

Nissan Engineering Standard

N E S	Geometrical tolerancing - Tolerancing of form, orientation, location and run-out
-------	--

Minor corrections of mistakes on P3 in 2004.
Quotation "NES D0190" is corrected to
"NES D0402" in English version.

Introduction

This Standard has been prepared as a translation of ISO 1101 issued in 1983 without changing the technical details and it is established as NES according to the current format. The standard name is "Geometrical tolerancing - Tolerancing of form, orientation, location and run-out" and the details of the Standard have been partially changed in accordance with conditions at Nissan.

Contents	Page
1. SCOPE AND FIELD OF APPLICATION	1
2. GENERAL	1
3. SYMBOLS	4
4. TOLERANCE FRAME	5
5. TOLERANCED FEATURES	6
6. TOLERANCE ZONES	7
7. DATUMS	8
8. RESTRICTIVE SPECIFICATIONS	10
9. THEORETICALLY EXACT DIMENSIONS	11
10. PROJECTED TOLERANCE ZONE	11
11. MAXIMUM MATERIAL CONDITION	11
12. DEFINITIONS OF TOLERANCES	12
13. DETAILED DEFINITIONS OF TOLERANCES	13
13.1 Straightness tolerance	13
13.2 Flatness tolerance	13
13.3 Circularity tolerance	14
13.4 Cylindricity tolerance	14
13.5 Profile tolerance of any line	14
13.6 Profile tolerance of any surface	14
13.7 Parallelism tolerance	15
13.8 Perpendicularity tolerance	18
13.9 Angularity tolerance	19
13.10 Positional tolerance	20
13.11 Concentricity and coaxiality tolerance	22
13.12 Symmetry tolerance	22
13.13 Circular run-out tolerance	23
13.14 Total run-out tolerance	25

Applicable and Reference Standards: See page 27.

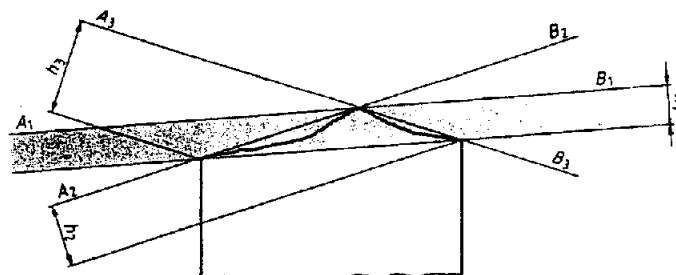
1. SCOPE AND FIELD OF APPLICATION

- 1.1 This Standard gives the principles of symbolization and indication on technical drawings of tolerances of form, orientation, location and run-out, and establishes the 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. GENERAL

- 2.1 The geometrical tolerance applied to a feature defines the tolerance zone within which the feature (surface, axis and median plane) is to be contained. (See 2.7 and 2.8.)
- 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 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 of a parallelepiped.

- 2.3 The tolerated 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).
- 2.4 Unless otherwise specified as in clauses 8 and 10, the tolerance applies to the whole length or surface of the considered feature.
- 2.5 The datum feature is a real feature of a part, which is used to establish the location of a datum (See NES D0402).
- 2.6 Geometrical tolerances which are assigned to features related to a datum does 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 necessary to specify tolerances of form for datum features.
- 2.7 Straightness or flatness of a single tolerated feature is deemed to be correct when the distance of its individual points from a superimposed surface of ideal geometrical form is equal or less than the value of 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.

- 2.8 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:

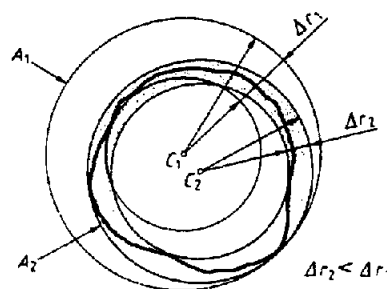


Figure 2

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.

3. SYMBOLS

Table 1 - Symbols for toleranced characteristics

Features and tolerances		Toleranced characteristics	Symbols	Subclauses
Single features Single or related features	Form tolerances	Straightness	—	13.1
		Flatness		13.2
		Circularity		13.3
		Cylindricity		13.4
		Profile of any line		13.5
		Profile of any surface		13.6
Related features	Orientation tolerances	Parallelism		13.7
		Perpendicularity		13.8
		Angularity		13.9
	Location tolerances	Position		13.10
		Concentricity and coaxiality		13.11
		Symmetry		13.12
	Run-out tolerances	Circular run-out		13.13
		Total run-out		13.14

Table 2 - Additional symbols

Descriptions		Symbols	Clauses
Toleranced feature indications	direct		5
	by letter		6.4
Datum indications	direct		7
	by letter		
Datum target			NES D0402
Theoretically exact dimension			9
Projected tolerance zone			10
Maximum material condition			11

4. TOLERANCE FRAME

4.1 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 tolerances;
- tolerance value in the unit used for linear dimensions. This value is preceded by the sign ϕ 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).

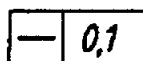


Figure 3

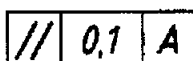


Figure 4

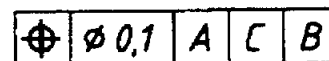


Figure 5

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

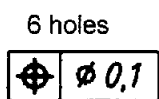


Figure 6

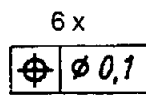


Figure 7

4.3 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).

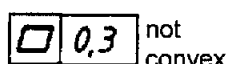


Figure 8

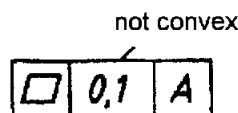


Figure 9

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

○	0,01	
//	0,06	B

Figure 10

5. 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 dimension line) when the tolerance refers to the line to surface itself (See figures 11 and 12.).

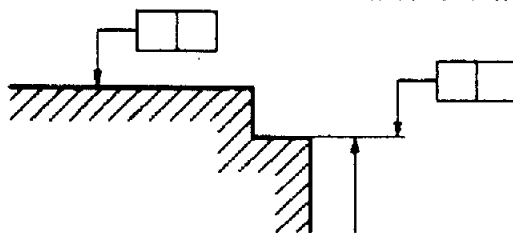


Figure 11

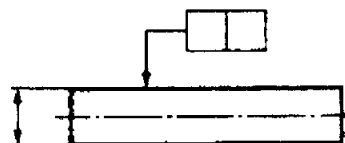


Figure 12

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

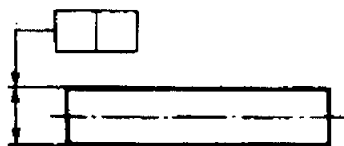


Figure 13

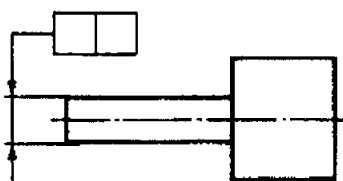


Figure 14

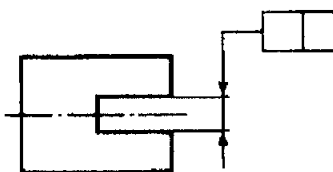


Figure 15

- 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.)

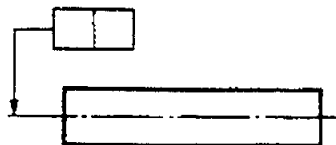


Figure 16

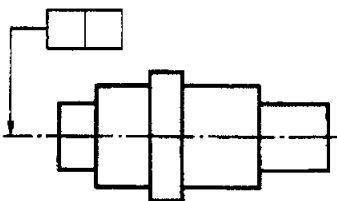


Figure 17

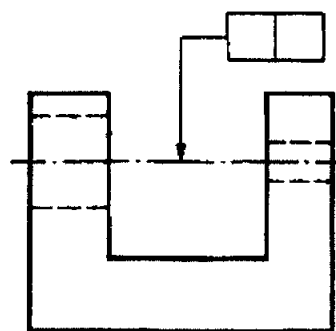


Figure 18

Note - Whether a tolerance should be applied to the contour of a cylindrical or symmetric feature or to its axis or median plane respectively depends on the functional requirements.

6. TOLERANCE ZONES

6.1 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 tolerated, unless the tolerance value is preceded by the sign ϕ (See figures 19 and 20).

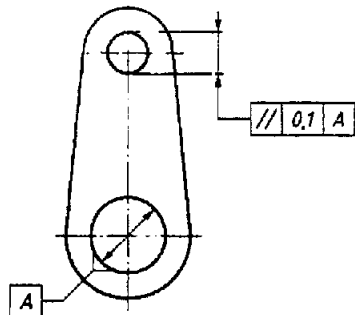


Figure 19

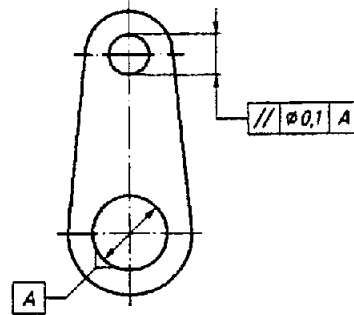


Figure 20

6.2 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).

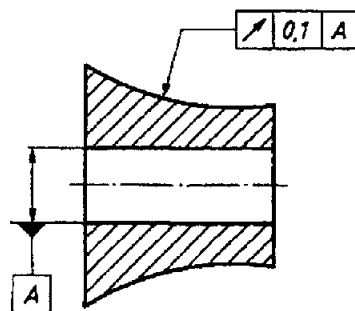


Figure 21

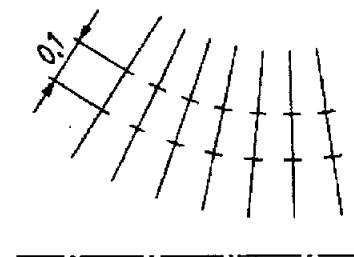


Figure 22

6.3 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).

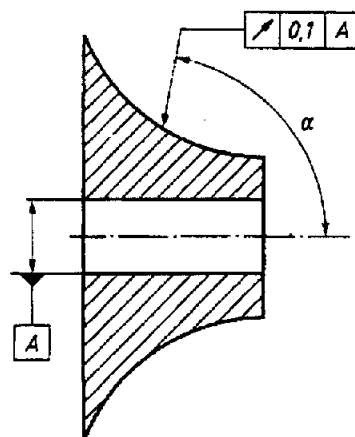


Figure 23

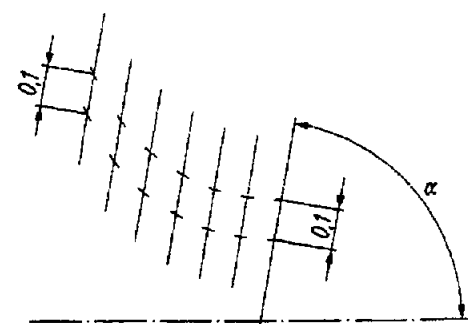


Figure 24

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

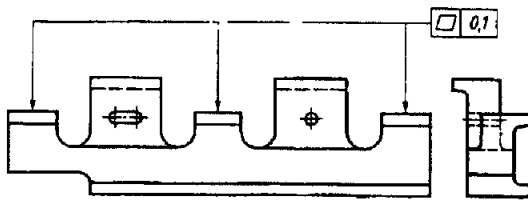


Figure 25

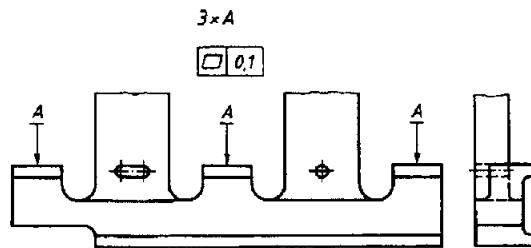


Figure 26

6.5 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.)

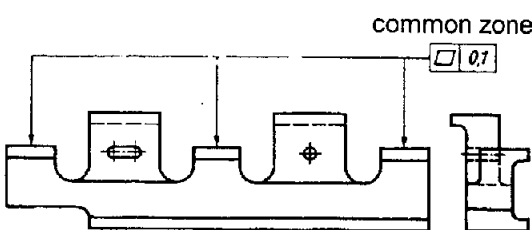


Figure 27

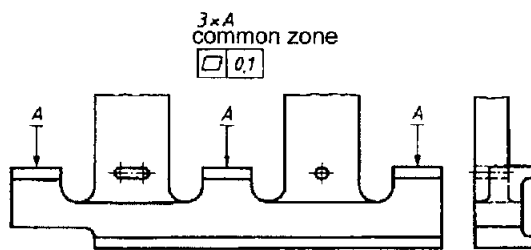


Figure 28

7. DATUMS

7.1 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.)

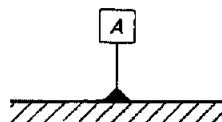


Figure 29

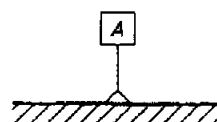


Figure 30

7.2 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.).

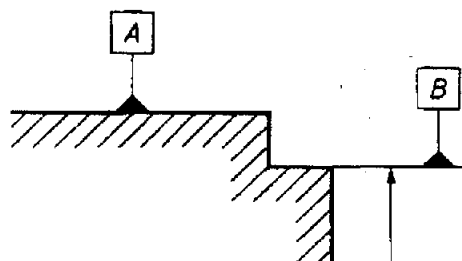


Figure 31

- as an extension of the dimension line when the datum 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).

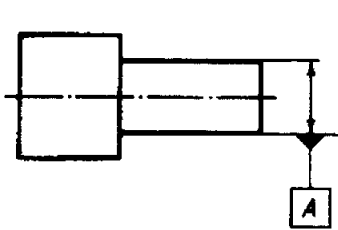


Figure 32

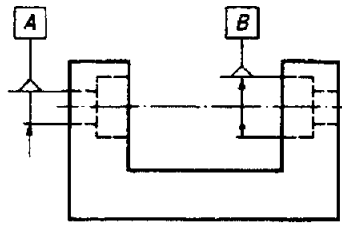


Figure 33

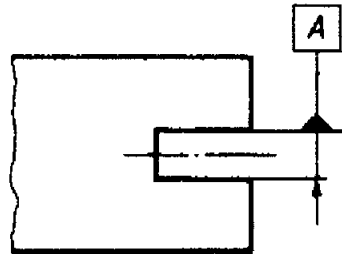


Figure 34

- on the axis or central 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).

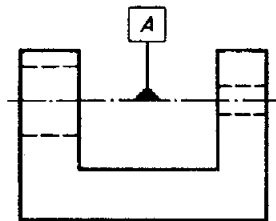


Figure 35

7.3 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).

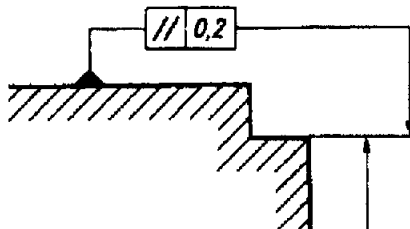


Figure 36

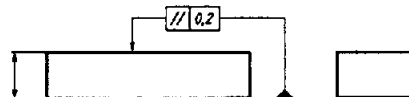


Figure 37

7.4 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 compartment (See figure 40.)

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

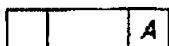


Figure 38

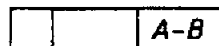


Figure 39



Figure 40

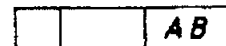


Figure 41