Chapter 10

Application of New Generation Geometrical Product Specifications—Position Tolerancing

Pavlina Kalcheva Toteva-Lyutova, Stoyan Dimitrov Slavov and Krassimira Stoyanova Koleva

Additional information is available at the end of the chapter

http://dx.doi.org/10.5772/65412

Abstract

The geometrical product specifications (GPSs) from new generation are composed of several standards issued by the ISO/TC 213. They are related to the way of denoting the requirements in the design engineering drawings, such as drawing indication, definition of tolerance and values of specifications, characteristic, parameters and definitions of actual features. They also include requirements relating to compare verification, measure instrument and calibrate size, distance, radius, angle, form and position of geometrical features, roughness profile, waviness profile, primary profile, surface imperfection and edges. A lot of new and mathematical terms, the size system, indications of dimensions other than linear sizes by using geometrical tolerances, uncertainty series, etc. are introduced in this chapter. The aim of this chapter is to explain the new requirements of new generation standards for geometrical product specifications related to positional deviation. The advantages and disadvantages of the possibility to indicate the accurate requirements for location of surfaces and axes are discussed.

Keywords: ISO, standards, specification, verification, dimensioning, tolerancing, position

1. Introduction

Locational deviation is such type of deviation, which determines the deviation of a feature (for example, surface, line and point) from its nominal location. The location is relevant to one or more (other) datum feature(s). The locational deviation also comprises the form deviation and the orientation deviation (for example, the surface, axis or median face; see Figure 1) [1]. The locational deviations are assessed over the entire feature, when not otherwise specified. Locational deviations are derived analogously to the size, form and orientation deviations.

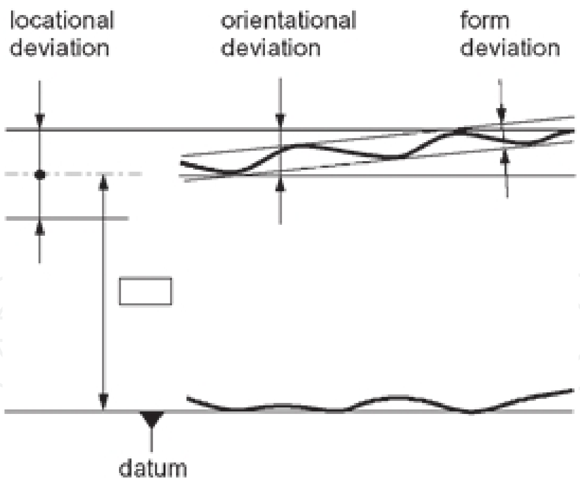


Figure 1. Form deviation, orientational deviation and locational deviation [1].

The definition of the positional tolerance is given in ISO 1101 [2], and methods for positional tolerancing are described in ISO 5458 [3].

In the method of positional tolerancing, the location of features is determined by theoretical exact dimensions and positional tolerances (points, axes, median faces and plane surfaces) relative one or more datum(s) to each other. In this case, symmetrical arrangement of the tolerant zone around the theoretically correct position is given.

2. Tolerancing of features by position

The location of features is one of the most frequently used applications of dimensions on technical drawings. Tolerancing may be either by coordinate tolerances applied to the dimensions or by geometric (positional) tolerancing.

Positional tolerancing is especially useful when applied on a maximum material condition (MMC) basis to groups or patterns of holes or other small features in the mass production of parts. This method meets functional requirements in most cases and permits assessment with simple gauging procedures.

2.1. Tolerancing methods

The location of a single hole is usually indicated by rectangular coordinate dimensions extending from suitable edges or other features of the pan to the axis of the hole. Other dimensioning methods, such as polar coordinates, may be used.

There are two standard methods of tolerancing the location of holes: coordinate and positional tolerancing.

1. Coordinate tolerancing (Figure 2 a) refers to tolerances applied directly to the coordinate dimensions or to applicable tolerances specified in a general tolerance note.

http://dx.doi.org/10.5772/65412

2. Positional tolerancing (Figure 2 b –d) refers to a tolerance zone within which the centre line of the hole or shaft is permitted to vary from its true position. Positional tolerancing can be further classified according to the type of modifying associated with the tolerance positional tolerancing, reciprocity requirement (RPR);positional tolerancing, maximum material condition basis (MMC) andpositional tolerancing, least material condition basis (LMC).

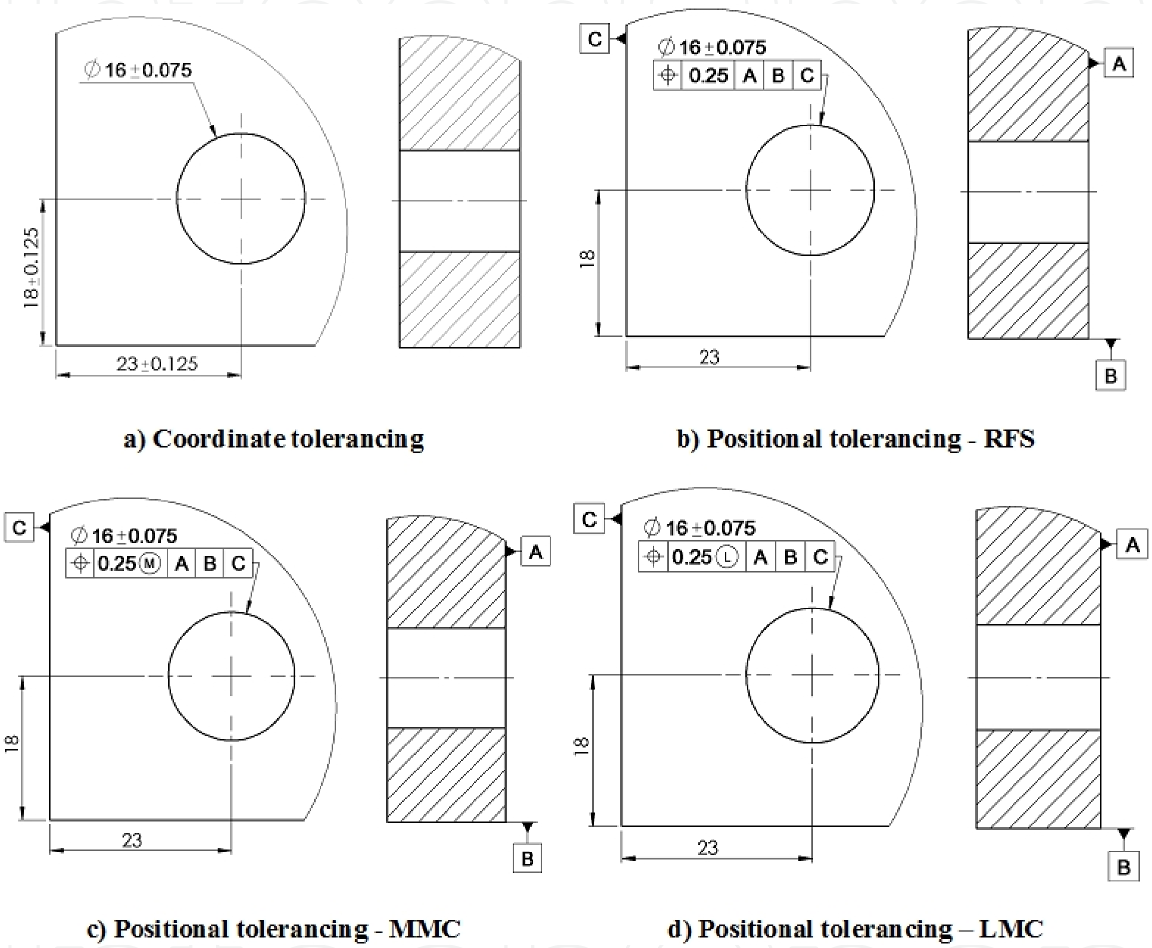


Figure 2. Overview of the standard methods of tolerancing the location of holes and the geometric tolerance parameters and symbols.

These positional tolerancing methods are part of the system of geometric tolerancing.

When the MMC or LMC modifying symbol is not shown in the feature control frame, it is understood that RPR applies.

Figure 3 a –d shows the tolerance zones by this way of tolerancing.

Any of these tolerancing methods can be substituted for another, although with differing results. It is necessary, however, to first analyse the widely used method of coordinate tolerancing to explain and understand the advantages and disadvantages of the positional tolerancing methods.

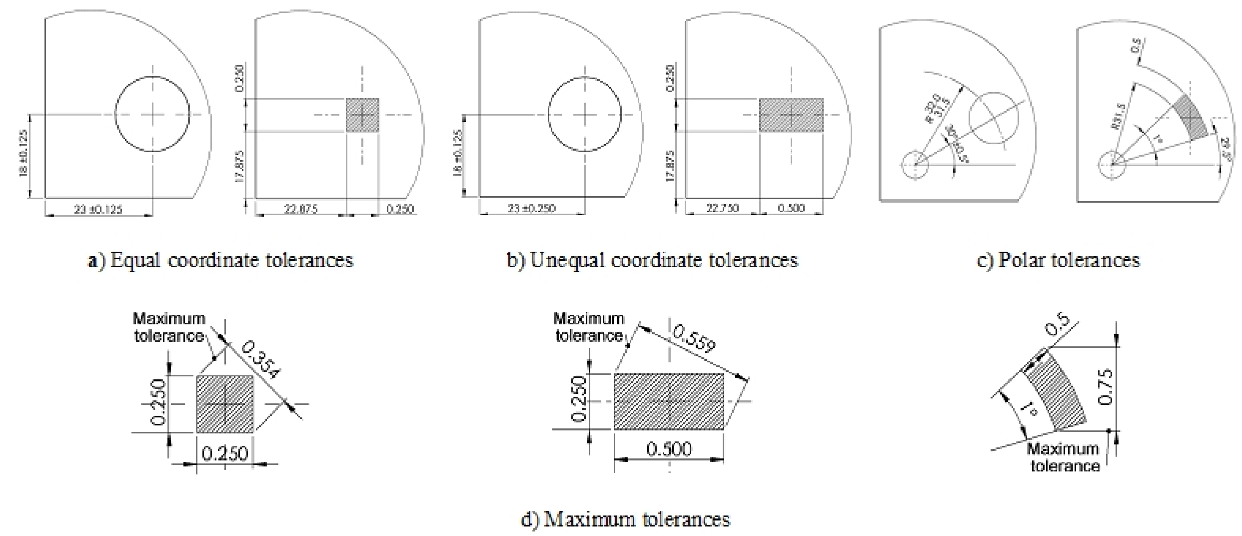


Figure 3. Overview of the coordinate and polar tolerances.

2.1.1. Advantages of coordinate tolerancing

The advantages of direct coordinate tolerancing are as follows:

It is simple and easily understood and, therefore, is a method commonly used.

It permits direct measurements to be made with standard instruments and does not require the use of special-purpose functional gauges or other calculations.

2.1.2. Disadvantages of coordinate tolerancing

There are a number of disadvantages to the direct tolerancing method:

It results in a square or rectangular tolerance zone within which the axis must lie. For a square zone, this permits a variation in a 45° direction of about 1.4 times the specified tolerance. This amount of variation may necessitate the specification of tolerances that are only 70% of those that are functionally acceptable.

It may result in an undesirable accumulation of tolerances when several features are involved, especially when chain dimensioning is used.

It is more difficult to assess clearances between mating features and components than when positional tolerancing is used, especially when a group or a pattern of features is involved. It does not correspond to the control exercised by fixed functional GO gauges often desirable.

2.1.3. Main advantages of positional tolerancing

In comparison with dimensional coordinate tolerancing, the positional tolerancing has the following more important advantages:

http://dx.doi.org/10.5772/65412

Function-related tolerancing with the largest possible tolerances is possible because functional relationships are better and directly indicated, and relationships to one or more datum(s) can be indicated unequivocally.

It is possible to indicate cylindrical tolerances. In comparison with tolerances of rectangular cross section, 57% larger tolerances are possible. In most cases, this is related to the function-related tolerancing, for example, mating of cylindrical surfaces (such as bolts with holes).

A simple application of the maximum material requirement with additional gain of tolerance is possible when using positional tolerancing.

2.2. Position tolerance according to ISO 1101

In ISO 1101 [2], the position tolerance of a point (see Figure 4), a flat surface or a median plane (see Figure 5) and a line (see Figure 6) are given.

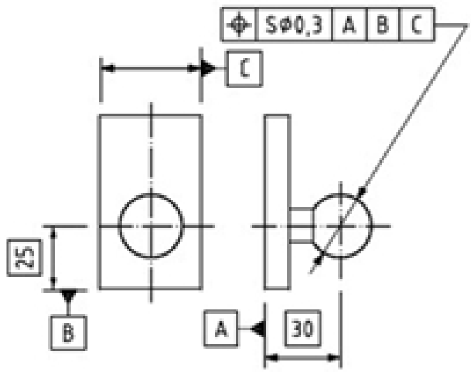


Figure 4. Position tolerance of a point.

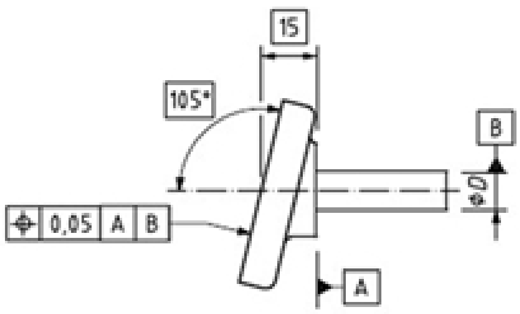


Figure 5. Position tolerance of a flat surface or a median plane.

The definition of the actual centre of a sphere has not been standardized in ISO 1101. According to this standard, the actual (extracted) centre of the sphere must be within a spherical zone (with diameter 0.3), and the centre of this zone shall coincide with the theoretically exact position of the sphere, according to datum planes A and B and to datum median plane C (see Figure 4).

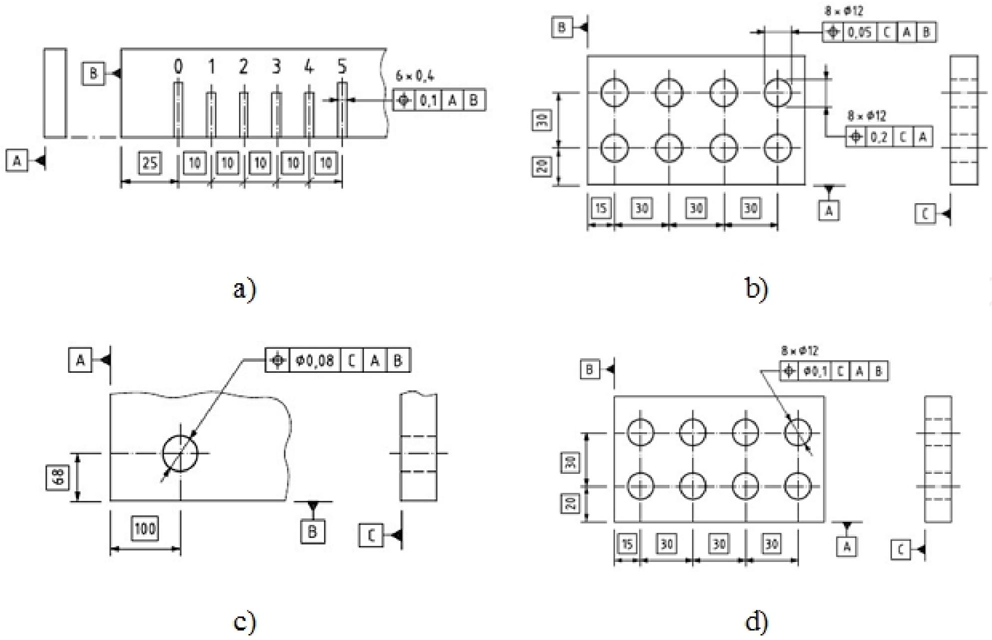


Figure 6. Position tolerance of a line.

The actual (extracted) centre line of each of the scribe lines shall be distributed between two parallel planes 0.1 apart, which are symmetrically located about the theoretically exact position of the considered line, according to datum planes A and B (see Figure 6a).

The actual (extracted) middle line of each hole shall be located between two pairs of parallel planes. These planes are positioned 0.05 and 0.2 apart, respectively, in the direction specified and perpendicular to each other. Each pair of parallel planes is orientated according to the datum system and symmetrically distributed about the theoretically exact position of the considered hole, according to datum planes C, A and B (see Figure 6b).

The actual (extracted) centre line must be within a cylindrical zone of diameter 0.08. Their axis must be coinciding with the theoretically exact position of the considered hole, according to datum planes C, A and B (see Figure 6c).

The actual (extracted) centre line of each hole must be within a cylindrical zone of diameter 0.1. Their axis must be coinciding with the theoretically exact position of the considered hole, according to datum planes C, A and B (see Figure 6d).

The actual (extracted) surface shall be distributed between two parallel planes 0.05 apart, which are symmetrically located about the theoretically exact position of the surface, according to datum plane A and datum axis B (see Figure 5).

3. Material condition basis

In ISO 2692:2014 [4], the maximum material requirement, the least material requirement and the reciprocity requirement are defined. The correlation between dimensional and geometrical precision of the element is determined by functional requirements, such as providing opportunities for assembling parts with clearance (in this case, requirements for maximum material are established) or, if necessary, to restrict the minimum wall thickness between the elements of one and the same part (in this case, it implies a requirement for a minimum material).

3.1. Reciprocity requirement (RPR)

Reciprocity requirement (RPR) is only possible after a toleranced feature has been used. RPR should be included as an additional requirement on drawings when maximum material requirement (MMR) or least material requirement (LMR) is used. In this case, the RPR is indicated by the symbol R placed after the symbol M, or the symbol R placed after the symbol L.

Thereby, the size tolerance of the feature is replaced by the collective requirements MMR and LMR by using the additional requirement RPR. The size can take full advantage of the maximum material virtual condition (MMVC) and the least material virtual condition (LMVC) by means of RPR. The choice of distribution of variation allowance between dimensional and geometrical tolerances based on manufacturing capabilities is possible when RPR is used. The indication "0 M" can be expressed with RPR whilst maintaining the same work piece characteristics.

3.1.1. Maximum material requirement and RPR

In Figure 7a and b, examples are shown how the RPR can be indicated on drawings by the symbol R placed after the symbol M itself placed after the geometrical tolerance of the derived feature of the feature of size in the tolerance indicator to alter the maximum material requirement for the surface(s).

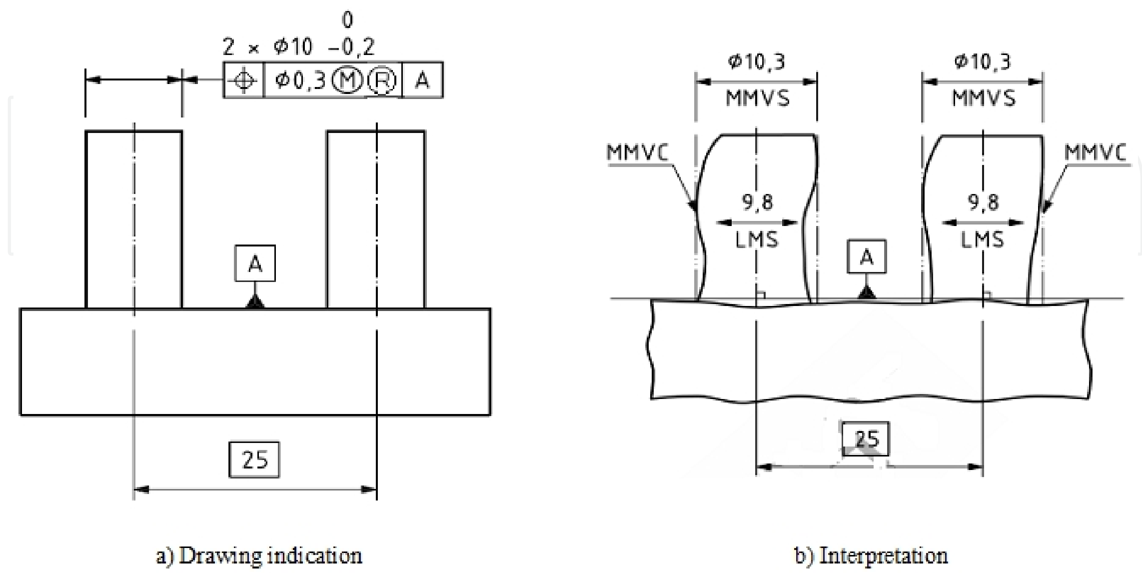


Figure 7. Examples of MMR for two external cylindrical features based on size and position (location) requirements.

An example with the assembly comprising a plate with two holes 25 mm apart is illustrated in Figure 7b. There is a requirement about holes to be perpendicular to the contact surface of the plate. The following interpretation is according to rules and definitions given in ISO 2692:2014:

a. The extracted feature of the toleranced pins shall not violate the maximum material virtual condition (MMVC), which has the diameter of 10.3 mm.

b. The extracted feature of the toleranced pins shall have everywhere a local diameter larger than LMS = 9.8 mm. The RPR requirement allows the size tolerance to increase.

The location of the two MMVCs is theoretically correct —at a distance of 25 mm relative to each other and perpendicular to the datum A. The RPR allows the dimensional tolerance increasing when the geometrical deviation does not take full advantage of the maximum material virtual condition (MMVC).

3.1.2. Least material requirement and RPR

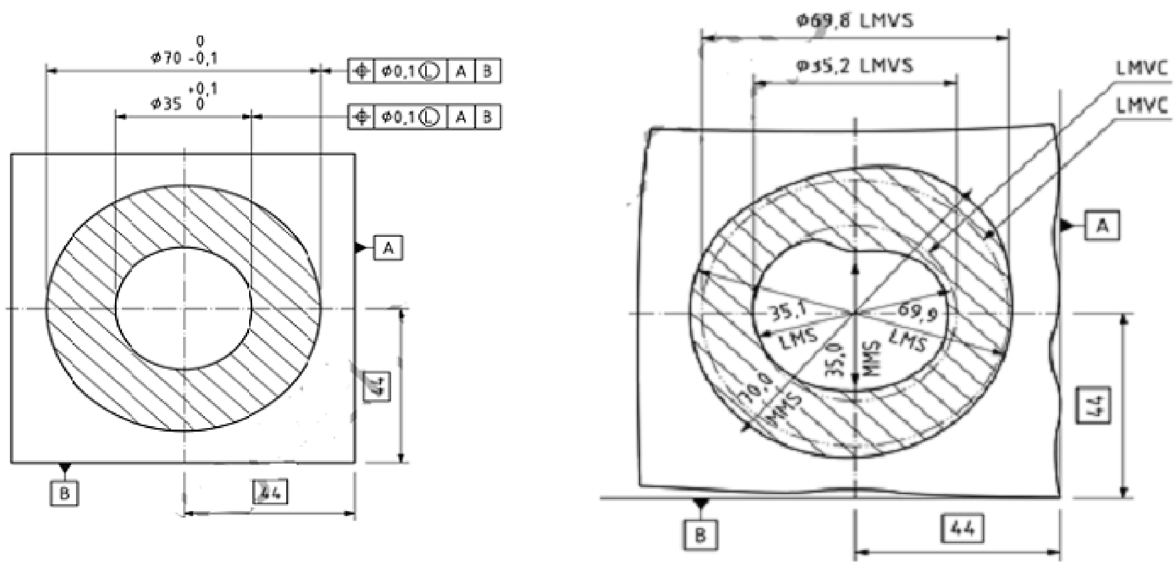
In this case, the RPR indicated on drawings by the symbol R placed after the symbol L itself placed after the geometrical tolerance of the derived feature of the feature of size in the tolerance indicator to alter the least material requirement for the surface(s).

3.1.3. Related datum features with maximum material requirement

The maximum material requirement for datum features results in the following three independent requirements:A requirement for the surface non-violation of the maximum material virtual condition (MMVC). The maximum material virtual condition (MMVC) of the related datum feature shall not be violated by the extracted (integral) datum feature from which the datum is derived. A requirement for MMS when there is no geometrical tolerance or when there is a geometrical tolerance not followed by the symbol M. When the related datum feature has no geometrical tolerance or has a geometrical tolerance of form not followed by the symbol M, the size of the maximum material virtual condition (MMVC) of the related datum feature is the maximum material size (MMS). A requirement for MMS when there is a geometrical tolerance of form followed by the symbol M. When the datum feature has a geometrical tolerance of form and this tolerance is followed by the symbol M (see Figure 8), the size of the maximum material virtual condition (MMVC) of the related datum feature is the maximum material size (MMS) plus (for external features of size) or minus (for internal features of size) the geometrical tolerance.

The symbol M is placed on drawings after the datum letter(s) in the tolerance indicator, when MMR applies to the datum feature. If the datum is obtained from a feature of size, the use of symbol M after the datum letter is only possible indication. The corresponding sequence of letters identifying the common datum is indicated within parentheses, when maximum or least material requirement applies to all elements of the collection of surfaces of a common datum. The sequence of letters identifying the common datum is not indicated within parentheses, when maximum or least material requirement applies only to one element of the

http://dx.doi.org/10.5772/65412



a) Drawing indication b) Interpretation

Figure 8. Example of LMR for two concentric cylindrical features (internal and external) both controlled by size and location (position) to the same datum systems A and B.

collection of surfaces of a common datum. In this case, the requirement applies only to the feature identified by the letter placed just before the modifier, and it specifies for the surface(s) in the following rules.

3 1.4. Related datum features with least material requirement

When the toleranced feature applies to the least material requirement (LMR), the indication on the drawing is the symbol L placed after the geometrical tolerance of the corresponding feature of the size in the indicator of the tolerance.

The symbol L is applied to the tolerancing of the features on both sides of the wall to fully control the minimum wall thickness. LMR can be implemented in two different ways, as follows.

In Figure 8, it shows the location tolerance for the two different sides of the wall, which can refer to the same datum axis or datum system. The symbol L, in this case, applies to the two toleranced features.

The intended function of the part illustrated in Figure 8 is the ability to resist internal pressure and prevent breakout.

The interpretation is based on the following rules and definitions given in ISO 2692:2014:

a. The diameter of least material virtual condition is LMVC = 69.8 mm, and the LMVC of the external feature shall be fully contained in the material.

b. Everywhere a local size of the extracted feature of the external feature shall be smaller than MMS = 70.0 and larger than LMS = 69.9 mm.

c. The diameter of the internal feature of the least material virtual condition is LMVC = 35.2 mm, and the LMVC shall be fully contained in the material.

d. Everywhere a local size of the extracted feature of the internal feature shall be larger than MMS = 35.0 mm and smaller than LMS = 35.1 mm.

e. The least material virtual condition (LMVC) of both the external and the internal features shall be in a theoretically exact orientation and the location relative to the datum system at a position of 44.44 mm.

It is permissible to have reference between location tolerances of the derived features for both sides of the wall as the datum. The tolerance for the toleranced feature and the datum letter is followed by the symbol L in this case.

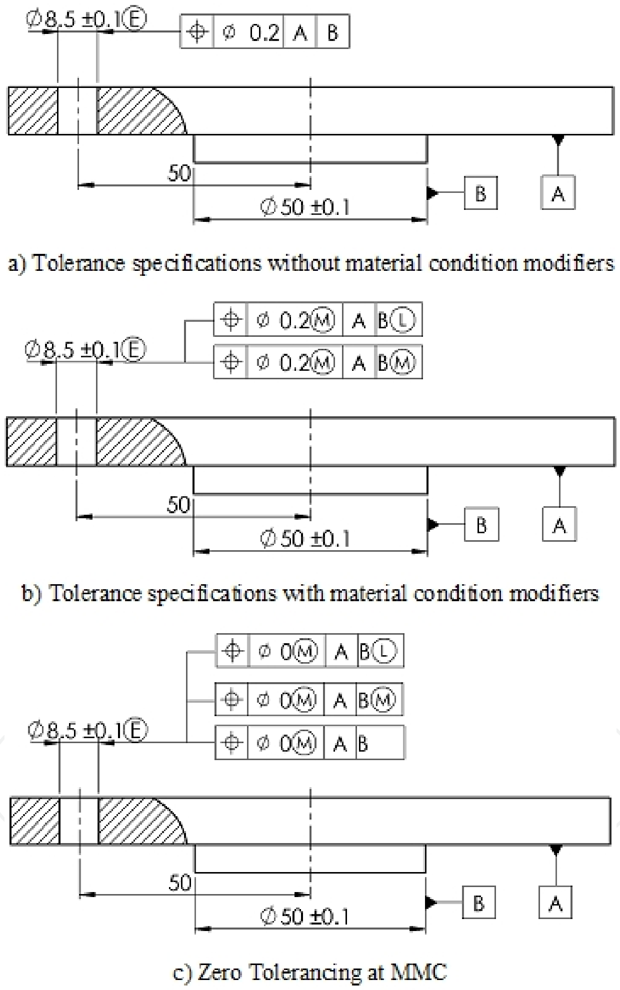


Figure 9. Examples of application of the new concepts to indicate the positional deviation.

If the datum is obtained from a feature of size, the use of the symbol L after the datum letter is only possible. The corresponding sequence of letters identifying the common datum is indicated within parentheses, when maximum or least material requirement applies to all elements of the collection of surfaces of a common datum. The sequence of letters identifying the common datum is not indicated within parentheses, when maximum or least material requirement applies only to one element of the collection of surfaces of a common datum. In this case, the requirement applies only to the feature identified by the letter placed just before the modifier.

Figure 9 shows practical examples of application of the new concepts to indicate the positional deviation in the drawings. In Figure 9 a, dimensions are indicated by E, if required a high mounting requirement is used the symbols for MMC and LMC (see Figure 9b) and O M inform to envelop requirements with connection to form and location tolerances (see Figure 9c). Some instructions about selecting tolerance of position control modifiers are given in Figure 10.

Modifier	Commonly used in these functional applications	Relative cost to produce and verify
Ⓜ	• Assembly; • Location of a non-critical FOS;	Lowest
Ⓛ	• Minimum wall thickness; • Minimum part distance; • Minimum machine stock; • Alignment;	Greater than MMC; Less than RFS
RFS invoked by showing no modifier	• To control a symmetrical relationship; • When the effects of bonus or datum shift will be detrimental to the function of the part; • To control minimum machine stock; • Centering; • Alignement;	Highest

Figure 10. Instruction about selecting tolerance of position control modifiers.

ISO 5458 [2] describes the principle of positional tolerancing for the location of regular and irregular features. However, for clarity, only regular-shaped features such as holes, bolts, studs or pins, parallel-sided slots, keys and keyways have been shown.

In Table 1, some specification modifiers [2] are given.

Description	Symbol
Combined zone	CZ
Separate zones	SZ
Orientation only	><
Simultaneous requirement N° i	SIMi

Table 1. Specification modifiers for linear size

By default, based on the independency principle defined in ISO 8015 [4], a geometrical specification of form, orientation, location or run-out, without modifier, applied to *n* geometrical features (*n* being greater than one) is equivalent to *n* independent geometrical specifications: each geometrical feature shall be considered individually and each specification shall be considered individually (independent between them). The resulting independent tolerance zones correspond to an implicit indication of the separate zone (SZ) modifier: the "all around" modifier does not create itself a united feature or a pattern.

When positional tolerancing is applied to several geometrical features and all the non-redundant degrees of freedoms of the tolerance zones are unlocked, either the separate zone (SZ) modifier or the combined zone (CZ) modifier shall always be indicated in the second compartment of the tolerance indicator. To create one homogeneous pattern, the CZ modifier shall be indicated in the tolerance section of the tolerance indicator. To create a new level of pattern defined as a homogeneous pattern of more than one homogeneous pattern, an additional CZ modifier shall be indicated after the sequence defining the previous level of pattern. One CZ in the tolerance section defines a single pattern. The sequence CZ in the tolerance in the tolerance section defines a pattern of patterns of patterns (pattern of Level 3).

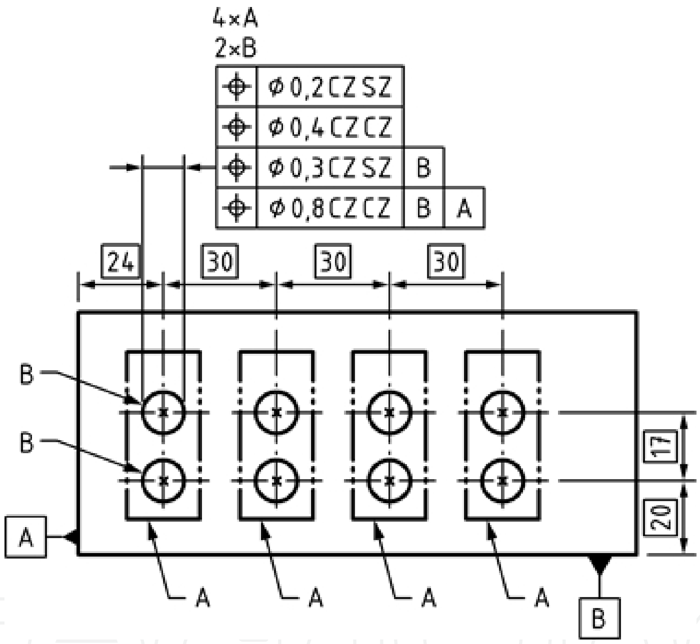


Figure 11. Example of identification of a pattern for repetition.

Figure 11 shows four specifications: The first specification (CZ SZ without datum) manages four independent patterns. For each pattern, the specification considers the following: As a toleranced feature, the collection of two extracted axes As a tolerance zone, the combined zone composed of two cylindrical zones of diameter 0.2 mm constrained in location between them at 17 mm. The second specification (CZ CZ without datum) manages four dependent patterns, resulting in only one specification considering the following: As a toleranced feature, the collection of eight extracted axes As a tolerance zone, the combined zone composed of eight cylindrical zones of diameter 0.4 mm constrained in location between them at 17 mm in a direction and 30 mm in a perpendicular direction. The third specification (CZ SZ with datum

http://dx.doi.org/10.5772/65412

B) manages four independent patterns constrained in location from B. For each pattern, the specification considers the following: As a toleranced feature, the collection of two extracted axes As a tolerance zone, the combined zone composed of two cylindrical zones of diameter 0.3 mm constrained in location between them at 17 mm and constrained from datum B at 20 mm. The fourth specification (CZ CZ with datum systems B and A) manages four dependent patterns, constrained in orientation from datum B (perpendicularly) and in location from datum A, resulting in only one specification considering the following: As a toleranced feature, the collection of eight extracted axes As a tolerance zone, the combined zone composed of eight cylindrical zones of diameter 0.8 mm constrained in location between them at 17 mm in one direction and 30 mm in a perpendicular direction and constrained from the datum systems B and A, respectively, at 20 and 24 mm.

To avoid ambiguity, when a positional specification applies to several features and not all of the non-redundant degrees of freedom for the tolerance zones are locked, either an SZ modifier or a CZ modifier is always indicated.

All types of geometrical characteristic symbol can be used to establish a geometrical specification in a pattern. However, to create a pattern specification, a CZ modifier can be indicated in the second compartment of the tolerance indicator.

4. Conclusion

The tolerances of position can be used to control the theoretically exact location of features and simulate mating part relationships. They may be modified to MMC and LMC to ensure flexibility in verification and simulation and may be used to control features in coaxial relationships, to provide symmetrical controls of features relative to a centre plane and to ensure generous margins of cost savings.

In the present work, the new requirements of new generation standards for geometrical product specifications (GPSs) related to positional deviation, the advantages and disadvantages of the possibility to indicate the accurate requirements for location of surfaces and axes and some instructions about selecting tolerance of position control modifiers are given.

Reviewed standards for GPS can only be used effectively after the relevant staff has been trained to use and interpret the symbol language. GPS should primarily be used for new drawings and new projects. Only the problematic ones amongst the old drawings should be considered for translation into GPS.

Engineering drawings with incomplete or incorrectly indicated tolerances can result in a lot of issues, such as questions for the production planning, manufacturing and/or inspection engineers, necessity of reworking, occurrence of defects in products, etc. Only engineering drawings with correct and completely indicated tolerances can provide manufacturing of precise as necessary and economic as possible work pieces. This is an important point in achieving business competitiveness.

Author details

Pavlina Kalcheva Toteva-Lyutova[1*], Stoyan Dimitrov Slavov[1] and Krassimira Stoyanova Koleva[2]

*Address all correspondence to: pavlina_toteva@abv.bg

1 Department of Technology of Machine Tools and Manufacturing, Technical University of Varna, Bulgaria

2 Department of Technical Mechanics, Technical University of Varna, Bulgaria

References

[1] Henzold Georg. Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection: A Handbook for Geometrical Product Specification Using ISO and ASME Standards. 2nd ed. Butterworth-Heinemann; 2006. 416 p.

[2] ISO 1101. Geometrical product specifications (GPS) —Geometrical tolerancing —Tolerances of form, orientation, location and run-out (ISO 1101:2012, including Cor. 1:2013). CEN/TC290; 2013. 114 p.

[3] ISO/DIS 5458. Geometrical product specifications (GPS) — Geometrical tolerancing — Positional and pattern tolerancing. Forthcoming.

[4] ISO 2692. Geometrical product specifications (GPS) —Geometrical tolerancing — Maximum material requirement (MMR), least material requirement (LMR) and reciprocity requirement (RPR) (ISO 2692:2014). CEN/TC 290; 2014. 57 p.

[5] ISO 5459. Geometrical product specifications (GPS) —Geometrical tolerancing —Datums and datum systems (ISO 5459:2011). CEN/TC 290; 2011. 91 p.

[6] ISO 8015. Geometrical product specifications (GPS) —Fundamentals —Concepts, principles and rules (ISO 8015:2011). CEN/TC 290; 2011. 20 p.