

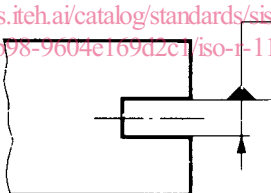


FIG. 16

If the tolerance frame cannot be connected in a clear and simple manner with the datum feature, a capital letter (different for every datum feature) is used (see Fig. 17 and 18).

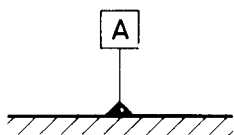


FIG. 17

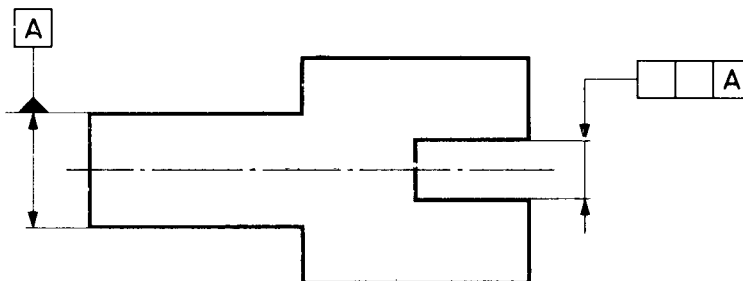


FIG. 18

This capital letter is enclosed in a frame connected to the datum feature as described in the first part of this clause.

- 4.4 If two associated features are identical, or there is no reason to choose one as a datum feature, the tolerance is indicated as in Figure 19.

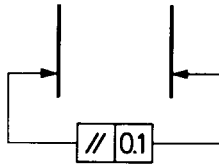
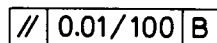


FIG. 19

- 4.5 If the tolerance is applied to a specified length, lying anywhere, the value of this length should be added after the tolerance value and separated from it by an oblique stroke.

In the case of a surface, the same indication is used. This means that the tolerance applies to all lines of the specified length in any position and any direction (see Fig. 20).



iTeh STANDARD PREVIEW
(standards.iteh.ai)

FIG. 20

- 4.6 If, to the tolerance on the whole feature, another tolerance of the same type but smaller and restricted on a limited length is added, the latter tolerance should be indicated below the former (see Fig. 21).

<https://standards.iteh.ai/catalog/standards/sist/30b9116b-b313-4c40-9b98-9604e169d2c1/iso-r-1101-1-1969>

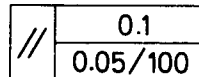


FIG. 21

- 4.7 If the tolerance is applied to a restricted part of the feature only, this should be dimensioned as shown in Figure 22 (in accordance with ISO Recommendation R 129, *Engineering drawing – Dimensioning*, clause 2.5).

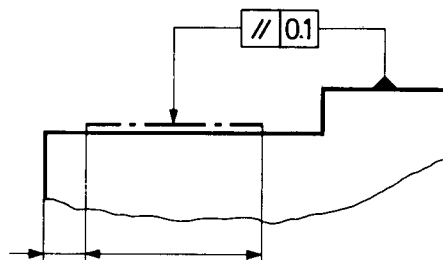


FIG. 22

- 4.8 The indication of "maximum material principle" (see ISO Recommendation R . . . *, *The maximum material principle*) is shown by the symbol \textcircled{M} placed after
- the tolerance value (see Fig. 23),
 - the datum letter (see Fig. 24),
 - or both (see Fig. 25),

according to whether the maximum material condition is to be applied to the tolerated feature, the datum feature, or both.

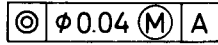


FIG. 23

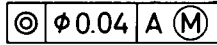


FIG. 24

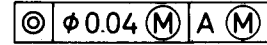
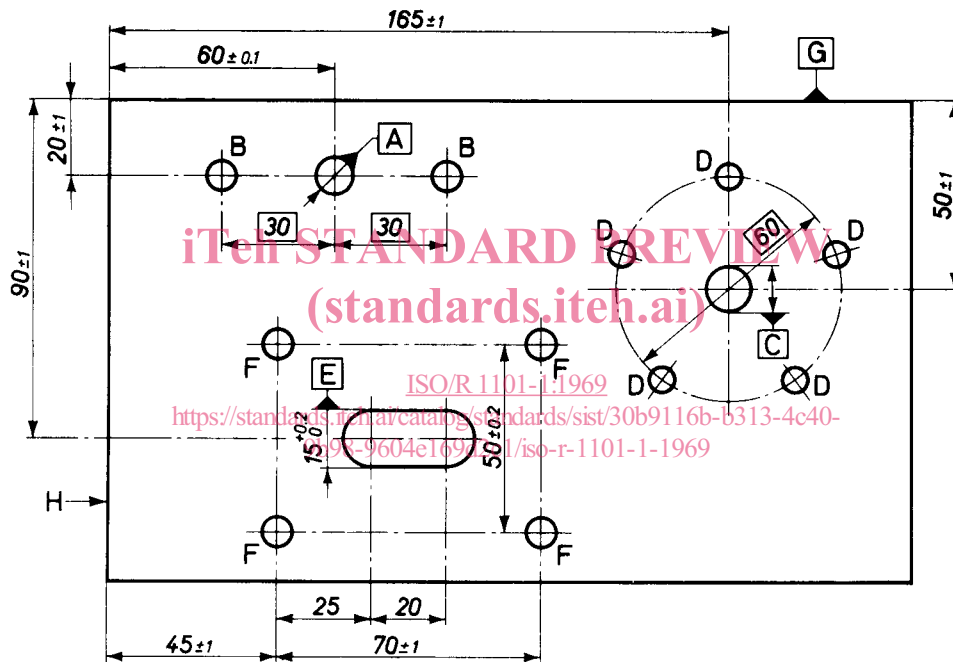


FIG. 25

- 4.9 If tolerances of position or of profile are prescribed for a feature, the dimensions determining the true position or the profile must not be tolerated. If tolerances of angularity are prescribed for a feature, the dimensions determining the angle must not be tolerated.

These dimensions are enclosed thus : $\boxed{30}$. The corresponding actual dimensions of the part are subject only to the specified position tolerance, profile tolerance or angularity tolerance.

- 4.10 As an alternative to showing the tolerances individually on a drawing, they may be grouped in a table (see Fig. 26).



Group	Letter	Holes		Tolerances	
		Dimension	Number		
1	A	$\phi 10^{+0.1}_0$	1	Datum \textcircled{M}	$\textcircled{\phi}$
	B	$\phi 8^{+0.5}_0$	2	Tol. $\phi 0.8$ \textcircled{M}	
2	C	$\phi 12^{+0.2}_0$	1	Datum \textcircled{M}	$\textcircled{\phi}$
	D	$\phi 7^{+0.5}_0$	5	Tol. $\phi 0.6$ \textcircled{M}	
3	E	-	-	Datum \textcircled{M}	\equiv
	F	$\phi 8^{+0.5}_0$	4	Tol. 0.1 \textcircled{M}	
4	G	-	-	Datum	\perp
	H	-	-	Tol. 0.05	

FIG. 26

NOTE. - The dimensions which are not enclosed in a frame or individually tolerated are subject to the general tolerance.

* In course of preparation.