ISO GPS AND ASME GD&T STANDARDS – DIFFERENCES AND SIMILARITIES IN DEFINITIONS OF MEASURANDS

Zbigniew Humienny[1)], Paweł Zdrojewski[2)]

1) Warsaw University of Technology, Faculty of Automotive and Construction Machinery Engineering, Institute of Machine Design Fundamentals, Narbutta 84, 02-524 Warsaw, Poland (✉ zbigniew.humienny@pw.edu.pl)

2) Łukasiewicz Research Network – Institute of Aviation, Aleja Krakowska 110/114, 02-256 Warsaw, Poland (pawel.zdrojewski@ilot.lukasiewicz.gov.pl)

Abstract

Geometrical tolerances as tricky measurands are indicated. Crucial differences between the ISO and ASME geometrical tolerancing standards are discussed. It is demonstrated that, in many cases, both systems have different default rules. Moreover, for some identical graphical indications, interpretations are different. On the other hand, the standards contain similar arrangements in many cases. It is underlined that nowadays, due to the progressing globalisation, it is necessary to know these standards, bearing in mind that suppliers or customers specify requirements according to provisions from particular standards implemented in their companies. The above justifies the need for research exploring differences and similarities in both systems of standards. It is shown that the ISO GPS system standards, due to default independency principle, prefer to set production as cheaply as possible, while ASME, due to default provisions (*e.g.* Rule #1, simultaneous requirement) puts stress on controlling product geometry more strictly, which is sometimes unnecessary.

Keywords: geometrical tolerancing; geometrical product specifications; ISO GPS system; ISO 1101; ASME Y14.5; engineering design; GD&T.

1. Introduction

The fundamental issue in measurements is the correct and unique definition of the measurand, *i.e.* the quantity intended to be measured. It is stated in [1] that "*Metrology is not just a process of measurement that is applied to an end product. It should also be one of the considerations taken into account at the design stage. According to the Geometrical Product Specification (GPS) model, tolerancing and uncertainty issues should be taken into account during all stages of design, manufacture and testing.*" In order to define a geometrical characteristic as a measurand and to communicate between the designer, the manufacturer and the metrologist, graphic language is used, which must be understandable for all three parties. In the case of technical product documentation, this language is defined in the standards (Fig. 1). Unfortunately, quite often,

designers are unaware of significant differences between the ISO GPS system – International Organization for Standardization – Geometrical Product Specifications [2–4] and the ASME Geometric Dimensioning and Tolerancing (GD&T) system [5] that sometimes use identical graphical notation to set different measurands.

Fig. 1. (a) Percentage of drawings that use a specific standard in international and US companies; (b) use of standards in US companies; (c) use of standards in multinational companies; (d) use of standards in Polish companies [18].

This paper aims to show and explore the main differences between the ISO GPS system and the ASME GD&T system because, for example, in the aviation or automotive industry, incorrect interpretation of the specification may be a factor posing a threat to the life and health of users. The paper [6] is focused on comparing the influence of different evaluation-datum axes on the measuring value of aero-engine rotor coaxiality. The process of providing flight safety consists of correct operation of many people and systems, all of which are subject to the risk of error. Usually, a catastrophe results from a chain of errors, and it is enough to break one of its links to avoid the disaster. Design and manufacturing errors can be limited by standards, regulations and guidelines, which result from knowledge gained over many years of experience [7].

The differences between the ISO GPS and ASME Y14.5 geometrical tolerancing systems are addressed in [8–11]. However, many publications have some inconsistencies, which may mislead the readers. This paper contains many graphical examples and focuses on differences in indications and interpretations which have not been previously fully addressed in other publications. The topic discussed in this paper is highly significant in industrial practice [12], which is also emphasised in [10], presenting the significance of understanding the differences in standards and verification of features because that may inflate the risk of a wrong decision to accept or reject a workpiece.

The uncertainty evaluation of coordinate measurement based on the point-plane distance model is presented in [13]. As an example of application, the case of the position of the centre of a ball with respect to the datum plane is considered. The issue of uncertainty assessment is crucial for conformity assessment, but during measurement the first question is how the ball centre that determines actual ball location shall be established from the cloud of points collected on an actual ball surface – the default ISO GPS and ASME approaches are different.

Geometrical tolerances given according to any tolerancing system are part of *Product Manufacturing Information* and imply the verification of geometrical characteristics. The possibility of applying *Artificial Intelligence* (AI) in the metrology of geometrical quantities with the usage of coordinate measurement technology is discussed in [14]. It is underlined in [14] that AI will be able to assist a metrologist. However, effective application of AI requires that a designer has selected and consequently applied a particular tolerancing system, and therefore, the differences

between the ISO GPS system standards and ASME Y14.5 standards shall be investigated and un-covered. It shall be mentioned that the actual tolerancing systems are insufficient for all industrial applications and their further development taking into account new measuring technologies [15] is required. The fundamental deficit of the ISO GPS system and ASME Y14.5 is discussed in [16], and a path for enlarging the systems is shown.

Each drawing shall list the standard according to which the drawing was made. Since some companies use older editions of standards, it is helpful to be aware of changes and imperfections in different editions. The basic standard used mainly in the USA and Canada is ASME Y14.5 – issued by the American Society of Mechanical Engineers. The latest release of ASME Y14.5 is from 2018 [17]. On average, it is updated every dozen years (ANSI Y14.5M-1982; ASME Y14.5M-1994; ASME Y14.5-2009; ASME Y14.5-2018). It is estimated that half of the companies are using the 2009 release, and ¼ of the companies are still using the edition from 1994. Implementing the 2018 release is on the way [18].

Another graphical language that engineers use when creating technical documentation is the ISO GPS system. In the case of ASME, there is one main document, while in the case of ISO, there are several standards for geometrical dimensioning and tolerancing. Usually, for the average user, the primary standard is ISO 1101 [19], where form, orientation, location and runout tolerances are defined. ISO 286-1 [20] is also quite popular is also, describing the ISO coding system for linear sizes commonly known as the ISO system of limits and fits. Less known is ISO 8015 [21], which sets fundamentals, concepts, principles and basic rules; as well as ISO 14405-1 [22] with over ten definitions of dimensions and ISO 2692 [23], which is used to control specific functions of workpieces where size and geometry are interdependent. Also, the series of standards ISO 17450 [24, 25], that describe general concepts for geometrical specifications and verification, is practically unknown in industry.

There are sometimes doubts when referring to the ISO GPS system because the entire ISO GPS system, unlike the American approach, is described in several standards. In ISO 8015, it is stated that it is enough to indicate only one ISO standard to invoke the entire GPS system. There is no need to refer to all standards. The frequently mentioned standard is ISO 8015 or, alternatively, ISO 2768 [26], which gives the values for general dimensional and geometrical tolerances. The second part of ISO 2768 has recently been replaced by ISO 22081 [27] with a completely new concept of general specifications. In the case of ASME system usage in the drawings, a particular edition of ASME standard, *e.g.* ASME Y14.5-2018, shall be listed.

2. Differences in indicating and interpreting specifications

The differences in indicating and interpreting geometrical specifications result in different definitions of measurands. Especially dangerous is the case when indications in ISO and ASME drawings are graphically identical (same symbols in the same configuration) and a less educated drawing user is not aware that the meaning of the indications is different due to the application of another tolerancing system. Bellow, such cases are identified and discussed to help the metrologist establish the correct measurements.

2.1. Position tolerance

The significant difference between the standards is in the application of position tolerance which is frequently used in design practice. According to ASME, it can only be used to locate the features of size (FOS) [2, 17, 24]. FOS are defined by size, *i.e.* characteristics such as the diameter

of a cylindrical surface or the distance between two opposed parallel surfaces. In ISO [19], the position tolerance is recommended to be used for flat surfaces (Fig. 2a). Therefore, if it is required to achieve the same functional goals according to ASME, the position specification in Fig. 2a shall be substituted by a surface profile tolerance (Fig. 2b).
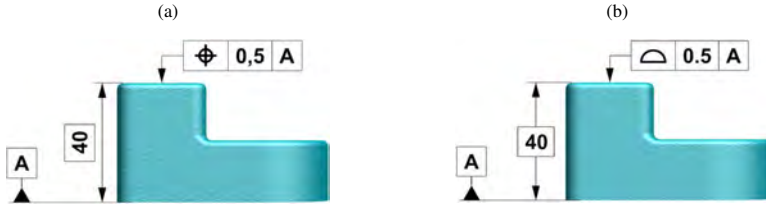

Fig. 2. (a) ISO – position tolerance applied to define the location of a plane (indication recommended in ISO and not permitted in ASME); (b) ASME specification equivalent to the requirement given in (a). [authors' drawing]

Another difference is shown in Fig. 3 – according to the ASME standard, two holes are considered as the pattern, so the angle between them is 180° by default. In the ISO GPS system, both holes with the indication "2x" according to the independency principle [21] are considered independently (Fig. 4). To obtain the same relation between the holes as in ASME, the modifier CZ (*combined zone* [19, 28, 29]) shall be added in the ISO drawing (Fig. 3a). According to ISO 1101 and ISO 5458, two independent tolerance zones can be obtained (Fig. 4) by changing the modifier from CZ to SZ (*separate zones* [28]).
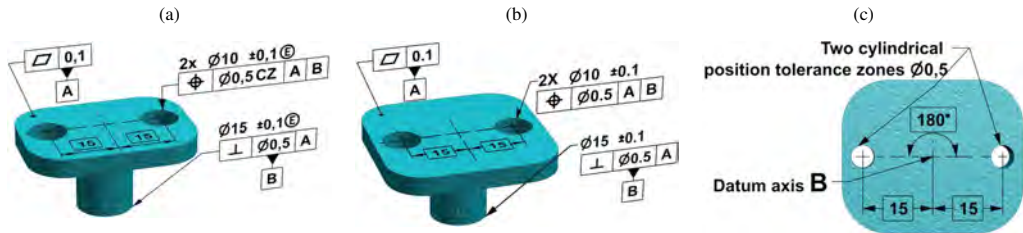

Fig. 3. Position tolerance specification according to (a) ISO and (b) ASME; (c) interpretation of the position tolerances for the specifications in drawings (a) and (b). [authors' drawing]
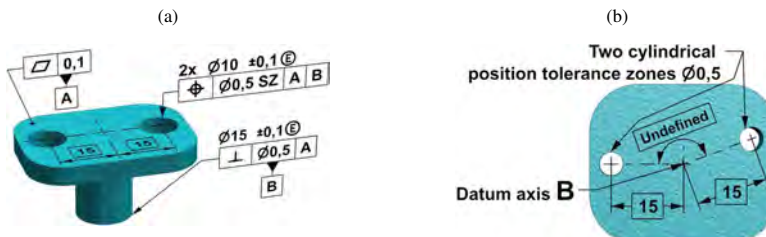

Fig. 4. (a) ISO indication; (b) permissible location of holes according to specification (a). [authors' drawing]

It is worth noting that in the latest edition of ISO 5458:2018 [28], it is stated that the modifier SZ or CZ, or CZR (*combined zone rotational only*) shall always be used when position tolerance is specified for several geometrical features that have at least one unlocked degree of freedom.

The provision in ISO 1101:2017, which is the latest edition of this standard, allows the use of the SZ indication optionally (by default, the tolerance zones are considered separately). Therefore, until ISO 1101:2017 is updated, this issue will have some inconsistency.

According to ASME, the repeated geometrical features to which the same datum system and modifiers apply by default form a pattern (Fig. 5, Fig. 6), and these features shall be analysed simultaneously [17]. It means that when the modifier (*maximum material requirement*) [23, 30] is used, the considered geometrical features shall be checked using one gauge (Fig. 6c). When the simultaneous verification is not required according to the expected workpiece function, the *note separate requirement* (SEP REQT) shall be added (Fig. 7). The approach is opposite to the ISO GPS system standards – the independency principle applies by default. So, when a designer wants geometrical features to be considered together, the modifier CZ or SIM (*simultaneous requirement*) shall be added [19, 28].
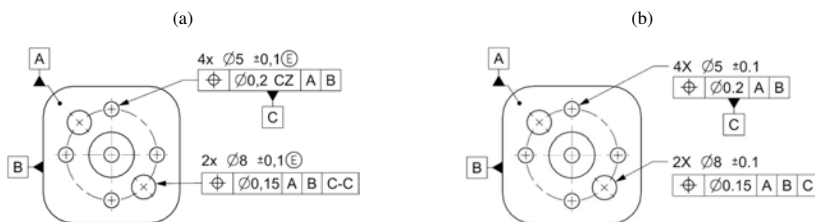

Fig. 5. Indication of a common datum created from a group of features according to (a) ISO and (b) ASME.
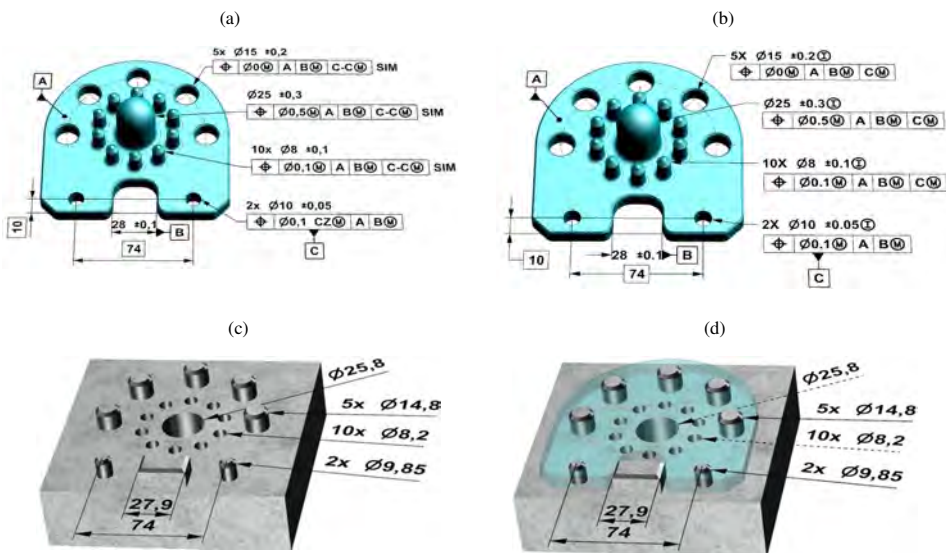[authors' drawing]

Fig. 6. Specifications that require simultaneous verification of geometrical features (a) ISO and (b) ASME; (c) gauge;
(d) workpiece on the gauge. [authors' drawing]

Another difference in notation can also be observed in Fig. 5 and Fig. 6. In the case of ISO, when datum A consists of several geometrical features, the reference to this common datum is made with the letters A-A (regardless of how many planes or holes it concerns, always two letters
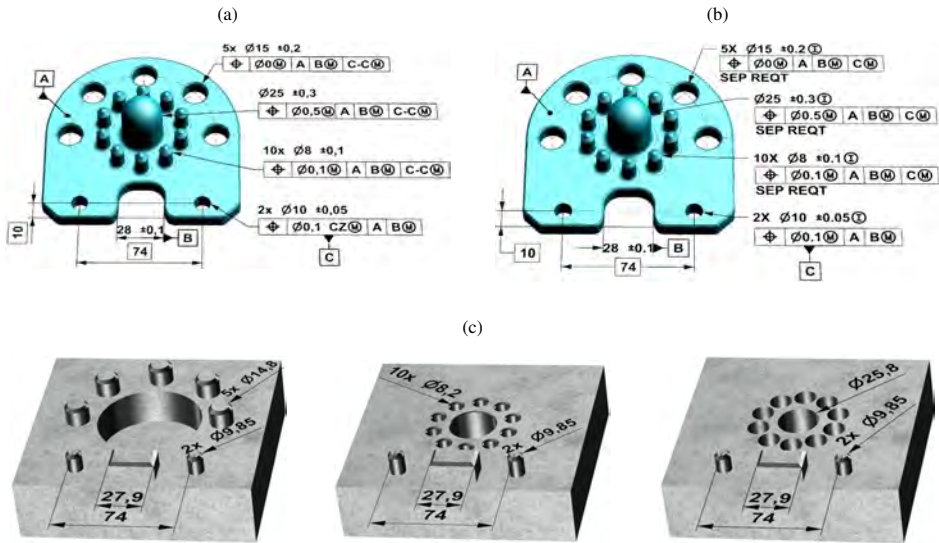
Fig. 7. Specifications that require independent verifications of each pattern of geometrical features (a) ISO and (b) ASME; (c) gauges. [authors' drawing]

separated by a hyphen shall be indicated), and in ASME reference to several planes or holes/pins, *etc*. from which a common datum is defined, it is indicated by a single letter.

2.2. Coplanarity and flatness tolerances

According to ASME, when two or more plane surfaces are drawn in the same plane, they shall be considered as one plane when the surface profile tolerance is applied (Fig. 8). According to the default ISO independency principle, each surface shall be considered separately. Thus, the symbol CZ after the tolerance value shall be indicated to achieve the same functional purpose when the ISO GPS system is used. The CZ modifier indicates that the collection of two extracted surfaces creates the toleranced feature. The toleranced feature shall be contained in the combined tolerance zone established by two pairs of parallel planes a distance 0.1 mm apart that are coplanar [19, 30].
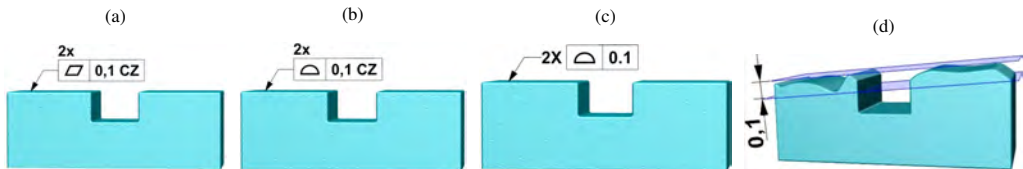

Fig. 8. Equivalent specifications according to (a, b) ISO and (c) ASME when two nominally coplanar surfaces shall be coplanar; (d) interpretation. [authors' drawing]

Both flatness and surface profile tolerances may be used in ISO when nominally flat surfaces are considered independently (Fig. 9). However, flatness tolerance is recommended [31] because flat and curvilinear surfaces with a large radius may appear as a straight line in the drawing. In ASME, when nominally flat surfaces are considered independently, flatness tolerance shall be used.
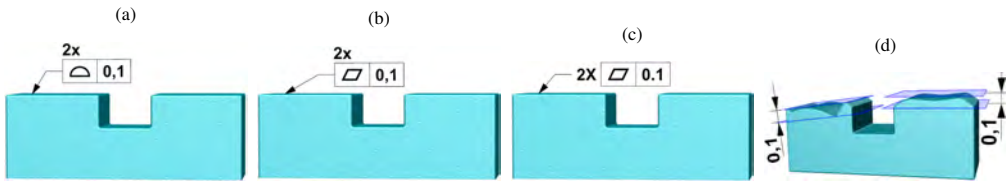
Fig. 9. Equivalent specifications when two nominally coplanar surfaces are considered independently. (a, b) ISO and (c) ASME; (d) interpretation of specification. [authors' drawing]

2.3. Surface profile tolerance in ISO

The surface profile specification in [31] defines the tolerance zone as the space between two equidistant surfaces, which are enveloping spheres of the diameter of 0.2 mm, whose centres are located on a surface, which is theoretically exact feature defined with respect to the datum system A, B and C (Fig. 10). The profile tolerance, as defined in the ISO GPS system, seems to be the most versatile tolerance and shall be applied to limit geometrical deviations of free-form surfaces. Measurements of free-form surfaces shall be performed with the use of coordinate measuring machines on the basis of a CAD model [32].
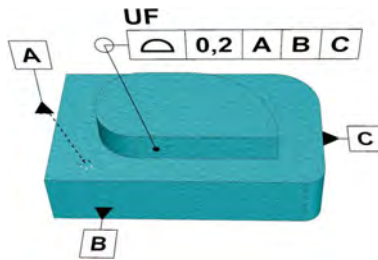

Fig. 10. Indication of surface profile tolerance as location tolerance according to ISO. [authors' drawing]

The surface profile tolerance with an unequally disposed tolerance zone is shown in Fig. 11. The extracted side surface of the boss shall be contained between two surfaces enveloping spheres of the diameter of 0.2 mm, whose centres are on the envelope of spheres with a diameter of 0.1 mm (fixed by the absolute value after the specification element UZ) rolled on the theoretically exact surface (defined by theoretically exact dimensions) with the direction of the offset indicated by the sign: the "–" sign means inside material and the "+" sign means outside the material.
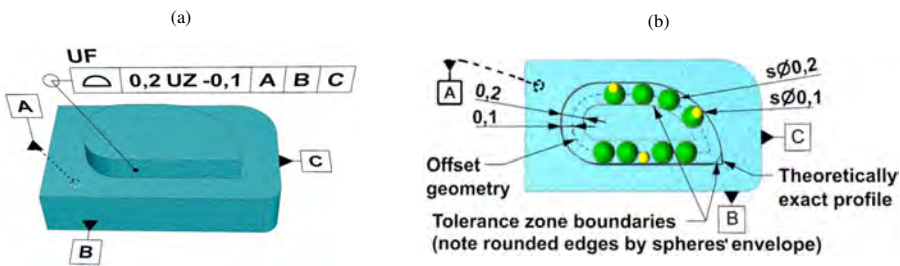

Fig. 11. (a) Indication of unequally disposed surface profile tolerance for united feature according to ISO; (b) interpretation. [authors' drawing]

2.4. Surface profile tolerance in ASME Y14.5-2018

The surface profile specification [17] defines the tolerance zone, obtained by offsetting on both sides by 0.1 mm from the true profile defined with respect to the datum system A, B and C (Fig. 12). The tolerance zone is divided bilaterally to both sides of the true profile. The ASME tolerance zone is similar but not identical to the ISO tolerance zone defined in Fig. 10. In ASME, sharp transition edges of the true profile produce sharp edges of the tolerance zone. In ISO, the tolerance zone is defined by two equidistant surfaces enveloping spheres of diameter 0.2 mm, the centres of which are situated on a surface having the theoretically exact geometrical form that transforms sharp edges of the theoretically exact profile into rounded transition edges between the surfaces establishing the tolerance zone.
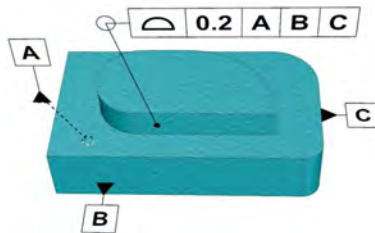

Fig. 12. Indication of surface profile tolerance as location tolerance according to ASME.
[authors' drawing]

The asymmetric surface profile tolerance zone in ASME is indicated with the modifier (Fig. 13). The value after the modifier determines how much additional material shall be added to the true profile (theoretical outline of the surface). The specification in Fig. 13 is approximately equivalent to the ISO specification shown in Fig. 11. A slight difference, which may be neglected in most applications, occurs in the transition area between the boss's flat side surface and the sector of the cylindrical surface.
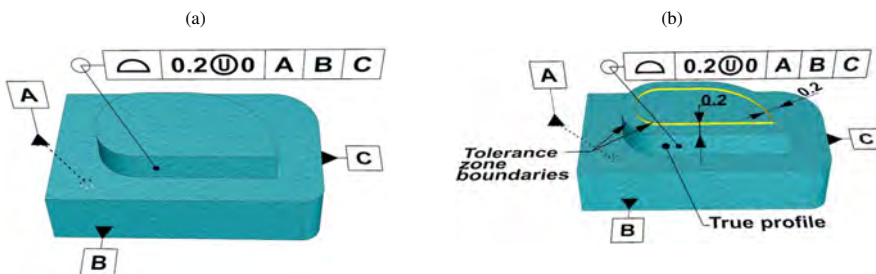

Fig. 13. (a) ASME inside material surface profile tolerance specification; (b) interpretation – tolerance zone with unilateral tolerance in the direction that removes material. [authors' drawing]

When the functional goal is the tolerance zone located entirely on one side outside the material, the specification shall be as in Fig. 14. The second value following the modifier unequally disposed indicates the tolerance zone in the direction that would allow additional material to be added to the true profile.
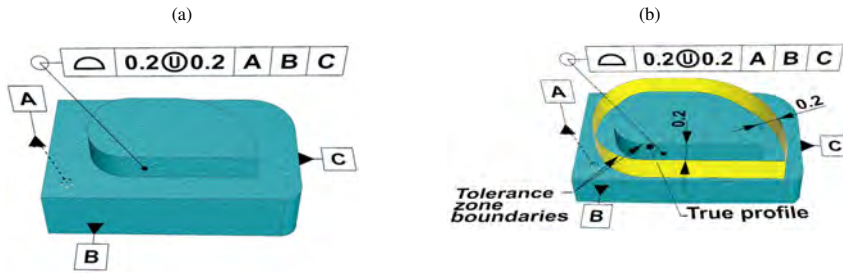
Fig. 14. (a) ASME outside material surface profile tolerance specification; (b) interpretation – tolerance zone with unilateral tolerance in the direction that adds material. [authors' drawing]

2.5. Rule #1 and the envelope principle (limitation for form deviation)

For the feature of size, like a shaft that shall rotate in the hole, the common functional requirement is that the dimensional tolerance (*e.g.*, shaft diameter) limits its form deviations. The shaft with the maximum material dimension shall have a perfect form along the entire cylinder length.

In ISO, the *envelope requirement* [20] applies only when the modifier is specified (Fig. 15). This means that the shaft shall pass through a ring gauge with a hole as long as the shaft's length and diameter equal to the shaft's upper limit of size given in the specification. If the shaft diameter is smaller than its *maximum material size* (MMS), then the form deviation may be larger. The shaft shall be made flawlessly if it is in its *maximum material condition* (MMC). This rule also applies to holes and grooves.
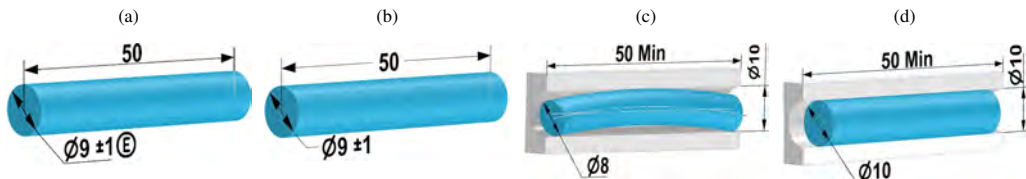

Fig. 15. Shaft dimensioned according to (a) ISO and (b) ASME; interpretation for the actual shaft of diameter (c) 8 mm and (d) 10 mm. [authors' drawing]

In ASME, approximately equivalent specification is enforced by default *Rule #1* (often called the perfect form at the MMC required). To fully reflect in ISO the rules prevailing in ASME, the specification of diameter limits shall be like in Fig. 16a, *i.e.*, using the modifier GN – *minimum circumscribed* association criterion and LS – *local size* defined by the sphere. Both modifiers are defined in [22]. To cancel Rule #1, the *independency modifier* [17] shall be specified after the dimension limits.

The ISO specification in Fig. 17a is graphically identical to the ASME specification in Fig. 16b but has a significantly different meaning. According to the ISO, the two-point dimensions in any cross-section shall be within the range specified in the drawing. The default application of the independency principle determines that the dimension tolerance does not limit the form deviations of the shaft (Fig. 17b).

(a)

(b)

Fig. 16. (a) Shaft with limit dimensions specified according to the ISO GPS system that defines identical requirements as the default ASME specification; (b) ASME specification. [authors' drawing]

(a)

(b)

Fig. 17. (a) ISO specification for shaft diameter; (b) shaft that fulfils the specification. [authors' drawing]

Specification of the straightness tolerance for the shaft axis cancels Rule #1. Of course, it shall be remembered that this requirement does not change the shaft dimension limits, but due to the rejection of Rule #1, the shaft diameter shall be verified by a two-point measurement.

This allows for larger form deviations. Thus, in the given example, a shaft with a two-point dimension equal to 10 mm may have a straightness deviation of 0.5 mm, and if it is made with a diameter of 8 mm, the straightness deviation will be accepted with a value up to 2.5 mm (Fig. 18).
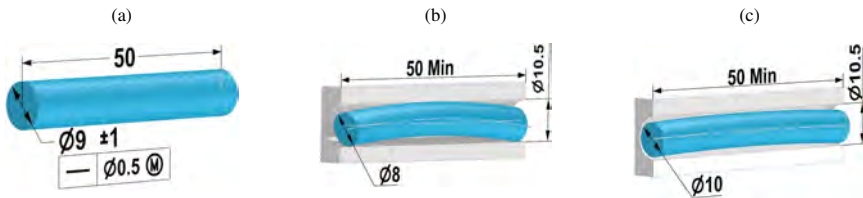
(a)

(b)

(c)

Fig. 18. (a) ASME specification – shaft diameter with the straightness tolerance of the axis; (b, c) interpretation – ring gauge of diameter 10.5 mm accepts smaller shafts with larger form deviations. [authors' drawing]

2.6. Roundness tolerance

One more difference between the standards regarding roundness tolerance (ISO) is worth mentioning. The term circularity tolerance is used in ASME, but it is not an issue. The fundamental dissimilarity is that ANSI/ASME B89.3.1 [33] specifies by default (*i.e.* without any additional indications) that for roundness measurement, the following conditions apply: MRS (*minimum radial separation*) as assessment criterion, the filter of 50 UPR (*undulations per revolution)* and the tip radius of 0.25 mm. In ISO 1101, by default, the reference feature association is the minimax (Chebyshev) association without constraints, *i.e.* minimum zone reference circles [34] which is another name for the MRS condition. Currently, the default roundness tolerance measurement conditions regarding numerical filtering and mechanical filtering (tip radius) are not specified in the ISO GPS system standards. However, ISO 1101 includes symbols that a designer can use to set up the measurements. The CB specification element indicates that a ball tip shall be used.

The value "0,25" means that a ball with a radius of 0.25 mm shall be used, and since it is followed by a "–", this is a long-wave-pass filter, removing wavelengths shorter than the cut-off value (higher UPR values). The "G50-" specification element indicates the use of a Gaussian filter with a nesting index of 50 UPR (Fig. 19) [17].

Fig. 19. Equivalent tolerance specifications. (a) Roundness tolerance (ISO); (b) circularity tolerance (ASME). [authors' drawing]

2.7. Perpendicularity tolerance

In the case of perpendicularity tolerance, to obtain the interpretation identical to ASME, the modifier ([19] – *minimum circumscribed feature*) shall be added (Fig. 20a). Without the modifier, the interpretation looks like in Fig. 20c, marked in red. In ISO, by default, the controlled feature is the extracted median line or median surface. The extracted median line in ISO is not a straight line, as it is established by the centres of the individual cross-sections of the pin, made along its length. The full definition of the extracted median line is given in ISO 17450-3 [25]. In ASME, the tolerance applies to the axis of the cylinder circumscribed on the extracted surface of the pin. Thus, the axis is the straight line (Fig. 20c). In Fig. 20c, the perpendicularity deviation assessed according to the ISO definition is smaller than that assessed according to the ASME definition. The reverse case is also possible.
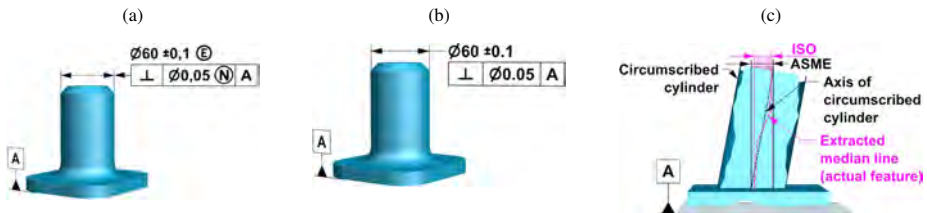

Fig. 20. Equivalent perpendicularity tolerance specification according to (a) ISO and (b) ASME; (c) actual (real) axis (ISO) and the axis of the circumscribed cylinder (ASME). [authors' drawing]

2.8. Composite and multiple single-segment position tolerances

Another difference is the *composite feature control frame* defined in ASME (Fig. 21). Such indication was used in ISO 5458:1988. Now, it is not defined in the ISO GPS system. The same, as well as more precisely dedicated, control on features within a pattern is obtained in ISO using

Fig. 21. Composite feature control frame, ASME. [authors' drawing]

the symbol $><$ *orientation constraint only* [19] after letters indicating datums in the bottom line of a stacked tolerance indication (Fig. 25a). In ASME, the stacked tolerance indication is called a multiple single-segment feature control frame.

The examples below show how, thanks to the specification of selected modifiers, the functional goal that the designer cares about may be achieved and, thus, how production costs may be optimised by using appropriate tools of the ISO and ASME standards to increase or decrease the tolerance value where it is needed. In addition, it can be seen that the approach to tolerancing according to ISO shall be a conscious choice of the author of technical product documentation – who must decide whether the tolerance zones for the hole pattern shall be separate SZ or combined CZ (Fig. 22 and Fig. 23). For the actual parts shown in Fig. 22 to Fig. 26, the dark blue cylindrical tolerance zones of diameter 0.2 mm illustrate the meaning of blue (top) indications. Respectively, the red cylindrical tolerance zones of diameter 0.1 mm illustrate the meaning of red (bottom) tolerance indicators.
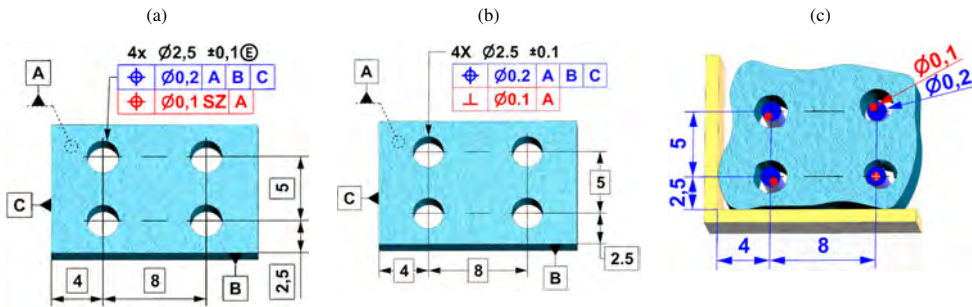

Fig. 22. Equivalent specifications (a) ISO and (b) ASME; (c) visualisation of the tolerance zones according to ISO and ASME – top view. [authors' drawing]

Approximately equivalent ISO and ASME specifications are shown in Fig. 22. The constraints for the position tolerance zones of diameter 0.2 mm (perpendicularity with respect to datum A, theoretically exact dimensions 2.5 mm and 7.5 mm with respect to datum B as well as theoretically exact dimensions 4 mm and 12 mm with respect to datum C) are the same (Fig. 22c). The difference is in the toleranced feature – according to ISO, the extracted median line of the actual hole shall be contained in each tolerance zone, and according to ASME, the axis of the cylinder inscribed to each actual hole shall be within the respective tolerance zone. Of course, the default interpretations of the hole diameter specifications are also different, as was explained above, therefore the modifier is added in the ISO specification.

The different symbols for ISO and ASME indications of geometrical characteristics are specified in the bottom line of the stacked tolerance indication for the tolerance zones of diameter 0.1 mm. In Fig. 22a, position tolerance is applied (that logically follows up the specification in the top line) with indication SZ to emphasise the independency of the requirement for each hole. So the tolerance zone for each hole shall be considered individually with respect to datum A, which only orients each tolerance zone perpendicularly. In Fig. 22b, perpendicularity tolerance is applied. Therefore, both specifications for tolerance 0.1 mm have the same meaning.

The top line in the indication in Fig. 23a is identical to that in Fig. 22a, so constraints for the cylindrical tolerance zones of diameter 0.2 mm in both drawings are the same. In Fig. 23b, made in line with ASME provisions, the composite control frame is applied. The top line in composite indication constrains the tolerance zones of diameter 0.2 mm in rotation and translation with

respect to the specified datum system. Therefore, the meaning of the top lines of the requirements in Fig. 23a and Fig. 23b is the same. According to the bottom line in Fig. 23a, the toleranced axis of each hole shall be contained within a cylindrical tolerance zone of diameter 0.1 mm perpendicular to datum A (tangent plane). Due to the presence of the modifier CZ, the axes of four tolerance zones form the rectangular pattern – are located as the edges of the rectangular prism with the rectangular base 8 mm × 5 mm. Theoretically exact dimensions, implicit $\text{TED}_A = 90°$, and explicit TEDs 8 mm and 5 mm fix perpendicularity to datum A and the interrelationship between the axes of the holes. The bottom line in composite indication controls the relative position between each feature in the pattern (feature-to-feature relationship) and constraints orientation with respect to datum A. So both indications that significantly differ visually establish identical patterns of the tolerance zones (Fig. 23).
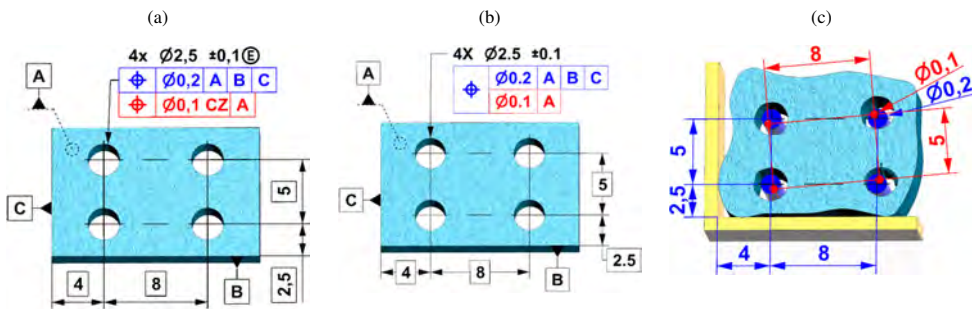

Fig. 23. Equivalent specifications (a) ISO and (b) ASME; (c) visualisation of the tolerance zones according to ISO and ASME – top view. [authors' drawing]

The constraints for the tolerance zones set by top lines of ISO and ASME specifications in Fig. 24 are identical because, in both cases, the datum system established by datums A|B|C locks all possible degrees of freedom for the position tolerance zones and the theoretically exact dimensions with respect to these datums are the same. In the bottom line in Fig. 24, in both specifications, the datum system is established only by two datums, A and B. To explain the meaning of the bottom line in Fig. 24b, it is necessary to recall the default simultaneous requirement formulated in ASME according to which the four holes shall be considered as the pattern. In ISO, according to the independency principle, by default, every GPS specification for a feature or relation between features shall be fulfilled independently of other specifications except when it is stated by a special indication. So, the modifier CZ is applied to lock the tolerance zones of diameter 0.1 mm between themselves by TED = 8 mm.

The top lines in indications in Fig. 25 are identical to those in Fig. 23, so constraints for the cylindrical tolerance zones of diameter 0.2 mm in both drawings are the same.

The complementary symbol >< in the bottom line (Fig. 25a) indicates that datum B is only used to lock the orientation degree of freedom for the toleranced pattern of four holes. The axes of four cylindrical tolerance zones of diameter 0.1 mm that form the pattern may move parallelly with respect to datum B (rotation of the pattern is not allowed). The notation for ASME that fixes in the same way the constraints for the tolerance zones of diameter 0.2 mm and 0.1 mm is given in Fig. 25b. The composite feature control frame that contains a single position tolerance symbol is followed by two segments (one above the other), each containing the required datum references that are applied. The top segment constraints in rotation and translation the tolerance zones of diameter 0,2 mm with respect to the datum system A|B|C. The bottom segment controls
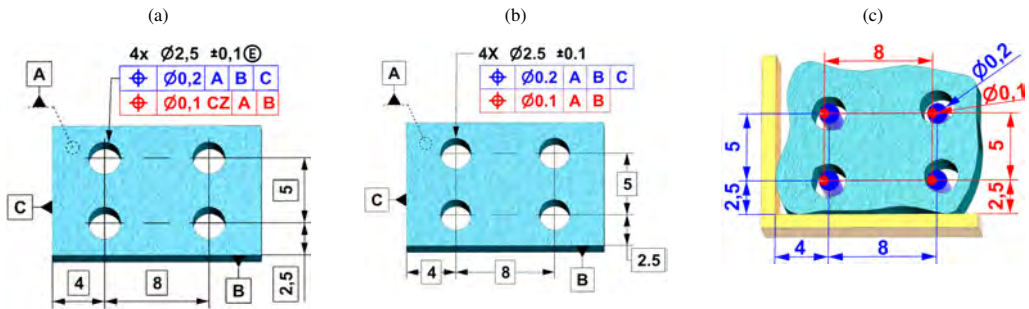
Fig. 24. Specification according to (a) ISO and (b) ASME; (c) visualisation of the tolerance zones according to ISO and ASME – top view. [authors' drawing]
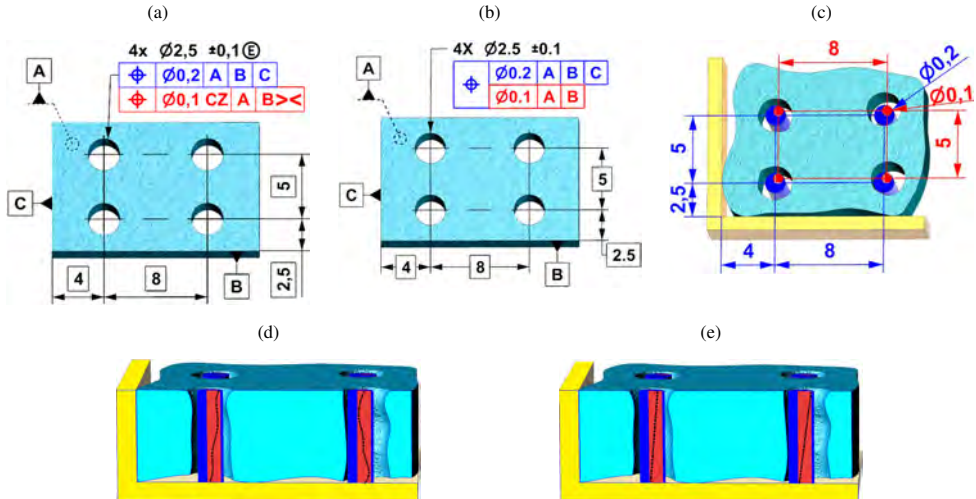

Fig. 25. (a) ISO specification with the modifier >< for orientation constraint only; (b) ASME; (c) visualisation of the tolerance zones according to ISO and ASME – top view; interpretation (d) ISO and (e) ASME. [authors' drawing]

tolerance the zones of diameter 0.1 mm within the pattern (feature-to-feature relationship). Linear theoretically exact dimensions with respect to the datum system A|B are not applied. Only angular theoretically exact dimensions with respect to the datum system A|B are used.

In Fig. 26a, to obtain the interpretation entirely identical to that in the ASME (Fig. 26b), specified are additionally: the modifier [19], which means that the axis of the maximum inscribed feature is the tolerance feature, modifier ><, which means that the specified datum establishes the orientation constraint only, and the envelope requirement . That eliminates the differences regarding the toleranced feature discussed above during the analysis of specifications given in Fig. 22–25.

It shall be underlined that according to the ISO GPS system by default the position tolerance applies to the derived median line (Fig. 25d), and according to ASME, to the axis of the inscribed cylinder (Fig. 25e and Fig. 26d) – the respectively circumscribed cylinder for the pin. It means that according to the ISO GPS system the position tolerance limits deviations of the toleranced
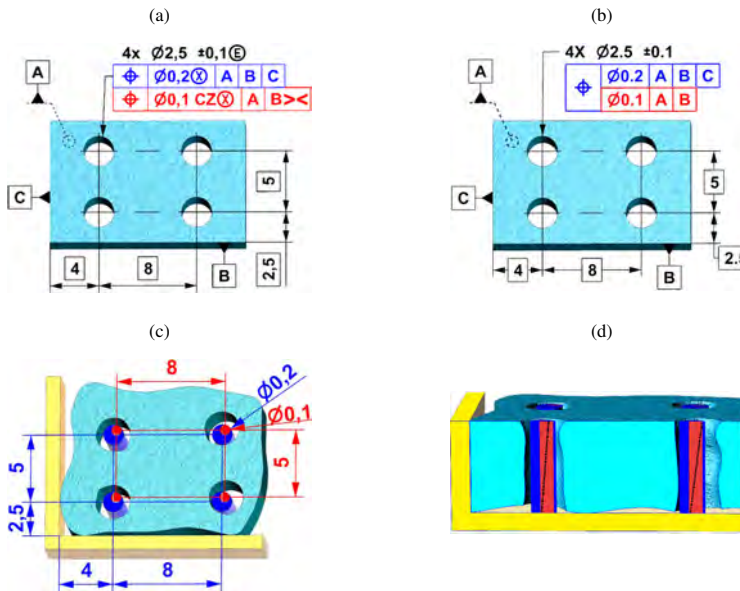
Fig. 26. Specification according to (a) ISO and (b) ASME; (c) visualisation of the tolerance zones according to ISO and ASME – top view; (d) interpretation – axis of the cylinder inscribed in each actual hole shall be within the respective cylindrical tolerance zones of diameters 0.2 mm and 0.1 mm. [authors' drawing]

axis straightness and the ASME specification does not set any requirements for the considered axis straightness. If the plate shown in Fig. 22–25 had a groove located by the position tolerance, then the ISO GPS system tolerance would apply to its derived median surface and ASME position tolerance would apply to the associated feature – the symmetry plane.

3. Conclusions

The primary function of production metrology in mechanical engineering is to assess whether the actual deviations of size, form, orientation, and location are within or outside the tolerances set by a designer. The deviations of the geometrical characteristics were identified as tricky measurands, and it is underlined here that the correct understanding of geometrical specifications is required, which strongly depends on the particular tolerancing system applied.

The crucial differences between the ISO Geometrical Product Specification system standards and the ASME Y14.5:2018 standard were discussed. It is demonstrated that, in many cases, both systems have different default rules. Moreover, for some identical graphical indications interpretations are different. On the other hand, the standards contain similar provisions in many cases, which may create the erroneous impression that there is no need to distinguish between the two tolerance systems because they are identical. Therefore, a lack of complete knowledge of the standards that make up the two tolerance systems, which has been noticed by the authors in the industry, leads to misunderstandings between the supplier and the customer, which often increases costs and delays delivery times.

The main difference between the discussed standards is that in the ISO GPS system, the designer decides whether a given workpiece will meet functional goals without imposing additional

requirements (*e.g.*, the default application of the independency principle, which among others, means two-point size for features of size that is quite often functionally unaccepted). At the same time, ASME GD&T takes the opposite approach. If there is no additional indication next to the tolerance, the part's verification is more stringent. The ISO GPS system standards automatically strive to make the production of parts as cheap as possible, and more details are accepted. When the product requires tighter geometrical tolerances due to expected functionality, device efficiency and reliability improvement, assurance of proper assembly or life cycle extension, it shall be obtained by additional specification, *e.g.*, envelope requirement and/or the modifier CZ/SIM. In ISO, a designer tightens the requirements where they are really needed and has to consider whether it is worth reducing the tolerances.

Over the last decade, both geometrical tolerancing systems have been significantly improved in the latest editions of several ISO GPS system standards and ASME Y14.5 standard. The standards give the tools to specify the same limits for geometrical deviations. The designer shall be aware of the differences and similarities between the standards that are the subject of the above study.

References

[1] Flack, D., & Hannaford, J. (2005). Measurement Good Practice Guide No. 80: Fundamental Good Practice in Dimensional Metrology. *National Physical Laboratory, Hampton Road, Teddington, Middlesex.*

[2] Tornincasa, S. (2020). *Technical drawing for product design: Mastering ISO GPS and ASME GD&T*. Springer Nature. https://doi.org/10.1007/978-3-030-60854-5

[3] Henzold, G. (2020). *Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection: A handbook for geometrical product specification using ISO and ASME standards*. https://ci.nii.ac.jp/ncid/BA87798291

[4] Humienny, Z. (2021). State of art in standardization in the geometrical product specification area a decade later. *Cirp Journal of Manufacturing Science and Technology*, *33*, 42–51. https://doi.org/10.1016/j.cirpj.2021.02.009

[5] GD&T Basics. (2023, May). Retrieved from https://www.gdandtbasics.com/gdt-symbols/

[6] Zhang, M., Liu, Y., Wang, D., & Tan, J. (2022). A coaxiality measurement method for the aeroengine rotor based on common datum axis. *Measurement*, *191*, 110696. https://doi.org/10.1016/j.measurement.2022.110696

[7] Babiasz, E. (2011). Regulations, standards and guidelines as a safety factor in synthesis and design of avionic systems. *Works of the Institute of Aviation*, *211*, 5–16. http://ilot.lukasiewicz.gov.pl/prace_ilot/?spis_zeszytow/211_2011/01.html

[8] Humienny, Z. (2021b). Can ISO GPS and ASME tolerancing systems define the same functional requirements? Applied Sciences, 11(17), 8269. https://doi.org/10.3390/app11178269

[9] Morse, E. (2016). Tolerancing standards: A comparison. *Quality*, *55*(8), 40–43.

[10] Baker, J., & Sesselmann, M. (2016). Analysis of differences between perpendicularity measurements applying ISO 1101/2012 and ASME Y14. 5-2009 and its impacts. In *Transdisciplinary Engineering: Crossing Boundaries (pp. 1009–1018).* IOS Press. https://ebooks.iospress.nl/volumearticle/45489

[11] Engineers Edge. (2023, May). Retrieved from https://www.engineersedge.com/asme-iso-differences.htm

[12] Schuldt, J., & Gröger, S. (2022). The assessment of the ISO GPS system implementation with GPS maturity model. *Procedia CIRP*, *114*, 197–202. https://doi.org/10.1016/j.procir.2022.10.027

[13] Płowucha, W. (2020). Point-plane distance as model for uncertainty evaluation of coordinate measurement. *Metrology and Measurement Systems*, *27*(4), 625–639. https://doi.org/0.24425/mms.2020.134843

[14] Wieczorowski, M., Kucharski, D., Śniatala, P., Pawlus, P., Królczyk, G., & Gapiński, B. (2023). A novel approach to using artificial intelligence in coordinate metrology including nano scale. *Measurement* 217, 1243–1252. https://doi.org/10.1016/j.measurement.2023.113051

[15] Gapiński, B., Wieczorowski, M., Mietliński, P., & Mathia, T. G. (2022). Verification of computed tomograph for dimensional measurements. In: Diering, M., Wieczorowski, M., Harugade, M., Pereira, A. (eds.) *Advances in Manufacturing III. MANUFACTURING 2022. Lecture Notes in Mechanical Engineering*. Springer. 142–155. https://doi.org/10.1007/978-3-031-03925-6_13

[16] Weckenmann, A., & Hartmann, W. (2015). A model- and simulation-based approach for tolerancing and verifying the functional capability of micro/nano-structured workpieces. *Measurement* 76, 70–79. https://doi.org/10.1016/j.measurement.2015.08.010

[17] American Society of Mechanical Engineers. (2018). *Dimensioning and tolerancing. Engineering product definition and related documentation practices* (ASME Y14.5-2018).

[18] Own study based on own survey conducted in the engineering community in Poland and Krulikowski, A. (2023, May) Survey Results on the Usage of ISO GPS and ASME Y14.5 Standards. https://krulikowskiconsulting.com/survey-results-on-the-usage-of-iso-and-asme-y14-5-standards/

[19] International Organization for Standardization. (2017). Geometrical product specifications (GPS) – Geometrical tolerancing – Tolerances of form, orientation, location and run-out (ISO Standard No. 1101:2017). https://www.iso.org/standard/66777.html

[20] International Organization for Standardization. (2010). Geometrical product specifications (GPS) – ISO code system for tolerances on linear sizes – Part 1: Basis of tolerances, deviations and fits (ISO Standard No. 286-1:2010). https://www.iso.org/standard/45975.html

[21] International Organization for Standardization. (2011). Geometrical product specifications (GPS) – Fundamentals – Concepts, principles and rules (ISO Standard No. 8015:2011). https://www.iso.org/standard/55979.html

[22] International Organization for Standardization. (2016). Geometrical product specifications (GPS) – Dimensional tolerancing (ISO Standard No. 14405-1:2016). https://www.iso.org/standard/65202.html

[23] International Organization for Standardization. (2021). Geometrical product specifications (GPS) – Geometrical Tolerancing – Maximum Material Requirement (MMR), Least Material Requirement (LMR) and Reciprocity Requirement (RPR) (ISO Standard No. 2692:2021). https://www.iso.org/standard/74592.html

[24] International Organization for Standardization. (2011). Geometrical product specifications (GPS) – General concepts, Part 1: Model for geometrical specification and verification (ISO Standard No. 17450-1:2011). https://www.iso.org/standard/53628.html

[25] International Organization for Standardization. (2016). Geometrical product specifications (GPS) – General concepts – Part 3: Toleranced features (ISO Standard No. 17450-3:2016). https://www.iso.org/standard/62309.html

[26] International Organization for Standardization. (1989). General tolerances – Part 1: Tolerances for linear and angular dimensions without individual tolerance indications. – Part 2: Geometrical toler-

ances for features without individual tolerance indicators (ISO Standard No. 2768-1,-2:1989). https://www.iso.org/standard/7748.html

[27] International Organization for Standardization. (2021). Geometrical product specifications (GPS) – Geometrical tolerancing – General geometrical specifications and general size specifications (ISO Standard No. 22081:2021). https://www.iso.org/standard/72514.html

[28] International Organization for Standardization. (2018). Geometrical product specifications (GPS) – Geometrical Tolerancing – Pattern and combined geometrical specification (ISO Standard No. 5458:2018). https://www.iso.org/standard/65559.html

[29] Marczak, P. (2020, October 5). *Individually or as a pattern?* LinkedIn. https://www.linkedin.com/pulse/individually-pattern-pawe%25C5%2582-marczak/?trackingId=C9gpo7x2lfpQJApo54%2FhPA%3D%3D

[30] Tang, Z., Huang, M., Sun, Y., Zhong, Y., Qin, Y., Huang, J. (2019). Coaxiality evaluation based on double material condition. *Measurement*, *141*, 287–295. https://doi.org/10.1016/j.measurement.2019.04.005

[31] International Organization for Standardization. (2017). Geometrical product specifications (GPS) – Geometrical Tolerancing – Profile tolerancing (ISO Standard No. 1660:2017). https://www.iso.org/standard/63218.html

[32] Poniatowska, M., & Werner, A. (2012). Simulation tests of the method for determining a CAD model of free-form surface deterministic deviations. *Metrology and Measurement Systems, 19*(1), 151-158. https://doi.org/10.2478/v10178-012-0014-y

[33] American National Standards Institute. (2023). *Measurement of out-of-Roundness* (NSI/ASME B89.3.1-1972 (R2003)).

[34] Rossi, A., Antonetti, M., Barloscio, M., & Lanzetta, M. (2011). Fast genetic algorithm for roundness evaluation by the minimum zone tolerance (MZT) method. *Measurement*, *44*(7), 1243–1252. https://doi.org/10.1016/j.measurement.2011.03.031

Paweł Zdrojewski received the M.Sc. degree from Warsaw University of Technology (WUT), Poland. Since 2011 he has held engineering positions in the Institute of Aviation and General Electric where he was involved in projects: LM9000, Hybrid Electric, GE9X, ILR 33-Bursztyn, TLPD (Throttleable Liquid Propulsion Demonstrator) and many others. Co-author of several patents and author of many solutions in jet engines.

Zbigniew Humienny Ph.D., D.Sc. works at the Institute of Machine Design Fundamentals, at the Faculty of Automotive and Construction Machinery Engineering, Warsaw University of Technology (WUT), Poland. He has published as author or co-author over 150 papers in scientific journals and conference proceedings. In 1998 he was nominated as an expert of the Polish Committee for Standardization in the Technical Committee ISO/TC 213 "Dimensional and geometrical product specifications and verification". He is Chairman of the Technical Committee TC 48 "Machine Design Fundamentals" of the Polish Committee for Standardization. He was the project contractor and coordinator of a European Commission grant within the Leonardo da Vinci programme and head of the WUT team in the European Erasmus+ programme project. He conducts research in the field of dimensional and geometrical tolerancing and geometrical tolerancing verification. He closely cooperates with industry and has conducted a number of vocational trainings dealing with geometrical tolerancing, mainly in the automotive, aviation and defence industries.