International Standard 1101

INTERNATIONAL ORGANIZATION FOR STANDARDIZATION•МЕЖДУНАРОДНАЯ ОРГАНИЗАЦИЯ ПО СТАНДАРТИЗАЦИИ•ORGANISATION INTERNATIONALE DE NORMALISATION

Technical drawings — Geometrical tolerancing — Tolerancing of form, orientation, location and run-out — Generalities, definitions, symbols, indications on drawings

Dessins techniques — Tolérancement géométrique — Tolérancement de forme, orientation, position et battement — Généralités, définitions, symboles, indications sur les dessins

First edition — 1983-12-01

UDC 744.4 : 621.753.1 **Ref. No. ISO 1101-1983 (E)**

Descriptors : technical drawings, form tolerances, tolerances of position, graphic methods, generalities, definitions, symbols.

Price based on 24 pages

Foreword

ISO (the International Organization for Standardization) is a worldwide federation of national standards bodies (ISO member bodies). The work of developing International Standards is carried out through ISO technical committees. Every member body interested in a subject for which a technical committee has been authorized has the right to be represented on that committee. International organizations, governmental and non-governmental, in liaison with ISO, also take part in the work.

Draft International Standards adopted by the technical committees are circulated to the member bodies for approval before their acceptance as International Standards by the ISO Council.

International Standard ISO 1101 was developed by Technical Committee ISO/TC 10, *Technical drawings*, and was circulated to the member bodies in December 1980.

It has been approved by the member bodies of the following countries :

Australia	Hungary	Romania
Austria	India	South Africa, Rep. of
Brazil	Iraq	Spain
Canada	Italy	Sweden
China	Japan	Switzerland
Czechoslovakia	Korea, Dem. P. Rep. of	United Kingdom
Denmark	Korea, Rep. of	USA
Finland	Netherlands	USSR
France	New Zealand	
Germany, F.R.	Norway	

The member body of the following country expressed disapproval of the document on technical grounds :

Belgium

This International Standard cancels and replaces ISO Recommendation R 1101/1-1969, of which it constitutes a technical revision.

A more detailed, bilingual (English, French) version of Table 1 "Symbols for toleranced characteristics" and Table 2 "Additional symbols" has been made up in A4 plastic-coated format. This represents an extract of this International Standard, which lends itself to everyday use on the shop floor.

This extract is available separately.

Contents

This page intentionally left blank

Technical drawings — Geometrical tolerancing — Tolerancing of form, orientation, location and run-out — Generalities, definitions, symbols, indications on drawings

0 Introduction

For uniformity all figures in this International Standard are in first angle projection.

It should be understood that the third angle projection could equally well have been used without prejudice to the principles established.

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

1 Scope and field of application

1.1 This International Standard gives the principles of symbolization and indication on technical drawings of tolerances of form, orientation, location and run-out, and establishes the appropriate geometrical definitions. Hence the term "geometrical tolerances" will be used in this document as synonymous with these groups of tolerances.

1.2 Geometrical tolerances shall be specified only where they are essential, that is, in the light of functional requirements, interchangeability and probable manufacturing circumstances.

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

2 References

ISO 128, *Technical drawings — General principles of presentation.*

ISO 129, *Engineering drawings — Dimensioning — General principles, definitions, methods of execution, and special indications.* [1]

ISO 1660, *Technical drawings — Dimensioning and tolerancing of profiles.*

ISO 2692, *Technical drawings — Geometrical tolerancing — Maximum material principle.* [2]

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

ISO 7083, *Technical drawings — Symbols for geometrical tolerancing — Proportions and dimensions.*

ISO 8015, *Technical drawings — Fundamental tolerancing principle.* [3]

3 General

3.1 A geometrical tolerance applied to a feature defines the tolerance zone within which the feature (surface, axis, or median plane) is to be contained (see 3.7 and 3.8).

3.2 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 area within a circle;

— the area between two concentric circles;

1) At present at the stage of draft. (Revision of ISO/R 129-1959.)

2) At present at the stage of draft. (Revision of ISO 1101/2-1974.)

3) At present at the stage of draft.

— the area 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 planes or two parallel planes;

— the space within a parallelepiped.

3.3 The toleranced feature may be of any form or orientation within this tolerance zone, unless a more restrictive indication is given, for example by an explanatory note (see figures 8 and 9).

3.4 Unless otherwise specified as in clauses 9 and 11, the tolerance applies to the whole length or surface of the considered feature.

3.5 The datum feature is a real feature of a part, which is used to establish the location of a datum (see ISO 5459).

3.6 Geometrical tolerances which are assigned to features related to a datum do not limit the form deviations of the datum feature itself. The form of a datum feature shall be sufficiently accurate for its purpose and it may therefore be necessary to specify tolerances of form for the datum features.

3.7 The straightness or flatness of a single toleranced 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 line or surface shall be chosen so that the maximum distance between it and the actual surface of the feature concerned is the least possible value.

Example :

Figure 1

Possible orientations of the line or surface :	$A_1 - B_1$		$A_2 - B_2$		$A_3 - B_3$
Corresponding distances :	h_1		h_2		h_3
In the case of figure 1 :	h_1	<	h_2	<	h_3

Therefore the correct orientation of the ideal line or surface is $A_1 - B_1$. The distance h_1 is to be equal to or less than the specified tolerance.

3.8 For the definition of circularity and cylindricity, the location of the two concentric circles or coaxial cylinders shall be chosen so that the radial distance between them is the minimum.

Example :

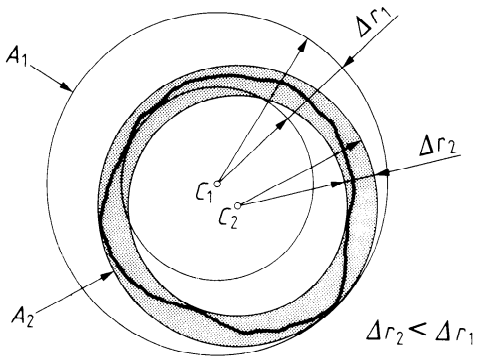


Figure 2

Possible location of the centres of the two concentric circles or the axes of the two coaxial cylinders and their minimal radial distances.

Centre (C_1) of A_1 locates two concentric circles or two coaxial cylinders.

Centre (C_2) of A_2 locates two concentric circles or two coaxial cylinders with minimal radial distance.

Corresponding radial distances : Δr_1 Δr_2

In the case of figure 2 : $\Delta r_2 < \Delta r_1$

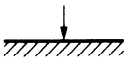
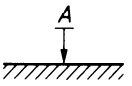
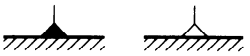
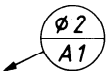
Therefore the correct location of the two concentric circles or the two coaxial cylinders is the one designated A_2. The radial distance Δr_2 should then be equal to or less than the specified tolerance.

4 Symbols

Table 1 — Symbols for toleranced characteristics

Features and tolerances		Toleranced characteristics	Symbols	Subclauses
Single features	Form tolerances	Straightness	—	14.1
		Flatness	▱	14.2
		Circularity	○	14.3
		Cylindricity	⌭	14.4
Single or related features		Profile of any line	⌒	14.5
		Profile of any surface	⌓	14.6
Related features	Orientation tolerances	Parallelism	//	14.7
		Perpendicularity	⊥	14.8
		Angularity	∠	14.9
	Location tolerances	Position	⌖	14.10
		Concentricity and coaxiality	◎	14.11
		Symmetry	⌯	14.12
	Run-out tolerances	Circular run-out	↗	14.13
		Total run-out	⌰	14.14

Table 2 — Additional symbols

Descriptions		Symbols	Clauses
Toleranced feature indications	direct		6
	by letter		7.4
Datum indications	direct		8
	by letter		
Datum target			ISO 5459
Theoretically exact dimension			10
Projected tolerance zone			11
Maximum material condition			12

5 Tolerance frame

5.1 The tolerance 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 figures 3, 4 and 5) :

— the symbol for the characteristic to be toleranced;

— the tolerance value in the unit used for linear dimensions. This value is preceded by the sign ø if the tolerance zone is circular or cylindrical;

— if appropriate, the letter or letters identifying the datum feature or features (see figures 4 and 5).

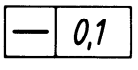


Figure 3

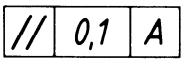


Figure 4

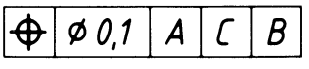


Figure 5

5.2 Remarks related to the tolerance, for example "6 holes", "4 surfaces" or "6x" shall be written above the frame (see figures 6 and 7).

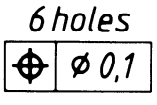


Figure 6

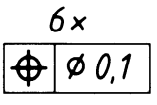


Figure 7

5.3 Indications qualifying the form of the feature within the tolerance zone shall be written near the tolerance frame and may be connected by a leader line (see figures 8 and 9).

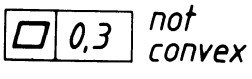


Figure 8

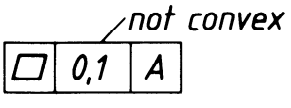


Figure 9

5.4 If it is necessary to specify more than one tolerance characteristic for a feature, the tolerance specifications are given in tolerance frames one under the other (see figure 10).

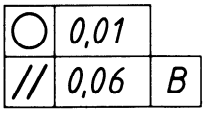


Figure 10

6 Toleranced features

The tolerance frame is connected to the toleranced 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 clearly separated from the dimension line) when the tolerance refers to the line or surface itself (see figures 11 and 12).

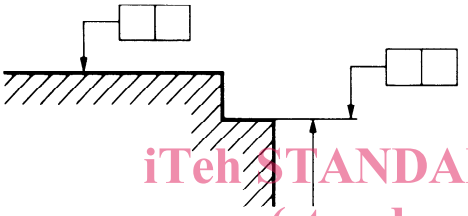


Figure 11

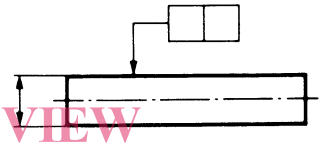


Figure 12

— as an extension of a dimension line when the tolerance refers to the axis or median plane defined by the feature so dimensioned (see figures 13 to 15).

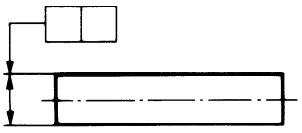


Figure 13

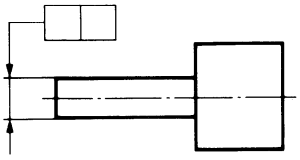


Figure 14

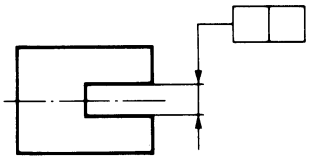


Figure 15

— on the axis when the tolerance refers to the axis or median plane of all features common to that axis or median plane (see figures 16, 17 and 18).

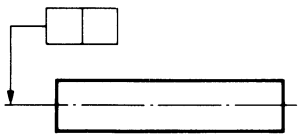


Figure 16

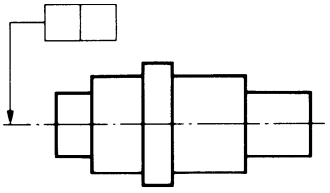


Figure 17

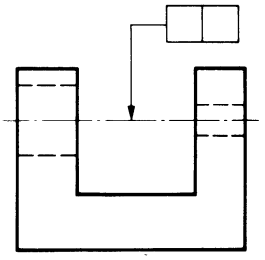


Figure 18

NOTE — Whether a tolerance should be applied to the contour of a cylindrical or symmetrical feature or to its axis or median plane respectively depends on the functional requirements.

7 Tolerance zones

7.1 The width of the tolerance zone is in the direction of the arrow of the leader line joining the tolerance frame to the feature which is toleranced, unless the tolerance value is preceded by the sign ø (see figures 19 and 20).

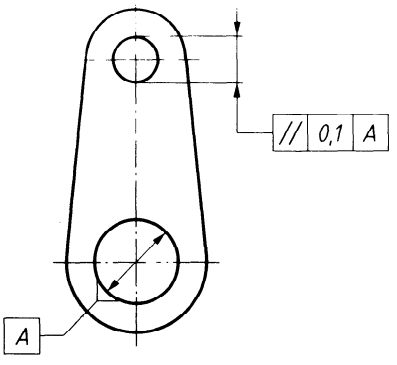


Figure 19

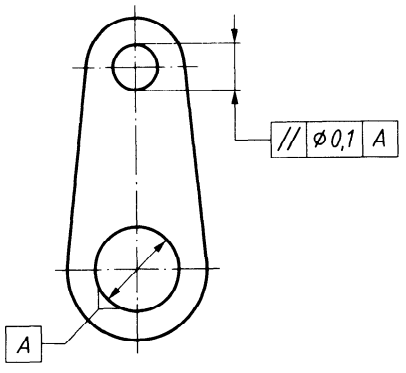


Figure 20

7.2 In general, the direction of the width of the tolerance zone is normal to the specified geometry of the part (see figures 21 and 22).

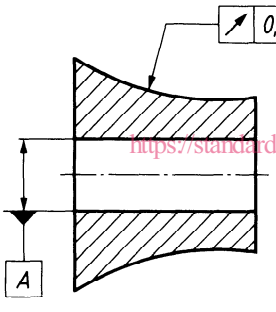


Figure 21

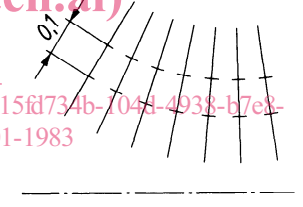


Figure 22

7.3 The direction of the width of the tolerance zone shall be indicated when desired **not** normal to the specified geometry of the part (see figures 23 and 24).

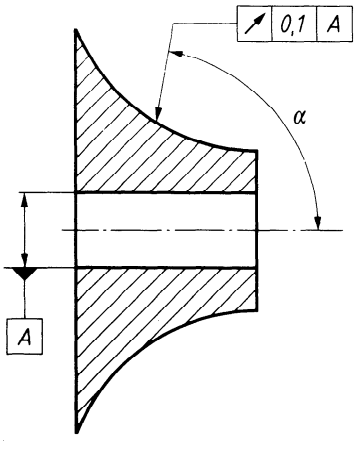


Figure 23

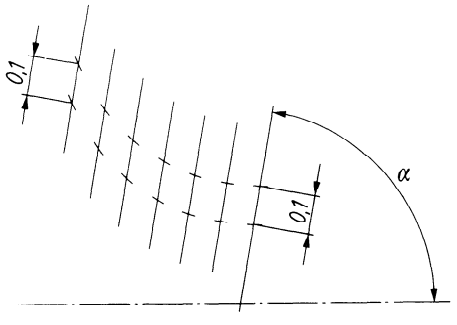


Figure 24

7.4 **Individual tolerance zones** of the same value applied to several separate features can be specified as shown in figures 25 and 26.

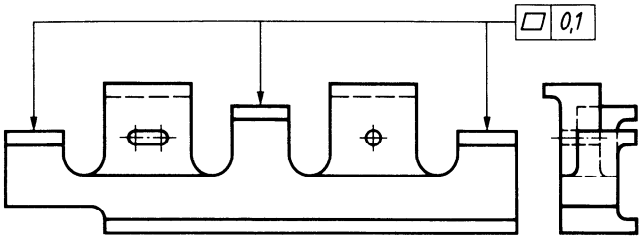


Figure 25

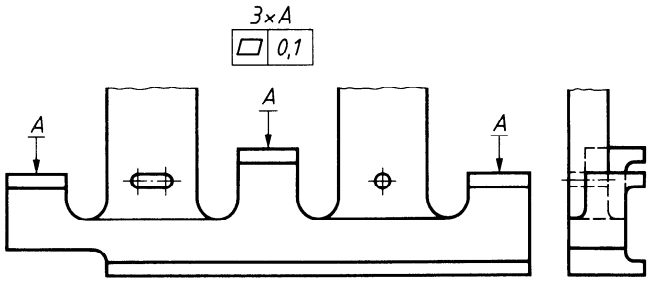


Figure 26

7.5 Where a **common tolerance zone** is applied to several separate features, the requirement is indicated by the words "common zone" above the tolerance frame (see figures 27 and 28).

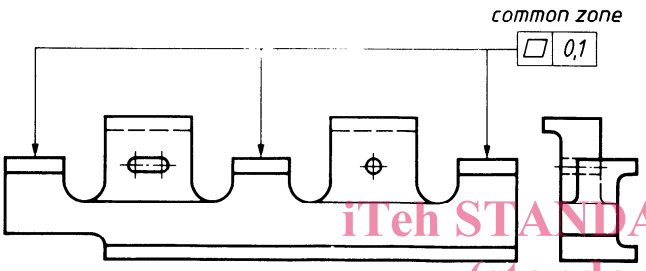


Figure 27

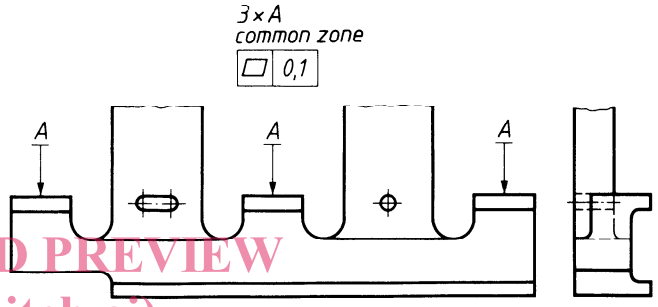


Figure 28

8 Datums

8.1 When a toleranced feature is related to a datum, this is generally shown by datum letters. The same letter which defines the datum is repeated in the tolerance frame.

To identify the datum, a capital letter enclosed in a frame is connected to a solid or blank datum triangle (see figures 29 and 30).

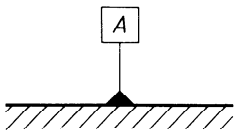


Figure 29

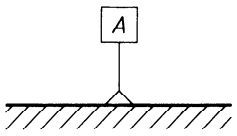


Figure 30

8.2 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 feature is the line or surface itself (see figure 31).

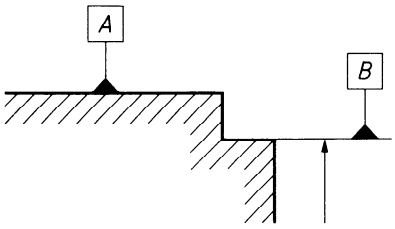


Figure 31

— as an extension of the dimension line when the datum feature is the axis or median plane (see figures 32 to 34).

NOTE — If there is insufficient space for two arrows, one of them may be replaced by the datum triangle (see figures 33 and 34).

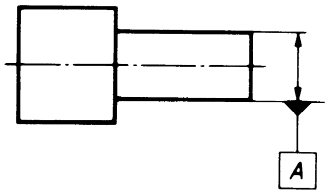


Figure 32

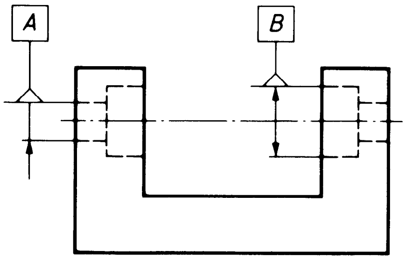


Figure 33

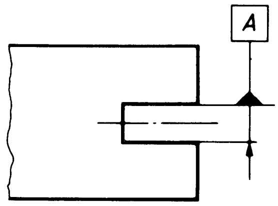


Figure 34

— on the axis or median plane when the datum is :

a) the axis or median plane of a single feature (for example a cylinder);

b) the common axis or plane formed by two features (see figure 35).

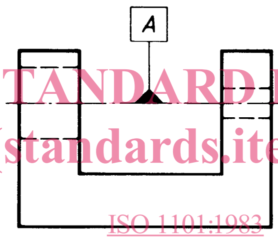


Figure 35

8.3 If the tolerance frame can be directly connected with the datum feature by a leader line, the datum letter may be omitted (see figures 36 and 37).

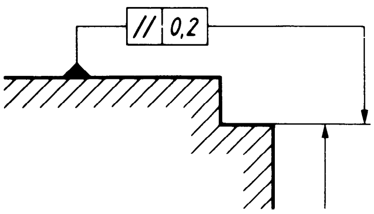


Figure 36

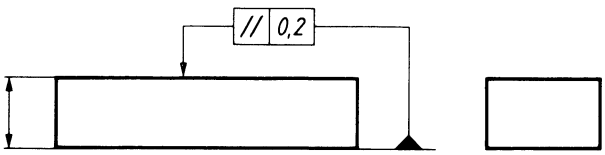


Figure 37

8.4 A single datum is identified by a capital letter (see figure 38).

A common datum formed by two datum features is identified by two datum letters separated by a hyphen (see figure 39).

If the sequence of two or more datum features **is** important the datum letters are placed in different compartments (see figure 40), where the sequence from left to right shows the order of priority.

If the sequence of two or more datum features **is not** important the datum letters are indicated in the same compartment (see figure 41).

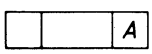


Figure 38

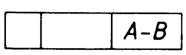


Figure 39

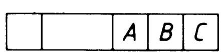


Figure 40

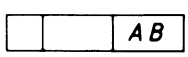


Figure 41