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GPS APPLICATION IN THE DESIGN OF GEARBOXES

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Abstract: The integrated geometrical product specification (GPS) system for workpiece geometry specification and verification is an improved engineering tool for product development and production. The goal of the GPS system is to provide tools for cost-effective management of variability in products and processes. This can be achieved by using a more precise way of expressing the functional requirements of the workpiece, complete and well-defined specifications and integrated verification approaches. The intended function of the product is ensured by controlling the geometry and material properties of the workpiece parts, which make up the product. GPS is a language just for checking geometry, and further development is based on computational mathematics and correct, consistent logic using general sets of rules that can be applied to all types of specifications. This article deals with the application of GPS rules in the design of gearboxes.

Key words: gearbox, geometrical product specifications, tolerancing, drawing

1. INTRODUCTION

Gear mechanisms allow the transmission of torque from the drive shaft to the driven shaft. They are mainly used for gears with a constant gear ratio and a small shaft axial distance. Spur gears are among the most widely used gearing mechanisms in a wide range of mechanical engineering practice. They are the basic elements by which the transmission and transformation of mechanical energy and motion are carried out in machines. The basic part of the gear is the gear train, which consists of a pair of gears – driving and driven – that mesh together. We can say that gears have become a symbol of engineering [1]. At the same time, gears are also one of the most commonly used types of automotive gears.

The quality of gears is mostly determined by their geometric design. If the geometric design is faulty, even the use of the highest quality materials will not ensure the reliability of the gearing. Conversely, sometimes, an excellent geometric design of the gearing can save the cost of expensive materials. Adequate attention must be paid to the design of gears. In the geometric design of gearing, it is necessary to choose basic geometric parameters. The basic parameters of the spur gear include the number of teeth, the size of the module, the pressure angle and the helical angle. But the choice of corrections and modifications of gearing also come into play here. Next, it is necessary to choose clearances in the gearing. Geometric tolerances of form, orientation and position play their role here.

The design process starts with a list of functional requirements and continues with technical specifications. At this stage, the designer begins to create solutions to the problem and can view the product – the pre-driving mechanism the designer wants to produce [2, 3]. The design can be realised by using conventional methods like engineering drawings or computer-aided design (CAD). CAD is a software tool that provides the computer-aided designer with the ability to produce fast and accurate drawings. The CAD system allows the designer to create a three-dimensional representation of a part or product and displays views from different angles to check for functionality and appearance (Fig. 1).

To meet the design requirements and specifications of the product, the designer nowadays uses the CAD system, finite element analysis (FEA) and mathematical modelling [4]. In order to resolve all possible aspects to the benefit of all involved in the process, it is necessary to seek an answer to the question of whether it is possible to design a part in such way to make its manufacture easier. An integral part of this is the creation of drawing documentation [5].

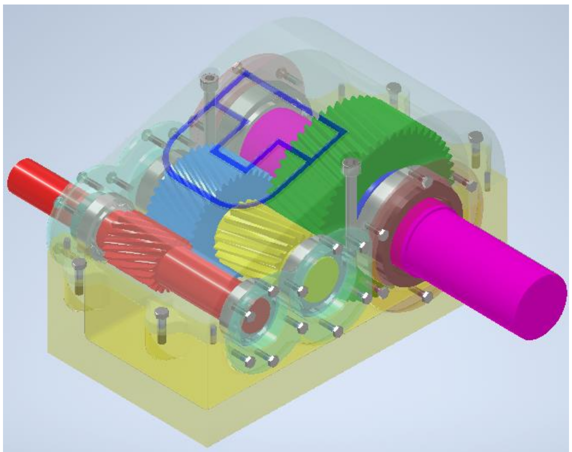

Fig. 1. Example of a gearbox CAD model. CAD, computer-aided design

Insufficient drawing documentation leads to production delays and rework of assembly procedures on the production floor. Regarding the conventions and symbols used in production documentation, international and national standards are used, and there is no reason to duplicate this proven methodology, rather an overview is offered of what is important in communicating how to translate knowledge into purposeful action at the level of the production drawing board. The goal of technical drawing is to depict the requirements for a functional design with clear and relevant information so that the product is manufactured, and these requirements are verified [6–8]. The methods used in the design process should be clear and concise so as not to cause confusion in the interpretation of the design and, consequently, should enable the interpretation of the design requirements by all those involved in the process. The geometric specifications of the gears also play an important role in the optimisation of gear transmissions [9].

Production produces parts that are not entirely accurate and that show deviations from nominal values and from each other. The parts are measured to compare them with the specification [10–12]. The geometrical product specification (GPS) defines, on an engineering drawing, the shape (geometry), dimensions and surface characteristics that provide the optimum function of the part, along with the variance around the optimum that will still provide satisfactory function.

2. GEOMETRICAL PRODUCT SPECIFICATION

The GPS describes the conditions that the component or part of the component must meet. These conditions are expressed from the geometric characteristics between or on the geometric elements. The geometric elements as a whole are considered for non-ideal surfaces of the component as a whole (skin model). The non-ideal surface of the component is modelled by the closed surface of the material–surrounding interface of the component. The interface is defined by a data set of all points belonging to the interface.

The integrated GPS system for specification and verification of workpiece geometry is an enhanced engineering tool for product development and production. The system is essential because companies in an international environment are rapidly adopting advanced technologies, new manufacturing processes, new materials and new products by outsourcing certain activities. The goal of an improved GPS system is to provide tools for cost-effective management, but not variability, in products and processes [13]. This can be achieved by using a more precise way of expressing the functional requirements of the workpiece, complete and well-defined specifications, and integrated verification approaches. This improved GPS has clarified current practice and harmonised the work of other relevant International Standards Organisation (ISO) and Technical Commissions (TCs). This harmonisation, for example, allows better integration with 3D CAD/CAM/CAQ systems.

The intended function of the product is provided by controlling the geometry and material properties of the workpiece component(s), which make up the product. GPS is a geometry-only control language, and further development is based on computational mathematics and correct, consistent logic using general rule sets that can be applied to all types of specifications [14]. The challenge for the future is to enrich the GPS language to allow the expression of requirements that cover a wide range of workpiece functions. The introduction of an improved GPS system is a prerequisite for continuous improvement of product quality and reduction of time to market.

A default global specification is available for each type of GPS, based on simplicity and overall cost reduction. Additionally, there are many simplified designations to cover commonly occurring workpiece functions, for example, notation to accommodate kinematic pairs.

2.1. Geometric dimensioning

Geometric dimensioning and tolerancing (GD&T) is a system for defining and communicating engineering tolerances. It is a set of instructions designed specifically for quoting and designing so that the part is correctly interpreted and allows the designer's intent to be translated into all stages of the product cycle. It provides instructions for drawing and dimensional inspection.

GD&T uses markers and computer-aided three-dimensional solid models in the drawing documentation that explicitly describe the nominal geometry and its allowable deviations. GD&T finds its widest application in mass production, where interchangeability of manufactured parts without counterparts is necessary.

The basic idea of GD&T is to determine the base of a component element or assembly group. This, of course, refers to the actual position and functional connections. The bases are selected as starting points for the dimensioning and the use of tolerances or tolerance zones. Functional bases must be selected and they are simply one of the elements of the component that determine the actual position of the component in the manufactured assembly (product). The use of any other base system, that is, axes – changes the overall tolerance.

2.2. Geometric tolerancing

Geometric tolerances are determined by the functional requirements of the product and the functional surfaces of the product; can be influenced by the manufacture and inspection of finished products; and are used in addition to dimensional tolerances and used to control more precise profiles and shapes of products. They are used only when the profile or shape has a specific function and errors could impair its performance.

Standard ISO 1101 contains basic information and specifies requirements for geometric tolerancing of products. Geometric tolerances are divided into the following categories:

- form tolerance (does not require a reference – base);
- orientation tolerance (requires a reference – base);
- position tolerance (requires a reference – base);
- runout tolerance (requires a reference – base).

Standard ISO 22432:2011 gives more detailed taxonomy of features and links and the concept of the surface model between nominal features, specification features and verification features. More elements such as geometric specifications, characteristics and conditions can be found in standard ISO 25378:2011. Requirement of geometric characteristics and condition is defined by each and every GPS (Fig. 2).

Differences from the nominal shape are called deviations. Geometric tolerances determine the deviation of a workpiece element from its shape, orientation or position and from the theoretically accurate (ideal) shape, orientation or position without reference to the dimensions of the elements.

However, the surface of the parts is also a separate element, for example, a cylinder or two parallel planes. Geometric tolerances are applied independently of the actual local dimensions of the individual elements, and they should only be prescribed if justified, for example, from a functional point of view.

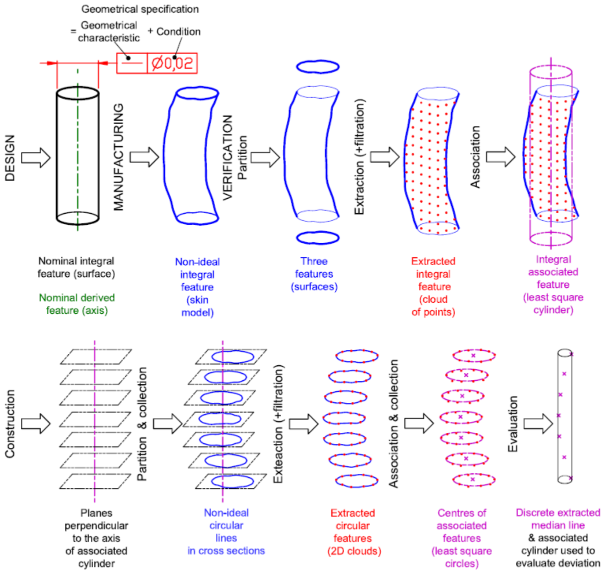

Fig. 2. Definition of the basic idea of GPS
GPS, geometrical product specification

3. APPLICATION OF GEOMETRIC SPECIFICATION OF PRODUCTS IN THE DESIGN OF GEAR COMPONENTS

Gears are most often composed of non-standardised and standardised components. Non-standardised components include shafts, gears, bottom and top of the gear case, lids, spacers and separator rings. Standardised components include in particular fasteners, seals, wedges and screws and also bearings and seals. For all non-standardised gearbox components, it is necessary to create drawing documentation, which is used for the actual production of the component and for its inspection. In addition to the shape and dimensions, the drawings must contain other data, including the definition of tolerances, which play an important role in the assemblability of the final product. These geometric specifications when creating drawing documentation using CAD applications can be defined in the library of the relevant program or defined manually. In this case, they were manually defined according to the standards.

3.1. Condition of assemblability of components

The installation or assembly of the manufactured machine components depends on a combination of two interrelated effects, namely, dimension and tolerance. Dimensions are obtained by measuring a finite number of points of element feature of size – the quantity of accuracy, geometric deviations of extracted features and their derived geometric features; for example, a set of holes (in a pair of plates) and the rivets that connect them – quantity of accuracy.

The smallest clearance occurs when each of the dimensional elements to be joined has the dimension of the maximum of the material and when the geometric deviations of the dimensional elements (e.g., deviation of shape, orientation and position) and their derived geometric elements (centre line or median plane) are also larger.

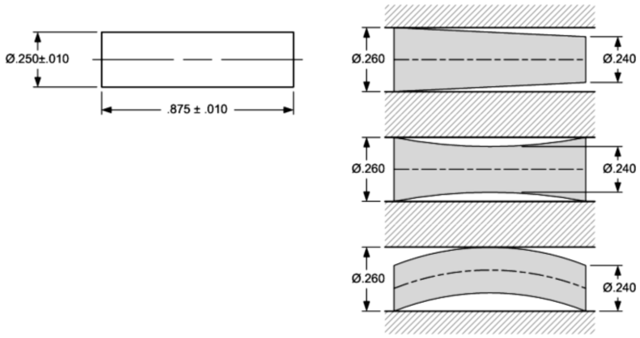

Fig. 3. Typical setting situations

The largest clearance occurs when the dimensions of the kinematically associated dimensional elements have the largest deviation from the dimension of the material maximum (e.g., the shaft with the smallest diameter and the hole with the largest diameter) and when the geometric deviations of the dimensional elements (e.g., deviations in shape, orientation and position) and their derived elements are zero.

It follows that when the actual dimensions of the parts to be joined do not reach the maximum dimension of the material, the geometric tolerance of the dimensional elements and their derived elements can be increased without compromising the assemblability of the parts. Typical situations that may arise when a pin is seated in a gear case bore, as shown in Fig. 3.

The position tolerances (position, concentricity, symmetry) for some functional cases are not sufficient to ensure their function. The projected tolerance zone indicates these functional requirements more clearly on the drawing. This notation means that the prescribed tolerance zone for the toleranced element is projected into the direction of the facing (associated) counterpart. The length of the projected tolerance zone and its position with respect to the workpiece are pre-written with a marking.

For practical reasons, the exact theoretical value of the projected length is chosen to be equivalent to the functional length of the counterpart.

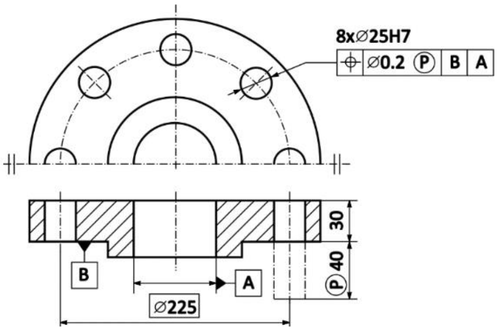

Fig. 4. Example of dimensioning a hole on the gearbox cover

The standard is applied in those specific cases where geometric tolerance is required in the space outside the component itself,

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most often in a second component that forms a fit with the component to be toleranced. The tolerance zone is moved from the part to a specific position, the length of which must be dimensioned.

The requirement for an offset tolerance zone is expressed by writing the letter P in a circle after the numerical tolerance value in the tolerance box. The outline of the displaced toleranced element (pin, bolt, pin, etc.) should be marked with a thin dotted line with two dots, the position of the element should be coded, and the P in the circle should be entered before the numerical value of the dimension. A typical example is the dimensioning of a hole on the gear case lid (Fig. 4). The prescription will ensure, for example, the correct position of the pins that are pressed into the holes 25H7.

In Fig. 5, an example of the application of the offset tolerance zone prescription in conjunction with geometric location tolerance is selected. The screw (position number 3 in Fig. 5) is loosely seated in the upper part of the gear case body (position number 2 in Fig. 5) and is screwed into the lower part of the gear case body (position number 1 in Fig. 5). When assembled, clearance in the holes of component 2 must be guaranteed. In the case of a geometric location tolerance prescription without an offset tolerance zone prescription, the screwed-in bolt may take a position that will not allow the components to be assembled. The prescription of a displaced tolerance zone means that the prescribed cylindrical positioning tolerance of the T = 0.2 mm axis with a length of 27 mm is displaced to component 2. This ensures that parts 2 and 3 can be assembled with the same value of geometric location tolerance (T = 0.2 mm).

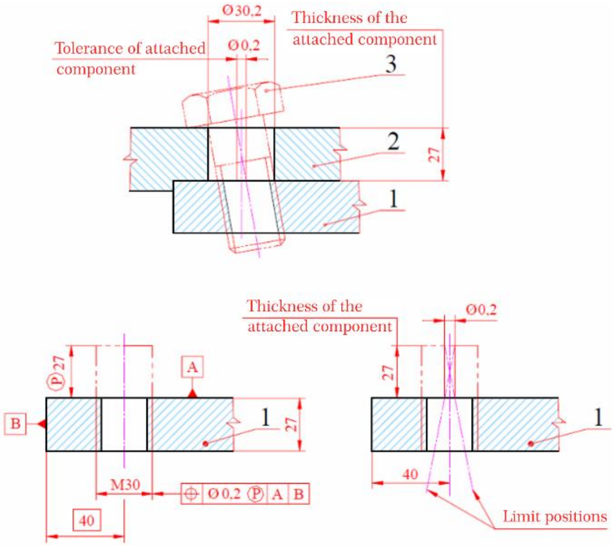


Fig. 5. Example of application of the offset tolerance zone prescription for gearbox assembly

The envelope inspection condition may be applied to a single element (e.g., the surface of a cylinder) or to a surface formed by two parallel planes (e.g., a groove).

The envelope condition is prescribed on the drawing by a mark E in a circle placed after the tolerance of the corresponding length dimension (Fig. 6a).

The condition is advantageous for the elements that will together form the fit. The prescription of the enveloping area condition (by the entry of the E mark in the ring and the entry TOLERATION ISO 8015) ensures that despite the validity of the independence of dimensions and geometry, no part of a cylindrical element or of an element formed by two parallel planes can exceed the enveloping area of the correct geometric shape with a dimension at the maximum limit of the material (e.g., due to geometric variations of shape and orientation). This means that the geometrically correct shape of the element must be respected for the dimensions of the element at the maximum limit of the material.

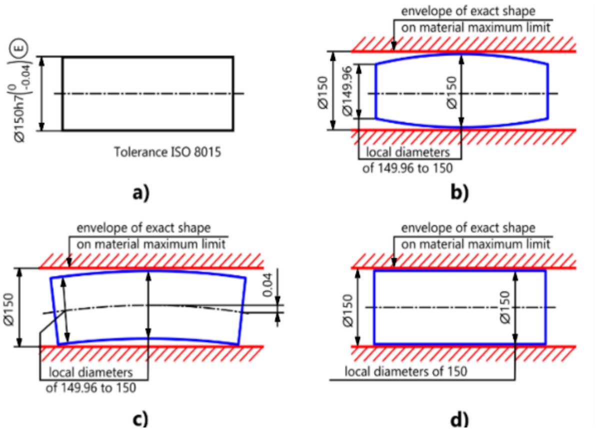


Fig. 6. Prescription of the envelope condition for the shaft: (a) marking on the drawing, (b) deviation of geometric shape, (c) deviation of geometric shape and (d) shaft with diameters at the maximum limit of the material

For elements of the character of the shaft (Fig. 6), which is one of the main elements of the gearbox, for example, the condition of the enveloping area of the cylindrical shaft prescribed in the drawing of the element according to Fig. 6a), is thereby determined:

- that the entire toleranced element (e.g., cylindrical shaft) must lie inside the envelope of the correct geometric shape with a dimension equal to the upper limit dimension of the toleranced element (material maximum), that is, 150 mm;
- that each actual local shaft dimension must be within the tolerance zone IT7 = 0.04 mm and can vary from 149.96 mm to 150 mm (Figs. 6b, c). The lower limit shaft dimensions (149.96–150 mm) can only be checked by two-point measurement.
- this means that if the shaft has actual local diameters at the maximum of the shaft material, that is, with an upper limit dimension of 150 mm, the shaft must be exactly cylindrical (Figs. 6d).

3.2. Position tolerances

Position tolerances include position, concentricity and symmetry. Position tolerances have their application on production drawings of the lids (Fig. 4), as well as on drawings of the top and bottom parts of the gear case.

The position tolerances, which are determined by the geometric elements related to the base, do not define the shape tolerances of the geometric elements of the base itself. It should be noted that the position tolerances include shape tolerances and orientation tolerances of the toleranced feature.

Fig. 7 is an example of the tolerance of the position of the centre of a sphere related to a basic system formed by three planes.

The tolerance zone is defined by the limits of a sphere with diameter t = 0.3 mm, with the tolerance value prefixed by the mark Ø. The centre of the spherical tolerance zone is given by the arrangement of the theoretically accurate dimensions relative to the bases A, B and C. The selected centre of the sphere is to be located within a spherical centre zone of diameter 0.3 mm, which is identical to the theoretically accurate location of the sphere with respect to the base formed by the planes A, B and C.

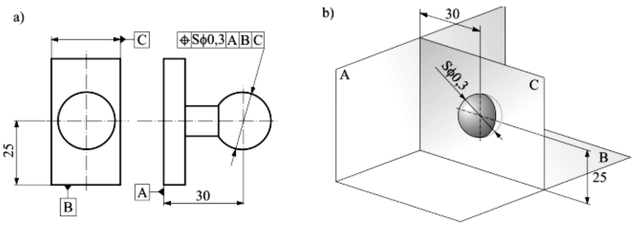


Fig. 7. Location of the centre of the sphere related to the base system formed by the three planes: (a) prescription on the drawing and (b) tolerance zone

Fig. 8 shows an example of the tolerance of the axis location related to the system formed by the three planes. It is an example of tolerancing the position of the hole on the gearbox mounting flange, which is used to attach the gearbox to the base – the frame. The tolerance zone is defined by the cylinder or diameter t, with the tolerance value prefixed by the mark Ø. The axis is given by the theoretically exact dimensions with respect to the bases C, A and B. In the example, the extracted (derived) axes should be in a cylindrical zone with an axis diameter of 0.08 mm, which coincides with the theoretically exact location of the hole, relative to the base formed by planes C, A and B.

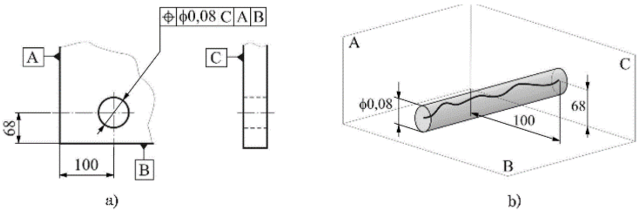


Fig. 8. Location of the axes related to the system formed by the three planes: (a) prescription on the drawing and (b) tolerance zone

Another element used in shaft tolerancing is the tolerance of concentricity. In Fig. 9, the extracted (true) centreline of the toleranced cylinder must be inside a cylindrical zone 0.08 mm in diameter whose axis is identical to the common A-B base line.

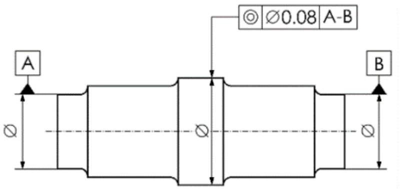


Fig. 9. Example of concentricity tolerance

Another tolerance used on gear shaft drawings is the tolerance for the symmetry of the midplane.

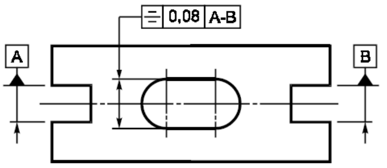


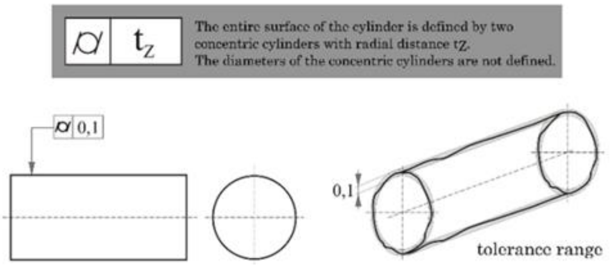
Fig. 10. Example of symmetry tolerance

In Fig. 10, the extracted (true) midplane must be located between two parallel planes spaced apart by a tolerance value of 0.08 mm, which are symmetrically arranged to the common A-B base plane.

3.3. Form tolerance

The irregularity of surfaces or profiles can be divided into shape, waviness and roughness according to the ratio of the distance between the irregularities (deviations, waves, cracks, etc.) and their depth. Form deviation is the value of the deviation of the actual shape of the workpiece from its nominal designed shape as indicated on the drawing, or defined in the standard as the maximum permissible distance of points of the actual surface from the envelope surface.

The cylindricity tolerance defines the limits of the straightness deviation, the deviation of the cylinder forming lines (axes), the roundness deviation of the cylinder cross-sections and the deviation of the parallelism of the opposite forming lines of the elements. The tolerance zone is defined by two concentric cylinders. When checking the deviation of the form, the complete surface of the real cylinder should not exceed the pitch of the two concentric cylinders, which are spaced apart by a minimum distance, without considering the size of the cylinder (Fig. 11).



The entire surface of the cylinder is defined by two concentric cylinders with radial distance tZ. The diameters of the concentric cylinders are not defined.

Fig. 11. Definition and explanation of cylindricity deviation

The cylindricity tolerance is used on production drawings of shafts and drawings of gear case bodies, to tolerate the cylindricity of holes.

3.4. Runout tolerance

In general, runout refers to the description of workpiece surfaces that have a rotationally symmetrical shape and their deviations from a theoretically accurate circular shape. On production drawings of gear shafts, the tolerance of the total circular runout finds its application. Tolerance of total runout is characterised by the fact that the locations of the zones for the different cuts are strictly related – they all have the same initial zero point.

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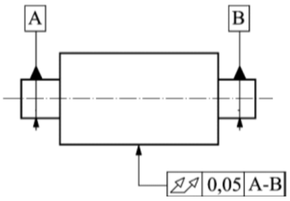

Fig. 12. Total runout tolerance

The total runout is indicated by a mark with two arrows in the tolerance box (Fig. 12). The zone is the volume defined by two concentric cylinders whose axes are coincident with the dotted axis defined by the basic system and whose radii differ by a prescribed tolerance value. For the workpiece, the actual area must be within the volume between two concentric cylinders whose radii differ by 0.05 mm and whose axes are coincident with the general datum formed by the straight line A-B.

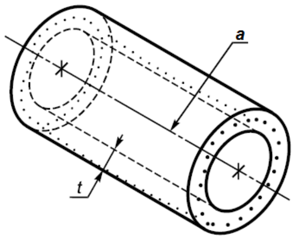

Fig. 13. Definition of tolerance zone: a – base A-B

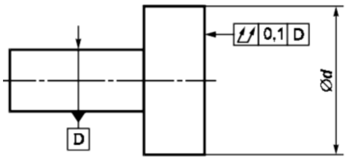

Fig. 14 Designation and explanation of axial total runout

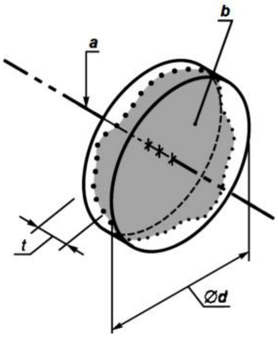

Fig. 15. Definition of tolerance zone: a – base D, b – extracted surface

The tolerance zone is bounded in Fig. 13 by two concentric cylinders whose radial distance is equal to the value of t and whose axes are identical to the baseline.

In addition to radial runout, axial runout is also used, an example of which is shown in Fig. 14.

In Fig. 14, the extracted (true) surface must be located between two parallel planes whose radial distance is equal to a tolerance value of 0.1 mm and which are perpendicular to the axis of the base D.

The tolerance zone is bounded in Fig. 15 by two parallel planes spaced apart by t, which are perpendicular to the base.

3.5. Example of gearbox shaft tolerance

Fig. 16 shows part of the gearbox input shaft fit. The example of shaft dimensioning will illustrate the dimensioning rule for individual tolerances on the manufacturing drawing of the shaft.

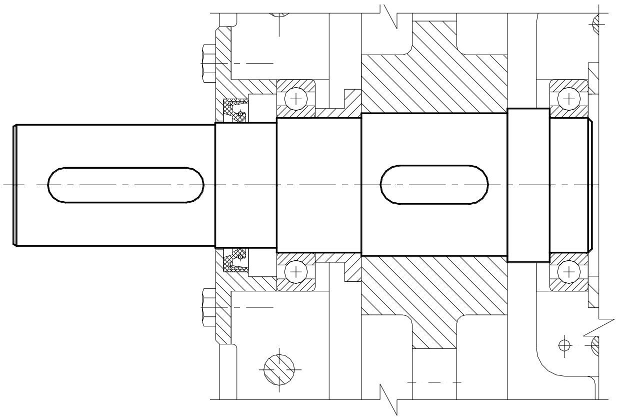

Fig. 16. Gearbox input shaft arrangement

Fig. 17 illustrates the prescribed tolerances for the individual functional dimensions, which are selected as follows. Under position number 1, this is the prescription for the cylindricity tolerance: its value is ‘10 μm = 0.01 mm’ and is determined from the standard for the nominal dimension Ø63m6 (nominal dimension range from 50 mm to 120 mm) and precision grade 6.

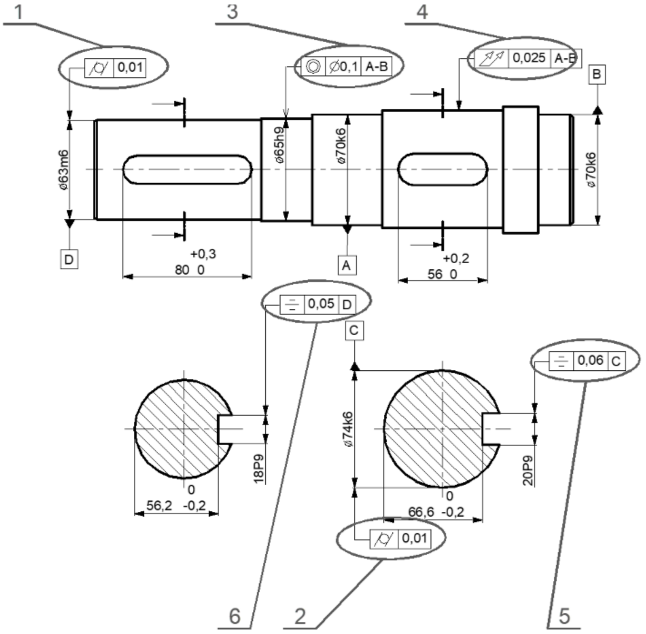

Fig. 17. Prescription of geometric tolerances on the shaft – example

Under position number 2 is the cylindricity tolerance. The value of the cylindricity tolerance “10 μm = 0.01 mm’ is determined according to the standard for the nominal dimension Ø74k6 (nominal dimension range from 50 mm to 120 mm) and accuracy grade 6.

Under position number 3 is the tolerance of concentricity. The value of the concentricity tolerance is ‘100 μm = 0.1 mm’ for the nominal dimension, that is, for the diameter of the considered rotating surface Ø65h9 and accuracy grade 9.

An example of a total circumferential runout tolerance prescription is given under position number 4. The tolerance value of the total circumferential runout is ‘25 µm = 0.025 mm’ for the nominal dimension, that is, for the diameter of the rotating surface under consideration Ø74k6 and accuracy grade 6.

An example of a symmetry tolerance prescription is indicated by position numbers 5 and 6. In the first case, the value of the symmetry tolerance is ‘60 µm = 0.06 mm’ and is determined for the nominal dimension of the groove width for the tight pen 20P9. In the second case, the value of the symmetry tolerance is ‘50 µm = 0.05 mm’ and is determined by standard for the nominal dimension, that is, for the dimension between the faces forming the element 18P9.

4. CONCLUSIONS

Based on the analysis of the application of the basic principles of GPS, the drawing documentation of each non-normalised gearbox element must satisfy the following rules:

- All dimensions must have a tolerance. Every element on a manufactured component is subject to variation, and therefore, tolerances must be prescribed. Upper and lower tolerance limits are prescribed directly for the dimension, or a block of basic parallel dimensions is used. For basic dimensions, geometric tolerances are indirectly prescribed by means of a tolerance box. The only exceptions are minimum and maximum dimensions, semi-finished products or starting points.
- Dimensioning and tolerancing must fully define the nominal geometry and tolerance.
- Technical drawings define the requirements for finished (complete) components. It is required that every dimension and tolerance that define the finished part are indicated on the drawing. If additional dimensions are required but not necessary, they may be marked as reference ones.
- Dimensions should be applied to elements and arranged in such a way as to represent the function of the element.
- Descriptions of manufacturing processes should be avoided. The geometry should be described without explicitly defining the method of manufacture.
- If a dimension is needed in the manufacturing process but is not required in the final geometry (due to dilation or other reasons), then it should be marked as informative.
- For maximum clarity, all dimensions and tolerances should be placed so that they are at the dimension and extension lines and at the correct elements.
- If it common practice to check geometry by limit gauges or by a marked code (e.g., a blank material code), then such dimensions should have the limit gauge or code indicated in brackets or below the dimension.
- Unless otherwise indicated on the drawing, all prescribed dimensions and tolerances apply at 20°C.
- Unless otherwise expressly stated, all dimensions and tolerances apply to the un-preloaded condition.
- Dimensions and tolerances refer to the overall length, width and depth of the element.

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