UDC 621.713.1 doi: 10.20998/2078-7405.2024.100.04

INTERPRETATION WORKFLOW OF GEOMETRIC TOLERANCES

Balazs **Mikó** [0000-0003-3609-0290]

Óbuda University, Budapest, Hungary
miko.balazs@bgk.uni-obuda.hu

Received: 2 April 2024 / Revised: 19 April 2024 / Accepted: 25 May 2024 / Published: 15 June 2024

Abstract. *The application of the principles of the geometric product specification (GPS) has increasing importance in the machining industry. The tolerancing process should harmonize the relationship between tolerancing, machining process, measuring and design requirements. The aim of the article is to present the interpretation process and the suggested workflow of geometric tolerances. The dimensional and geometric tolerances are compared, and some critical problems of geometric tolerancing are demonstrated by case studies.*

Keywords: *geometric tolerance; GPS; GD&T; degree of freedom; datum feature*

1 INTRODUCTION

In the production of machine components, we always experience deviations due to a number of factors. Deviations may be singular or recurring, recurring deviations may be permanent or random. The causes of deviations can be explained by changes in the raw material used, changes in the condition of the manufacturing equipment, changes in technology, human factors, environmental influences or the measurement process.

There are several ways to manage deviations from plan. The first is to define a manufacturing technology, a series of processes in the process planning that will gradually, step by step, produce the desired part geometry with the expected accuracy. The second solution is to define tolerances, which are the allowable deviation. We can define macro and micro precision for the geometry design. The macro accuracy of a part can be defined by dimensional and geometric tolerances. Dimensional accuracy refers to the distance between points, while geometric tolerances refer to shape and positional accuracy. A comparison of dimensional and geometric tolerances is provided in Table 1.

The ISO standards define the specifications for dimensional tolerances of mechanical parts [1] by the abbreviated name GPS - geometric product specification. The American ASME standard uses the term GD&T - geometric dimensioning and tolerancing. The aim and the basic principles of the two sets of standards are the same, but some details can be different. In case of interpretation of the engineering drawing, the detailed understanding of the relevant standards is essential.

The aim of the research is to investigate the tolerancing process, the relationship between tolerancing, machining process, measuring and design requirements. The aim of the article is to present the interpretation process and the suggested workflow of geometric tolerances. Some critical problems are demonstrated by case studies.

Table 1 Comparison of dimensional and geometric tolerances

	Dimensional Tolerances	**Geometric Tolerances**
Definition	Specify the acceptable variations in the numerical values of dimensions, such as length, width, height, or diameter. They focus on the size of individual features.	Specify the allowable variations in the form, orientation, location, and relationship of features. Geometric tolerances are concerned with the shape and spatial arrangement of features
Representation	Usually represented by numerical values indicating the acceptable range of dimensions. For example, a dimension is specified as "25 ± 0.05 mm".	Represented using symbols and feature control frames, indicating the desired geometric relationship and allowable variations.
Focus	Primarily concerned with the size of features, ensuring they fall within specified limits.	Focus on the form and arrangement of features, ensuring that they relate to each other correctly in terms of shape, orientation, and location.
Application	Applied to individual measurements of length, width, height, etc. For example, specifying the length of a shaft or the diameter of a hole.	Applied to relationships between features, ensuring proper alignment, orientation, and form.
Complexity	Generally simpler to specify and interpret, involving straightforward numerical values	Can be more complex as they involve a combination of symbols, datum, and specific instructions to define relationships between features accurately.
Interchangeability	Mainly contribute to the interchangeability of parts by specifying acceptable size variations.	Contribute to both interchangeability and functional relationships between parts by ensuring proper form, orientation, and alignment.

Cost Considerations	Tight dimensional tolerances can increase manufacturing costs, as they require more precise machining and measurement.	Can contribute to cost savings by allowing more flexibility in certain dimensions while ensuring critical relationships for functionality.

2 TOLERANCING PROCESS

The tolerance process should identify the type and value of tolerances required. In this process, several conflicting aspects need to be taken into account. The primary consideration is the function of the part, the role of each geometric element [2][3]. The second aspect is to take into account the manufacturing capabilities [4][5]. The third aspect is the analysis of the assembly relationships and the impact of the propagation of manufacturing defects. The fourth aspect is the consideration of the measurement possibilities [6][7][8]. Finally, these requirements must be indicated on the drawing by means of engineering symbols [1], for which clarity and compliance with engineering standards are essential. The tolerancing of the part therefore sets expectations for the manufacturing and measurement process, taking into account the functional requirements, and thus determines the cost of the part.

For geometric tolerances, whether we are in the design, manufacturing preparation or measurement phase, the following analysis process should be carried out to clarify the interpretation of the tolerance (Figure 1). The interpretation of geometric tolerances is based on standard drawing symbols (Figure 3).

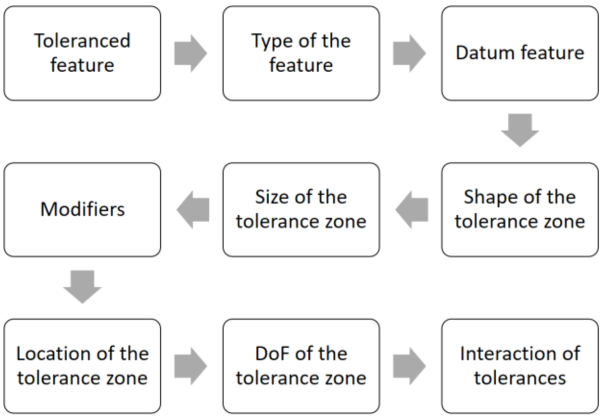


Figure 1. Tolerance analysing process

The first question is what is the *toleranced feature*. The leader and an arrow show this from the tolerance frame (Figure 3a). The direction of the leader can have a modifying effect on the meaning, so accurate application is important. The notation can only point to a physically existing feature, not to a theoretical geometric element (e.g. centre line, plane of symmetry). Indeed, if a centre line is the centre line of several geometric elements (e.g. a stepped hole), it is not possible to determine to which element the specification applies. If the indicator arrow continues along a dimension line, the centre line, plane of symmetry of the marked element is the one to which the specification applies.

Another important aspect of the toleranced element is whether the measuring is technologically feasible. For example, a hole in a sheet metal part theoretically has an axis, but due to the short length of the hole, this can only be measured with a higher uncertainty. The fundamental cause of the measured deviation will be the measuring process. It is therefore necessary to consider the metrological process in the tolerancing process.

The *tolerance type* is indicated by the symbol in the first field of the tolerance frame. ISO 1101 defines 15 types of geometric tolerances, shown in Figure 2. The interpretation of each tolerance is defined in the standard.

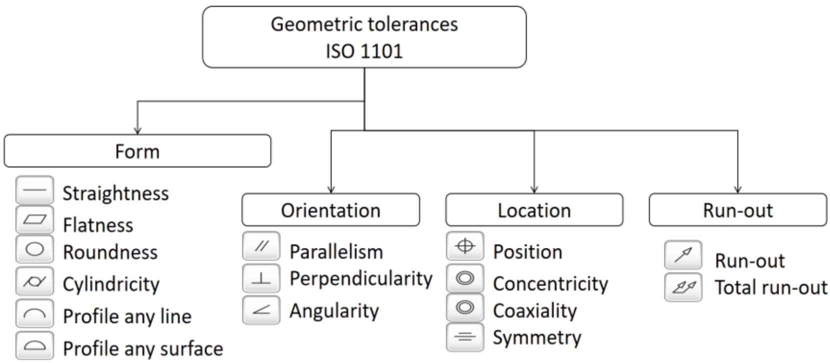
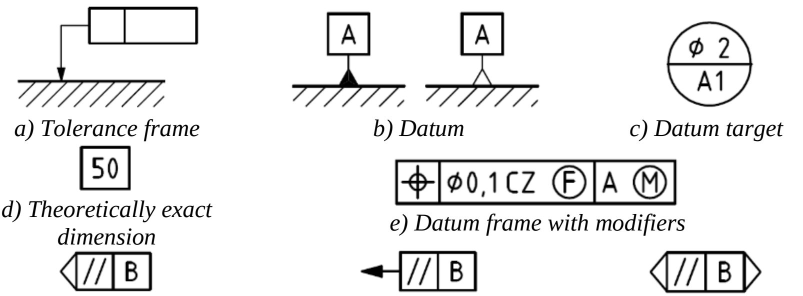


Figure 2. Type of geometric tolerances based on ISO 1101

The next step is to define the *datum feature*. In the tolerance frame, the datum are marked on the right-hand side and must be indicated on the drawing. The datum feature is connected to the surface by a triangle, not an arrow (Figure 3b). The notation rules are the same as for the marking of the toleranced feature, so the marking should not point to a theoretical geometric element. An exception to this is when the points forming the base are also given with base locations (ISO 5459). The locations of the points are given with theoretical dimensions (Figure 3d), which have no tolerance, giving the theoretical dimensions necessary to interpret the tolerance.

For some geometric tolerances, no datum can be used, for others the use of a datum is necessary, while in some cases both with and without a datum can be used (profile any line, profile any surface, position), but the interpretation is modified. If a datum is required, it must be checked that (1) the datum feature is independent of the toleranced element; (2) its indication is on the drawing; (3) it corresponds to the type of tolerance (e.g. perpendicularity tolerance is indeed perpendicular to the toleranced element).

If several datum are applied to a tolerance, the order of these is important. We can talk about primary, secondary and tertiary bases. These are important for the degrees of freedom of the tolerance field (see later). The primary base is usually a planar geometric element, which is, for all intents and purposes, the supporting surface of the element being toleranced. The secondary base is an axis or straight element that gives directional constraint, while the tertiary base is a point element.



a) Tolerance frame *b) Datum* *c) Datum target*
d) Theoretically exact dimension *e) Datum frame with modifiers*
f) Inspection plane *g) Direction feature* *h) Orientation plane*

Figure 3. Some drawing symbols related to geometric tolerances

The next step is to determine the *shape of the tolerance zone*. Depending on the type and nature of the toleranced feature, the tolerance zone can be either planar or three-dimensional. A tolerance field is two parallel elements that envelops the toleranced feature along its entire extent. Where a diameter (Ø) is indicated in the metre of tolerance, the tolerance zone is circular or cylindrical. The type of tolerance and the nature of the toleranced feature determines whether the tolerance can be used with a diameter mark only, without a diameter mark, or with both. In the case of circular and cylindrical tolerances, the tolerance field is in the form of a circular ring or cylindrical ring.

The *size of the tolerance zone* is always given in millimetres (mm) and is the width or diameter of the aforementioned zone. The application of the diameter symbol is depends on the type of the tolerance and the nature of the toleranced feature.

The interpretation of the tolerance can be modified or clarified using a number of *modifiers*. These modifiers can refer to the definition of the associated geometry (C, G, N, X, T, E); the relationship between several features (CZ, SZ, UF, SIM), the offset of the tolerance field (P, UZ, OZ), the measurement condition for elastic pieces (F), the relationship between geometric and dimensional tolerances (M, L, R). The use of these modifiers provides a high degree of flexibility in the definition of tolerances, but requires careful analysis during measurement. The modifiers cannot be used in all cases and the guidelines of the standard should be followed.

Based on the type of geometric tolerance and the nature of the toleranced element, the *location of the tolerance zone* can be determined. In this context, account shall be taken of any datum, the location and orientation of the drawing markings, or, in their absence, the indicative notations which are mandatory under the standard (Figure 3f, Figure 3g, Figure 3h).

Geometric tolerances are not fixed tolerance zones at a given location, but have a different number of *Degrees of Freedom* (DoF) depending on the type of tolerance, the nature of the element being toleranced and the number of possible bases. These provide the flexibility to adapt to the specific design requirement, determining the most appropriate tolerance at the design stage.

The final step investigated the *interaction* effects between each tolerance. If a toleranced feature is associated with more than one tolerance, the relationship between them is examined. It is possible that the same requirement has been specified in several different ways, so that one of the tolerances is redundant. For example, there are both perpendicularity and parallelism tolerances on the axis of a hole. Specifications may also be found that are conflicting. This means that if one specification is satisfied by the part, the other is not. This can be due to an incorrect choice of tolerance type or tolerance value. The analysis shall determine the value of the degree of freedom of each tolerance field. The tolerance with more degrees of freedom should have a smaller tolerance zone value.

3 CASE STUDIES

The current chapter demonstrates the interpretation process of the geometric tolerancing by case studies.

Figure 4 shows a position tolerancing of a hole. The toleranced feature is the axis of the hole, because the indicator arrow continues along a dimension line. The position tolerance required datum features, which define the theoretical location of the hole. The theoretical position is 55 mm and 70 mm from the *A* and *B* datum.

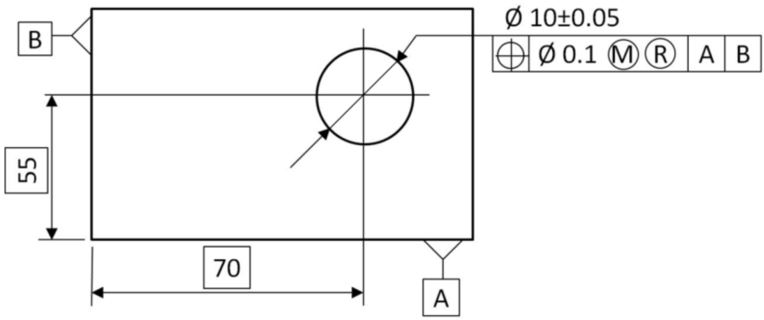
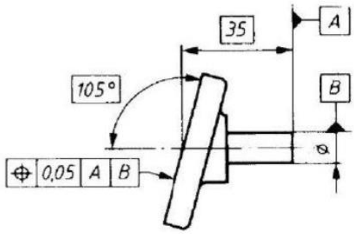


Figure 4. Case study #1 – Position tolerance of a hole with modifiers

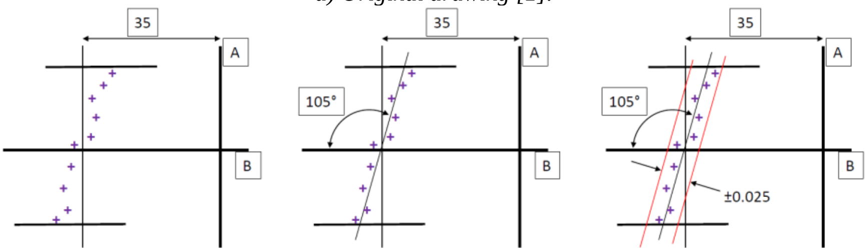
The shape of the tolerance zone is a circle or a cylinder, because of the diameter indicator (Ø), and the size is 0.1 mm. The shape of the tolerance zone can also be interpreted as a circle or a cylinder, a kind of uncertainty in the definition of tolerance. The shape of the tolerance zone depends on how deep the hole is. For a sheet metal part, it can be interpreted as 2D geometry, in which case the tolerance field is a circle. If the depth of the hole allows measurements to be made on several levels, the tolerance field is a cylinder as a 3D element.

The tolerance frame contains two modifiers. The first is the maximum material principle, which means, than the current value of the geometric tolerance depends on the dimeson of the hole. The tolerance of 0.1 mm is valid for the maximum material condition (M). In this case, the part contains the most material, so this is the lower limit for a hole (9.95). If the diameter of the manufactured part is larger than this, the position tolerance is increased by this value. If the diameter of the hole is 10.05 mm (upper limit), the value of the tolerance zone is 0.2 mm. The result is a virtual hole in which no hole's contour will intersect any of the elements of the produced series. The reciprocity modifier (R) extends this by allowing the dimensional tolerance of the hole to increase if the hole's position error is smaller than the specified position tolerance. Therefore, if the current position error is 0 mm, the minimum value of the diameter can be 9.85 mm. This is the theoretical value of the previous mentioned virtual hole. The tolerance zone located in the given position, there is no degree of freedom, so this type of geometric tolerance is close to the dimensional tolerancing.

Figure 5a presents an example from the ISO 1101 standard for the position tolerance of a plane surface. The two datum features are the face surface on the right side (A) and the axis of the shaft (B). Let's determine the location of the tolerance zone!



a) Original drawing [1].



b) *c)* *d)*

Figure 5. Case study #2 – Location of the tolerance zone

First, a parallel plane is drawn at a distance of 35 mm from datum *A* (Figure 5b). After that this plane is rotated 105° about the horizontal, an *A* datum parallel axis passing through the intersection of bases *A* and *B* (Figure 5c). The resulting plane is offset by ±0.025 mm. This indicates the limit of the tolerance zone (Figure 5d). The position tolerance has no degree of freedom, so the points of the toleranced surface must be within this fixed range.

In some cases it is possible to change the degree of freedom of the tolerance zone by specifying the datum. This provides a high degree of flexibility during the design process. The larger the number of degrees of freedom in the tolerance zone, the more stringent the specification. The „profile any surface" tolerance is often used for general tolerancing of parts bounded by free-form surfaces (e.g. injection moulded plastic parts).

Figure 6 shows the "profile any surface" tolerance for different bases. If no base is defined (Figure 6a), the 3D tolerance zone around the part has six degrees of freedom, three linear and three rotational. The surface points of the part must be covered by displacements along these three axes. If a planar datum is defined (Figure 6b), the number of degrees of freedom is reduced, leaving two linear and one rotational possible displacements. This tolerance zone is more strict. If a secondary datum is added (Figure 6c), the tolerance field is reduced by two more degrees of freedom, leaving only one linear displacement possibility. With the addition of a tertiary datum (Figure 6d), all degrees of freedom are removed.

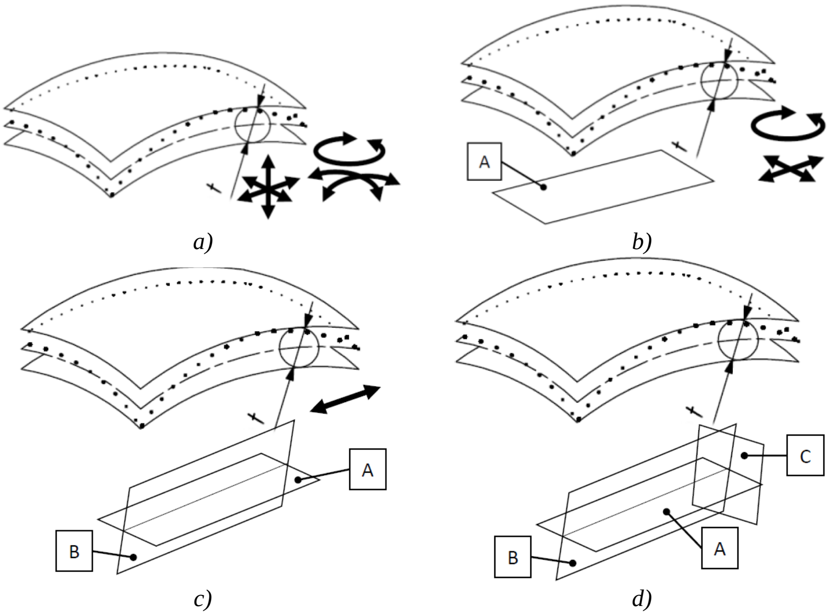


Figure 6. Case study #3 - Profile any surface with datum

The fourth case study shows an example of multiple tolerancing (Figure 7). The drawing in the second example (Figure 5a) is extended with two additional tolerances, an angularity tolerance and a flatness tolerance.

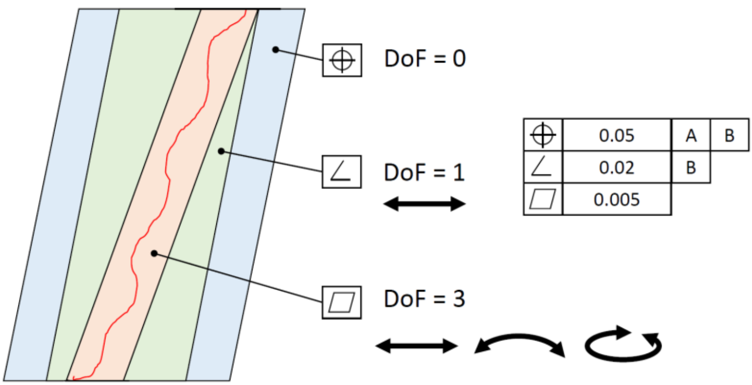


Figure 7. Case study #4 – Multiple tolerances

Position tolerance has no degree of freedom, as we saw earlier. However, the angularity tolerance has one degree of freedom, it can move along the datum *B*.

However, this is only a true degree of freedom if the value of the angularity tolerance is less than the position tolerance, so the angularity tolerance is within the position tolerance. Otherwise, both tolerances cannot be satisfied. It is advisable to adjust the ratio between the two tolerances between 1:3 and 1:5, so that the effect of a smaller tolerance is noticeable in operation, measurement and production.

The flatness tolerance has no datum, so it has three 3D degrees of freedom: one linear and two rotational DoF. The value of the flatness tolerance must be smaller than the value of the angularity tolerance, as explained above. The degrees of freedom can only be used if the other tolerance zone does not constrain it.

4 CONCLUSION

The use of geometrical tolerances is nowadays essential in the design of mechanical engineering components. Consequently, the correct interpretation of the engineering drawings is also required when planning manufacturing and measurement plans.

In this paper, a nine-step workflow has been presented to provide a systematic interpretation of each engineering drawing item. The proposed nine steps are:

1. Define the tolerance feature.
2. Determine the type of tolerance.
3. Determine the datum.
4. Determine tolerance zone shape.
5. Determine the size of the tolerance zone.
6. Interpretation of modifiers.
7. Determine the location of the tolerance zone.
8. Determine the degree of freedom of the tolerance zone.
9. Analyse the interaction between tolerances.

In the future research the machining and measuring circumstances are investigated, in order to support the manufacturing process planning. The geometric tolerance based machine tool accuracy and the GPS based machinability of materials will be investigated and developed.

References. 1. ISO 1101-2017 Geometrical product specifications (GPS) - Geometrical tolerancing - Tolerances of form, orientation, location and run-out. **2.** *Sun W.; Gao Y.* (2022) The rule-based specification of the datum-based model for geometric dimensioning & tolerancing. Procedia CIRP 114:189–196 https://doi.org/10.1016/j.procir.2022.10.026 **3.** *Fengxia Z.* et. al (2015) Research on the intelligent annotation technology of geometrical tolerance based on geometrical product specification (GPS). Procedia CIRP 27: pp. 254 – 259 https://doi.org/10.1016/j.procir.2015.04.074 **4.** *Corrado A.; Polini W.* (2021) Model of geometrical deviations in milling with three error sources. Manufacturing Technology. 21(5): pp. 575–584. https://doi.org/10.21062/mft.2021.078 **5.** *Sultana J., Sztankovics I.* (2023) Roundness error and topography of hard turned surfaces. Cutting & Tools in Technological System 98: pp. 83–92 https://doi.org/10.20998/2078-7405.2023.98.08 **6.** *Gosavi A; Cudney E.* (2012) Form error in precision metrology: a survey of measurement techniques. Quality Engineering 24: pp. 369–380 https://doi.org/10.1080/08982112.2011.652583 **7.** *Li Y; Gu P.* (2004) Free-form surface inspection techniques state of the art review. Computer-Aided Design 36: pp. 1395–1417 https://doi.org/10.1016/j.cad.2004.02.009 **8.** *Armillotta A.* (2013) A method for computer-aided specification of geometric tolerances. Computer-Aided Design 45: pp. 1604–1616 https://doi.org/10.1016/j.cad.2013.08.007

Балаш Міко, Будапешт, Угорщина

ІНТЕРПРЕТАЦІЯ РОБОЧОГО ПРОЦЕСУ ВИЗНАЧЕННЯ ГЕОМЕТРИЧНИХ ДОПУСКІВ

Анотація. *Застосування принципів геометричної специфікації продукту (ГСП) набуває все більшого значення в обробній промисловості. Процес визначення допуску повинен гармонізувати взаємозв'язок між допуском, процесом механічної обробки, вимірювальними та проектними вимогами. У разі інтерпретації інженерного креслення важливе значення має детальне розуміння відповідних стандартів. Метою дослідження є вивчення процесу визначення допуску, взаємозв'язку між допуском, процесом механічної обробки, вимірювальними та проектними вимогами. Метою статті є представлення процесу інтерпретації та запропонованого робочого процесу геометричних допусків. Деякі критичні проблеми демонструються на прикладах кейсів. Процес допуску повинен визначати тип і значення необхідних допусків. У цьому процесі необхідно враховувати кілька суперечливих аспектів. Першочерговим фактором є функція деталі, роль кожного геометричного елемента. Другим аспектом є врахування виробничих можливостей. Третій аспект - аналіз складальних взаємозв'язків і впливу поширення виробничого браку. Четвертим аспектом є розгляд можливостей вимірювання. Нарешті, ці вимоги повинні бути позначені на кресленні за допомогою інженерних символів, для чого необхідна чіткість і відповідність інженерним нормам. Таким чином, допуск деталі встановлює очікування щодо процесу виготовлення та вимірювання з урахуванням функціональних вимог і, таким чином, визначає вартість деталі. У цій роботі представлений дев'ятиетапний робочий процес, що забезпечує систематичну інтерпретацію кожного елемента інженерного креслення. Запропоновані дев'ять кроків: визначення ознаки допуску, типу допуску, форми зони допуску, розміру зони допуску, тлумачення модифікаторів, визначення місця розташування зони допуску, ступіня свободи зони допуску, аналіз взаємодії між допусками.*

Ключові слова: *геометричний допуск; ГСП;геометричні розміри та допуски; ступінь свободи; вихідна ознака.*