TOMSK POLYTECHNIC UNIVERSITY

A. B. Kim

METROLOGY, STANDARDISATION AND CERTIFICATION

Tomsk Polytechnic University Publishing House
2011

UDC 621-187+681.2 (075.8)
BBC 00000
K00

Kim A.B.
K00 Metrology, standardisation and certification: study aid / A.B. Kim; Tomsk Polytechnic University. – Tomsk: TPU Publishing House, 2011, 100 p.

This book is devoted to dimensional and geometrical tolerancing principles, standardisation and certification aspects and measurement instruments applied in mechanical and manufacturing engineering. The material of this book is written in accordance with the relevant state standards of Russian Federation.

The book is recommended for English-speaking students following the Bachelor Degree Program in Mechanical Engineering at Tomsk Polytechnic University.

UDC 621-187+681.2 (075.8)
BBC 00000

Linguistic Advisor
PhD (Philology)
Leading expert of the Centre for Education Quality Assurance, TPU
R.I. Tolbanova

Reviewers

PhD (Technical)
Associate professor of the
Department of Automated Mechanical Manufacturing Engineering, TPU
O.B. Shamina

PhD (Technical)
Head of the Center for Learning and Training of Teachers, Non-State Educational Institution “Open Youth University”
M.Yu. Dorofeeva

Preface

This textbook is written for students of mechanical engineering majors and learners who intend to apply tolerancing principles and measurement techniques in practice. This work is designed to be used as a textbook in engineering, manufacturing, inspection and design curriculums.

This book is composed of materials relevant to the syllabus "Metrology, Standardisation and Certification" that is taught at the department of Automated Mechanical Manufacturing Engineering and is a long-term teaching experience of the author. To facilitate understanding of the core material the textbook is supported with pictures and drawings, a set of examples of various fits and their specifications is also given.

The content of the book is organized into seven chapters, each devoted to different aspects of the course. The topics covered in the book are as follows: dimensional and geometric tolerancing, surface texture parameters, tolerance analysis, linear measurement methods and instrumentation, standardisation and certification. The material in this book is written in accordance with the state standards related to dimensioning, tolerancing, drawing specifications and metrology. The reader is not expected to know how to read engineering drawings.

CONTENTS

1. Principles of Dimensional Tolerancing

The vast majority of modern machines and devices are an assembly of many interacting components, which should provide successful performance of the whole machine. The violation of the dimensional accuracy or other parameters of components can greatly affect the machine performance and reliability. And since no industrial manufacturer produce all the range of components needed for the given machine assembly, it is essential for the manufacturer and suppliers to follow the same norms and standards. And in the first place we should mention standards that form the basis of interchangeability.

1.1 Interchangeability

In the past each pair of mating parts was produced individually and could not be interchanged with parts from other pairs of similar components. This was due to the absence of rules for assigning limitations to dimension variations which are in fact specifications needed to guide the manufacturing processes. This was highly inconvenient and led to standardisation in dimensional tolerancing which in turn formed a base for interchangeability. According to the basic principle of interchangeability every component made must meet design specifications within a specified range of allowable variation. Nowadays, for instance, any M20x1.5 mm nut is expected to fit any M20x1.5 mm bolt no matter where they were made.

Thus, by **interchangeability** we mean the ability of independently made components to substitute worn or faulty components of the same type without loss of function or suitability.

Benefits of interchangeability are as follows:

- Increased availability
- Possibility of cooperative low cost manufacturing
- Cost reduction due to increased volume of manufacture
- Quality assurance
- Effective maintenance
- Possibility of assembly automation

1.2 Manufacturing Accuracy and Deviations

Since every machining process may have inherent inaccuracies, in practice it is not possible to manufacture products to the exact dimensions. Among such inaccuracies we can mention machine tool imperfections, cutting tool wear, irregularity of workpiece material properties, human elements etc. Thus to provide interchangeability modern mass production requires parts to be machined to sizes which are limited to some degree. These size limitations are indicated in dimensions on engineering drawings.

For the purposes stated above the following terms are used.

Shaft. This term is used for specification of all outer elements of the component regardless of their shape. Shaft sizes are denoted by the lower case letters. Examples: d, d_{min}, d_{max}.

Hole. This term is used for specification of all inner elements of the component regardless of their shape. Hole sizes are denoted by the upper case letters. Examples: D, D_{min}, D_{max}.

Basic size. It is designated as **D** for hole and **d** for shaft. This is the exact theoretical size from which the limits are established through the applications of tolerances and deviations. The basic size is the same for both the hole and the shaft of a pair. The basic sizes are found from strength and stiffness calculations, manufacturability reasons are also taken into account (refer to Fig. 1.1).

Actual size. This is the size of the finished part obtained through measurement with permissible error.

Limits of size. These are two extreme permissible sizes for any particular dimension between which the actual size should lie. There are two limits of size: minimum limit of size and maximum limit of size (Fig. 1.1). Limits of size are designated as $\mathbf{D_{max}}$ and $\mathbf{D_{min}}$ for hole and $\mathbf{d_{max}}$ and $\mathbf{d_{min}}$ for shaft.

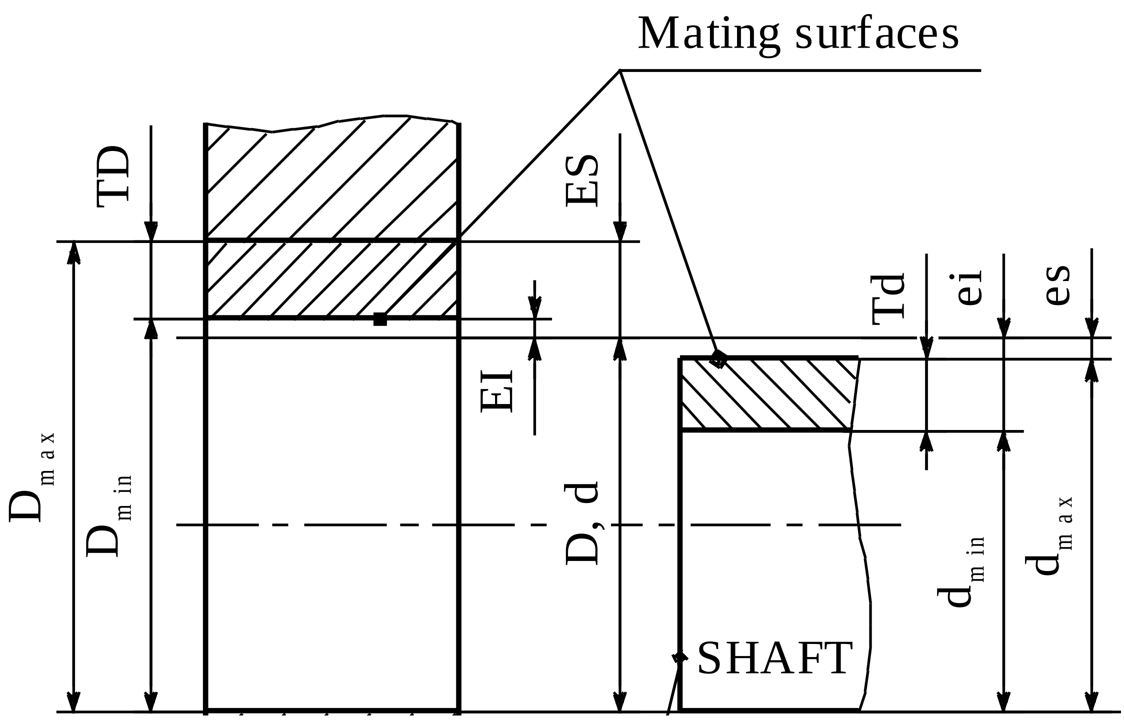


Fig. 1.1 Conventional diagram

To ease the dimensioning in a drawing the following terms defining limit deviations with relation to the basic size are implemented into State standards.

Lower Deviation is the algebraic difference between the minimum limit of size (of either hole or shaft) and the corresponding basic size. It is designated as **EI** for hole and **ei** for shaft.

Upper Deviation is the algebraic difference between the maximum limit of size (of either hole or shaft) and the corresponding basic size. It is designated as **ES** for hole and **es** for shaft.

Upper and lower deviations are applied with relation to the given basic size which is a **zero line** in graphical representation. When a deviation is above the zero line it is the positive quantity and when a deviation is below the zero line it is the negative quantity (refer to Fig. 1.2 and 1.3). Examples: +ES, -ES, +ei, -ei, -es etc.

Actual deviation is the algebraic difference between the actual size and the corresponding basic size. Actual deviation is positive if the actual size is bigger than the basic size and negative if the actual size is smaller than the basic size.

Each dimension on an engineering drawing includes a basic size and deviations which define the tolerance value for correct manufacturing, for example, 60±0.1. Hence, **tolerance** is the difference between maximum and minimum limits of size or absolute algebraic difference between the upper and lower deviations. It is designated as **TD** for hole and **Td** for shaft. The tolerance value is always positive. The manufacturing cost and product finish decrease with increasing tolerance value.

The tolerance positioned relatively to the basic size and restricted by the upper and lower deviations is called **tolerance zone.** It is used for graphical representation of dimensional tolerances as shown in Fig. 1.2 and 1.3.

Some examples of the graphical representation of tolerance zones are given below.

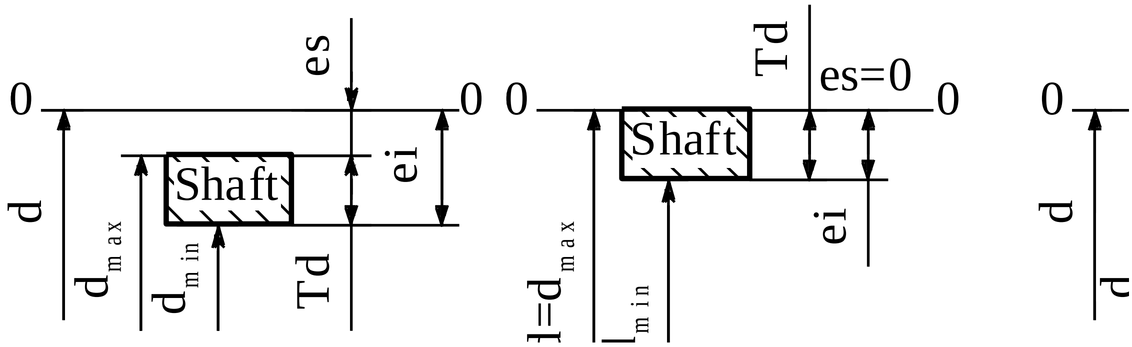


Fig. 1.2 Shaft tolerance zones

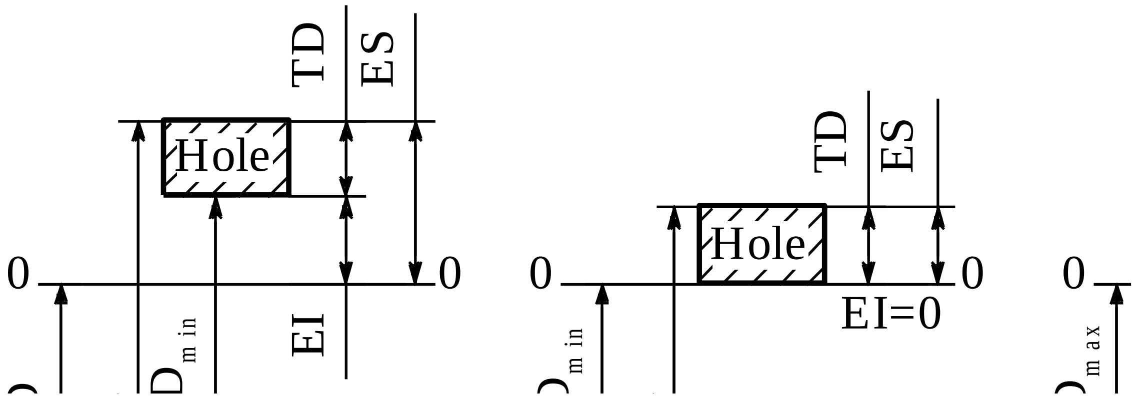


Fig. 1.3 Hole tolerance zones

1.3 Fits

Fit is the relationship between two mating parts before coupling; fit also defines the degree of freedom or relative displacement of these parts in the joint. In other words, fit identifies the algebraic difference between sizes of the parts before the assembly or defines the degree of looseness or tightness between coupling parts. Mating parts that make a fit are called a hole and a shaft, for inner and outer parts respectively, regardless of their shape. A collection of fits obtained through experience is set as a standard and is being used in industrial applications. Among general types of fits there are fits for smooth parts, rolling bearings, cones, threads and key joints. Fits used for smooth parts are discussed in this section; some other types are discussed in sections 1.6-1.9.

Depending on the mutual position of tolerance zones of a hole and a shaft the fit formed can be of three types: clearance fit, transition fit and interference fit.

1.3.1 Clearance Fit

Clearance fit (Fig. 1.4 and 1.5) always provides clearance between mating parts. Therein **clearance** is the difference between hole and shaft sizes before assembly, provided that the shaft is smaller than the hole. The clearance is designated by capital letter S. Maximum, minimum and mean clearances are calculated by the following equations:

$$S_{\max} = D_{\max} - d_{\min},\ S_{\min} = D_{\min} - d_{\max},\ S_m = (S_{\max} + S_{\min})/2 \qquad (1.1)$$

The term tolerance can also be applied to the system of fits. In the case of clearance the **fit tolerance** is the difference between maximum and minimum clearances as shown in equation below.

$$TS = S_{\max} - S_{\min} \qquad (1.2)$$

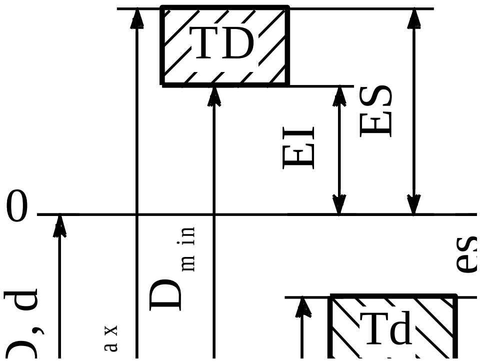


Fig. 1.4 Schematic illustration of clearance fit

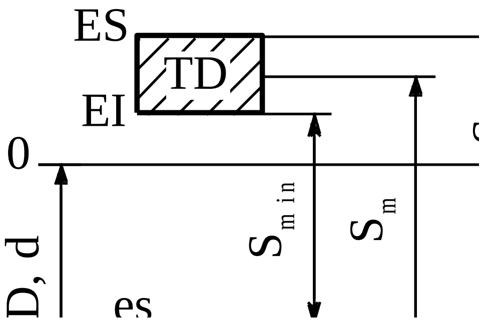


Fig. 1.5 Schematic illustration of maximum and minimum clearances

1.3.2 Interference Fit

Interference fit (Fig. 1.6 and 1.7) always provides interference between mating parts. **Interference** is the difference between shaft and hole sizes before assembly, provided that the hole is smaller than the shaft. The interference is designated by capital letter N. Maximum, minimum and mean interferences are calculated by the following equations:

$$N_{\max} = d_{\max} - D_{\min}\,,\ N_{\min} = d_{\min} - D_{\max}\,,\ N_m = (N_{\max} + N_{\min})/2 \quad (1.3)$$

The interference fit tolerance is the difference between maximum and minimum interferences as shown below.

$$TN = N_{\max} - N_{\min} \quad (1.4)$$

Since during the assembly the coupling parts undergo elastic deformation the interference value can be calculated as

$$N = \Delta D + \Delta d \quad (1.5)$$

where: ΔD and Δd are elastic deformations for hole and shaft, respectively.

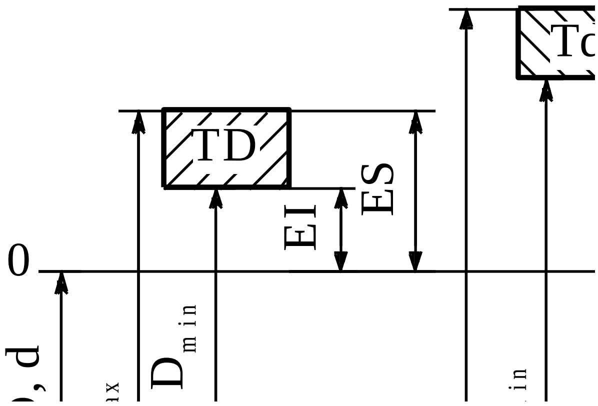


Fig. 1.6 Schematic illustration of interference fit

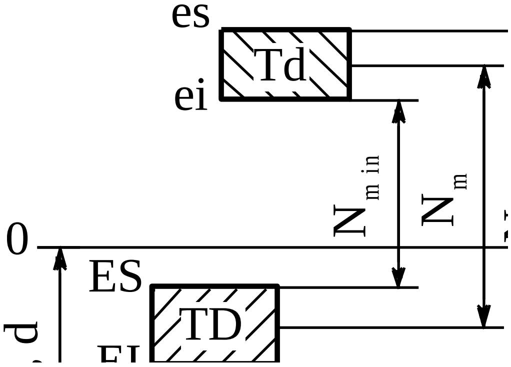


Fig. 1.7 Schematic illustration of maximum and minimum interference

1.3.3 Transition Fit

In **transition fit** both clearance and interference may occur between mating parts. Thus hole and shaft tolerance zones partly or completely overlap, eliminating minimum clearance and interference (refer to Fig. 1.8). Therefore only maximum clearance and maximum interference can exist in transition fit. The transition fit tolerance is the sum of the absolute values of the maximum clearance and maximum interference of the fit.

$$TS(TN) = S_{max} + N_{max} \tag{1.6}$$

For all types of fit the fit tolerance can also be calculated as:

$$TS(TN) = TD + Td \tag{1.7}$$

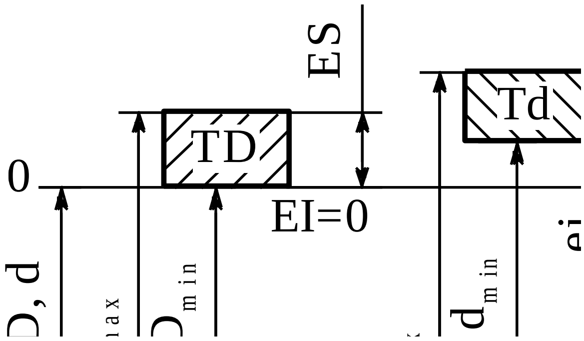


Fig. 1.8 Schematic illustration of transition fit

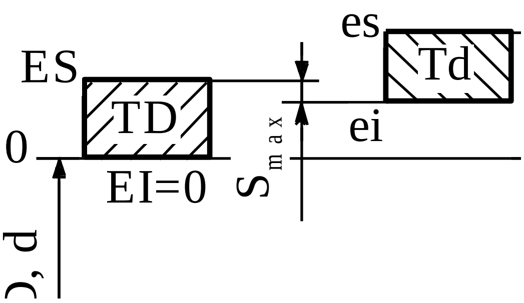


Fig. 1.9 Schematic illustration of the maximum interference and clearance

Since both clearance and interference may occur in a transition fit, fits of this kind are divided into three groups, depending on which one is more probable. Fig. 1.9 illustrates the situation when the interference probability is higher compared to that of the clearance. This clearly shows that the majority of the components will be assembled with interference fit rather than the clearance fit. From this diagram one can see that the higher interference probability is also characterized by simple correlation between the maximum interference and maximum clearance values, with maximum interference being bigger in the case. Equations (1.8) can be used to identify whether the transition fit has higher probability of interference or clearance or equal probability of both.

$$\begin{gathered} |S_{max}| > |N_{max}| \\ |N_{max}| > |S_{max}| \\ |S_{max}| = |N_{max}| \end{gathered} \tag{1.8}$$

The illustration of a transition fit with different interference/clearance conditions is given in Fig. 1.10.

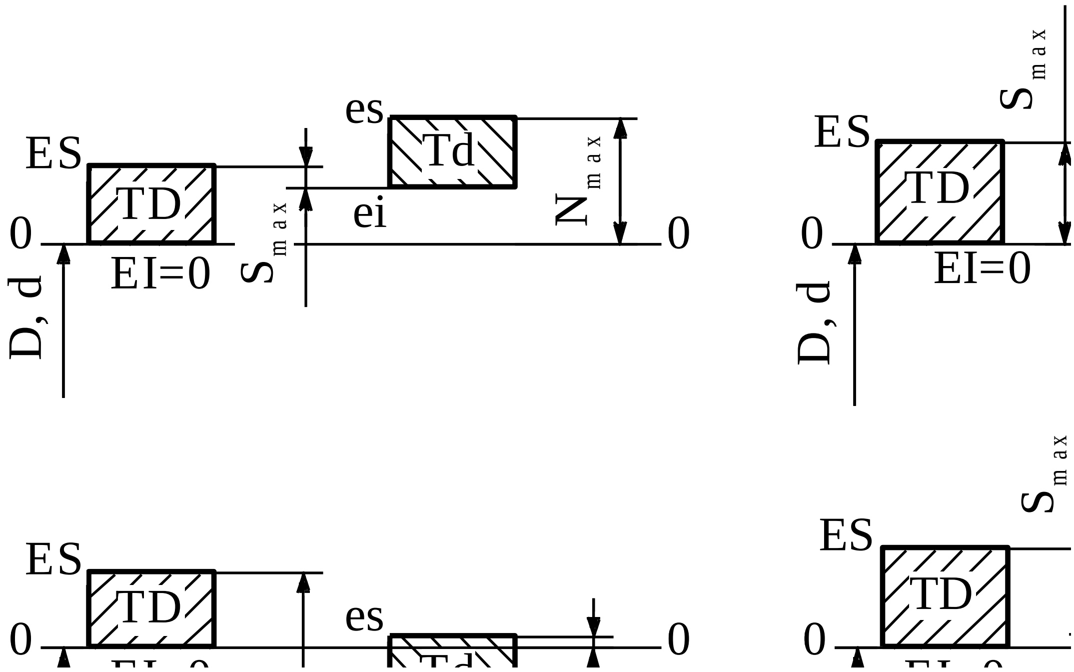


Fig. 1.10 Examples of a transition fit

1.3.4 Systems of Fit

Although a hole and a shaft can be coupled in different combinations of the tolerance zones, only two methods of coupling are recommended. These methods are called a hole-basis system of fit and shaft-basis system of fit. These systems are preferable due to constructional, technological and economical advantages they provide.

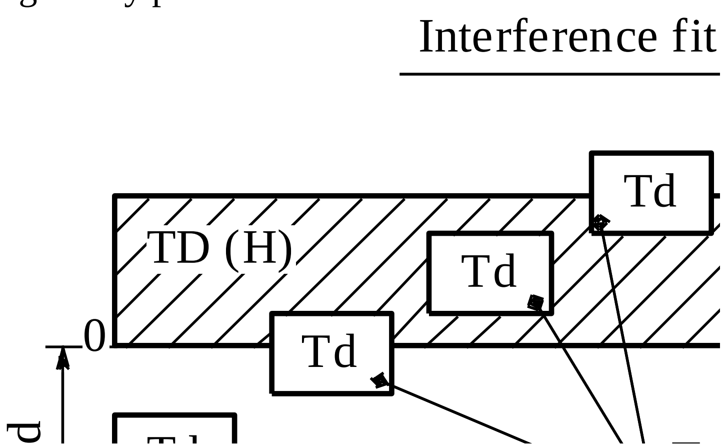


Fig. 1.11 Hole-basis system of fit

In the **hole-basis system of fit** the lower deviation of a hole is equal to zero and the desired clearances and interferences are achieved through the application of different tolerance zones for a shaft (refer to Fig. 1.11). This hole is called **basic hole** and designated by capital letter H. For the fits of this system the hole minimum limit of size is always equal to the basic size with the tolerance zone set right above the zero line. The hole-basis system is preferred since it is more economical to apply standard drills, core-drills, reamers, etc, as the application of these cutting tools will directly result in “H” tolerance zone. It is also more convenient to gauge a shaft than a hole, so application of the hole-basis system will help to reduce cost of measurement.

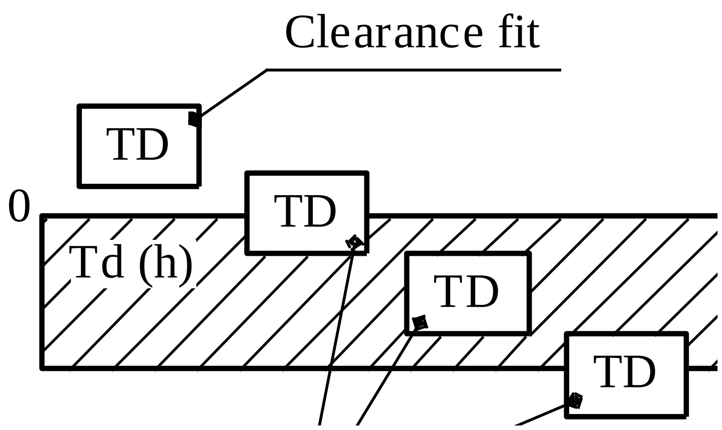


Fig. 1.12 Shaft-basis system of fit

In the **shaft-basis system of fit** the upper deviation of a shaft is equal to zero and desired clearances and interferences are achieved through the application of different tolerance zones for a hole (refer to Fig. 1.12). The shaft in this case is called **basic shaft** and designated by lower-case letter h. For the fits of this system the shaft maximum limit of size is always equal to the basic size with tolerance zone set right below the zero line.

With the hole-basis system being more economically preferable, the application of the shaft-basis fits in some cases may be the only applicable choice (refer to Fig. 1.13). In the example the pin is the basic shaft and the required fits are made with the application of corresponding tolerance zones for the piston and bushing holes. A problem with assembling of this coupling can arise, if one tries a hole-based system of fit in this case.

Another example of assigning the system of fit is shown in Fig. 1.14. Here, a standard item such as a ball bearing defines the choice of fit system. In this case the dimensions of neither outer nor inner rings of a bearing can be changed. Thus the fits needed for the operation of a unit are obtained by changing the tolerance zone positions of the mating parts.

It is also a common practice to apply shaft-basis fits when the “shaft” parts are to be manufactured from precise cold-drawn bar without post-machining.

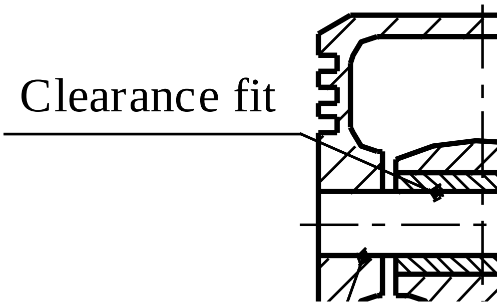


Fig. 1.13 Application of the shaft-basis system of fit

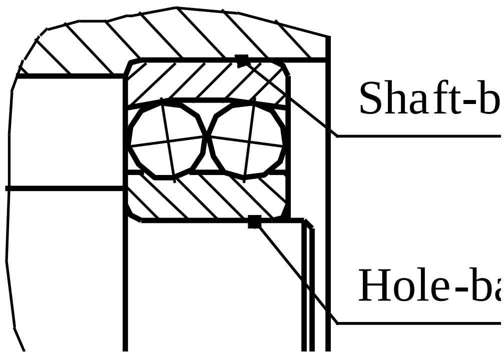


Fig. 1.14 Shaft-basis and hole-basis systems of fit applied for a standard ball bearing

In addition to the shaft- and hole-basis fits, off-system fits are also applied in practice. In this case other than “h” or “H” tolerance zones are used.

1.4 ISO Tolerance Grades

In order to meet the requirements of various products for accuracy of the components, the ISO system implements 20 grades of accuracy, which are called **tolerance grades** and denoted as IT (International Tolerance). Thus the tolerance of a dimension is marked as IT with the attached grade of accuracy, for example, IT01, IT0, IT1, IT2... up to IT18. Here, the bigger is the grade number, the bigger is the tolerance value for a given dimension. The given 20 IT grades allow an engineer to denote precisely the accuracy level needed for a component to perform well.

In order to cover the whole range of dimensions found in engineering applications, each IT grade defines a set of tolerances. Within any IT grade a tolerance value increases with increasing dimension value. This is consistent with practice since machining errors increase with the increase of cutting diameter, making it more challenging to comply with the same accuracy level for bigger dimensions. In ISO system the whole range of dimensions is subdivided into sub-ranges. The relationship between the dimension and accuracy is defined by the **standard tolerance factor**, which is denoted by letter ***i*** for the dimensions less than 500 mm, and ***I*** for the dimensions from 500 to 10000 mm. The standard tolerance factors are obtained from the following equation:

$$i = 0{,}45 \cdot \sqrt[3]{D} + 0{,}001 \cdot D \tag{1.9}$$

$$I = 0{,}004 \cdot D + 2{,}1 \tag{1.10}$$

where D is the geometric mean of the sub-range extreme dimensions, which is defined as $D=\sqrt{D_{\min}\cdot D_{\max}}$ or $D=\sqrt{3}$ for dimensions smaller than 3 mm.

Thus the specific tolerance for any IT grade can be calculated as

$$T=a\cdot i \qquad (1.11)$$

where ***i*** is a standard tolerance factor and ***a*** is a quantity of these factors for the grade. The tolerances are given in micrometers (μm). With Eq. (1.11) a series of tolerances for each tolerance grade are obtained.

It should be noted that all tolerances and deviations are applied to products and components with dimensions measured at a normal temperature, which is internationally set equal to +20 °C – the common workshop temperature.

1.5 Fundamental Deviations

As it was mentioned earlier (refer to Fig. 1.2-1.12) the tolerance zone can take different positions in relation to the zero line. In the ISO system the position of the tolerance zone is determined by a fundamental deviation. Among two deviations that form a tolerance zone the fundamental one is the nearest to the zero line. The ISO system specifies 28 classes of fundamental deviations for holes and 28 classes of fundamental deviations for shafts. Uppercase letters denote holes and lowercase letters denote shafts. Each letter specifies a class of fundamental deviations. The deviation value, within a class, depends on the basic size value and doesn't depend on the IT grade. A tolerance zone is denoted by a letter of fundamental deviation followed by a tolerance grade. An overview of fundamental deviations can be found in Fig. 1.15 and 1.16, and some examples of tolerance zones are shown in Fig. 1.17.

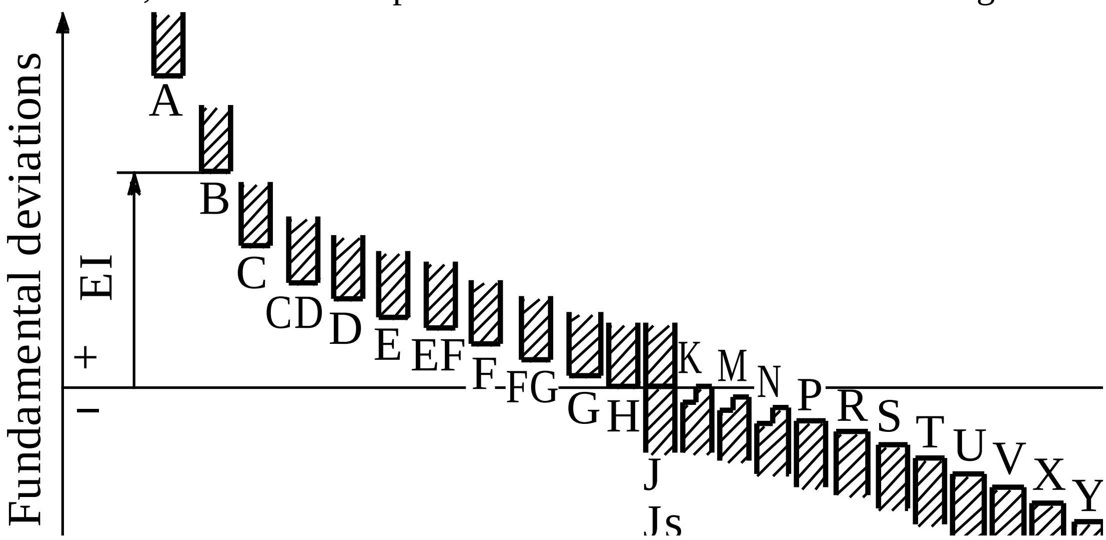


Fig. 1.15 Hole fundamental deviations

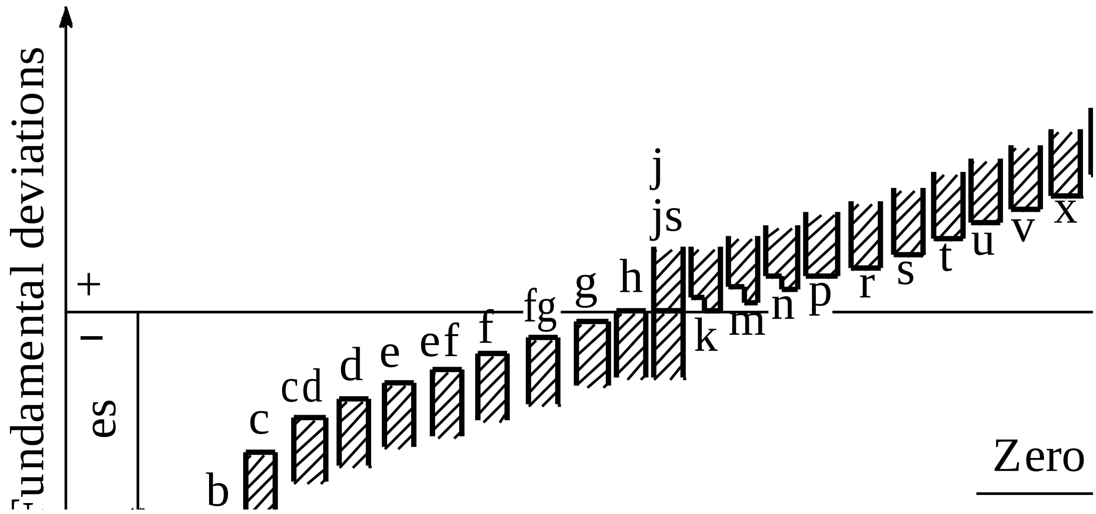


Fig. 1.16 Shaft fundamental deviations

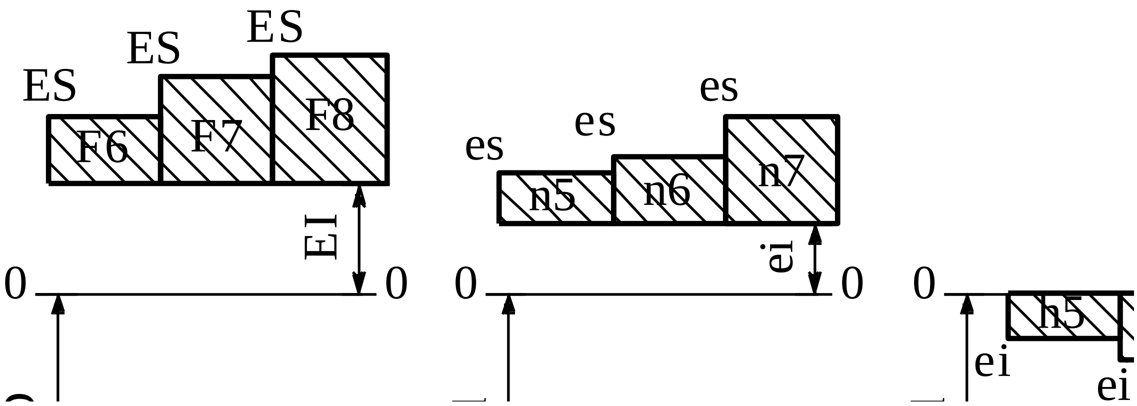


Fig. 1.17 Tolerance zones indication

Although there are extensive sets of tolerance grades and fundamental deviations, which can be used to compose tolerance zones by the mutual combinations, only a limited range of tolerance zones is found to be economically attractive. Some graphical examples of fits are given in Fig. 1.18.

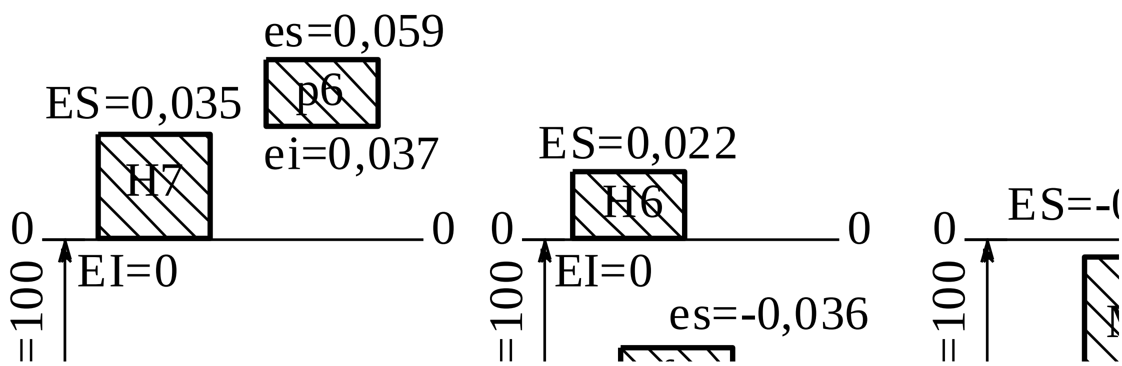


Fig. 1.18 Fits

1.6 Rolling Bearings Tolerancing

Rolling bearings are the most commonly used standard products, with the complete interchangeability principle for the outer and inner rings diameters ensured. The Russian Federation Standard, in line with ISO, implements six accuracy grades for bearings, which are, in ascending order of accuracy: P0 (normal), P6, P5, P4, T, P2 (the most precise). There are two extra grades, P8 and P7, of even lower accuracy, which are delivered as option.

Bearings are a standard industrial product and are not intended for additional machining of its surfaces in order to gain an appropriate fit so the fits are obtained through machining of housing and shaft surfaces to dimensions desired for the fit. Thereby fits for the outer ring, which is a basic shaft, are made in the shaft-basis system and fits for the inner ring, which is the basic hole, are made in the hole-basis system of fit. Although the bearing bore is a basic hole, the tolerance zone is positioned below the zero line instead of the above as for common case. This way of specifying the tolerance zone position is chosen to provide light-interference fits with the help of ordinary tolerance zones, such as n6, m6, k6 and js6, as applying a higher interference values on the flexible bearing rings will cause jamming of the rolling elements and consequent failure of the entire assembly. The examples of the tolerance zones for the bearing rings, shaft and housing are shown in Fig. 1.19. By combining tolerance zones of the bearing and either shaft or housing, a set of principal fits can be obtained.

The fits are selected in accordance with the conditions of the rings rotation, load rate, running accuracy and operating temperature.

The rotation conditions are specified by the rotation of one of the bearing rings with respect to the load applied and are denoted as local loading, circumferential loading and oscillation loading, as shown in Fig. 1.20.

In the case of the **local load** the bearing ring area exposed to the loading is local. In other words, the load of a ring is local if the ring is stationary with respect to the load direction. In order to distribute the load to the entire area of the ring, and therefore prolong its life, it should have a freedom to displace relative to its seating surface, which is achieved by the application of a loose fit.

With the **circumferential load** the ring of a bearing withstands the load by every point on its surface progressively. In other words the load of a ring is circumferential if the ring is rotating with respect to the load direction. In this case the load is distributed gradually along the whole circumference, thus the interference fit is necessary.

For **oscillation** loading, everything stated for local loading is true besides additional oscillating force is present.

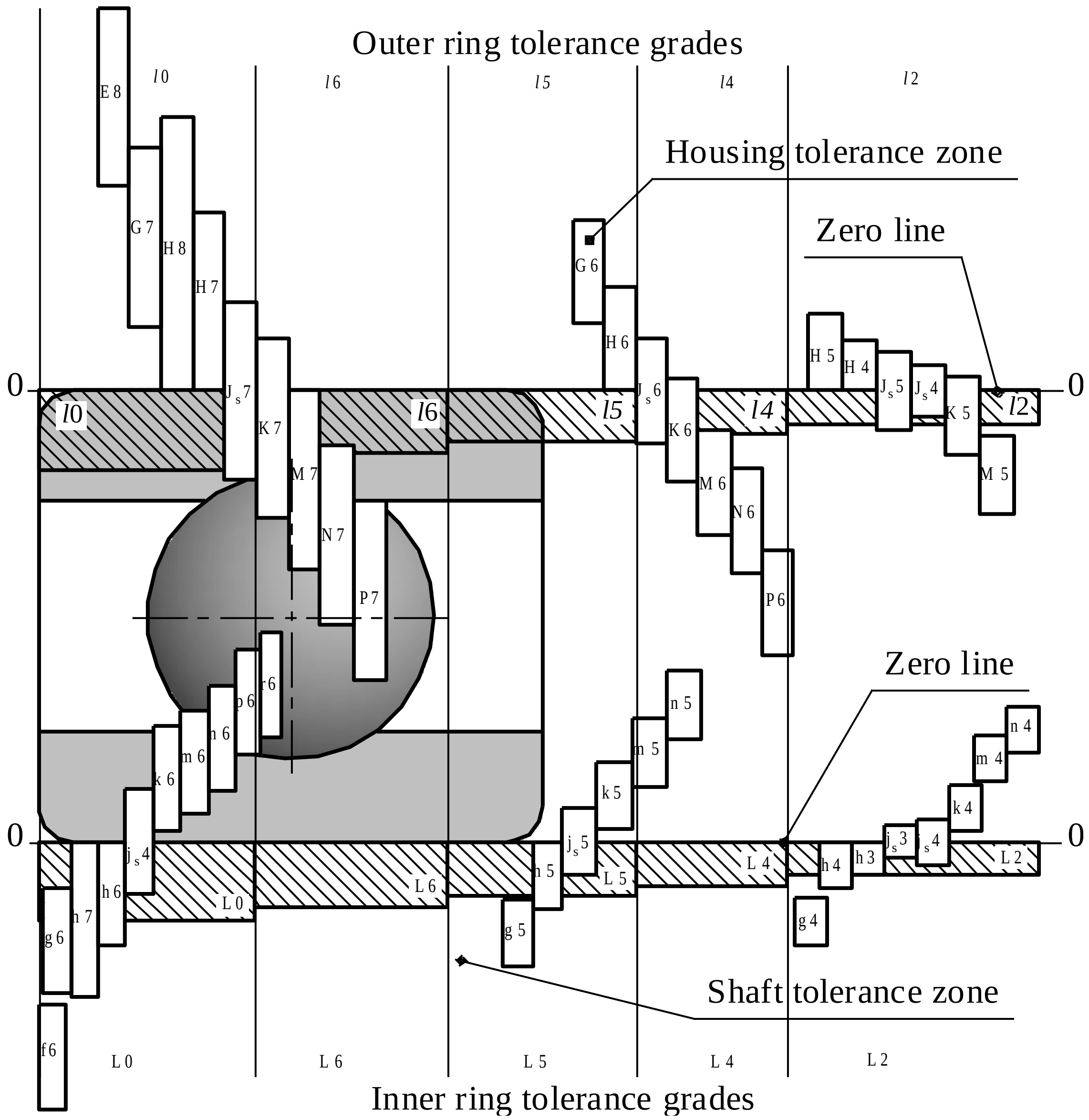


Fig. 1.19 Bearing fits

Since the bearing and mating components may have different temperatures in the course of operation, the dimensional changes caused by the different expansion coefficients should be taken into account, as it can lead to either bearing jamming or seating surface rubbing against the sliding ring.

Running accuracy is ensured by the appropriate seating surface finish with the high geometric accuracy, since the former can loosen the fit and the latter can cause bearing rings deformation with eventual assembly failure. As for the load rate, it should be noted that higher loads require tighter fits.

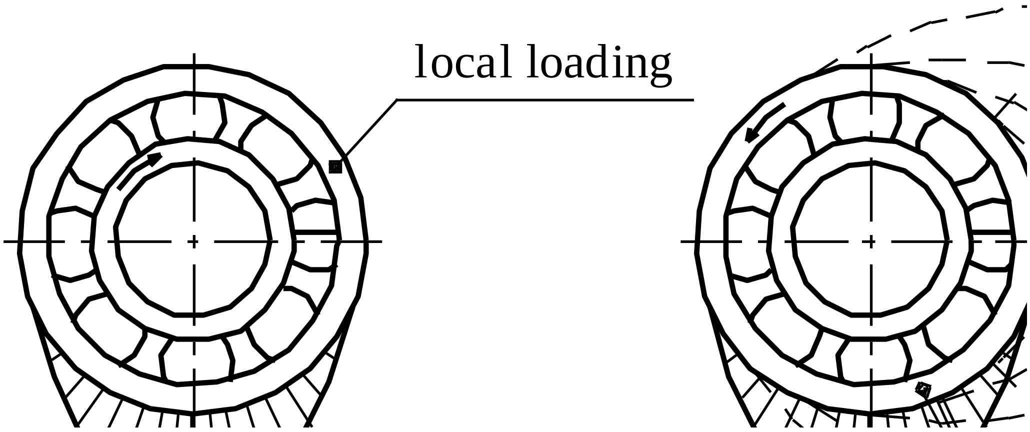


Fig. 1.20 Various rotation conditions

1.7 Limit Gauges Tolerancing

The component dimensional accuracy in mass production is often inspected by the limit gauges due to the simple form and relatively high performance. With the help of the limit gauges the cylindrical, tapered, threaded, splined and other types of components are inspected.

Inspection in this case implies the use of two limit gauges, which are the go and not-go gauges. The principle involved in the inspection of the components with the gauges is illustrated in Fig. 1.21. It can be seen that the inspection is based on a very simple principle, where gauges dimensions are the same as the limits of size with one gauge testing the maximum material limit and another one testing the minimum material limit. From here the names for the gauges originate, which are the **go** and **not-go**. Otherwise the limit gauges check whether the requirements for dimensional accuracy are fulfilled:

$$d_{\min} \le d_{actual} \le d_{\max}$$
$$D_{\min} \le D_{actual} \le D_{\max} \qquad (1.12)$$

Assuming the component is satisfactory, it will pass the go gauge and will not pass the not-go gauge. Otherwise the faulty component will either pass the not-go gauge or hang on the go gauge or both. Thus it can be noted that the limit gauges check whether or not the dimension of a feature is within the limits of size instead of measuring the actual dimension value.

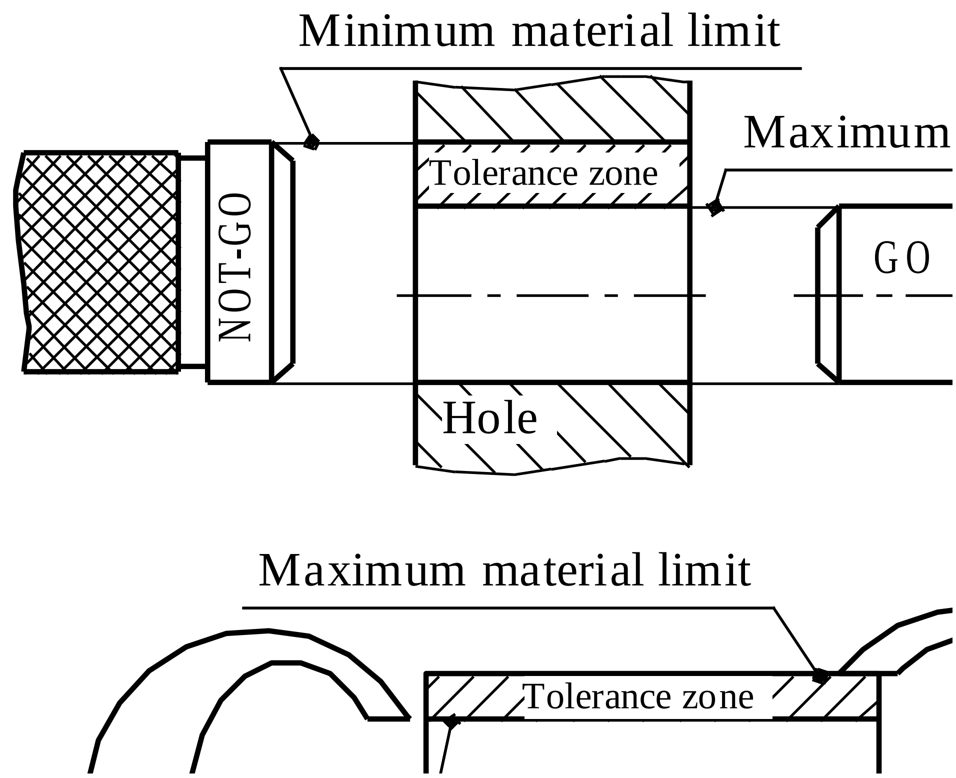


Fig. 1.21 Gauging principle

In fact the not-go gauge controls only the dimension of the component and the go gauge controls all the possible geometric deviations, such as dimension, form, location deviations, etc, that is actually consistent with the Taylor's principle. And as it can be seen from Fig. 1.21 this principle implies that the not-go gauge should be shorter compared to the component length and the go gauge should be almost the same length as the component in order to check the deviations in complex (refer to Fig. 1.22).

In practice the limit gauges are applied in inspection of components made within IT6-IT17 accuracy range. As limit gauges accuracy should be a few times higher than the accuracy of the component inspected, it is almost impossible to apply gauges for 01-5 IT grades dimensions, since the price and wear rate of a gauge will raise significantly.

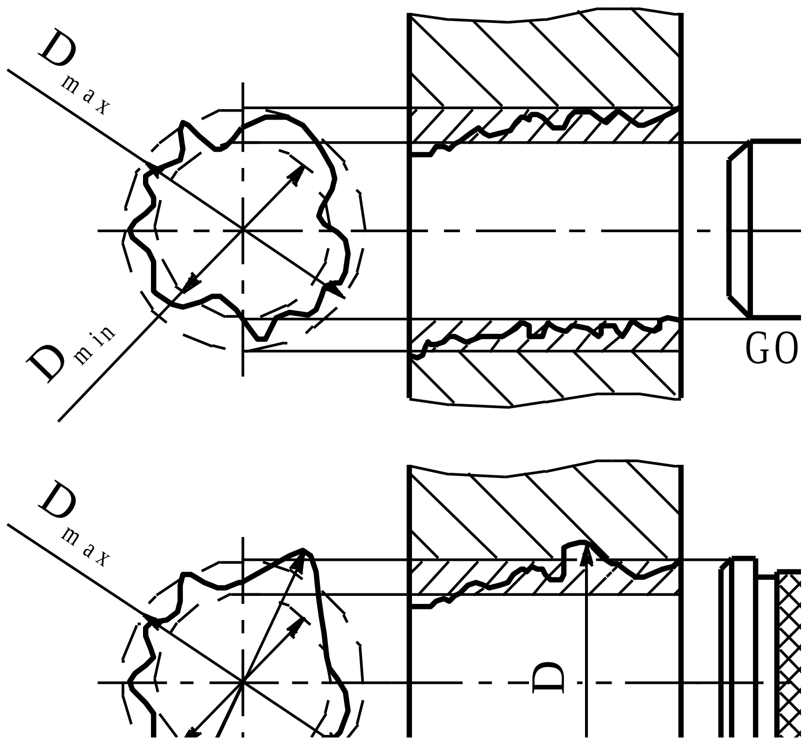


Fig. 1.22 Gauging

It is common practice to classify limit gauges as follows:

- On the basis of appearance:
  - Plug gauges
  - Snap or ring gauges
- On the basis of purpose:
  - Workshop gauges
  - Reference gauges
- On the basis of application:
  - Plain cylindrical gauges
  - Tapered gauges
  - Thread gauges
  - Special types of gauges
- On the basis of construction:
  - Single sided gauges (Fig. 1.23 a)
  - Double sided gauges (Fig. 1.23 b)
  - Solid gauges (Fig. 1.23 a and b)
  - Adjustable gauges (Fig. 1.23 c)

One of the most common limits gauges is the snap gauge (refer to Fig. 1.23) with the solid and adjustable designs available. The snap gauge is used to check outside diameters, thicknesses and lengths. Although the adjustable type is more universal, since it can be adjusted for other dimensions with the help of adjustable anvils, the accuracy of this gauge is slightly reduced thus limiting its use to dimensions of the IT8-IT17 grades.

Plain plug gauges are used for checking inner diameters and slots. The plug gauges of the single sided type, double-sided type, cylindrical form etc. are available.

Tapered gauges have two strips on the surface, one is **go** and another one is **not-go**, to check the size of tapered element by determining the distance to which the element travels along the gauge. The amount of taper is controlled with the help of paint or chalk marks applied on the gauge surface; in case the marks are equally disturbed after the slight slippage then the taper amount is correct.

Thread limit gauges are used for inspection of the threaded components. In this case the complex method of control is applied, which assume the checking of all the elements of the thread profile simultaneously, in other words the screwing capability of a thread is checked.

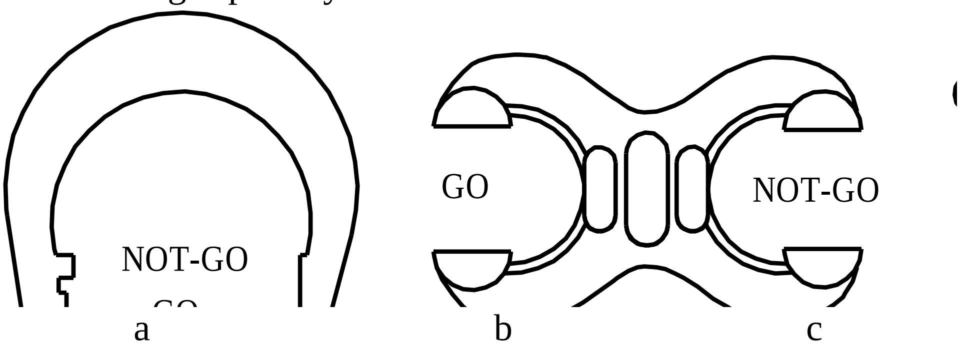


Fig. 1.23 Some of the limit gauges: (a) solid single sided gauge; (b)solid double sided gauge; (c) adjustable gauge

The principle involved in inspection of the components with limit gauges was illustrated earlier in Fig. 1.21. As one can see from the figure the dimension limits of size are the references for establishing the deviations of the gauge dimensions. A diagram describing the tolerancing principles for the plain plug gauge (a hole is being controlled) is shown in Fig. 1.24. Here, the gauge tolerance is denoted by the letter **H** for plug gauges, by $\mathbf{H_1}$ for snap gauges and $\mathbf{H_P}$ for reference gauges. The position of the not-go gauge tolerance is symmetrical to the line lying at the distance α from the hole minimum material limit. The position of the go gauge tolerance is symmetrical to the line lying at the distance **Z** from the hole maximum material limit. In order to prolong the go gauge life a certain amount of its wear is permitted. The wear loss is limited by the **wear limit** positioned at the distance **Y-**α from the maximum material limit of a hole.

Note: The α value is equal to 0 for the basic size smaller than 180 mm. The **Y** value is equal to 0 for the 9-17 IT grades. The maximum material limit is called **go limit**, and the minimum material limit is called **not-go limit**.

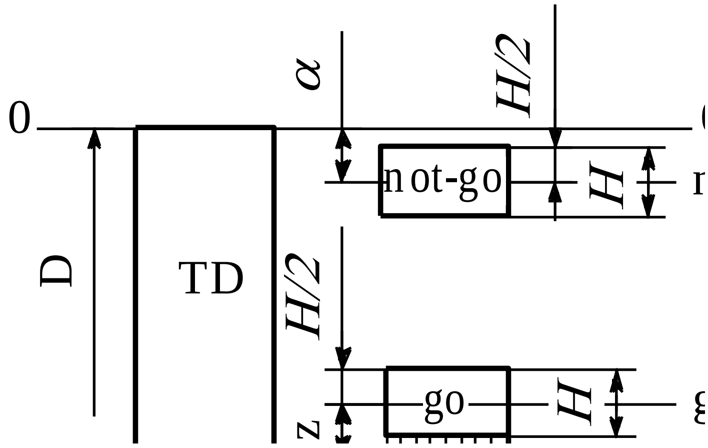


Fig. 1.24 Limit plug gauge tolerances

Plain snap gauge tolerancing follows the same principles as plain plug gauge. A diagram describing the tolerancing principles for plain snap gauge is shown in Fig. 1.25.

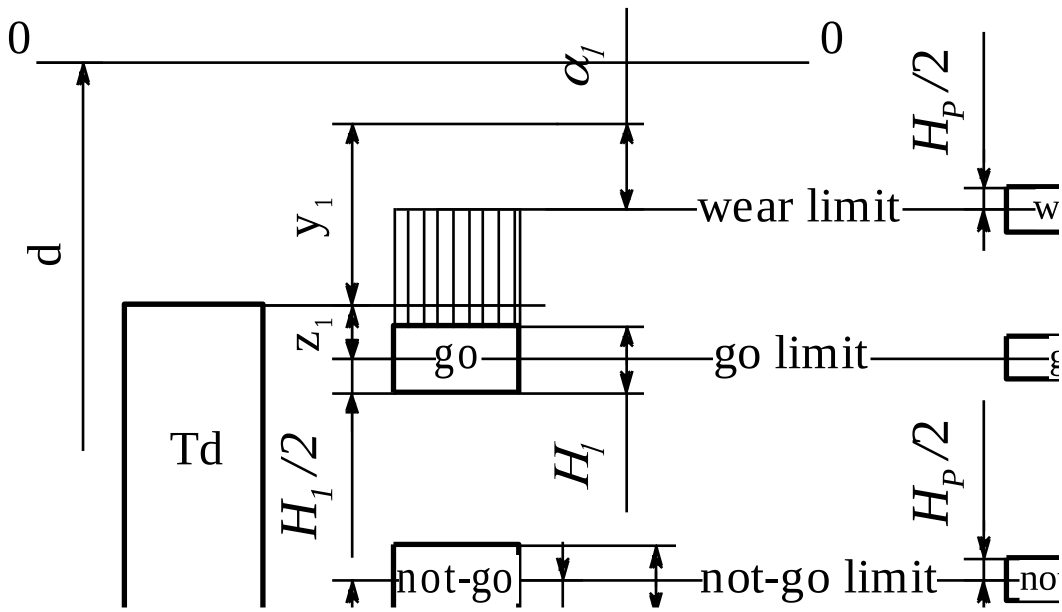


Fig. 1.25 Limit snap gauge tolerances

1.8 Screw Thread Tolerancing

A screw thread is a helical or spiral structure of a constant section on the surface of a cylinder or a cone, either external or internal. A threaded bolt is an example of the external thread and nut is an example of internal thread. Threads may be right-handed or left-handed, with the former being more common. A right-hand thread item advances away from the viewer when turned clockwise; and a left-hand thread item advances away from the viewer when turned counterclockwise.

Screw threads are widely used in engineering and can be classified as follows:

- On the basis of hand:
  - Left-handed
  - Right-handed
- On the basis of purpose:
  - Fastening
  - Translation
  - Special
- On the basis of the number of starts:
  - Single-start thread
  - Multi-start thread
- On the basis of the location:
  - External
  - Internal
- On the basis of the surface form:
  - Straight
  - Tapered
- On the basis of thread profile:
  - Triangular
  - Round
  - Square
  - Trapezoid
  - Special form

To discuss the principles of thread tolerancing a general-purpose metric screw thread is used.

The thread basic profile, from which all deviations are established, is the same for both the external and internal thread. The metric **thread profile** is based on an isosceles triangle with the thread angle equal to 60° (refer to Fig. 1.26). The **thread angle** is measured between adjacent flanks in the axial plane and is designated as α. Along with the thread angle for the metric threads the **half angle of thread** is also used. It is described as an angle between the flank line and a line perpendicular to the thread axis, designated as $\boldsymbol{\alpha_1}$. For threads with asymmetric thread profile a **flank angle** is also used, which is the angle between a flank line and a line perpendicular to the thread axis, designated as $\boldsymbol{\beta}$ or $\boldsymbol{\gamma}$. The theoretical triangle is usually truncated in order to have clearance between roots and crests of the mating threaded components and thus ensure tighter contact between sloped flanks. The height of the truncated triangle is called **thread overlap** and designated as H_1. The **height of basic triangle**, designated as H, relates to the pitch value P in the following way:

$$H = P \cdot \cos 30^\circ \approx 0{,}8660254 \cdot P \tag{1.13}$$

The relationship between thread overlap and height of the basic triangle is defined as:

$$H_1 = \frac{5}{8} \cdot H \tag{1.14}$$

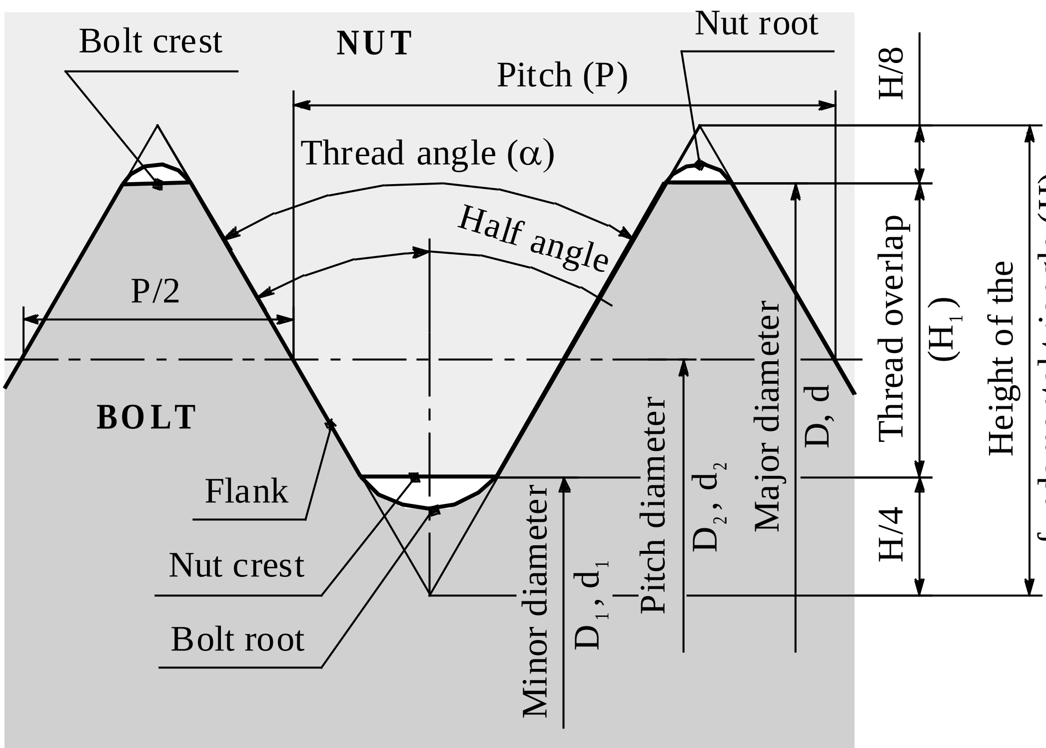


Fig. 1.26 Basic parameters of thread

Pitch is the distance between two corresponding flanks (left or right) points measured in the same line parallel to the thread axis in the axial plane and on the same side of the thread axis. The thread pitch is often confused with the lead, since for the most common single-start threads the pitch and lead are the same. For coarse threads the pitch value is determined from the following equation:

$$d(D) = 6 \cdot P^{1,3} \tag{1.15}$$

For a fine thread several different pitches correspond to the given major diameter.

Lead is the distance a threaded component advances axially, relatively to the mating component, in one complete turn, it is designated as P_n. The lead for a two-start thread is twice the pitch, for a three-start thread is three times the pitch, etc (Fig. 1.27).

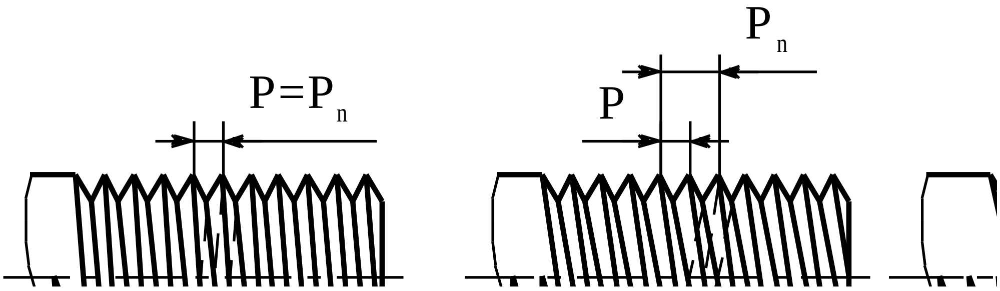


Fig. 1.27 Relation of pitch and lead for a thread

For a straight thread the **major diameter** is the diameter of a virtual cylinder, coaxial with the thread, which circumcircle crests of an external thread or roots of an internal thread. For a straight thread, **the minor diameter** is the diameter of a virtual cylinder, coaxial with the thread, which circumcircle roots of an external thread or crests of an internal thread. For a straight thread, the **pitch diameter** is the diameter of a virtual cylinder, coaxial with the thread, the surface of which would pass through the thread profiles so as to make the width of the groove equal to the widths of the thread and equal to one-half of the basic pitch.

The terms major and minor diameters are often confused because they are often associated with such terms as outside and inside diameters, which are opposite for male and female threads. In order not to confuse these two terms it is better to use major diameter and minor diameter terms.

Lead angle is the angle formed by a helix at the thread axis with a line perpendicular to the axis, designated as ψ.

The **length of thread engagement** is the length of the mutual contact of two coupling threads, measured parallel to the thread axis.

It should be noted that screw threads were the first objects for standardisation, as threaded components account for more than 60% of all manufactured products. In order to meet the requirements for accuracy of threaded components Russian State Standard specifies the following:

- For external thread:
  - limits of size and tolerances for the major and pitch diameters (Td, Td_2);

- upper limit of size for the minor diameter (es for d_1).
- For internal thread:
  - limits of size and tolerances for the minor and pitch diameters (TD_1, TD_2);
  - lower limit of size for the major diameter (EI for D).

It should be noted that screw threads are difficult to tolerance because of the complex geometric pattern of their form. The limits of size are established from the basic profile of a thread, which is the same for both the external and internal thread, in the direction perpendicular to the thread axis. The basic profile is a kind of a basic line used for the smooth components, but being of a complex form rather than just a straight line. Since the thread form can take different shapes (refer to thread classification) the basic profile and thus the total tolerance zone will follow the corresponding shape. By **total tolerance zone** a block of tolerance zones for separate thread elements is meant. This total tolerance zone, built on the basis of diametral tolerances, also specifies the deviations of other elements of a thread, such as a pitch and thread angle, which are actually not toleranced. The tolerance zones for external and internal threads are shown in Fig. 1.28 and Fig. 1.29 respectively. Fig. 1.30 illustrates the thread clearance fit.

Applying tolerance zones for threaded components follows the same principles as for the smooth components. The tolerance zone is specified by the tolerance grade and its position with respect to the basic line. As it can be seen from Fig. 1.28 and 1.29 the values for the deviations and tolerances are shown halved, this is due to the fact that these values are applied to a diameter, but the thread diagram usually shows only a half-thread.

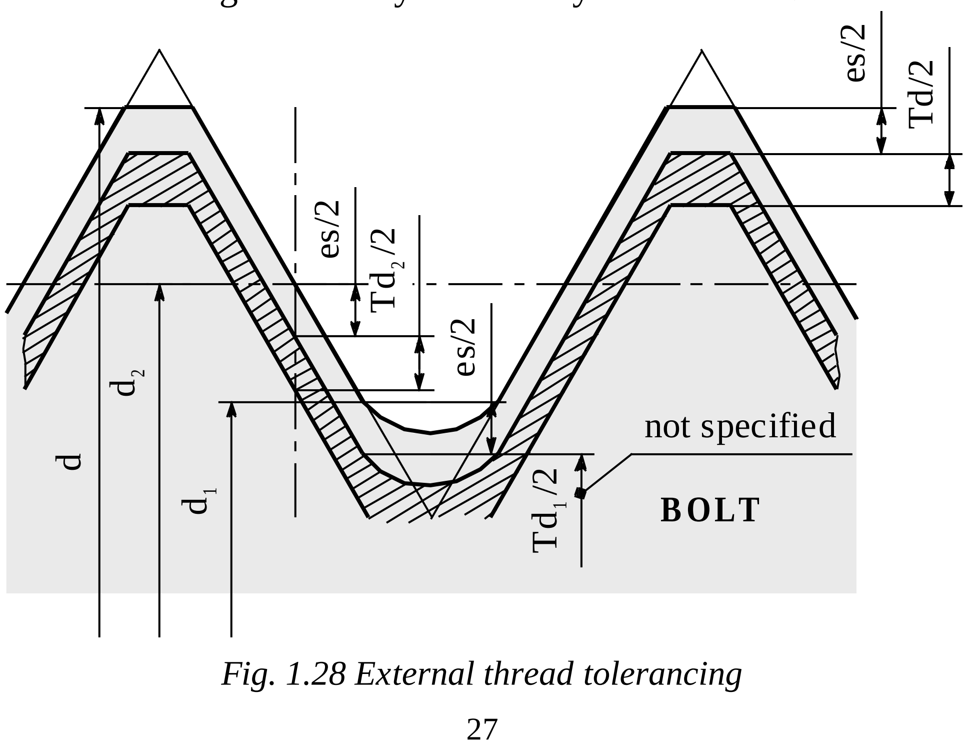


Fig. 1.28 External thread tolerancing

Tolerance values for crest diameters (the minor diameter of an internal thread and the major diameter of an external thread) relate to the accuracy grade and pitch value. Unlike the tolerances for the major and minor diameters the pitch diameter tolerance is a cumulative tolerance. The pitch diameter tolerance specifies not only the tolerance for the pitch diameter, but the deviations for a pitch and thread angle as well. From here the term **virtual pitch diameter** arises, which is a sum for a bolt or difference for a nut of the actual pitch diameter and diametric compensations for the pitch and flank angle errors. The calculated virtual pitch diameter helps to determine whether or not the two given threaded components couple.

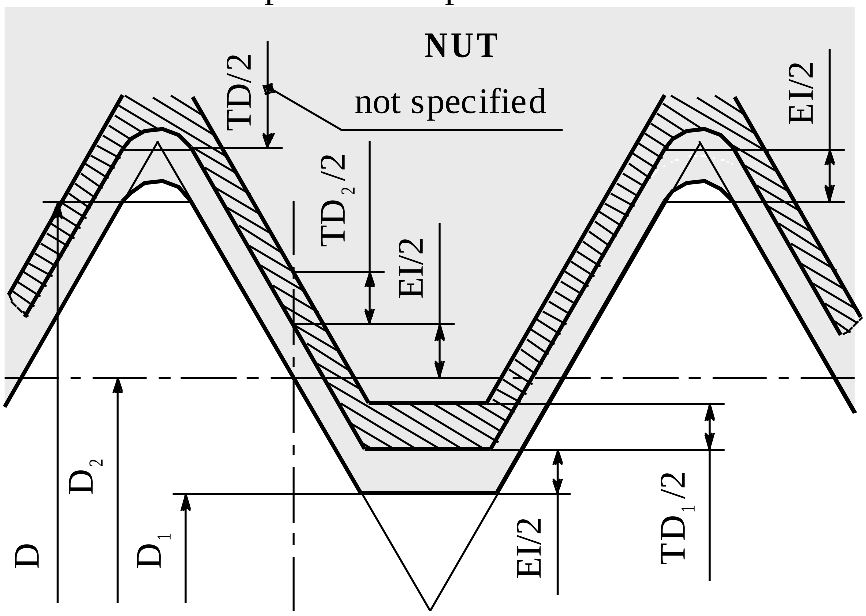


Fig. 1.29 Internal thread tolerancing

Although the same terms, like fit, clearance and interference, as for smooth components, are applied to the threaded components, the primary fit is the pitch diameter fit (mutual contact of two coupling threads by their flanks). Major and minor diameters fits always provide clearance even for interference and transition fits due to the truncation of the theoretical triangle.

As it was stated above in order to determine whether or not the threaded components couple it is needed to measure the actual values of the thread diameters, pitch and thread angle and, consequently, calculate the virtual pitch diameter. For all straight threads with straight-line flanks it is possible to compensate the pitch and thread angle errors by changing the virtual pitch diameter.

Pitch error is the difference between the actual pitch and nominal pitch, designated as ΔP. When the actual pitch is bigger than the nominal pitch it is positive value and when the actual pitch is smaller than the nominal pitch it is negative value. The pitch error includes local pitch error, progressive pitch error (relates to the length of thread engagement) and periodical pitch error (increases and decreases periodically).

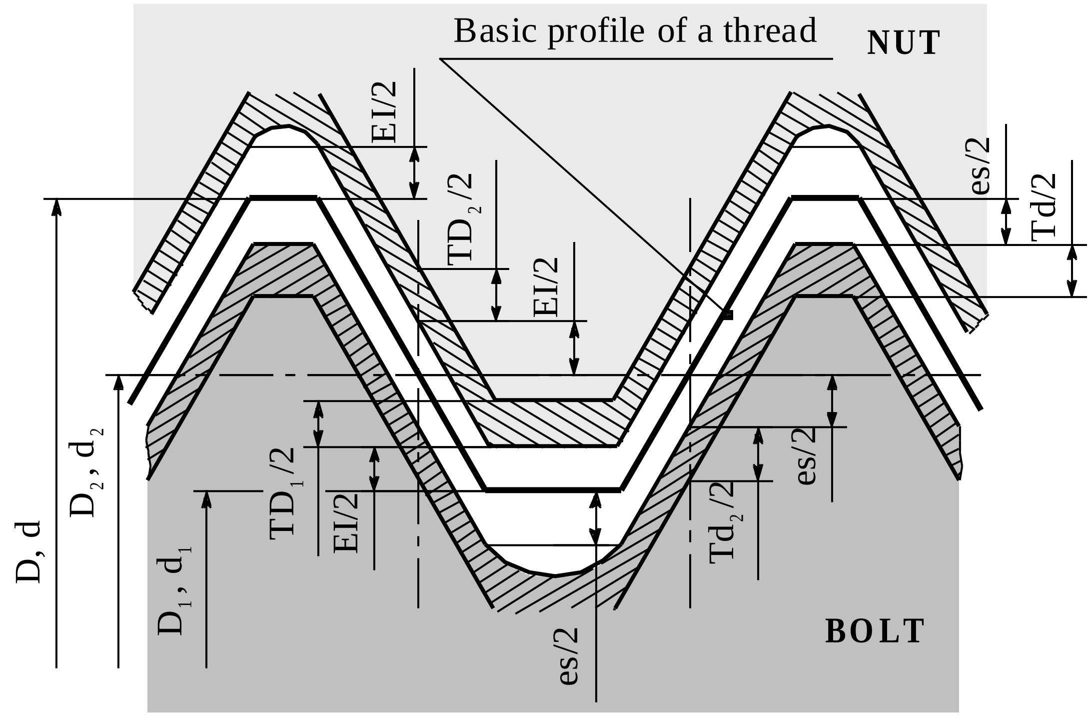


Fig. 1.30 Thread fit

Fig. 1.31 illustrates the case, where the external thread has a positive pitch error and, thus doesn't couple with the internal one. In order to correct the coupling it is necessary to reduce the pitch diameter of the external thread by a value that is called **pitch error diametric compensation** and designated as f_P. The f_P is measured in microns and defined as:

$$0{,}5 \cdot f_P = 0{,}5 \cdot |\Delta P_n| \cdot ctg(\alpha/2) \text{ or } f_P = |\Delta P_n| \cdot ctg(\alpha/2) \qquad (1.16)$$

For assessing the thread angle error a flank angle is usually used, since it allows determining not only the angle error, but the thread profile skew as well. **Half angle error** is the difference between the actual half angle of thread and nominal half angle of the thread, designated as $\alpha/2$. The half angle error for symmetric thread profile is defined as arithmetical mean of absolute values for half angle deviations of both left and right flanks:

$$\Delta\alpha/2 = 0{,}5 \cdot \left|\Delta(\alpha/2)_{right}\right| + \left|\Delta(\alpha/2)_{left}\right| \qquad (1.17)$$

Fig. 1.32 shows the case where the external thread has a half angle error and thus doesn't couple with the internal one. In order to correct the coupling it is necessary to reduce the pitch diameter of the external thread by a value that is called half **angle error diametric compensation** and designated as f_α. f_α is measured in microns and defined as:

$$f_\alpha \approx \frac{0{,}582 \cdot H_1}{\sin\alpha} \cdot \Delta\alpha / 2 \tag{1.18}$$

Taking into account Eq. 1.16 and 1.18 the virtual pitch diameter can be calculated as:

$$d_2^{virtual} = d_2^{actual} + f_P + f_\alpha$$
$$D_2^{virtual} = D_2^{actual} - f_P - f_\alpha \tag{1.19}$$

For the virtual pitch diameter the following definition can also be used. A v**irtual pitch diameter** is a diameter of a virtual thread without pitch and angle errors, that screw in the given manufactured thread without interference and clearance over the given thread engagement.

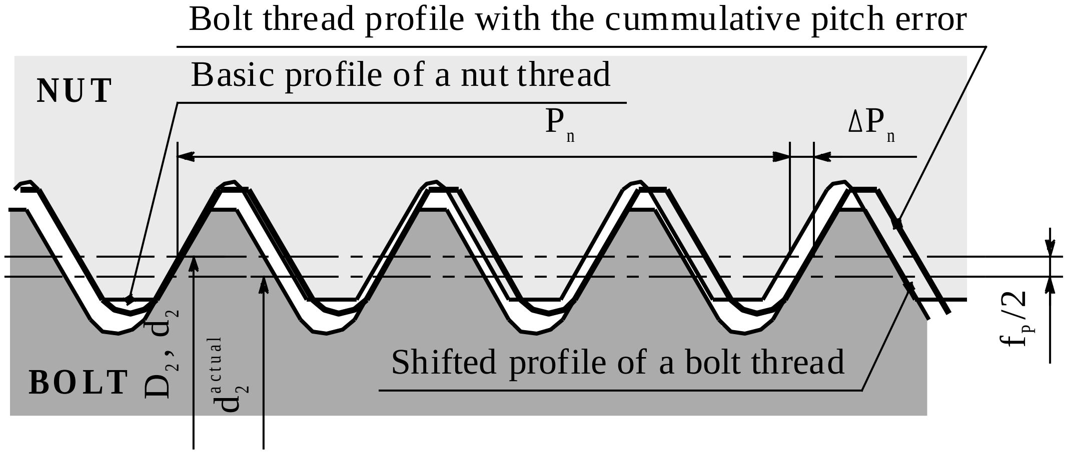


Fig. 1.31 Illustration of the pitch error

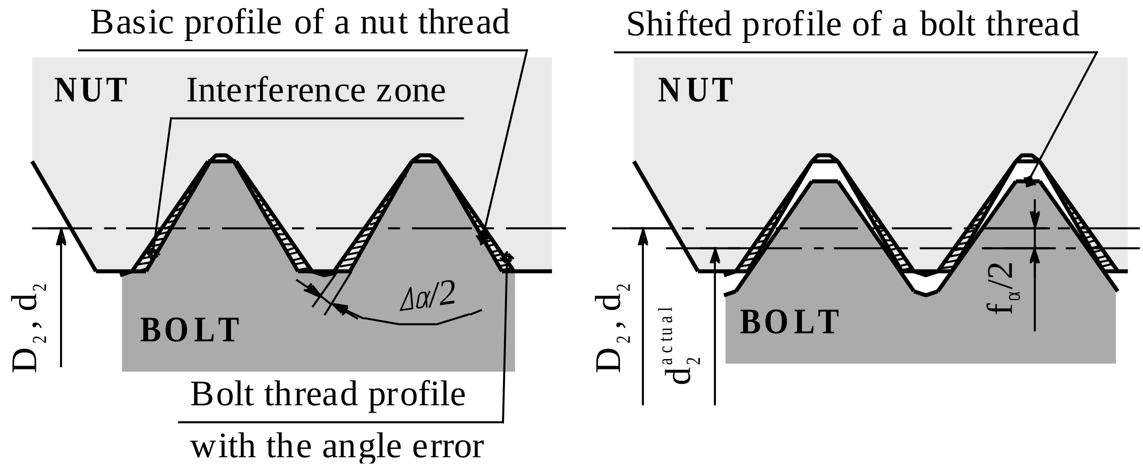


Fig. 1.32 Illustration of the flank angle error

The conventional diagram for the pitch diameter tolerance zone is shown in Fig. 1.33. This diagram illustrates the case when the external thread has a pitch diameter deviation as well as pitch and angle errors. Though the actual pitch diameter is within the tolerance zone, the virtual pitch diameter exceeds the upper limit of size so that the given external thread will not screw in the nut unless the virtual pitch diameter for a nut is bigger than the virtual pitch diameter for a bolt.

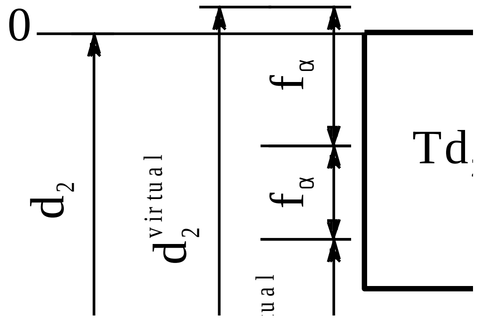


Fig. 1.33 Virtual diameter composition

The system of tolerances and fits implements a fewer number of fundamental deviations and tolerance grades for the threads. The thread deviation value is related to the pitch value and differs in its nature from the fundamental deviations for the smooth components.

The tolerance zone is specified by the tolerance grade number followed by the fundamental deviation letter, which is the opposite order as compared to the smooth components tolerance zone designation. Examples: 6h, 6H, 6G, etc. Since the tolerancing of a thread includes two diameters, the designation of the thread tolerance zone will consist of two tolerance zones. First comes the pitch diameter tolerance zone and then follows the tolerance for the crest diameter (minor diameter of an internal thread and the major diameter of an external thread). The latter is omitted in case these two tolerance zones are the same. There are five fundamental deviations for external threads, which are: d, e, f, g, h; and four fundamental deviations for internal threads, which are: E, F, G, H. For the external thread tolerance grades, there are three grades for the major diameter, which are 4, 6, 8; and eight grades for the pitch diameter: 3, 4, 5, 6, 7, 8, 9 and 10. For the internal thread there are five minor diameter grades, which are: 4, 5, 6, 7, 8; and six pitch diameter grades, which are: 4, 5, 6, 7, 8 and 9. The lower is the number, the smaller is the tolerance.

All the limit deviations and tolerances are specified to the length of a thread engagement, which is divided into the short (S), normal (N) and long (L). The shorter is the engagement length, the tighter is the tolerance.

1.9 Tolerancing of Cones and Wedges

The system of angle tolerances is applied to the components with the smaller side of an angle up to 2500 mm. Fig. 1.34 provides two examples of components with the sides of an angle designated as L_1 and L_2, where the angle tolerance should be related to the shorter side, which is L_1. A cone (refer to Fig. 1.35) is described by the diameter of long base (D), the diameter of a short base (d), cone length (*L*), cone angle (α) and slope angle ($\alpha/2$). These parameters are interrelated by the following equation:

$$(D-d)/L = 2tg(\alpha/2) = C \tag{1.20}$$

where *C* is a taper (conicity) and $C/2=i$ is inclination (slope).

Russian Standard implements 17 tolerance grades, designated as AT1, AT2… AT17, in descending order of accuracy. It is allowed by the standard to create extra grades of higher accuracy by sequential dividing AT1 value by a factor of 1.6. By the **tolerance angle** a difference between the limit maximum angle α_{max} and limit minimum angle α_{min} is meant.

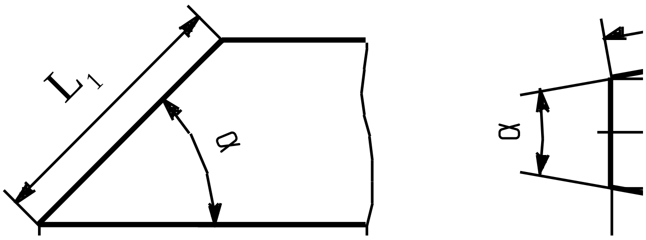


Fig. 1.34 Angle parameters

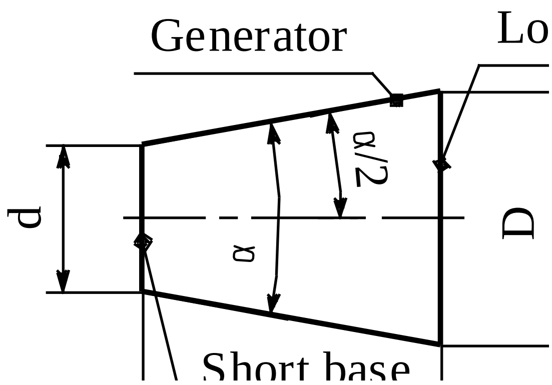


Fig. 1.35 Cone parameters

The following four types of tolerance are implemented by Standard: AT_α, AT_α', AT_h and AT_D (Fig. 1.36). **AT_α** is an angle tolerance expressed in angle units either radians or degrees. **AT_α'** is a rounded value (expressed in standardised values of degrees, minutes and seconds) of the angle tolerance

AT_α; it is used on drawings. **AT_h**, which is expressed in microns, is a tolerance expressed by a section of a line perpendicular to the smaller side of an angle on distance L1 from the angle corner. Angle tolerance AT_D, expressed in microns, is the tolerance applied to the difference of two diameters measured in two perpendicular to the cone axis sections on the specified distance L (refer to Fig. 1.36 b). The relationship between linear and angular types of the angle tolerances is expressed as:

$$AT_h = AT_\alpha \cdot L_1 \cdot 10^{-3} \tag{1.21}$$

where AT_h is expressed in μm, AT_α – in μrad, L_1 – in mm.

The angle tolerances are applied with respect to a cone length (L) for taper values smaller than 1:3; and with respect to a cone generator length (L_1) for taper values bigger than 1:3. In the first case the $AT_D \approx AT_h$ and in the latter case the AT_D value is calculated as:

$$AT_D = \frac{AT_h}{\cos(\alpha/2)} \tag{1.22}$$

where α is a basic cone angle.

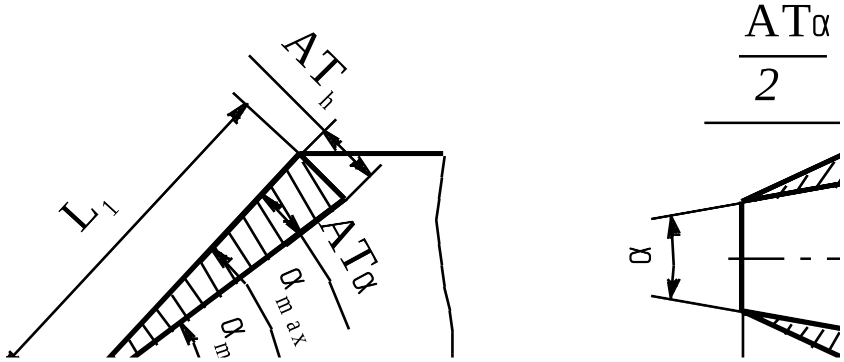


Fig. 1.36 Angle tolerances

The angle tolerances can be positioned as shown in Fig. 1.37 and Fig. 1.38, which illustrate tolerances for wedge angles and cone angles respectively. Other tolerance positions with respect to the basic angle are also allowed.

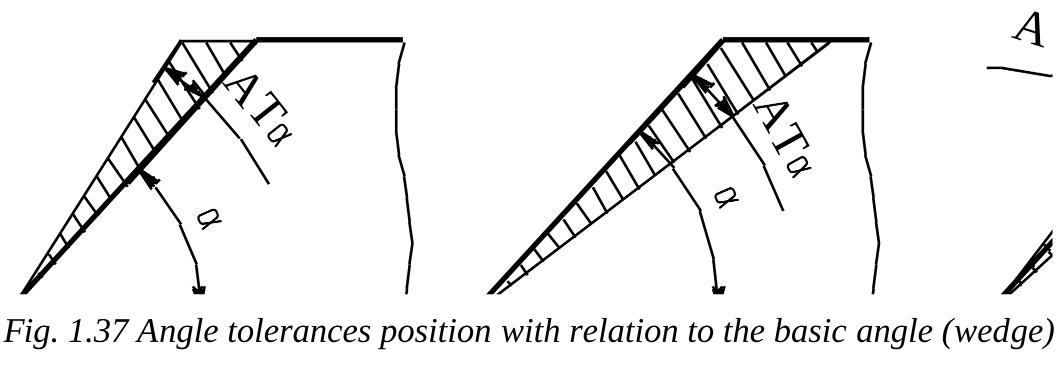


Fig. 1.37 Angle tolerances position with relation to the basic angle (wedge)

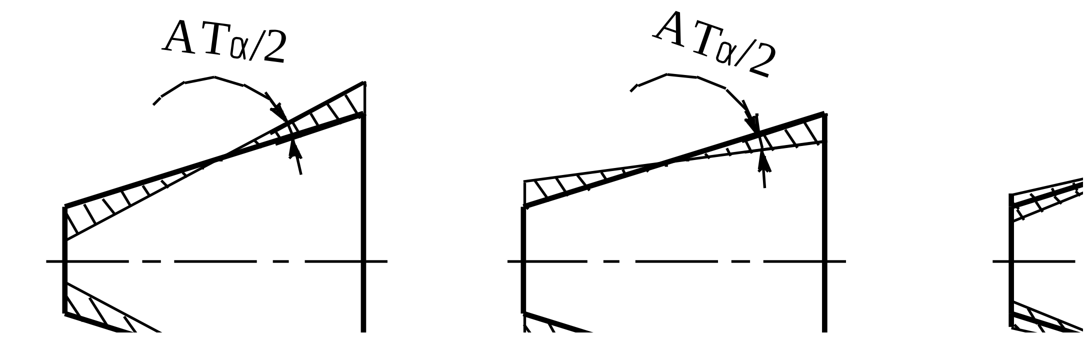


Fig. 1.38 Angle tolerances position with relation to the basic angle (cone)

Similar types of fit as for plain components are applicable to cone couplings, which are clearance, interference and transition fits. The specified fit is defined by the mutual axial position of the coupled components, and can be defined: a) by aligning base planes of conical components; b) by defining an axial displacement of the cones base planes; c) by defining an axial displacement of the cones from their initial positions; d) by specifying the pressing force value (refer to Fig. 1.39).

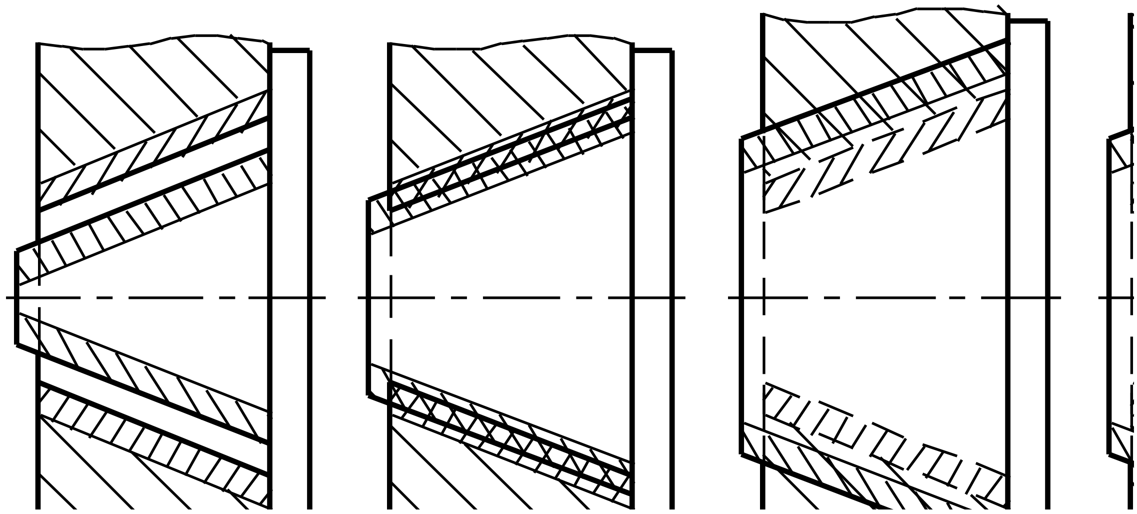


Fig. 1.39 Axial arrangements of mating cones

The cone tolerancing system provides the following four types of tolerance:

- **Tolerance of a cone diameter**, designated as T_D. This tolerance is constant in any section of a cone and depends on a diameter value (Fig. 1.40 a). The cone diameter tolerance covers the whole cone length (L).

– **Cone section diameter tolerance** is applied only to a defined section of a cone and is designated as T_{DS}.
– **Cone angle tolerance**, designated as AT, lies within the cone diameter tolerance zone unless other requirements are specified and defines the limit cone angles α_{min} and α_{max}, (Fig. 1.40 b).
– **Cone form tolerance** embodies straightness (T_{FR}) and roundness (T_{FL}) tolerances of the cone generator. If these tolerances are to be smaller than half of the cone diameter tolerance, then the special indication should be included in a drawing.

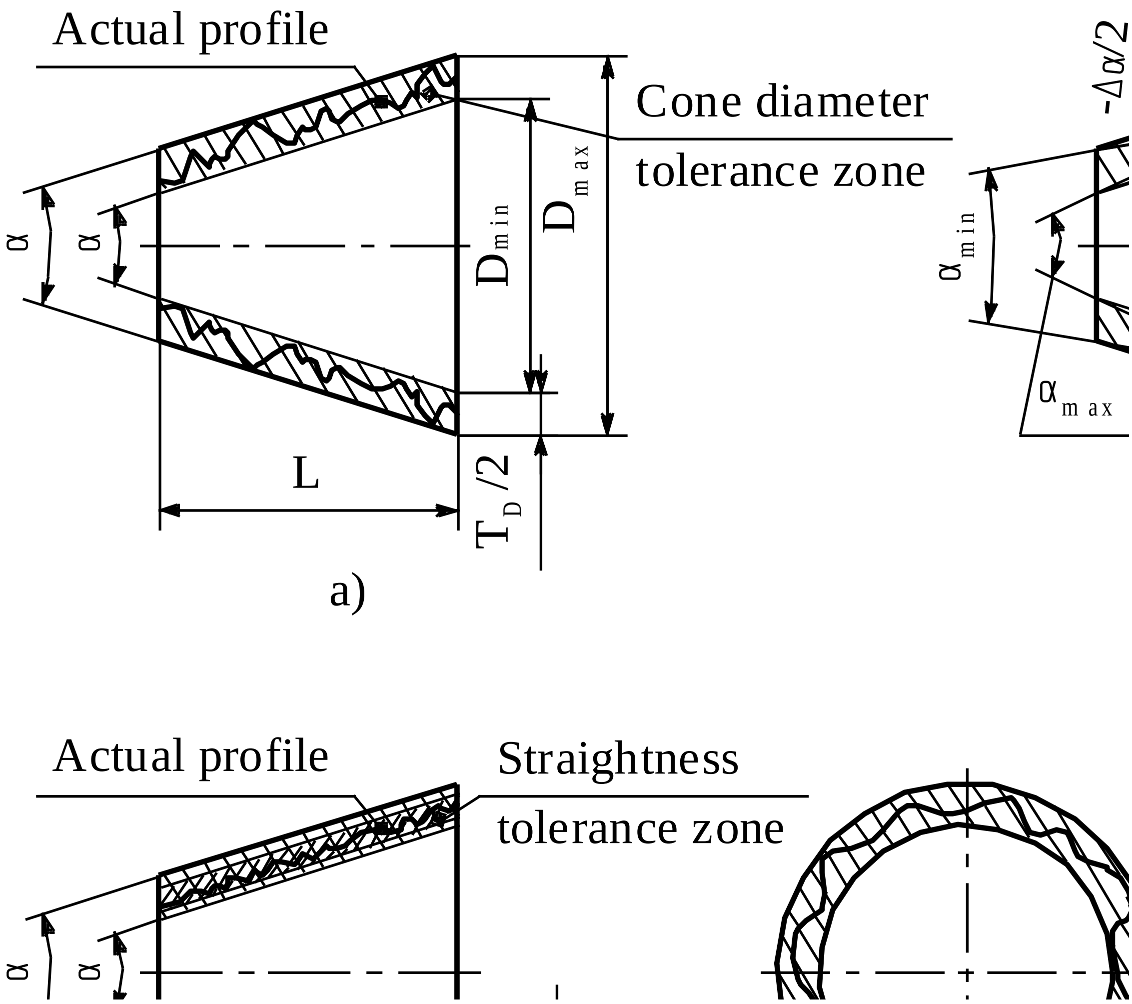


Fig. 1.40 Types of cone tolerances

In the majority of practical cases only the cone diameter tolerance (T_D) is used. The cone angle tolerance and cone form tolerance are included in the diameter tolerance and such deviations can occupy the whole tolerance zone. If stronger requirements are applied to the angle and cone form tolerances, then these can be reduced within the cone tolerance zone.

1.10 Indications on Drawings

This section contains information on indications of the toleranced dimensions mentioned in sections above. The designation of a dimension tolerance comprises the basic size, followed by the letter of a fundamental deviation and a number of the tolerance grade. Symbols indicating the nature of a feature of size, such as diameter, radius, sphere, square, etc., are placed in front of a dimension value if needed. A fit is described by the combination of a hole and shaft designations separated by a slash, with the hole designation given in the first place. Examples of designations for shafts and holes are shown in Fig. 1.41 and fit designation is given in Fig. 1.42.

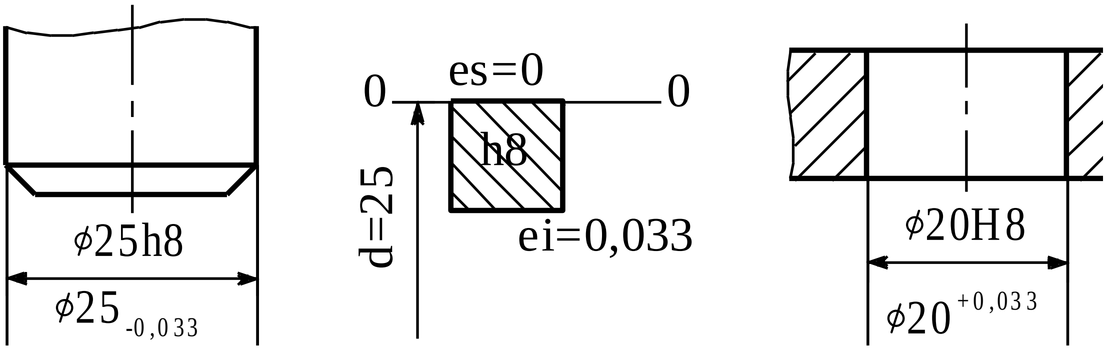


Fig. 1.41 Shaft dimensions and tolerances

The designation of dimensions or fits can be delivered in three forms. It can be expressed in a numerical form, in letters and as a combination of both. Literal form is used on assembly drawings, since it is more compact and still informative. Numerical form is used for working drawings, as the machine tool operator will take tolerance values directly from the designation rather than looking for it in the handbooks.

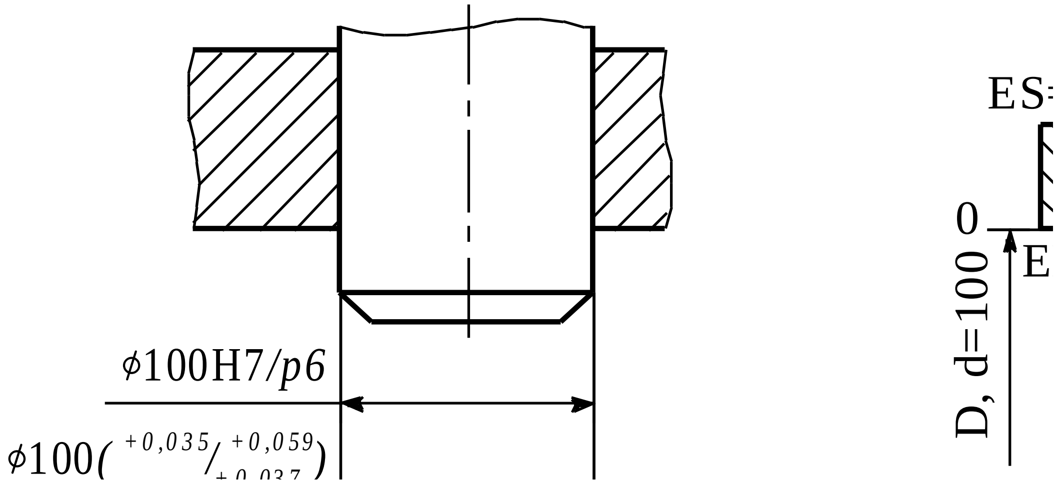


Fig. 1.42 Hole dimensions and tolerances

Fig. 1.43 shows common fits used for rolling bearings. The designation in this case follows the same principles as for the smooth component. The basic size is followed by the fundamental deviation, which is designated by the upper-case letter *L* for the bore and lower-case letter *l* for the outer ring with the tolerance grade number (Fig. 1.43 a). Russian Standard allows not specifying the tolerance zone for a bearing in the fit designation, since the rolling bearings are standardised items and are never remachined. Thus the fit designation includes tolerance zones for the housing and shaft only (as shown in Fig. 1.43 b).

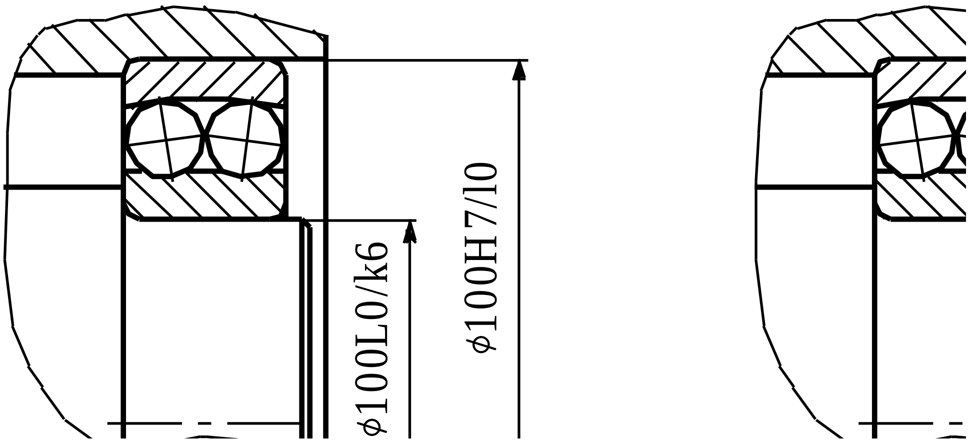


Fig. 1.43 Rolling bearing fits

The designation for the metric threads is rather complex; it begins with the capital letter M, Tr, S, etc., which defines the type of the metric thread whether it is metric, pipe, buttress or some other. Then the nominal size in millimeters is given. For fine threads the nominal size is followed by the "×" symbol, with the pitch value given in millimeters. If the thread is multi-started, then the symbol "×" will be followed by the lead in millimeters with pitch value given in brackets. In case of a left-hand thread the symbol "LH" should follow next. Right-hand threads are not designated by any symbol. Then the tolerance zone designation follows, separated by a dash. The tolerance zone of a pitch diameter comes first and then the crest diameter tolerance zone follows. If the length of a thread engagement differs from the nominal value it should be added to the designation on the last place. An example of external thread designation is shown in Fig. 1.44. This designation is the most complex form of indication and is rarely used in real cases, since the combination of these parameters is not so common.

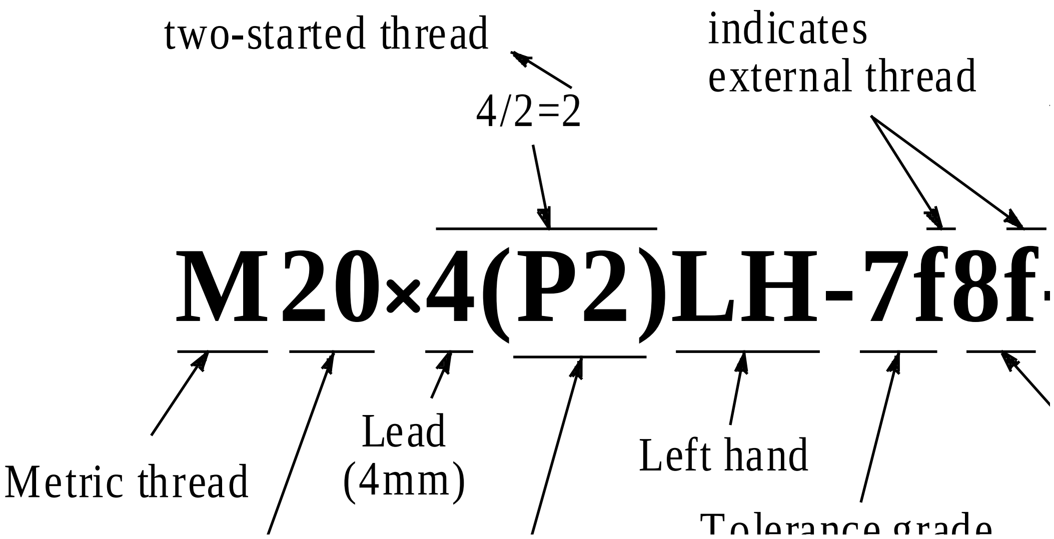


Fig. 1.44 External thread designation

Fig. 1 45 provides a simpler indication of a thread, an internal one in this case, though it is much shorter it is still informative.

The designation and description for a thread clearance fit is illustrated in Fig. 1.46.

M68-7

Fig. 1.45 Internal thread designation

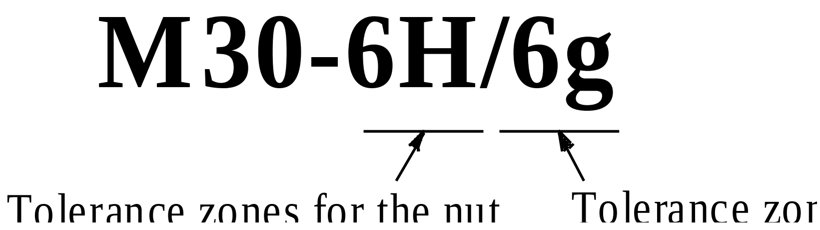


Fig. 1.46 Designation of the thread clearance fit

2. Principles of Geometrical Tolerancing

As manufacturing processes introduce some inaccuracies into the component geometric features, tolerances are applied not only to dimensions but to other geometrical features as well.

Any component comprises a set of superposed geometrical deviations, also called errors (Fig. 2.1), which can be defined as follows:

– 0^{th} order: size deviations (designated as Δd in the figure).
– 1^{st} order: location deviations (designated as *e* in the figure).
– 2^{nd} order: form deviations (designated as Δ_F in the figure).
– 3^{rd} order: waviness (designated as W in the figure).
– 4^{th} order: roughness (designated as Rmax in the figure).

These orders are refer to the expansion term numbers of the Fourier serie that describes harmonic components of a profile deviations.

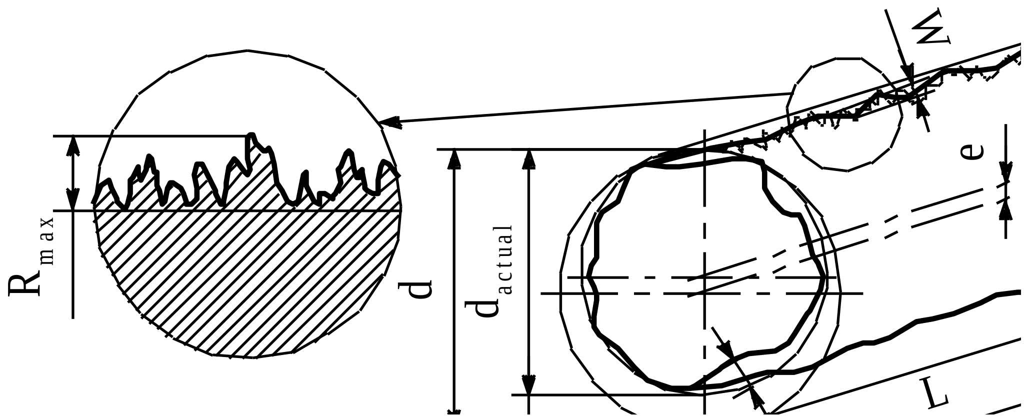


Fig. 2.1 Geometric deviations

Thus to ensure the overall accuracy of a component it is needed to specify both the macrogeometrical deviations of size, form and location, and microgeometrical deviations of waviness and roughness.

In this chapter the tolerancing of the geometric elements, such as form and location, is discussed; waviness and roughness parameters are explored in Section 3.

As the dimensional tolerances are applied with relation to the basic size, the geometrical tolerances are applied and deviations are controlled with relation to the geometrically ideal shape of a component feature. Thus an engineer usually specifies the geometrically ideal shape of a component defined maximum permissible deviations for these features.

2.1 Concept of Geometrical Tolerancing

According to functional requirements the geometric tolerances and deviations are divided into the following three groups:

- **Form deviations** define the deviation of the actual feature shape relatively to the geometrically ideal nominal shape.
- **Location deviations** define the deviation of the actual feature relatively to the geometrically ideal position.
- **Composite deviations** are the result of combination of form and location deviations.

Tolerancing and deviation indications are based on the principle of comparison of the actual feature form to its geometrically ideal counterpart or the actual position of a feature to its geometrically ideal position or reference. The geometrically ideal features are defined with the help of superimposed lines, circles, cylinders, planes and profiles. A **superimposed line** is a line that adjoins the actual profile of a feature on the outer side so that the deviation of an outermost point is a minimum (refer to Fig. 2.2 a). A **superimposed circle** is a minimum diameter circle circumscribed around the actual external surface of revolution (Fig. 2.2 b); or a maximum diameter circle inscribed in actual internal surface of revolution (Fig. 2.2 c). A **superimposed plane** is a plane that adjoins the actual surface of a feature on the outer side so that the deviation of an outermost point is a minimum (Fig. 2.3 a). A **superimposed cylinder** is a minimum diameter cylinder circumscribed around the actual external surface of revolution; or a maximum diameter cylinder inscribed in the actual internal surface of revolution (Fig. 2.3 b). Along with the superimposed features the mean features are also used. The **mean feature** is a feature of nominal shape dimensioned or positioned so that the sum of the squares of deviations of the actual feature from the mean feature is a minimum (Fig. 2.4).

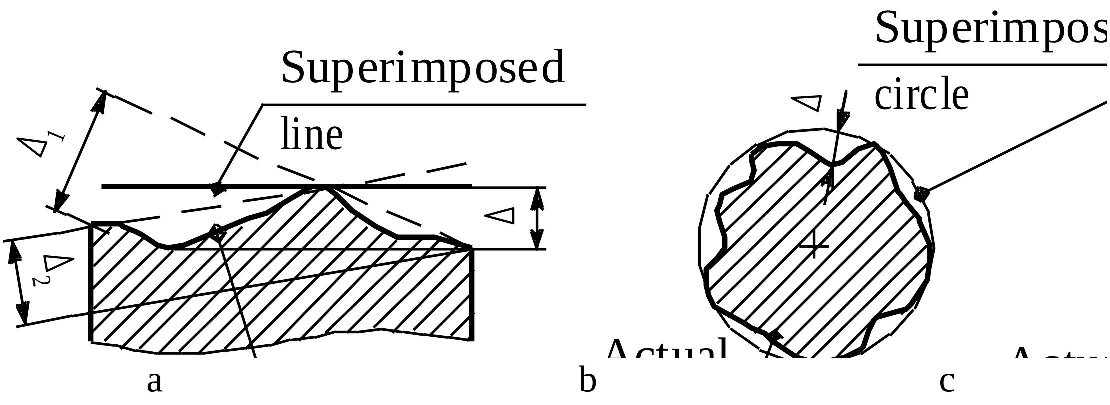


Fig. 2.2 Superimposed line and superimposed circle

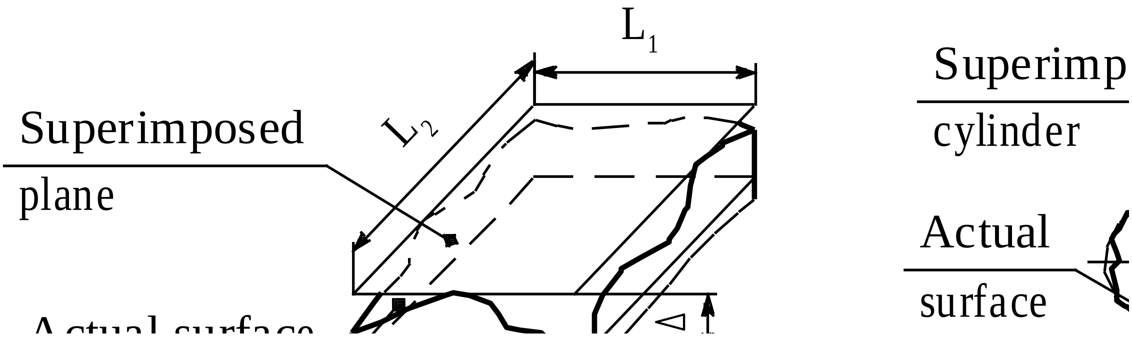


Fig. 2.3 Superimposed plane and superimposed cylinder

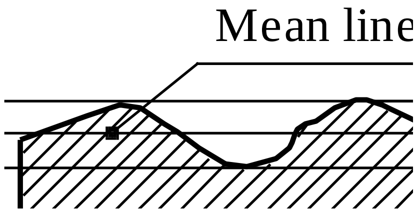


Fig. 2.4 Mean line

Russian Standard GOST 24643-81 implements 16 accuracy grades for geometrical tolerances. The tolerance values between any two adjacent grades differs by the factor of 1,6.

The following relative geometrical accuracy specifications are implemented:

A – normal relative geometrical accuracy (tolerances of form or location accounts for 60% of the dimension tolerance).

B – increased relative geometrical accuracy (tolerances of form or location accounts for 40% of the dimension tolerance).

C – high relative geometrical accuracy (tolerances of form or location accounts for 25% of the dimension tolerance).

2.2 Form Deviations

Form deviations of a feature are defined with respect to the nominal form of the same feature. A form deviation is described by the maximum distance from the actual feature points to the superimposed feature, measured at a right angle to this superimposed feature. The form tolerance defines a space in which the actual feature points should be included. The inspection is usually performed on a feature with the waviness excluded.

The following geometric characteristics are related to form deviations:

- **Straightness**
- **Flatness**
- **Roundness**
- **Cylindricity**

– **Longitudinal surface profile**
– **Form of a line**
– **Form of a surface**

Each geometrical requirement shall be met regardless of other geometrical requirements defined (principle of independency), unless otherwise specified. The control of geometrical deviations is carried out with surface plates, gauges, test arbors, interference glass, etc. as superimposed features.

2.2.1 Straightness

Deviations from straightness can be of planar or spatial type. By the deviation from **straightness in plane** we mean the maximum distance (Δ) from the actual line points to the superimposed line, measured within the specified length (Fig. 2.5 a). Among particular straightness deviations there are "convexity" and "concavity" (refer to Fig. 2.5 b and c). The tolerance zone is limited by two parallel lines positioned at a specified distance.

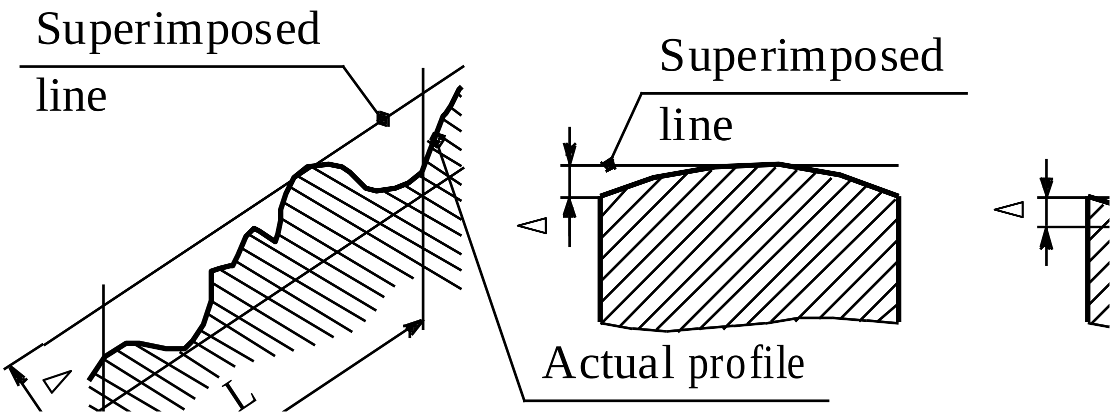


Fig. 2.5 Deviation from straightness in plane

By the deviation from **straightness in space** we mean the minimum diameter (Δ) cylinder of the specified length within which the actual line is contained (Fig. 2.6). The tolerance zone is limited by the cylinder of a specified diameter.

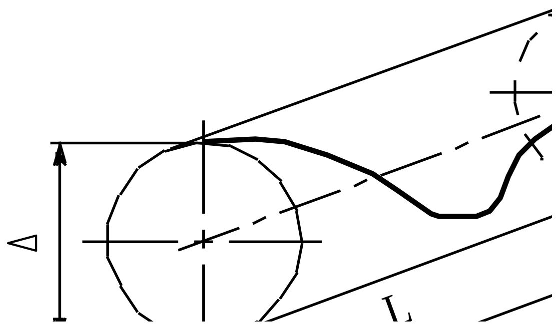


Fig. 2.6 Deviation from straightness in space

2.2.2 Flatness

By the deviation from **flatness** the maximum distance (Δ) from the actual surface points to the superimposed plane, measured within the specified area, is meant (Fig. 2.7). Among particular flatness deviations there are "convexity" and "concavity". All elements of the actual surface must lie within the tolerance zone limited by two parallel planes positioned at a specified distance.

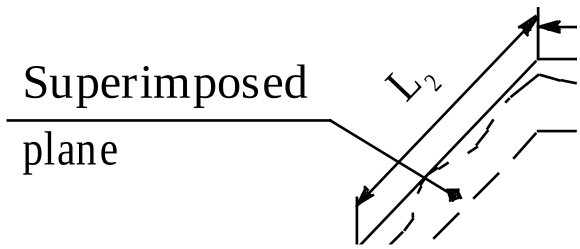


Fig. 2.7 Deviation from flatness

2.2.3 Roundness

By the deviation from **roundness** the maximum distance (Δ) from the actual profile points to the superimposed circle is meant (Fig. 2.8). Among particular flatness deviations there are "ellipticity" and "faceting". The tolerance zone is limited by two concentric circles with radii different by a specified amount. Roundness is always controlled in plane.

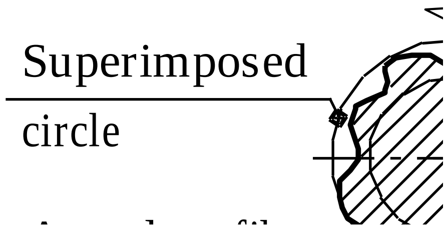


Fig. 2.8 Deviation from roundness

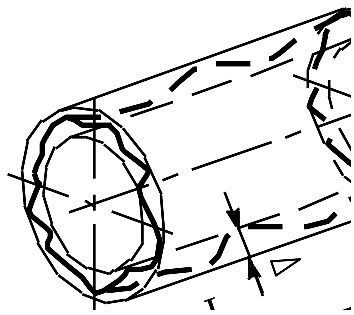


Fig. 2.9 Deviation from cylindricity

2.2.4 Cylindricity

By the deviation from **cylindricity** the maximum distance (Δ) from the actual surface points to the superimposed cylinder, measured within the specified length, is meant (Fig. 2.9). The tolerance zone is limited by two coaxial cylinders positioned at a specified distance. The cylindricity simultaneously specifies the requirements for roundness and straightness of a feature.

2.2.5 Longitudinal Section Profile

The longitudinal section profile refers to cylindrical axial sections only. By the deviation from **longitudinal section profile** the maximum distance (Δ) from the actual profile points to the superimposed profile of a cylinder in axial section, measured within the specified length, is meant (Fig. 2.10 a). Particular deviations from longitudinal section profile are "bow", "taper", "barrel" and "waist" (Fig. 2.10 b, c, d and e respectively).

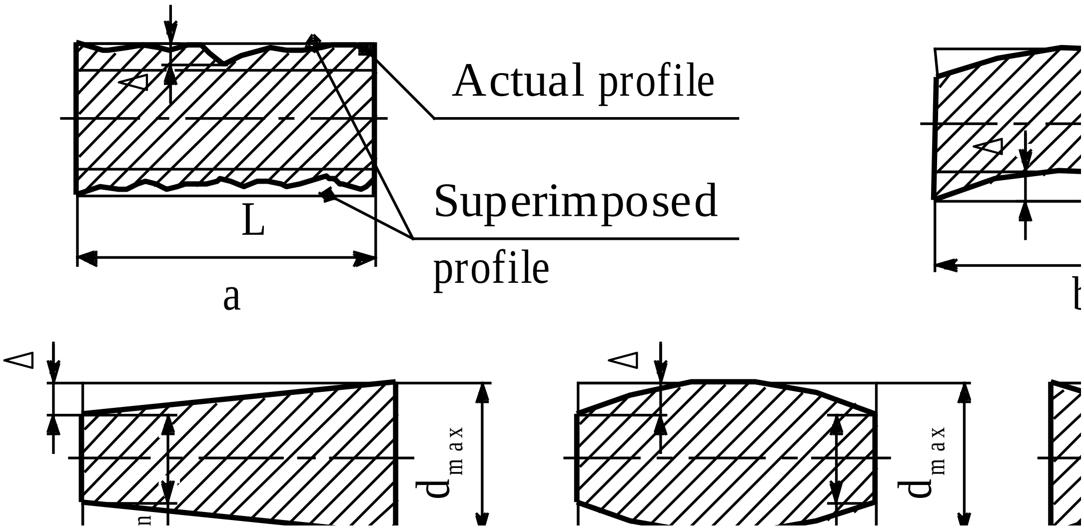


Fig. 2.10 Longitudinal section profile

2.3 Location Deviations

The performance of machines is affected not only by the dimension and form deviations, but the mutual position of component geometrical features are also of great importance. Geometrical requirements that specify positional accuracy are called location deviations.

Location deviation of a feature, such as surface or profile, is the deviation of the actual location of the feature from its nominal location. Inspection is carried out on features with excluded form deviations, thus the actual features are replaced by the ideal superimposed ones and the axes, symmetry planes and centers of the superimposed features are used instead of those of the actual features.

Another specific attribute of the location deviations is that they are always referenced to a datum.

By a **datum** a theoretical plane, line, point, or cylinder, which is derived from a corresponding physical feature of the component, is meant. As a datum any geometrical feature or a set of geometrical features of a component can be used. In case of a surface of revolution (cylinder, taper) or thread, its axis is used as a datum. Thus the inspection is to be made with respect to the corresponding datum feature.

The location deviations are as follows:

- **Parallelism**
- **Perpendicularity**
- **Angularity**
- **Coaxiality**
- **Cylindricity**
- **Symmetry**
- **Position**
- **Crossed axes**

2.3.1 Parallelism

Deviation from parallelism can be defined for a line or a surface with reference to another line or a surface. The following parallelism tolerances are found:

- **Parallelism tolerance of a line with reference to a datum line**; the tolerance zone specifies either a parallelepiped of the section $TPA_X \times TPA_Y$ or the diameter of a cylinder parallel to a datum line within the specified length (Fig. 2.11 a and b).
- **Parallelism tolerance of a line with reference to a datum surface**; the tolerance zone specifies the difference between maximum and minimum distances from a line to a datum plane within the specified length (Fig. 2.12 a).
- **Parallelism tolerance of a surface with reference to a datum surface**; the tolerance zone specifies the difference between maximum and minimum distances from a plane to a datum plane within the specified area (Fig. 2.12 b).

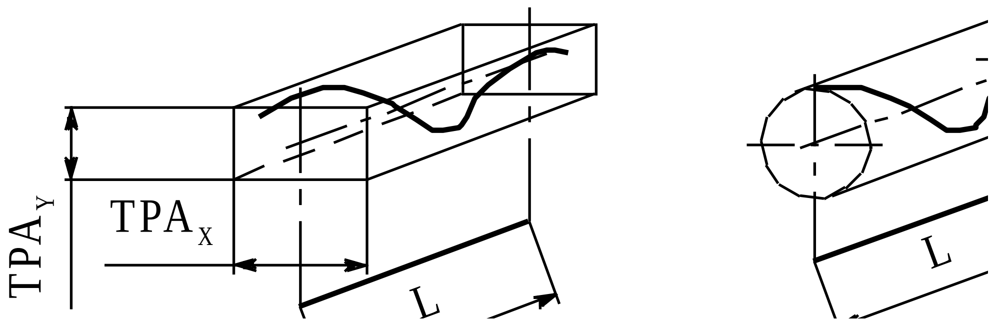


Fig. 2.11 Parallelism I

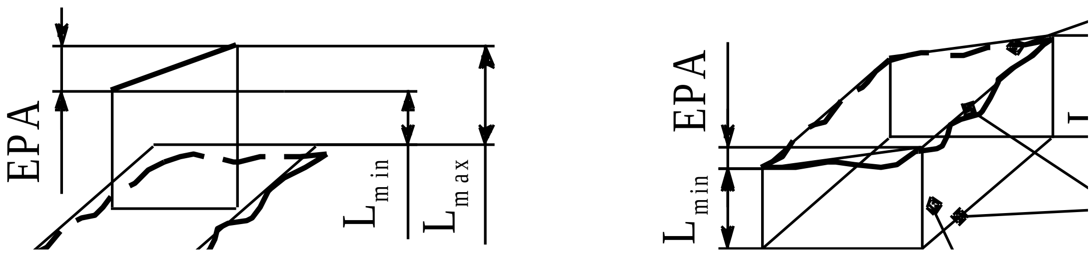


Fig. 2.12 Parallelism II

2.3.2 Perpendicularity

Deviation from perpendicularity can be defined for a line or a surface with reference to another line or a surface. The following perpendicularity tolerances are found:

- **Perpendicularity tolerance of a surface or a line with reference to a datum line**; the tolerance zone specifies the distance between two parallel planes at a right angle to a datum surface within the specified length (Fig. 2.13 a).
- **Perpendicularity tolerance of a surface with reference to a datum surface**; the tolerance zone specifies the distance between two parallel planes at a right angle to a datum surface within the specified length (Fig. 2.13 b).

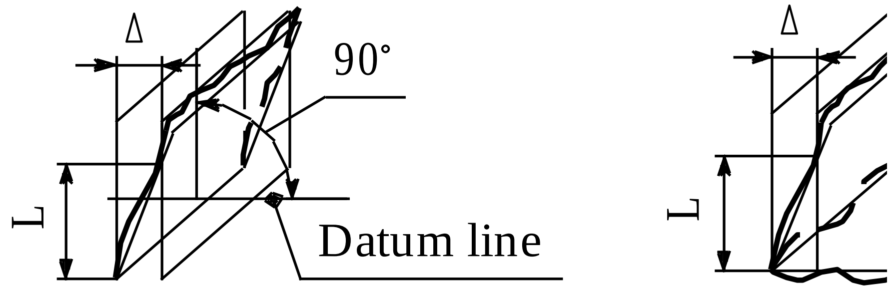


Fig. 2.13 Perpendicularity

2.3.3 Angularity

Deviation from angularity can be defined for a line or a surface with reference to another line or a surface. The following angularity tolerances are found:

– **Angularity tolerance of a surface or line with reference to a datum line or surface**; the tolerance zone specifies the distance between two parallel planes at a right angle to a datum surface within the specified length (Fig. 2.14 a).
– **Angularity tolerance of a line with reference to a datum line or surface**; the tolerance zone specifies the distance between two parallel planes at a right angle to a datum surface within the specified length (Fig. 2.14 b).

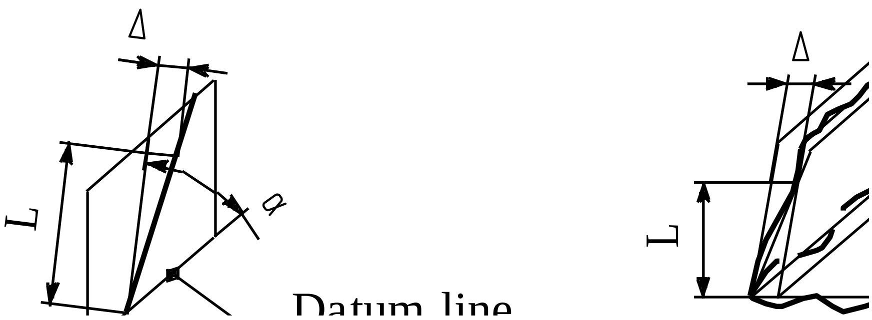


Fig. 2.14 Deviation from angularity

2.3.4 Coaxiality

In practice the two cases of coaxiality, depending on the datum used, can be found:

– **Coaxiality tolerance of a surface with reference to a datum surface axis**; the tolerance zone specifies either the maximum distance from the actual axis to a datum axis within the specified length (radial tolerance), or the diameter of a cylinder, coaxial with the datum axis, within which the actual axis should be included (Fig. 2.15 a). If the latter is the case, then the tolerance value is preceded by the symbol Ø.
– **Coaxiality tolerance of several surfaces with reference to a common datum axis**; the tolerance zone specifies the maximum distance from the actual axis to a common datum axis of two or more surfaces of revolution within the specified length. The **common datum axis** is an axis that intersects the given features axes at the median planes of these features (Fig. 2.15 b). If the tolerance zone is circular or cylindrical its value is preceded by the symbol Ø.

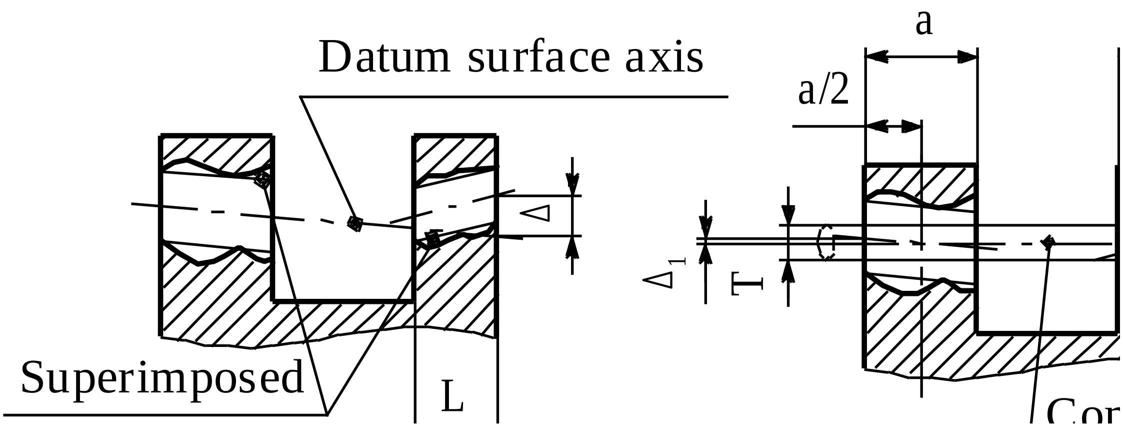


Fig. 2.15 Coaxiality

2.3.5 Symmetry

Deviation from symmetry can be defined with relation to a single datum feature or to a set of datum features. The symmetry deviations are as follows:

- **Symmetry deviation with reference to the datum feature**; the deviation is defined as the distance Δ (EPS) between the medium planes of a given feature and datum feature within the specified area $L_1 \times L_2$ (Fig. 2.16 a).
- **Symmetry deviation with reference to the common datum plane**; the deviation is defined as the distance Δ (EPS) between the medium plane of a given feature and common medium plane of two or more datum features within the specified area (Fig. 2.16 b).

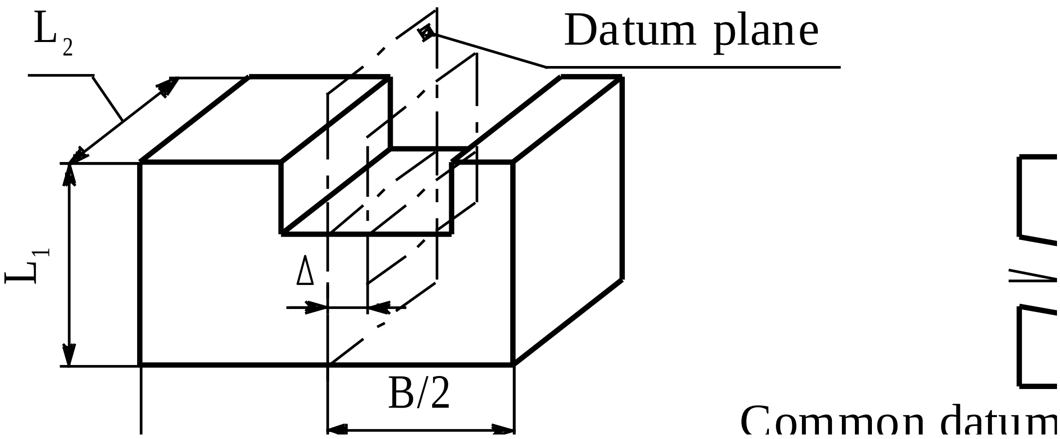


Fig. 2.16 Symmetry

2.3.6 Crossed Axes

The tolerance zone for this type of deviation specifies either the maximum distance between the axes of two features or the diameter of a cylinder coaxial with the datum axis, within which the other feature axis points should lie. Fig. 2.17 illustrates the concept.

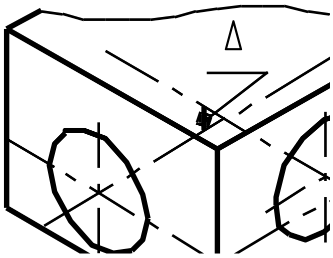


Fig. 2.17 Crossed axes

2.3.7 Position

Position deviation is the maximum distance between the actual position of a feature axis or median plane and theoretically exact position within the specified length.

The tolerance zone can be specified as follows:

– When the position is considered in a plane, the tolerance zone is defined as a distance between two parallel lines disposed symmetrically with relation to the exact position (Fig. 2.18 a).

– When the position is considered in space (Fig. 2.18 b and c), the tolerance zone is defined either as a diameter of a cylinder the axis of which is the exact position (tolerance value is preceded by the symbol R or Ø) or as a parallelepiped of section $TPP_1 \times TPP_2$ (tolerance value is preceded by the symbol T/2 or T).

It should be noted that the dimensions that specify the theoretically exact position are not toleranced and are considered as **theoretically exact dimensions**. The actual deviation of these dimensions is restricted by the position tolerance zone only. Thus, on engineering drawings the exact dimensions are given in frames. The same principle is also applied to the tolerancing of angularity, crossed axes, form of a line and form of a surface.

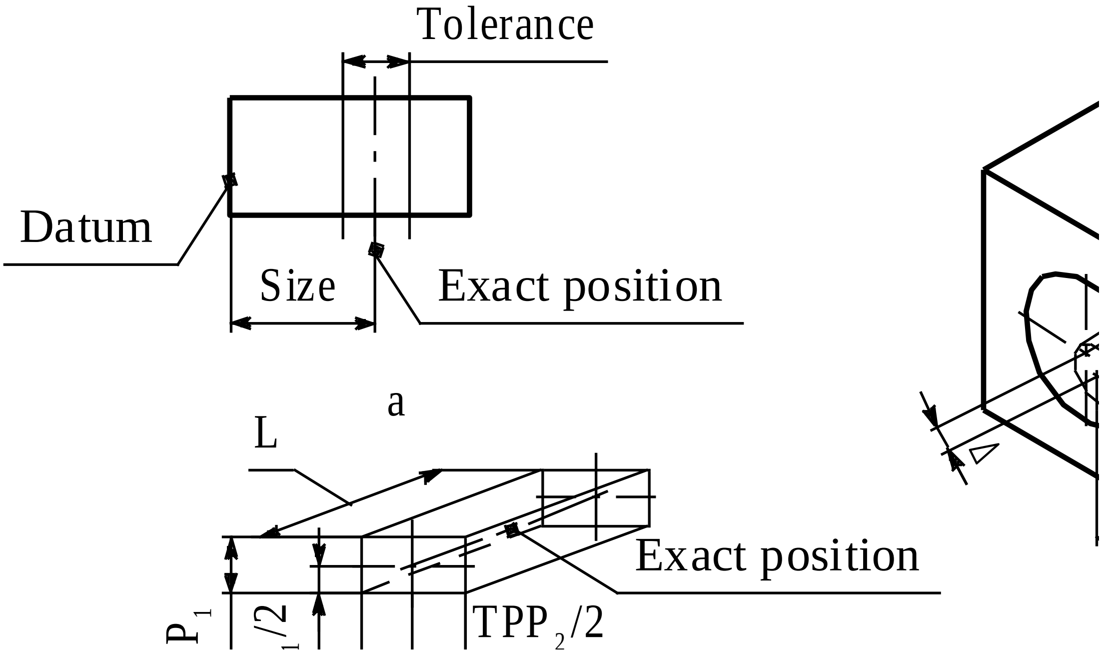


Fig. 2.18 Position tolerance

2.4 Composite Deviations

Composite deviations are the result of combination of the form and location deviations of a given surface or profile with respect to the datums.

According to GOST 24642-81 the following geometric characteristics are related to composite deviations:

– **Circular run-out**
– **Total run-out**
– **Form of a line**
– **Form of a surface**

2.4.1 Circular Run-out

Circular run-out deviations can be of radial or axial type. In the first case the inspection is made in the plane perpendicular to the axis datum, with the radii of surface elements controlled (Fig. 2.19 a). The **radial circular run-out** is composed of roundness with position deviations, and is usually defined for a complete revolution about the datum axis, but also can be applied to a sector of a definite angle only. The tolerance zone is limited by two concentric circles of specified diameters. The **axial circular run-out** is defined in the cylindrical section of a component face; with the sectioning cylinder concentric with the datum axis and of specified diameter (Fig. 2.19 b). The axial run-out is composed of flatness with perpendicularity deviations.

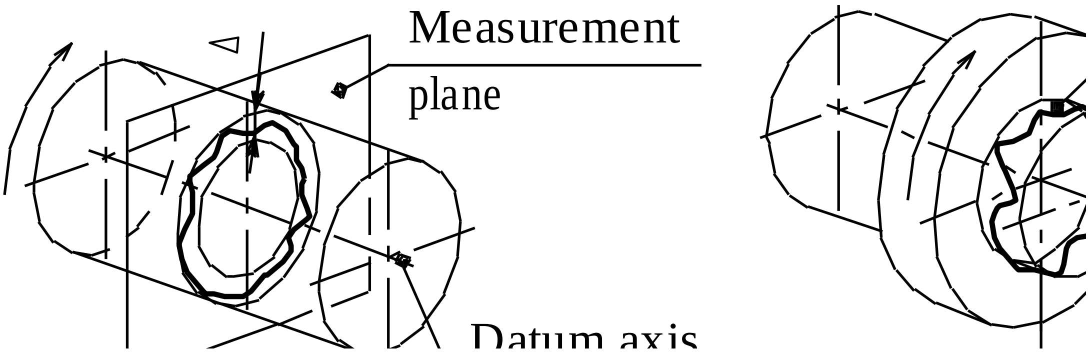


Fig. 2.19 Circular run-out

2.4.2 Circular run-out in a specified direction

Circular run-out in a specified direction is defined as the difference between maximum and minimum distances of an actual surface profile points, measured on the generator of a cone of specified angle and coaxial with the datum axis (Fig. 2.20). The direction of control is usually at a right angle to the inspected surface, unless another direction is specified.

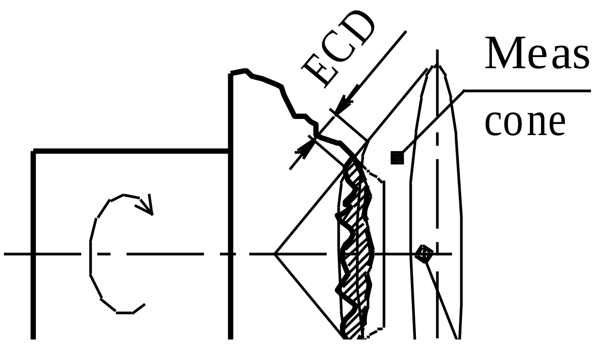


Fig. 2.20 Run-out in a specified direction

2.4.3 Total Run-out

Total run-out deviation can be radial or axial. **Radial total run-out** is a form of circular run-out but performed within the whole or specified length of a feature. The **axial total run-out** is defined as a composite axial circular run-out for the whole face surface of a feature.

The total radial run-out is a difference between maximum and minimum distances of the actual surface profile points to the datum axis within the length of inspection (Fig. 2.21 a). The radial total run-out is a combination of cylindricity with coaxiality.

Axial total run-out is the difference between maximum and minimum distances of the whole actual face surface to the plane perpendicular to the datum axis (Fig. 2.21 b).

The axial total run-out is composed of flatness and perpendicularity.

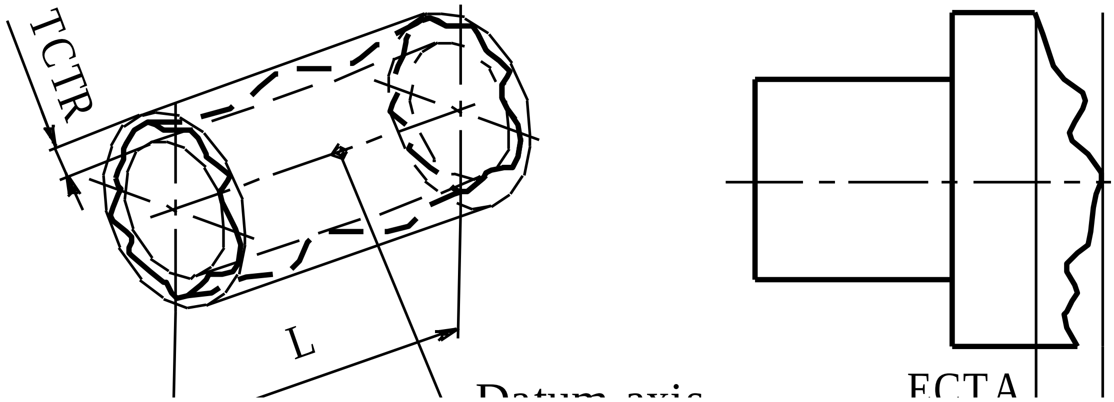


Fig. 2.21 Total run-out tolerance zones

2.4.4 Form of a Line

Form of a line specifies the maximum deviation of the actual profile points from the nominal profile, measured at a right angle to each point within the specified length (Fig. 2.22 a). The tolerance zone is limited by two equidistant lines at a specified distance and placed symmetrically with respect to the nominal profile (Fig. 2.22 b). This kind of tolerance is applied to irregular profiles, such as turbine blades. For three-dimensional objects a profile of a surface tolerance is used.

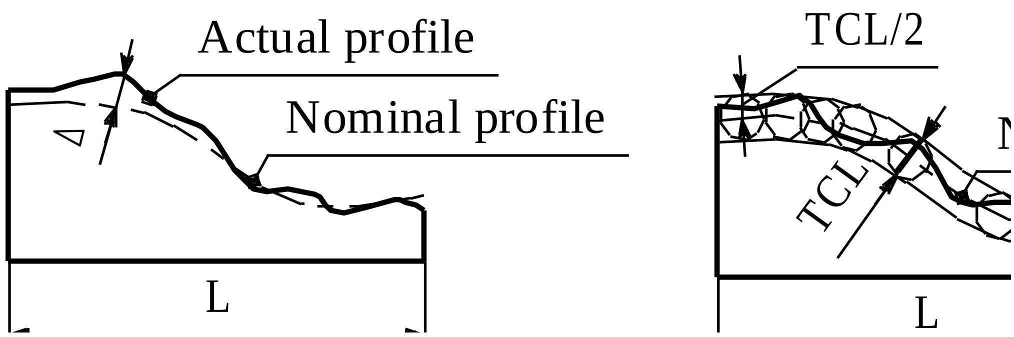


Fig. 2.22 Form of a line: deviation and tolerance zone

2.4.5 Form of a Surface

Form of a surface specifies maximum deviation of the actual surface points from the nominal surface, measured at a right angle to each point within the specified area (Fig. 2.23 a). The tolerance zone is limited by the two equidistant surfaces at a specified distance (tolerance value) and placed symmetrically with respect to the nominal surface. Fig. 2.23 illustrates the concept.

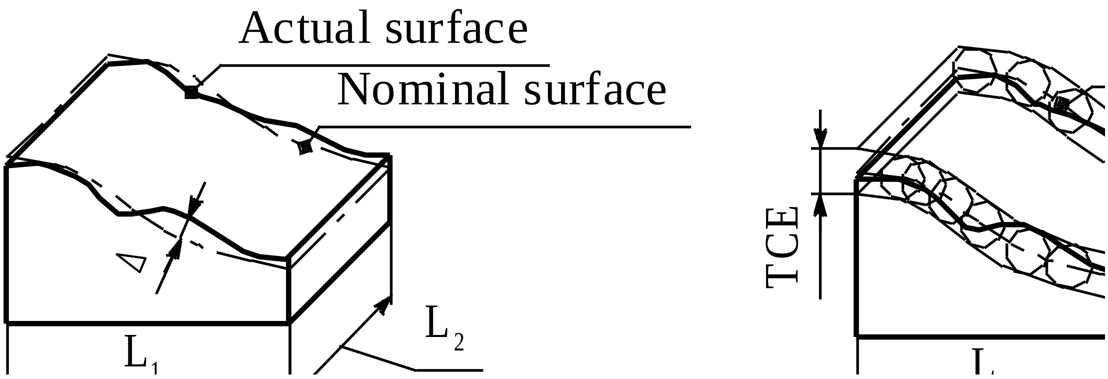


Fig. 2.23 Form of a surface: deviation and tolerance zone

2.5 Maximum Material Condition

The principle of independency states that dimensional and geometrical tolerances are applied independently relative to each other. Thus the geometrical deviations are not limited by the dimension tolerance zone, unless otherwise specified. In case a mutual dependency of dimensional and geometrical requirements is needed, the maximum material condition (MMC) should be used. The MMC states that the tolerance value of a form or location can be increased on the difference between the maximum material limit of size and the actual size. The dependency of dimensional and geometrical tolerances is denoted on drawings by the symbol Ⓜ. Fig. 2.24a illustrates an example of independently specified tolerance of holes concentricity. In this case, the assembly of the component with the counterpart can be hard to perform because of the worst collective effect of the geometric and actual sizes deviations of both components. And tolerancing shown in Fig. 2.24b can save the situation, since the concentricity tolerance can gain an extra value, defined as:

$$\Delta = (S_1 + S_2)/2 \tag{1.23}$$

where S_1 and S_2 are the clearances in couplings of the two elements, since the concentricity is defined as radial distances of the element points from the hole axis.

If both holes are at least material condition, i.e. of the maximum limit of size 18,3 and 9,2 mm, the additional gain in concentricity tolerance value is:

$$\Delta = (0{,}3 + 0{,}2)/2 = 0{,}25\,\text{mm}$$

And the concentricity tolerance value is now equal to 0,35, or 3,5 times higher, hence the manufacture of the component can be facilitated. The example shown in Fig. 2.24 b illustrates the case, when the dependence of the geometrical tolerance is established on both the specified size and datum size actual dimensions. This dependence can also be set individually on the actual dimensions of either the specified size or datum size.

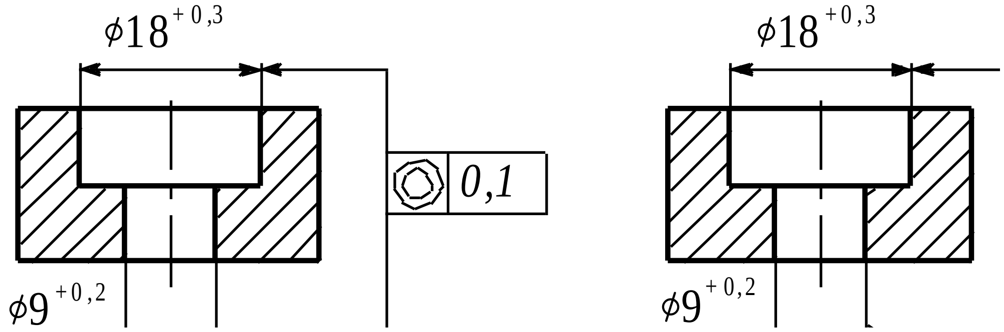


Fig. 2.24 MMC principle interpretation

2.6 Indications on Drawings

The from and location tolerances are denoted with the symbols shown in Fig. 2.25

Straightness	—	Coaxiality
Flatness	▱	Symmetry
Roundness	○	Position
Cylindricity	⌭	Crossed axes
Longitudinal		Circular run-
section profile	=	Total run-out
Parallelism	//	Form of a lin

Fig. 2.25 Geometrical tolerances symbols

Geometric tolerance requirements are indicated on a drawing with the help of a feature control frame. Feature control frame is a rectangular frame which is divided into compartments for geometrical symbols, tolerance value, modifiers and datum references. The feature control frame layout is given in Fig. 2.26.

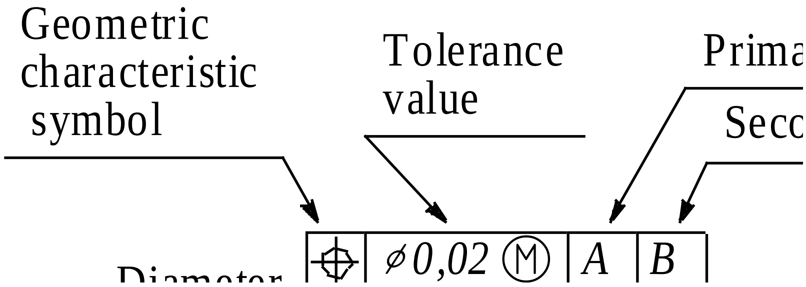


Fig. 2.26 Feature control frame

Modifiers in the second compartment describe the tolerance conditions, such as maximum material condition or a type of a tolerance zone. If the tolerance zone is circular or cylindric, then the symbol Ø should be placed before the tolerance value. If no symbol precedes the tolerance value, then the tolerance zone is limited by two parallel lines, planes or equidistant lines or surfaces.

It is also possible to specify more than one geometrical tolerance for the feature; in this case tolerance requirements are given in separate feature control frames one under another as shown in Fig. 2.27.

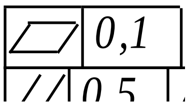


Fig. 2.27 Double feature control frame

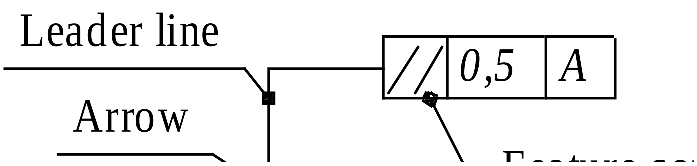


Fig. 2.28 Feature control frame connected to a feature

The feature control frame is connected to the toleranced feature by a leader line with an arrowhead in the way shown in Fig. 2.28. The leader line should be placed on the extension of the dimension line, if a tolerance refers to the axis or the median plane of the feature (Fig. 2.29 a). The leader line is placed on the outline of a feature and definitely off the dimension line if tolerance refers to the line or surface itself (Fig. 2.29 b). The leader line is placed on the axis if tolerance refers to the common axis or the median plane of the specified features (Fig. 2.29 c).

A datum feature used as a reference for some types of geometrical tolerances is indicated by a letter enclosed in square and connected to the datum feature by a datum triangle, as shown in Fig. 2.30. The same principles, illustrated in Fig 2.29, relative to connection of a feature control frame to a toleranced feature are applied to datum indications as shown in Fig. 2.31.

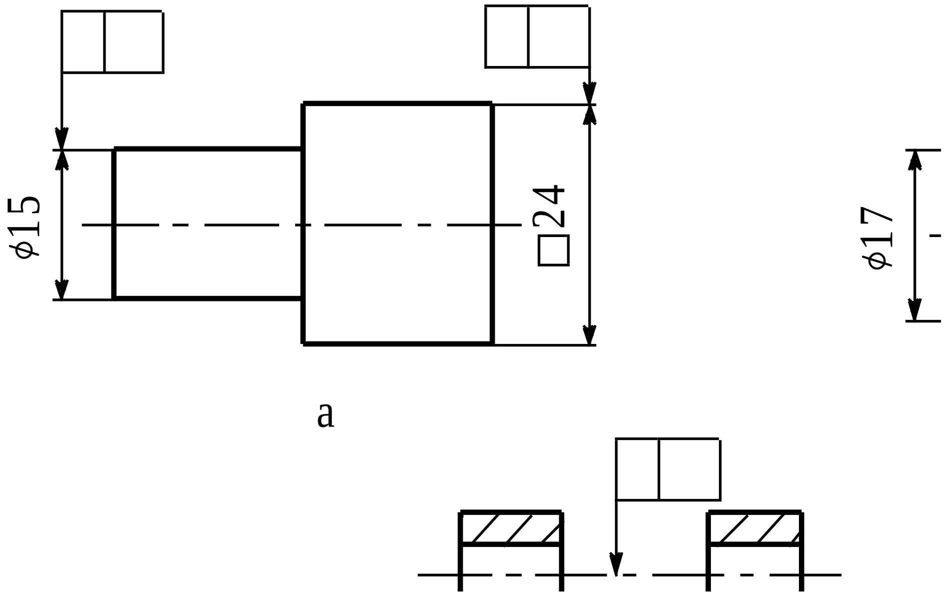


Fig. 2.29 Application of tolerance requirement to a feature

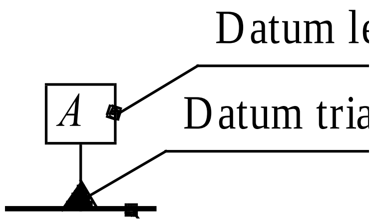


Fig. 2.30 Datum feature symbol

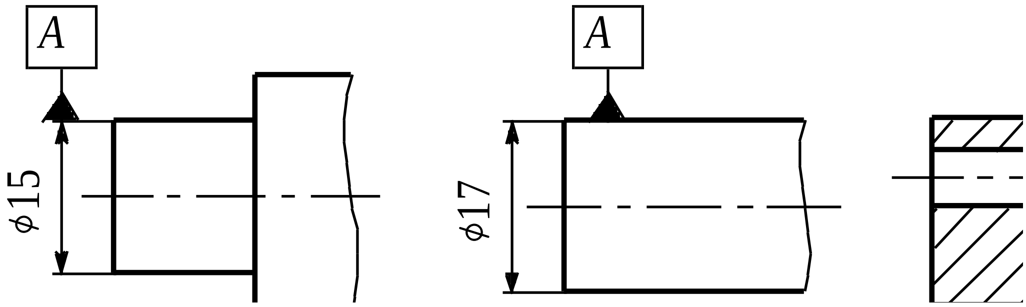


Fig. 2.31 Application of a datum indication to a feature

The datum on drawings can also be specified without indicating a letter. In this case the feature control frame is directly connected to a datum feature with the help of a leader line with a datum triangle (Fig. 2.32).

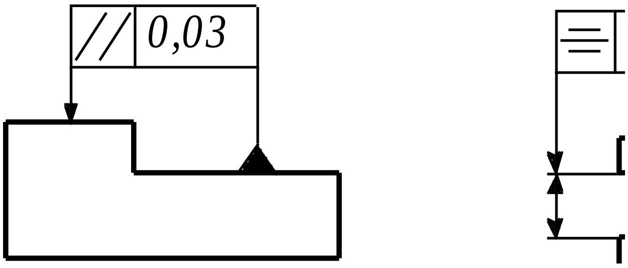


Fig. 2.32 Datum indicated without letter

The information on the specification of MMC principle on drawings was partly given in Section 2.5. The feature control frames for different conditions of MMC are given in Fig. 2.33. In case the MMC is applied only to a toleranced feature, then the symbol Ⓜ is placed in the tolerance value compartment (Fig. 2.33 a). If the MMC is applied only to a datum feature, then the symbol Ⓜ is placed after the datum letter in corresponding compartment (Fig. 2.33 b and c). If the MMC is applied to the toleranced feature and datum feature simultaneously, then the symbol Ⓜ is placed both after the tolerance value and the datum letter in the corresponding compartments (Fig. 2.33 d and e).

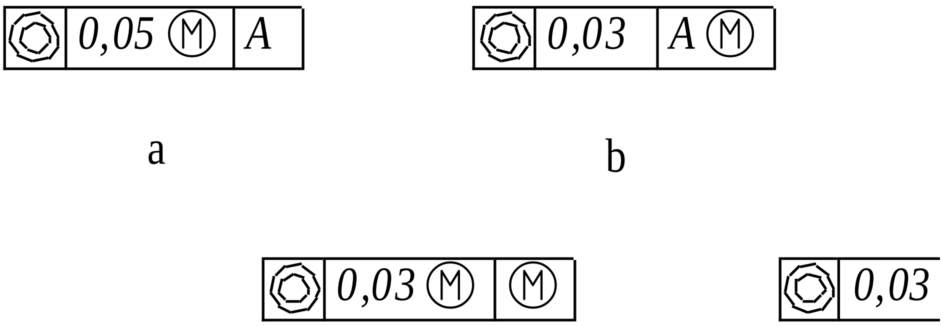


Fig. 2.33 MMC indications

3. Surface Texture

Among other geometrical deviations **surface texture** or **surface finish** plays a key role in determining product performance, especially when a product surface is in moving contact with the adjacent surface or when micro-geometrical deviations themselves can strongly affect the product functionality. The reasons for surface finish control are as follows:

1. **Friction behaviour** can be significantly improved by surface finish. High surface finish reduces friction forces and helps to maintain oil films on the contacting surfaces; consequently, the energy losses are reduced.
2. **Wear characteristics** relate to parameters of surface finish.
3. **Fatigue strength** of heavy loaded components can be affected by sharp irregularities of the surface texture, which act as stress concentrators.
4. **Corrosion resistance** also decreases with surface finish improvment.
5. **Appearance** of car bodies is an example of the cosmetic effect that is provided by high quality surface finish.
6. **Effective contact area** is of great importance for microprocessor industry or when heat conductivity of a surface should be as high as possible, e.g. transistor heatsinks.
7. **Microgeometry** of plastic tubes, for instance, can greatly reduce the damage of blood cells during blood transfusion or hemodialysis.
8. The performance of **sealing** components and devices is defined by surface finish, too.
9. **Noise requirements** are met via proper surface finish control.
10. **Painting and plating** process facilitation.

As it was mentioned in the beginning of Section 2, geometrical deviations of a component feature include errors of dimension, location and form, as well as waviness and roughness (refer to Fig. 3.1).

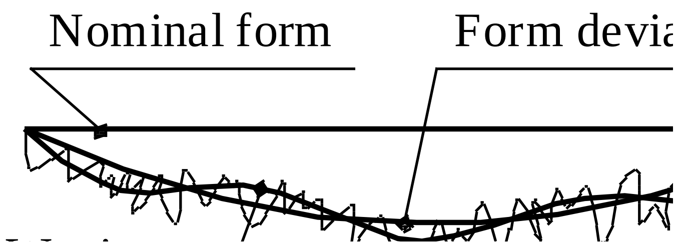


Fig. 3.1 Superimposition of geometrical errors

The surface finish is a combination of two geometrical characteristics: roughness and waviness. In order to distinguish form errors from waviness and roughness the following relations between irregularity heights and irregularity spacing are used (refer to Fig. 3.2).

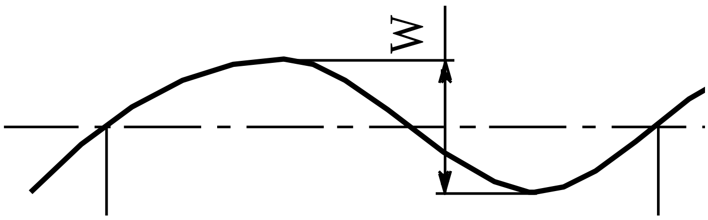


Fig. 3.2 Superimposition of geometrical errors

When the ratio is $\frac{W}{S} \geq 1000$, the irregularity is considered as a form error. When the ratio is $40 \leq \frac{W}{S} \leq 1000$, the irregularity is considered as waviness. When the ratio is $\frac{W}{S} \leq 40$, the irregularity is considered as roughness. And it is obvious that the point when roughness becomes waviness is ambiguous.

Since the manufacturing process defines the surface texture, roughness and waviness are distinguished in the following way:

– Roughness irregularities are the marks left by a cutting tool, spark, abrasive grain, etc.
– Waviness irregularities are the result of machine tool deflections and chatter.

Surface texture parameters are usually evaluated in a plane perpendicular to the inspected surface, thus the parameters describe the profile of a surface in a predefined direction. But with the help of modern optical 3D measurement technologies the surface can be evaluated as a whole 3D surface. In this case name of the 3D parameters starts with a capital letter S or V followed by a suffix. The names for the 2D parameters, described in this textbook, start with either capital R for roughness, or capital W for waviness parameters.

For inspection purposes the real surface profile is represented in the form of a profilogram. The profilogram is somewhat different from the real surface, since it is recorded with the help of tracing machines, which are not ideal and their capacities are limited.

Fig. 3.3 shows different close-ups of the surface for waviness and roughness, and a common profilogram is shown in Fig. 3.4.

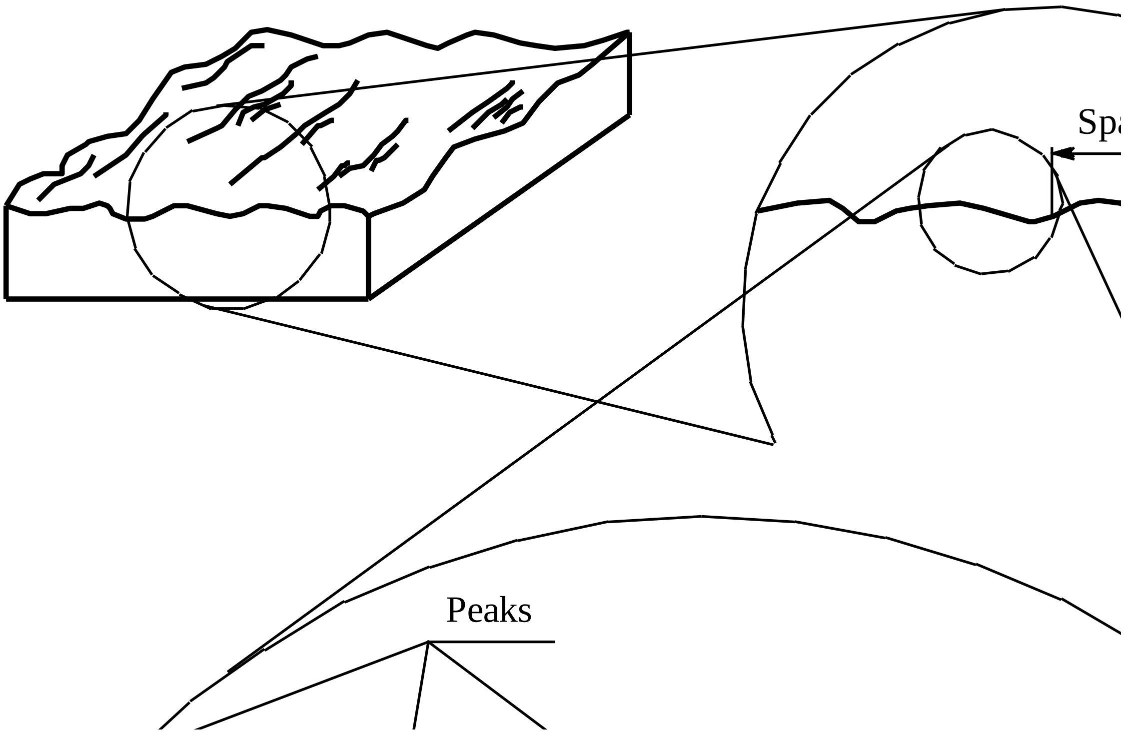


Fig. 3.3 Surface illustrated

Sampling length is the length over which parameters are measured. The sampling length should be long enough to reveal clearly the surface pattern needed to measure the parameters correctly, but not too long to exclude the influence of waviness (for roughness assessment) or form (for waviness assessment). Actually the sampling length for waviness measurements is usually limited by the profilometer maximum traverse, rather than the form error influence.

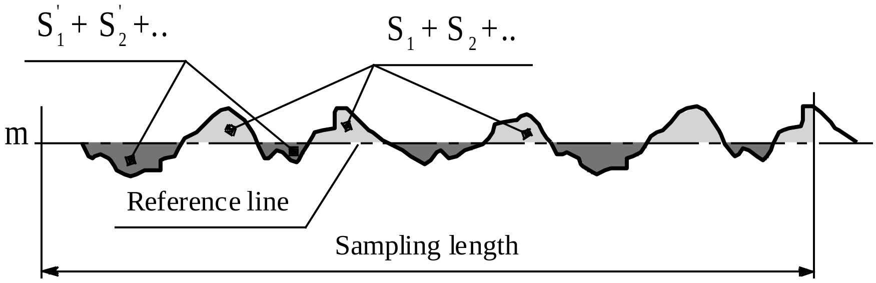


Fig. 3.4 Profilogram

The roughness parameters are measured relative to the **reference line**, which is equidistant to the surface profile and can be represented by either a **center line** or **least squares mean line**. The center line is positioned so that the area above the line (S_1+S_2+…) is equal to the area below the line (S_1'+S_2'+…). The least squares mean line is a line position so that the sum of square deviations of the profile from the line is a minimum.

3.1 Roughness

According to GOST 2789-73 surface roughness can be assessed by the six parameters, which are grouped with relation to the characteristics of the profile they describe.

1. **Amplitude parameters (vertical direction)**
   - Maximum roughness height, designated as R_{max}
   - Ten-point height, designated as R_z
   - Roughness average, designated as R_a
2. **Spacing parameters (horizontal direction)**
   - Mean spacing of adjacent local peaks, designated as S
   - Roughness peak spacing on the center line, designated as S_m
3. **Bearing parameter**
   - Bearing length ratio, designated as t_p

In order to measure the waviness and roughness parameters individually, surface profilogram, which is called raw profile (refer to Fig. 3.5 a), is filtered and two separate profiles of waviness (with roughness errors filtered out, refer to Fig. 3.5 b) and roughness (with waviness errors filtered, refer to Fig. 3.5 c) are obtained.

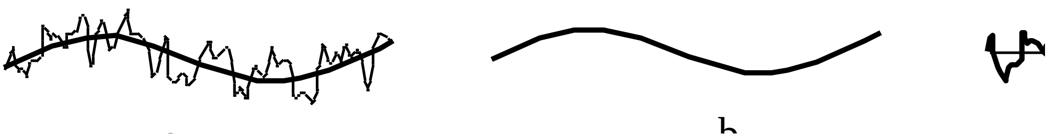


Fig. 3.5 Profile filtration

Maximum roughness height is defined as the distance between the highest peak, R_P, and the deepest valley, R_V, within the specified length, and measured in µm (refer to Fig. 3.6). R_{max} calculated as:

$$R_{max}=R_P+R_V \tag{3.1}$$

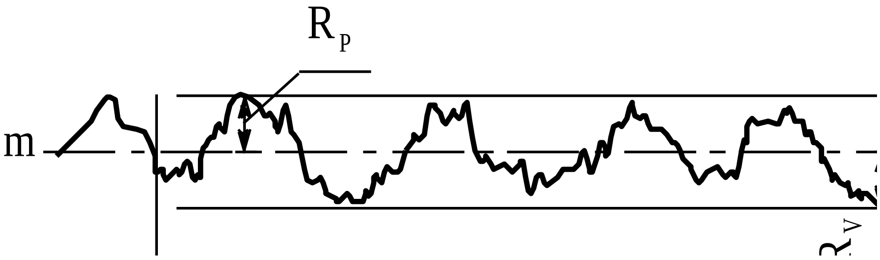


Fig. 3.6 Maximum roughness height

Ten-point height is defined as the mean distance between the five highest peaks, Y_P, and the five deepest valleys, Y_V, within the specified length (Fig. 3.7). The parameter is evaluated with reference to the center line, measured in μm.

$$Rz = \frac{1}{5}\left[\sum_{i=1}^{5}|yi| + \sum_{j=1}^{5}|yj|\right] \tag{3.2}$$

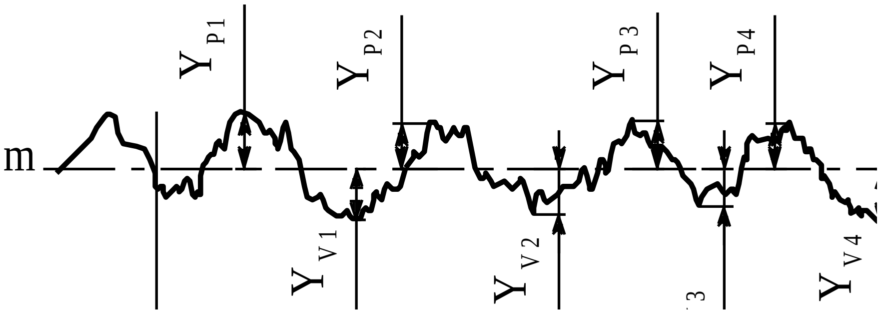


Fig. 3.7 Ten-point height

Roughness average is defined as the arithmetic average of the absolute values of the profile points departure (y_i) from the center line within the sampling length (illustrated in Fig. 3.8). The parameter is evaluated with reference to the center line, measured in μm. To calculate the R_a parameter, the following two equations can be used:

$$Ra = \frac{1}{n}\sum_{i=1}^{n}|yi| = \frac{1}{l}\int_{0}^{l}|y(x)|dx \tag{3.3}$$

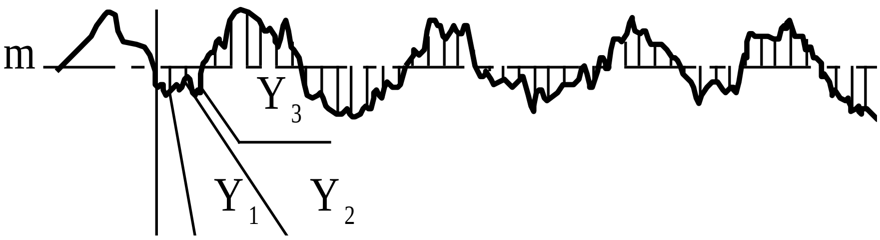


Fig. 3.8 Roughness average

The next two parameters are measured in the horizontal direction and relate to spacing between profile irregularities.

Roughness peak spacing on the center line is the average spacing, measured on the center line between adjacent crossings (in the same direction) within the sampling length (Fig. 3.9).

The peaks that go into count are the profile peaks that are separated by crossings of the profile line with the center line. This parameter is expressed in mm, and is obtained from:

$$S_m = \frac{1}{n}\sum_{i=1}^{n} Smi \tag{3.4}$$

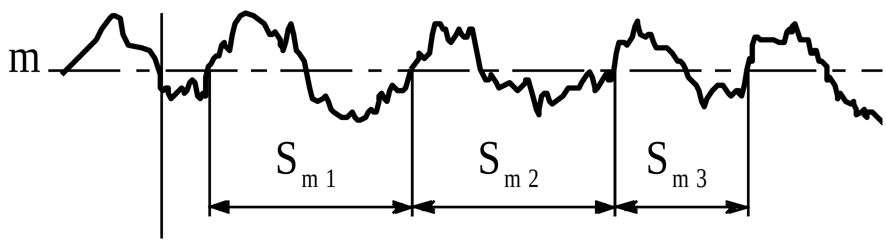


Fig. 3.9 Roughness peak spacing on the center line

Mean spacing of the adjacent local peaks is defined as the average spacing, S_i, between the adjacent local peaks measured within the sampling length.

As it is illustrated in Fig. 3.10 by local peaks all the peaks within the sampling length are meant. This parameter is expressed in mm, and can be calculated from the equation:

$$S = \frac{1}{n}\sum_{i=1}^{n} Si \tag{3.5}$$

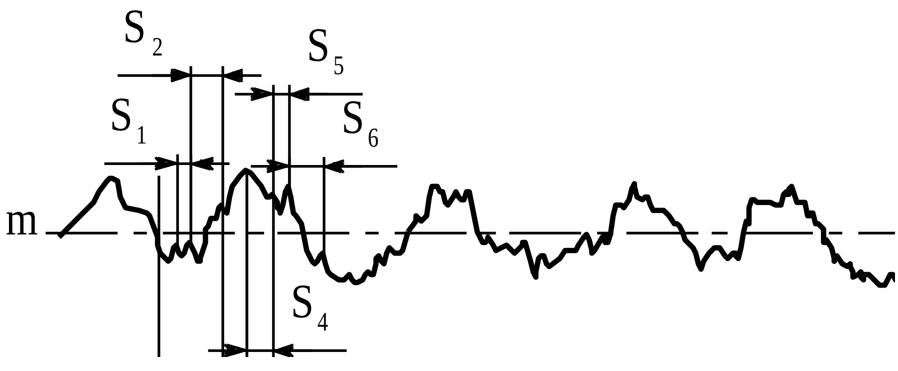


Fig. 3.10 Mean spacing of adjacent local peaks

Bearing length ratio is defined as the ratio of the bearing length on the specified level (p) in the profile to the sampling length. The calculation of the parameter is represented by the equation 3.7. The bearing surface length is designated as η, and is obtained from:

$$\eta_p = \sum_{i=1}^{n} bi \tag{3.6}$$

The level of the profile section is positioned at a certain depth with respect to the highest peak line, and is given in percents of the maximum roughness height.

$$t_p = \frac{1}{l}\sum_{i=1}^{n} bi \times 100\% \tag{3.7}$$

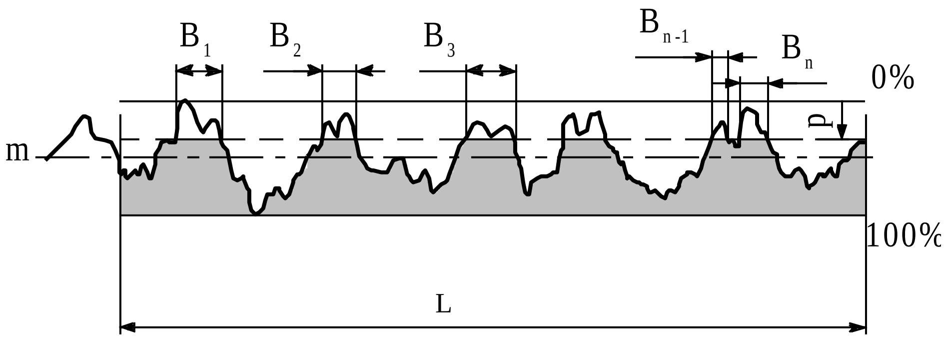


Fig. 3.11 Bearing length ratio

The bearing length ratio parameter t_p is written in the following form: $t_{40}70\pm20$. Where the section level is equal to 40% of R_{max}, the bearing length ratio is 70% of the sampling length and ±20% are the limit deviations of the parameter value.

As the machining patterns left on the surface have a distinct direction, which can affect the functionality of the given surface, this pattern directions are classified into the types shown in Fig. 3.12 and are called **lays**.

In general, the direction of surface roughness measurements should lie perpendicular to the lay. For multidirectional irregularities the measurements are performed in several directions and the parameter value is defined as the arithmetic average of these measurements. For the “particulate” type of lay the surface can be measured in any direction.

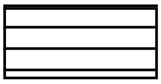
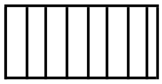
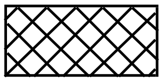
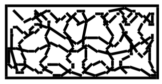
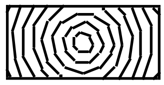
Type	Parallel	Perpen-dicular	Crossed	Multi-directional	Circular
Lay					

Fig. 3.12 Lay types

To denote the roughness parameters on a drawing, the symbols shown in Fig. 3.13 are used. The first symbol (a) is used when a surface can be produced by any type of the machining. The second symbol (b) is applied when material removal by machining is required. The third symbol can be used in two cases: (a) when removal of material is prohibited, i.e. such processes as casting, forging or rolling should be used; (b) when the surfaces specified by this symbol are not to be machined and, thus retain initial roughness.

The roughness parameters are placed in accordance with Fig. 3.14, where (a) is for roughness parameter(s); (b) is for method of machining; (c) is for lay.

Fig. 3.13 Roughness symbols

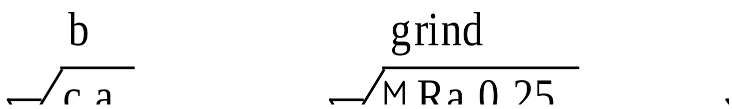


Fig. 3.14 Roughness symbol description

3.2 Waviness

The waviness parameters are designated by capital letter W and are as follows.

Waviness maximum height is the maximum peak-to-valley distance measured within the same wave of the sampling length L_W, designated as W_{max} (refer to Fig. 3.15).

Waviness height is the arithmetic average of the five peak-to-valley distances within the sampling length L_W, designated as W_z.

$$W_z =(W_1+ W_3+ W_3+ W_4+W_5)/5 \tag{3.8}$$

Waviness peak spacing is the arithmetic average of the length of the center line sections, between adjacent crossings (in the same direction) of the center line by the profile, designated as S_W.

$$S_W = \frac{1}{n} \cdot \sum_{i=1}^{n} S_{Wi} \tag{3.9}$$

The center line m_W is positioned in the same manner as the roughness center line.

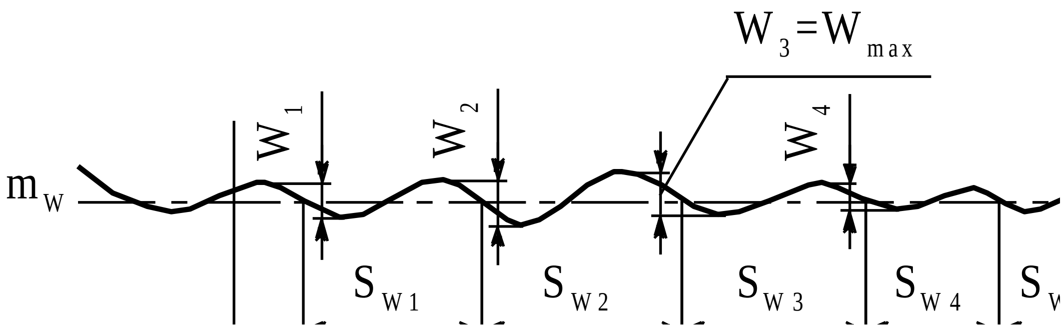


Fig. 3.15 Waviness profile

4. Tolerance Analysis

Tolerance analysis is the term used to describe the study of accumulated variation in dimensions of components and assemblies. As every dimension is supposed to vary to some extent, i.e. tolerance, so when a set of components is assembled into a product, variation of the assembly dimension can be as high as the number of components involved. To ensure the required assembly parameters and prevent components from mutual interference or failure, the tolerance values for the components dimensions should be assigned wisely. For these purposes tolerance analysis methods are intended.

The object of tolerance analysis is always a chain of dimensions that constitute a task. It can be individual component dimensions that define the accuracy of mutual position of the component features; or dimensions that specify the mutual position of components in an assembly. This chain can be of linear, two-dimensional or three-dimensional type. The sequence of dimensions that form a closed circuit is called a **dimensional chain** (refer to GOST 16319-80). The examples of a linear dimensional chain (a) and 2D chain (b) are shown in Fig. 4.1 below.

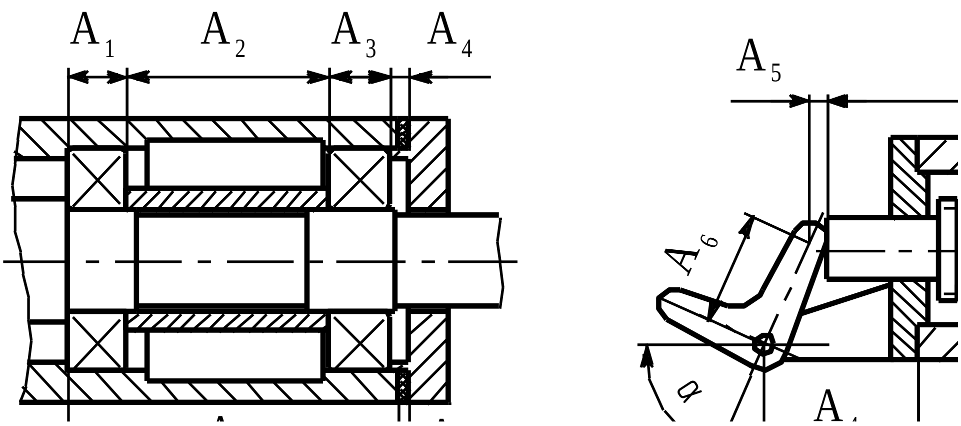


Fig.4.1 Dimensional chains

Since the dimensions in a drawing **never** close a circuit into a chain, leaving the last dimension to be formed by itself during the assembly or machining process, for instance, the extra dimension is added to the dimensional sequence to make a loop. This extra dimension is called **resulting dimension** (A_Δ in Fig. 4.1) and is used to perform analysis. The dimensions that constitute a chain are called **component dimensions** and can be either increasing or decreasing.

Increase of the **increasing dimension** (A_1-A_4 in Fig. 4.1) will result in an increase of the resulting dimension value. The increase of a **decreasing**

dimension (A_5 in Fig. 4.1) will result in a decrease of the resulting dimension value. The dimensional chain is usually represented in the way shown in Fig. 4.2.

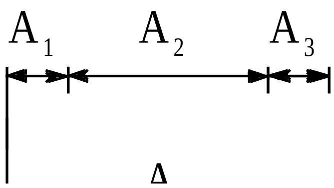


Fig.4.2 Dimensional chain diagram

4.1 Basic Concepts and Methods of Calculation

The tasks of the tolerance analysis can be of two types:

1. **Direct task** (product design stage) consists in the calculation of the tolerances and limits of size for component dimensions, with given basic sizes of the component dimensions and the basic size and limit deviations of the resulting dimension.
2. **Inverse task** (product control stage) consists in the calculation of the basic size and limit deviations for the resulting dimension, with the given basic sizes and limit deviations of the component dimensions.

According to GOST 16320-80 the following tolerance analysis methods are applied:

1. The complete interchangeability method (maximum-minimum method)
2. The incomplete interchangeability method (statistical method)
3. The method of group interchangeability
4. The adjustment method
5. The fitting method

The maximum-minimum method is the most common, since it ensures complete interchangeability in use. For this method the limits of size for a resulting dimension are the results of combinations of the maximum and minimum component dimensions. This method results in tighter tolerances and therefore higher costs of manufacturing. For dimensional chains with the number of dimensions more than six, it is more profitable to apply the statistical method. Methods of group interchangeability, in turn, are common in bearing manufacture. The adjustment methods are used when the dimensions of a machine tend to change over the period of operation. In this case the dimensional change is compensated with the help of spacers, gaskets or special adjustment devices. Application of the fitting method implies additional machining of one of the assembly components to the desired size to ensure the assembly. In this textbook only the first method is described.

The basic equation of the dimensional chain, used to calculate the basic size of the resulting dimension, is represented below:

$$A_\Delta = \sum_{j=1}^{n} A_j - \sum_{k=n+1}^{m-1} A_k \tag{4.1}$$

where m – is the number of components in a chain, including the resulting dimension; n – is the number of the increasing dimensions A_i; (m–n) – is the number of decreasing dimensions A_j.

The tolerance of the resulting dimension is equal to the sum of component dimensions tolerances and is calculated as:

$$TA_\Delta = A_\Delta^{\max} - A_\Delta^{\min} = \sum_{j=1}^{m-1} TA_j \tag{4.2}$$

The upper deviation of the resulting dimension can be obtained from:

$$ES(es)_{A_\Delta} = A_{\Delta_{\max}} - A_\Delta = \sum_{j=1}^{n} ES(es)_{A_j} - \sum_{k=n+1}^{m-1} EI(ei)_{A_k} \tag{4.3}$$

where ES and EI – are upper and lower deviations of the “hole” component dimensions respectively; es and ei – are upper and lower deviations of the “shaft” component dimensions respectively.

The lower deviation of the resulting dimension is defined as:

$$EI(ei)_{A_\Delta} = A_{\Delta_{\min}} - A_\Delta = \sum_{j=1}^{n} EI(ei)_{A_j} - \sum_{k=n+1}^{m-1} ES(es)_{A_k} \tag{4.4}$$

IT grades for the component dimensions of a chain are usually assigned of the same grade depending on the tolerance value of the resulting dimension. For this purpose the number of standard tolerance units is calculated from the equation:

$$a_{mean} = TA_\Delta / \sum_{j=1}^{m-1} i_j \tag{4.5}$$

where i – is a standard tolerance unit with respect to the dimension A_i value.

The standard tolerance unit, in its turn, can be obtained from the corresponding tables in hand-books or calculated as:

$$i = 0{,}45\sqrt[3]{A_j} + 0{,}001A_j \tag{4.6}$$

With the calculated number of standard tolerance unit a_{mean}, the IT grades for all component dimensions are defined. Then the sum of component dimensions tolerances is compared to the resulting dimension value, according to:

$$\Sigma TAj \leq [TA\Delta] \tag{4.7}$$

The difference between these values should be less than 3%. Otherwise it is necessary to correct the tolerance value for one or several component dimension.

Afterwards, the limit deviations are assigned for all component dimensions except one, limit deviations of which should be defined by Eq. 4.3 and 4.4.

4.2 2D and 3D Tolerance Stacks

Two-dimensional and three-dimensional stacks are analyzed by the same methods as linear chains. Spatial chains are brought into linear state in this case. Two-dimensional chain sizes are projected on a single line that is usually of the same direction as the resulting dimension. Dimensions of the three-dimensional chains are projected on two or three mutually perpendicular axes.

5. Engineering Metrology

Engineering metrology is the science of measurement, methods and principles that ensure accuracy, precision and traceability.

Measurement is the process of determining the value of a physical quantity with the help of means of measurement.

Means of measurement are classified as follows:

– **Standards** are used to keep and represent the value of a given physical quantity.
– **Gauges** represent specified values of a physical quantity and are used to check a given dimension.
– **Measuring instruments** are intended for the process of determining the value of a physical quantity.

Instruments used for measuring can be described by a set of the metrological characteristics, such as measuring range, measuring limits, indication range, resolution, accuracy, precision and measuring pressure.

5.1 Methods of Measurements

The process of measurement can be performed with the help of various methods, which are sets of predefined techniques, principles and means of measurement.

The methods of measurement are classified in the following way:

1. **The direct method** The value of a quantity is determined directly without measuring other elements and calculations.
2. **The indirect method** The value of a quantity is determined through the measurement of the related elements and calculations.
3. **The absolute method** is based on direct measurement of the fundamental quantity in relation to the definition of the quantity.
4. **The comparative method** is used when the quantity is compared to a known value of the same quantity.

The accuracy of measurement results is affected by measurement error and measuring instrument error. **Instrument error** is the difference between the instrument readings and the actual dimension. In practice it is difficult to distinguish these two errors and often they are described as the same error but obtained in different conditions. Thus the instrument error is the error obtained during calibration of the instrument; and the measurement error is the error obtained in working conditions.

Another classification of errors is based on the error nature. There can be systematic or random errors. A **systematic error** is constant for all measurements made with the given instrument and can result from faulty instrument. For instance, faulty position of the graduation mark or zero error reading will lead to permanent deviation in one direction from the actual value. It is obvious that systematic errors cannot be reduced or eliminated by repeated measurements. These errors are minimized by replacing the defective scale, by resetting the zero reading or by improving the instrument design.

Random errors unlike systematic ones are not constant and change with repeated measurements. For example, a set of measurements of the same component carried out with the same instrument will result in different values for the dimension. Since it is impossible to repeat the measurement in exactly the same manner, it can be stated that the actual size distribution is the result of statistical probability. The Gauss distribution is conventionally used to describe the distribution law.

Among errors affecting measurements there are faulty measurement or reading process, human elements, inappropriate physical environment, imperfect instrument etc.

Errors induced by inappropriate physical environment are reduced by setting standard temperature for measurements. As materials change in size with temperature change, it is essential to perform measurements at some constant temperature in order to get uniform results regardless of temperature change. The international standard temperature for dimensional measurements is stated as 20 °C, as common room temperature is 20 °C.

An example of faulty reading process is parallax error. This type of error arises when the pointer and scale are at some distance relative to each other, so when a reading is made at an acute angle to the scale, the pointer will take different positions relative to the scale, depending on the actual angle of view. Thus it is necessary to read the scale at a right angle to avoid parallax errors.

Design of the vernier caliper can induce error that is called Abbe error. The Abbe principle states that the object of measurement and scale should be collinear to reduce measurement errors. Since the scale of vernier caliper is usually parallel to the object, except measurements made with a depth bar, any errors in perpendicularity of the caliper jaws to the beam can cause error of measurements.

5.2 Gauge Blocks

Gauge blocks are reference standards or gauges reproduced with high accuracy and intended for practical linear measurements. The common application of gauge blocks consists in setting and calibrating limit gauges and measuring instruments, such as micrometers and dial gauges. Gauge blocks are manufactured with the length deviations within ±0,06 μm for small gauge blocks and up to ±16 μm for 1000 mm long gauge blocks. Measuring surfaces of gauge blocks are also inspected for flatness and parallelism, which in sum should not exceed 0,05 μm for gauge blocks smaller than 10 mm and 1,0 μm for a 1000 mm long gauge block. The accuracy of gauge blocks is checked with the help of interferometers.

Gauge blocks are available in sets and thus can be used in combinations to build block stacks of greater length. When the gauge blocks are properly joined, the measuring surfaces will adhere tightly due to the flatness and excellent surface finish. This process is also called **wringing**. When gauge blocks are wrung together the shearing force needed to separate them can be roughly equal to 98,1 N. In building up a given dimension, one should use as few blocks as possible starting with the gauge block of the smallest size.

In accordance with GOST 9038-90, gauge blocks are available of several accuracy grades, depending on the application: 00; 01; 0; 1; 2; 3.

Gauge blocks were invented by a Swedish machinist Carl Edvard Johansson in 1896, and to date gauge blocks are basic standard of length in industry and manufacture.

Gauge blocks are used for calibrating measuring instruments and setting instruments to the desired size for comparative measurements.

5.3 Measuring Instruments

The sections given below are teaching guides in laboratory classes. Each topic contains a description of measuring instrument and measuring principles used for laboratory work. The assignment for a laboratory work is given at the end of each section.

Students are supposed to write reports on completed laboratory works and make short reports with defending results of the works. The completed set of laboratory works serves as a permission for the exam of the course.

5.3.1 Measurements with Vernier Calipers

The objectives of the laboratory work are as follows:

1. To study the construction and measurement principle of a vernier caliper.
2. To measure the given component with vernier caliper.
3. To sketch a component with the actual dimensions denoted.

Vernier instruments are used for direct absolute measurements and lay-out procedures and constitute a wide group of instruments with a vernier scale. According to GOST 166-80, there are vernier calipers, vernier height gauges, vernier depth gauges, vernier gear tooth calipers, etc. Actually some types of vernier instruments have no vernier scale, in these cases a dial scale or digital display is used.

GOST 166-80 specifies three types of vernier calipers with the caliper shown in Fig. 5.1 being the most common. The vernier caliper shown in Fig. 5.1 consists of a flat beam with jaws fixed at right angle to the beam, and engraved main scale, having its markings 1 mm apart (pos. 1). On the movable frame (2) there are two jaws and vernier scale (3), a clamping screw (4) and depth bar (5). The upper jaws are intended for internal measurements and the lower jaws are for external measurements. The depth bar is used to measure depths and heights. In order to eliminate the clearance between the beam and movable frame there is a flat spring placed between them, right under the clamping screw. When the jaws are closed, then the reading is zero.

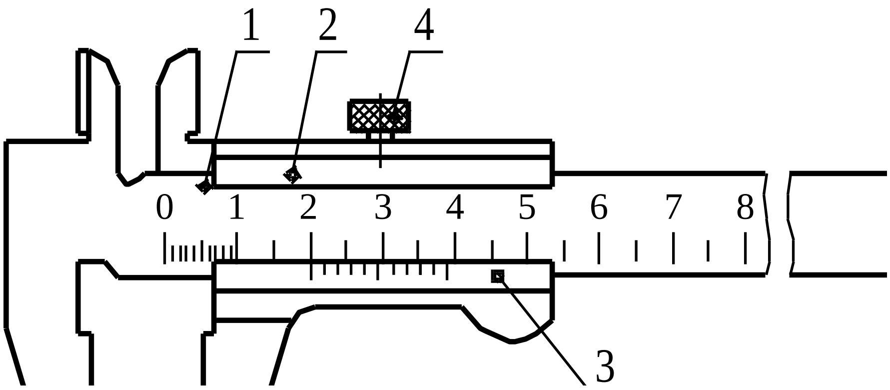


Fig.5.1 Vernier caliper

The vernier depth gauge is used to measure depths, heights, shoulders, etc. As shown in Fig. 5.2 the vernier depth gauge consists of a base (1), bar (2)on which the main scale is engraved, frame (3) with vernier scale (4), clamping screws (5), fine adjustment frame (6), fine adjustment screw (7). When the beam face coincides with the base face, then the reading is zero.

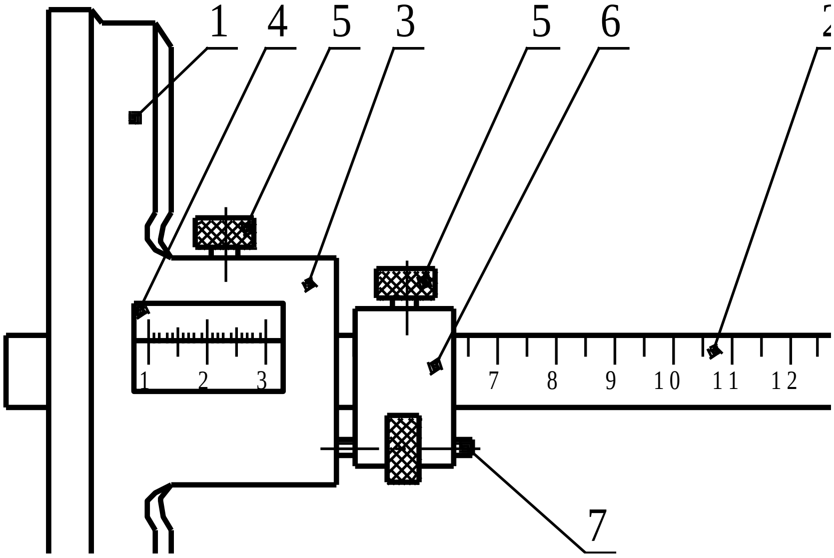


Fig.5.2 Vernier depth gauge

Vernier height gauge is used for measuring various height dimensions and carrying out layout works. It consists of a base block (1) with the vertical beam with the main scale (2) on which the movable frame (3) can slide. The frame carries a holder (4) for clamping various measuring blades (5) or layout blades (6). There are clamping screws (7) for fixing the position of a holder, movable and adjustment frames (8). When a blade tip touches the surface plate on which the vernier height gauge is positioned, the reading is zero.

The important characteristics of any vernier instrument are the measurement range and the accuracy of readings or resolution. The measurement range is the range of length values that a measuring instrument can measure. The measurement ranges for vernier instruments are 0-125 mm, 0-200 mm, 0-300 mm, 0-500 mm etc. By resolution the minimum graduation of the scale is meant. Common resolution values for vernier instruments are: 0,1 mm; 0,05 mm and 0,02 mm.

All vernier instruments have two scales: the main one and vernier. In order to read the measured dimension value these two scales are used. To define a dimension two readings should be made. The number of whole millimeters is read on the main scale as the value of the mark which is the nearest on the left of the vernier scale zero index. The vernier reading is defined in the following way. Firstly, identify which mark of the vernier scale coincide or match with any mark on the beam. To calculate the fractions of millimeter multiply the mark number by the vernier scale factor. The total reading is the sum of the main scale and vernier scale readings. Examples of various readings are shown in Fig. 5.4.

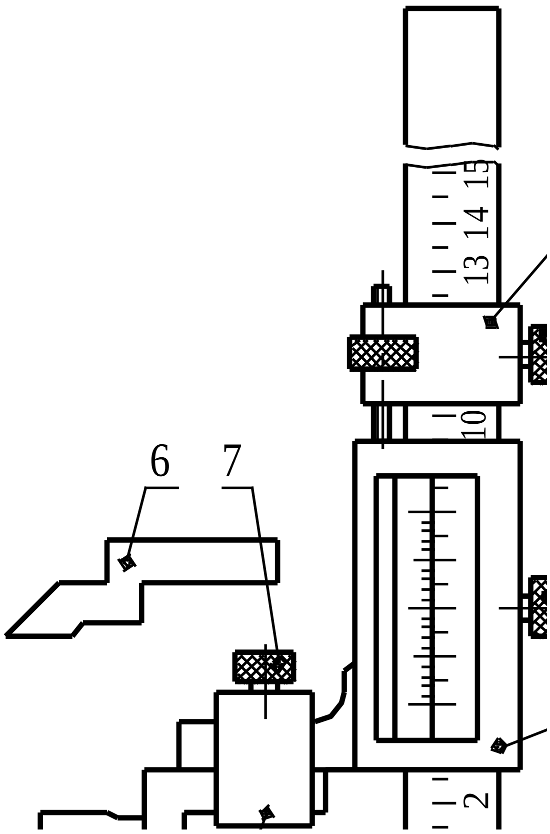


Fig.5.3 Vernier height gauge

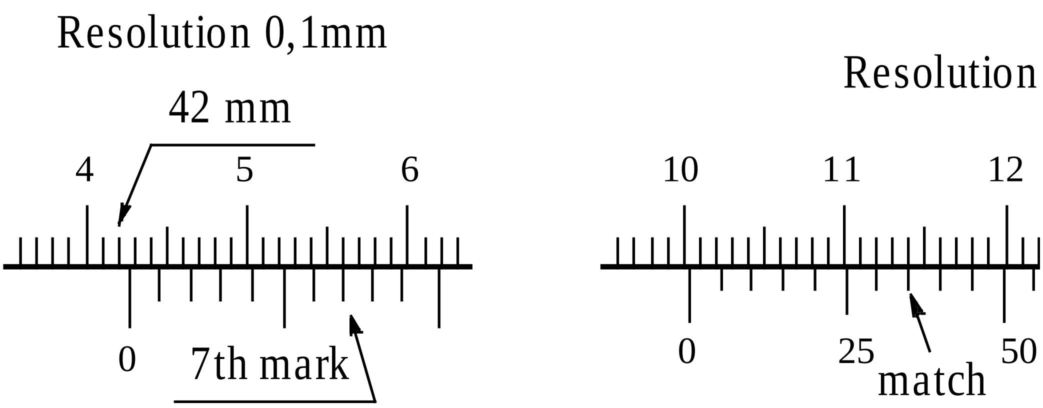


Fig.5.4 Vernier caliper readings

The principle of vernier measurement is based on one observation, which states that for a human eye it is easier to find two matching lines than to define the correct distance between them. So the marks on the vernier scale have a slightly smaller distance between them, compared to the main scale marks distance. The difference between these two distances is called **accuracy of readings** and represents resolution or the vernier scale factor, which is designated as ***c***. Another parameter used for any scale description is **graduation spacing** which specifies the distance between the adjacent graduations or marks on the scale. It is designated as ***a*** for the main scale and as ***b*** for the vernier scale. As vernier scale marks are shifted relatively to the marks on the main scale then the spacing parameter for the vernier scale is obtained from the equation:

$$b = \gamma \times a - c \tag{5.1}$$

where γ is the number of the main scale marks, with respect to which every vernier mark is shifted. Usually the γ number is taken equal to 1; 2; 4. For the vernier scales shown in Fig. 5.4 γ=2 and for the Fig. 5.5 γ=1.

The number of vernier scale marks is defined as:

$$n = a / c \tag{5.2}$$

And the length of the vernier scale is:

$$l = n \times b = (\gamma \times n - 1) \times a \tag{5.3}$$

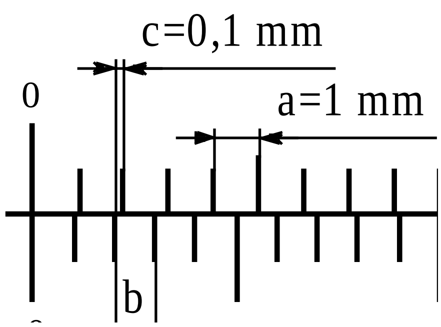


Fig.5.5 Vernier scale

Assignment:

1. Measure specified diameters and length of the component.
2. Make a sketch of the component and denote actual dimensions.
3. Denote type of the vernier calliper and its resolution value.

5.3.2 Measurements with Micrometers

The objectives of the laboratory work are as follows:

1. To study the construction and measurement principle of a micrometer.
2. To measure the given component with a micrometer.
3. To sketch a component with the actual dimensions denoted.

Various micrometers are used for linear two-point measurements of a higher degree of accuracy than it is possible with a vernier caliper. The micrometers use the principle of a screw and nut to amplify the small travels of a screw into visible rotations of a nut. When a screw is rotated for the full revolution, it advances by one pitch relatively to the stationary nut.

According to GOST 6507-78 there are:

1. **Outside micrometers** used for external dimensions (refer to Fig. 5.6).
2. **Inside micrometers** used for internal dimensions from 50 to 6000 mm (shown in Fig. 5.7 b).
3. **Depth micrometers** used for the same purposes as vernier depth gauges. The measuring range of the depth micrometers can be changed in 25 mm steps by installing measuring rods. Scales of the depth micrometer are reversed when compared with other micrometer types (Fig. 5.7 a).
4. **Lever-type micrometers** are precision micrometers enhanced by the application of levers in addition to the threaded screw and nut. This type has an accuracy of readings equal to 0,002 mm compared to the accuracy of other types, which is always 0,01mm.
5. **Thread micrometers** commonly used for measuring the pitch diameter of an external thread. Micrometers of this type have hollow anvils intended for holding V-shape and cone-shape special inserts.
6. **Micrometers with disc-type anvils** used for measuring the base tangent length ***W*** of a gear.
7. **Micrometers with a spherical anvil** intended for measuring the pipe wall thickness.
8. **Digital micrometers**

Although micrometer operation is based on a simple principle, the construction of a micrometer is rather complex, with all the elements aimed at raising the accuracy of measurements. The construction of an outside micrometer is shown in Fig. 5.6.

The base element of a micrometer is a U-shaped frame (1), in which the parts of a micrometer are installed. The U-form of a frame ensures high rigidity, necessary for accurate measurements, since frame deformations under measurement pressure will be minimized. In order to eliminate thermal deformations of the frame, heat-insulating coatings or covers are usually applied.

Measuring faces are presented by a stationary **anvil** (2) face and movable **spindle** (4) face. Anvil and spindle are usually carbide tipped with planar-parallel flat faces. Carbide tips ensure precision and long lifetime of a micrometer. The excellent surface finish of the flats is a result of grinding with consequent lapping.

The spindle is threaded with the **main nut** (5) installed in the **barrel** (3), which in turn is pressed into the frame and also provides additional support for the spindle to move evenly through the travel range. The barrel has a main scale with graduation marks 1 mm spaced. These marks are engraved both above and below the horizontal reference line and are shifted by 0,5 mm with respect to each other in order to make the 0,5 mm readings easier.

The main nut is actually split like a collet and is used with the **adjusting nut** (6) which has tapered contact with the main nut and can tighten the latter in order to eliminate the backlash between the nut and screw. Thus the change in the spindle motion will not affect the accuracy of readings.

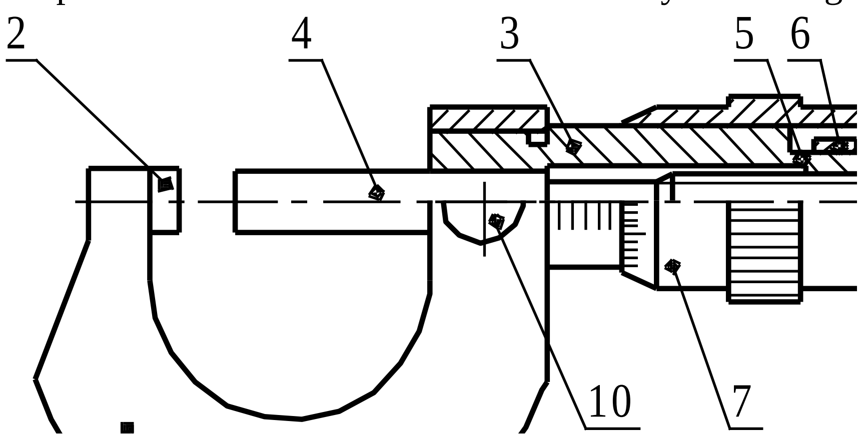


Fig.5.6 Micrometer construction

For size measurements the spindle carries a tubular jacket called **thimble** (7) fastened to the screw with the **thimble adjusting nut** (8). Since the thread used in micrometer has a pitch of 0,5 mm, the thimble will travel 0,5 mm for a complete revolution of the screw. The conical part of the thimble circumference is subdivided into 50 graduation marks, thus when the thimble is rotated for one division, the spindle travels 0,01 mm.

The thimble adjustment nut is used to control the position of the thimble with respect to the spindle and thus to correct the **zero error reading**. That is more convenient compared to the vernier caliper, as there is no need to re-grind the faces.

Since the thread can incorporate pitch errors, which increase with the increase of the screw length, all micrometers are divided into small measurement ranges of 25 mm, and hence the frame gap is also limited to approximately the same dimension. Large micrometers range can be changed with help of replaceable anvils.

The thimble adjusting nut has inbuilt **ratchet** (9) which is used to provide constant measuring pressure needed for accurate measurements and for keeping the screw and measuring surfaces from damage. When the applied pressure overcomes the nominated value, the ratchet slips with the clicking sound.

To fix the measured size for more accurate readings the spindle position can be locked with the help of locking devices, which can be screws, cam clamps or **locking nuts** (10).

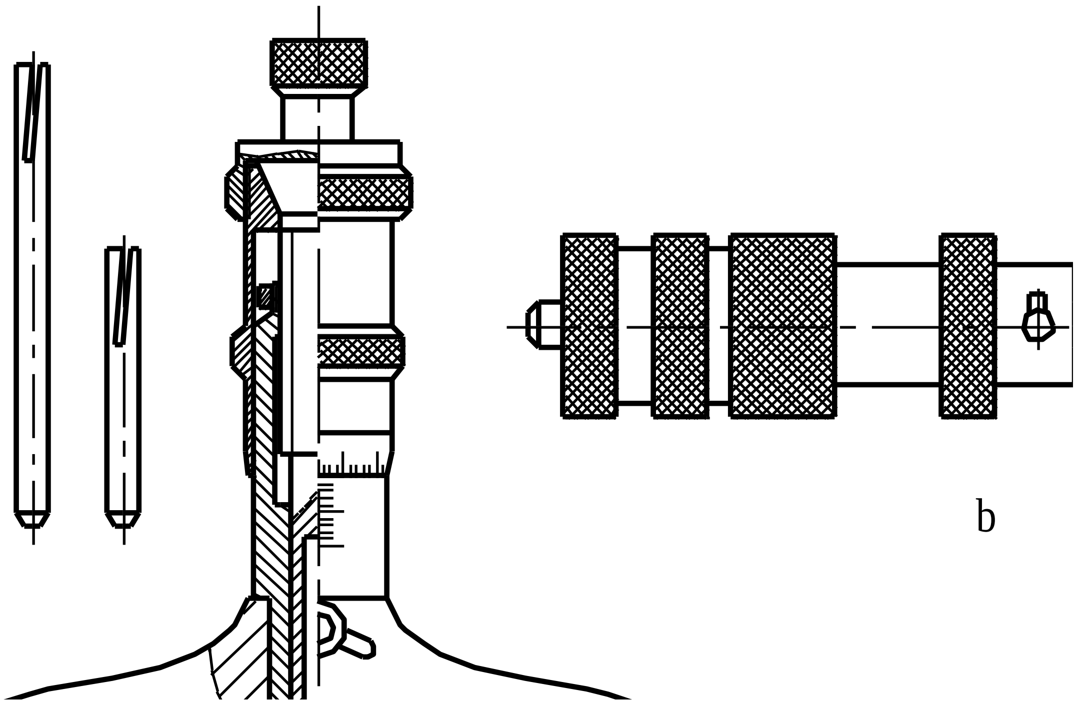


Fig.5.7 Micrometers: (a) depth micrometer and (b) internal micrometer

Measurements are performed with the help of the barrel and thimble scales. The thimble edge acts as an index mark for halves of a millimeter count on the main scale. In order to read a micrometer, check the value of a graduation mark which is the nearest to the thimble edge. Here, the lower scale is used to count whole millimeters and upper scale is used to count one-half of a millimeter if this mark is closer to the thimble. Then it is needed to add the value of the thimble mark which is in line with the barrel reference line. Several reading examples are shown in Fig. 5.8.

For the reading shown in Fig. 5.8 (a) the main scale reading is 6 mm while the thimble reading is 0; therefore the whole reading is 6,00 mm. For the example shown in (b) the last mark visible to the left of the thimble edge is 3 mm and the thimble graduation mark, which coincides with the reference line, is 0,46 mm, then the total reading is 3,46 mm. For example (c) the nearest graduation to the thimble on the lower scale is 8 mm. And there is an additional line visible on the upper scale mark, therefore 0,5 mm should be added to the reading. The thimble reading is 0,23 mm. Consequently, the reading is 8+0,5+023=8,73 mm. The reading for the example (d) is 17,98 mm.

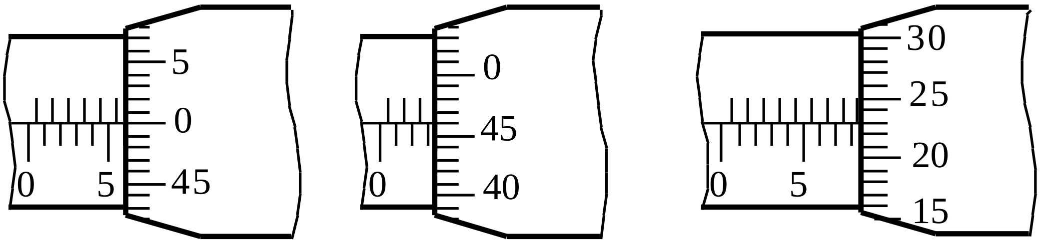


Fig.5.8 Micrometers readings

Although micrometers are easy to use and read, the following disadvantages should be mentioned:

1. Limited range of measurements, it is only 25 mm.
2. Since the range is narrow, a set of micrometers is needed to cover the whole range of practical dimensions.
3. Different types of micrometers are required for external and internal measurements.

Assignment:

1. Check zero reading of the given micrometer. Make adjustment if needed.
2. Measure specified diameters of the component.
3. Make a sketch of the component and denote actual dimensions.
4. Compare actual dimensions with corresponding limits of sizes.
5. Decide on applicability of the component.

5.3.3 Application of Dial Indicators

The objectives of the laboratory work are as follows:

1. To study the construction, measurement principles and characteristics of dial indicators and gauges.
2. To measure the given components with a dial bore gauge and dial snap gauge. Decide about the applicability of the components.

Dial indicators represent a group of linear displacement measuring instruments with a circular scale, called dial. Dial indicators provide more accurate measurements compared to vernier calipers or micrometers; and are used to measure smaller dimensions by the comparative method. The absolute method can also be used, but only for dimensions smaller than the measuring range of a given indicator.

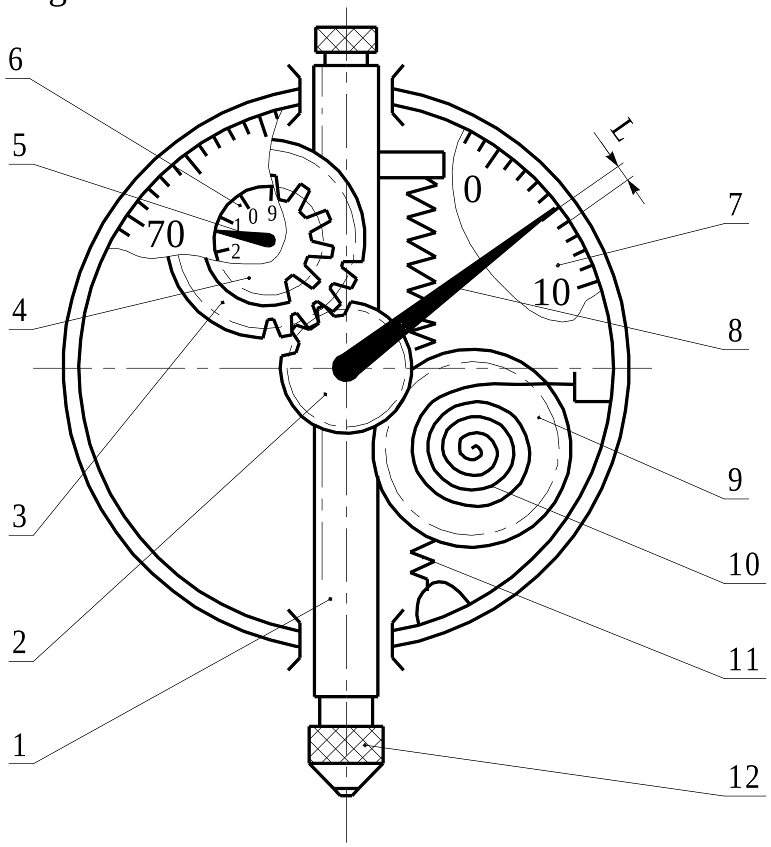


Fig.5.9 Dial indicator

Advantages of dial indicators are as follows:

1. Dial indicators are easy to read.
2. Constant measurement pressure.
3. Compact dimensions with a relatively long measuring range.
4. High resolution for precision measurements.

5. Temperature variations and wear affect the measurements accuracy to a lesser degree as compared to vernier calipers or micrometers.
6. With the help of auxiliary devices the dial indicators can be used in numerous applications.

Among various types of dial indicators, such as dial indicators, test type indicators, mikrokators, mikators, high-sensitivity mechanical indicators etc., the dial indicator shown in Fig. 5.9 is the most common.

Dial indicators used for linear measurements work on the principle of two-point measurements, one point is stationary and the other one is moving. The removable contact point (12) of a dial indicator is the moving point and a surface plate, as it is shown in Fig, 5,10a for example, is the other, i.e. stationary one. Under the measurement conditions the measuring spindle (1) slides in guiding bushing that pressed into the stern; and meshes with the pinion (4) by the rack cut on its periphery. This pinion carries the small pointer or hand (5) for counting whole millimeters and the gear (3), which meshes with the pinion (2). This pinion, in its turn, carries the long hand (8) for counting hundredths of a millimeter on the dial (7).

When the measuring spindle travels 1 mm, the long hand makes one complete revolution over the dial and the small hand indicates this revolution by shifting by one graduation mark.

Since the gear teeth mesh with clearance, the backlash eliminating devices, such as the hairspring (10), are incorporated in the dial indicator design to improve the accuracy of measurements. The hairspring loaded gear (9) meshes with the pinion (2) and provides single-tooth contact.

To ensure constant measurement pressure the spring (11), connected to the spindle (1) and dial housing, is used.

The gear mechanism imperfections results in cumulative error related to the spindle travel length. Hence the accuracy of measurements can be raised if the dial indicator is applied for measuring smaller displacements. Thus, for example, the measurement error for the 2 mm range is up to 15 μm and for the 10 mm range it is up to 22 μm.

In order to fulfil length measurements, the dial indicator should be used with various auxiliary devices. The most common setups for run-out inspection, include stands and columns (refer to Fig. 5.10). Fig. 5.10 can also be used to illustrate the absolute and comparative methods of the dial indicator application.

The dial indicator (1) is mounted on the stand (2) by its stern (3) and with the point (4) touching the table (5). To eliminate loose contact between the point and table the dial should be a little bit lowered to the table, so that the spindle is slightly compressed, say for 2,5 mm. Afterwards the outer frame or bezel (6) is rotated to place the zero index on the dial in line with the long pointer and secured with a clamp (8). Remember the position of the small hand. Then the spindle is retracted from the table surface and the object (7) is placed under the point position (Fig. 5.10 b). When the point touched the object face, the reading, for example 7,45 mm, is made. The height (h) of the object is 7,45-2,5=4,95 mm in this case. That is the way the absolute measurements are performed.

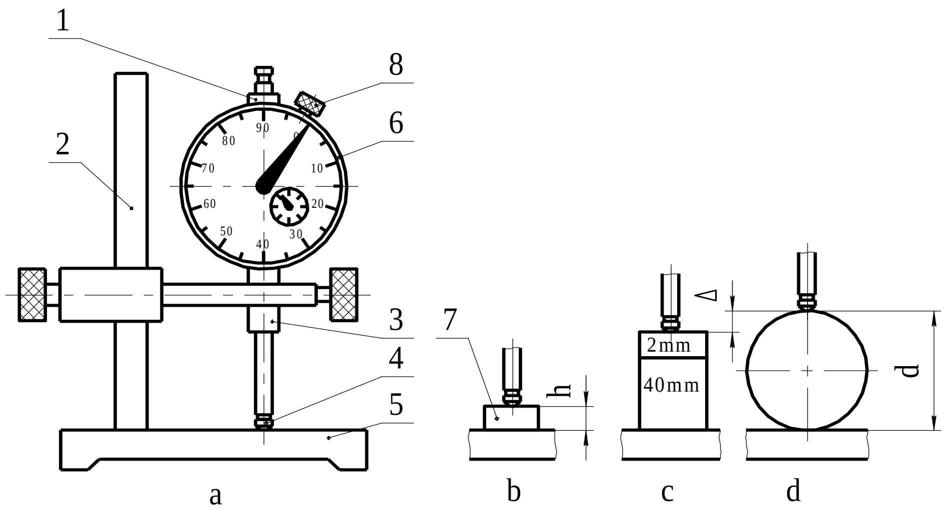


Fig.5.10 Absolute and comparative measurements

The comparative method is used when the size of an object exceeds the measurement range of the dial; or when the more accurate measurements should be made, and the cumulative error can be reduced by limiting the spindle travel. The comparative method is illustrated in Fig. 5.10 c and d.

In this case the dimension of the object is roughly determined with the help of a vernier caliper or a micrometer. Then the gauge block stack of the defined size is built and placed on the table for setting the dial indicator pointer to zero. When the position of pointers is remembered the gauge block stack is replaced by the object and the reading of the dial indicator is checked. The actual dimension of the object is calculated as the sum or difference of the gauge block size and the displacement (Δ) of the spindle relatively to the gauge block size. If the spindle shifted upward then the Δ is positive; of the spindle shifted downward then the Δ is negative.

For linear measurements the dial indicator and auxiliary device are usually made as a single tool. This is called an indicator gauge and is available in two basic types: dial snap gauge for external measurements (Fig. 5.11) and dial bore gauge for internal measurements (Fig. 5.12).

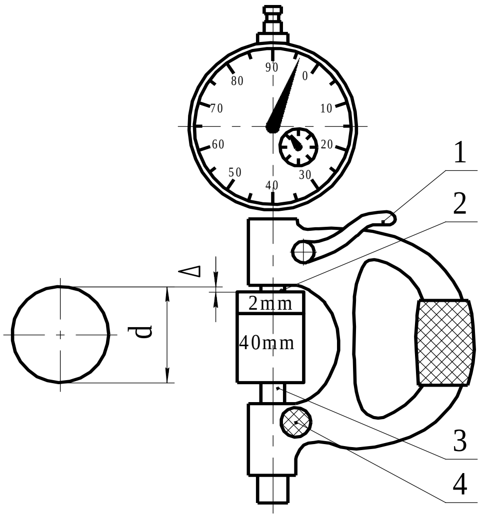


Fig.5.11 Use of a dial snap gauge

The snap gauge frame holds the dial indicator connected to the movable anvil (2) and the stationary replaceable anvil (3), which is clamped with the screw (4) and can be used to adjust the measurement ranges. The movable anvil is spring loaded to maintain constant measurement pressure and can be retracted with the help of the lever (1). The measurement procedure follows the same steps described for a comparative measurement with a dial indicator held in stand.

Dial bore gauges are used for comparative measurements of internal dimensions. A common construction of such a dial bore gauge is shown in Fig. 5.12.

Two-point measurements are fulfilled with the help of two anvils held in the housing (4). One is a stationary replaceable anvil (8) and the other one is a movable anvil (6) placed on the opposite side of the housing and acting on the double-arm lever (5) fixed on the pivot (3). The rod (2) placed inside the handle, transmits the displacements of the movable anvil to the dial indicator spindle. The gauge spring (1) and the dial indicator spring ensure the constant measurement pressure applied to the movable anvil via the rod. The anvil (8) is replaced with the other anvil with respect to the size of the hole and is secured with the locknut (7). Since aligning of a dial bore gauge inside the hole is more complex compared to a snap gauge, bore gauges are usually equipped with special centering devices, such as the centering shoe (9) equipped with two springs (10) for adequate contact pressure.

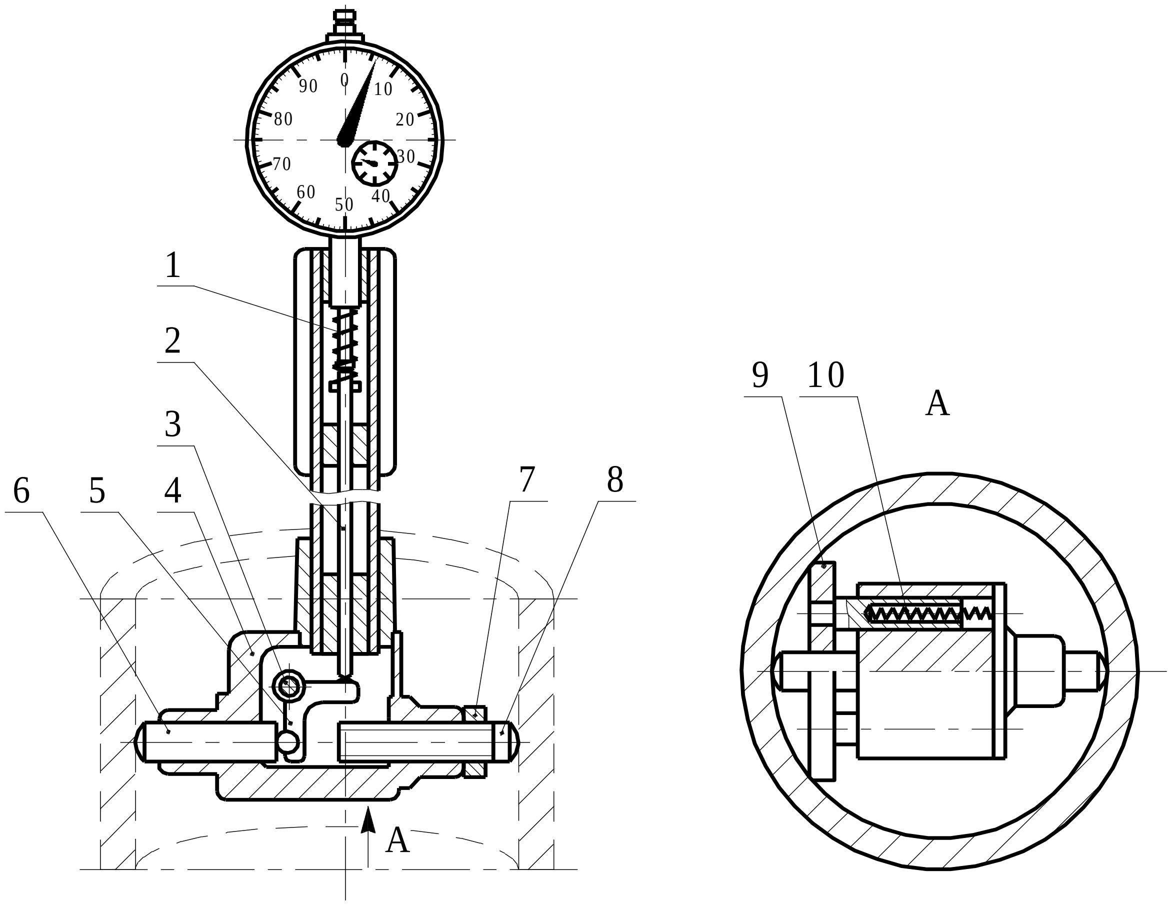


Fig.5.12 Dial bore gauge

Setting of a dial bore gauge (Fig. 5.13) to the required nominal size is usually performed with the help of a gauge block stack held with two beams (2) in the adjusting clamp (1). The setup procedure is as follows:

1. Determine approximately the hole diameter with the help of a vernier caliper.
2. Build a gauge block stack of the defined size and clamp it between two beams in the adjusting clamp to set the dial.
3. Install the appropriate extension anvil in the dial gauge and lock it securely with the nut. The anvil should be long enough to contact the beam and slightly compress the opposite anvil.
4. Insert the dial bore gauge between the beams and align it perpendicular to the beam faces. To find the perpendicular position, gently rock the indicator gauge back-and-fore in two perpendicular directions so that to find the minimum indication on the dial.
5. Rotate the bezel to align the zero index with the pointer. Secure its position with a clamp. The dial bore gauge is ready for measurements relative to the setting.

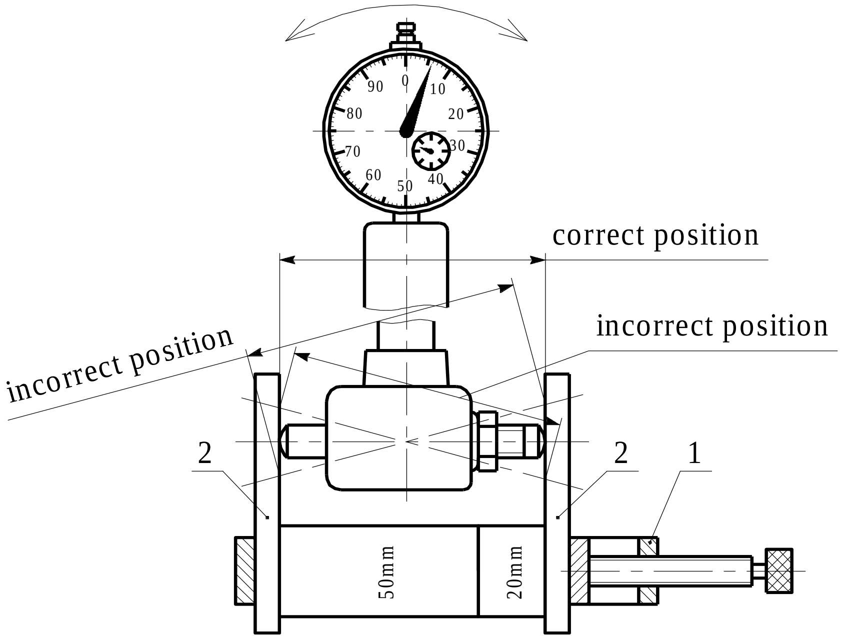


Fig.5.13 Dial bore gauge setup

The measurements are performed the same way. The dial gauge is gently inserted into the bore. The centering shoe aligns the dial gauge in diametric plane of the bore and the aligning in the axis plane is performed by vertical rocking the gauge. As the long pointer will swing clockwise and counterclockwise with the gauge position changed, the reading should be performed at the moment when the pointer changes the direction of rotation. The reading can be positive or negative, depending on the deviation of the actual bore size relative to the nominal size of the gauge block stack. The actual dimension of the bore is the sum or difference of the gauge block stack size and the displacement read by dial.

Assignment:

1. Set a dial snap gauge for measurements of specified external diameter of the shaft.
2. Carry out the measurements of the diameter in several planes along the component length. Make a sketch of the component and denote actual dimension.
3. Set a dial bore gauge for measuring specified diameter of the hole.
4. Carry out the measurements of the hole diameter in several planes along the hole length. Make a sketch of the component and denote actual dimension.

5.3.4 Inspection of a Limit Gauge

The objectives of the laboratory work are as follows:

1. To study the construction, measurement principles and characteristics of twisted stripe type indicators (GOST 6933-81).
2. To measure the given limit plug gauge with a mikrokator held in the C-1 stand.
3. Decide about the applicability of the plug gauge.

Mikrokators are precision measurement instruments with mechanical system of displacement amplification.

Mikrokators are commonly applied for comparative measurements of the 2-6 IT grades dimensions. To convert linear displacement of a spindle into pointer swinging, a twisted bronze stripe is used. Fig 5.14a illustrates a mikrokator diagram. The thin bronze strip (2) is fixed to the spring knee (1) and flat cantilever (4). The magnitude of the tensile force applied along the axis of the twisted bronze stripe is controlled via adjusting screws. The actuating spindle (7) is suspended by the two diaphragms (6) and is fixed to the spring knee (1). Displacement of the measuring spindle results in swinging of the spring knee and consequent stretching of the spring (2). Measurement pressure is produced by the conical spring (5). Stretching of the spring (2) rotates the quartz pointer (3) fixed to the middle of the twisted spring.

Assignment:

For comparative measurements mikrokators are mounted on stands (refer to Fig. 5.14 b) with a measurement table. The setting of a mikrokator to the required nominal size is performed with the help of a gauge block stack.

The laboratory work is carried out in the following sequence.

1. Read the specifications for the given plug gauge. With the help of relevant GOSTs define the values for the tolerance and fundamental deviation of a hole. Define the plug gauge parameters: H, z, y and α.
2. Draw a diagram of the hole and plug gauge tolerance zones.
3. Calculate the limits of size for the plug gauge. Denote them on the diagram.
4. Choose one of the plug gauge limits of size for setting the mikrokator for measuring the actual size of the plug gauge. Actually for easier gauge block stacking up, dimension a slightly different from the limits of size can be used.
5. Build a gauge block stack of a definite size.

6. Place the gauge block stack on the stand table under the spindle measurement point.
7. Set the mikrokator to the required size. For this purpose unlock the fixing screw (9) and by rotation of the nut (10) adjust the position of the arm (8). Then fix the screw (9). For fine tuning of the table position unlock the screw (2) and adjust the position of the table (3) with the nut (1). When the spindle point touches the gauge block secure the screw (2). The screw (7) can additionally adjust the dial scale position relative to the pointer position within the ±5 graduations.

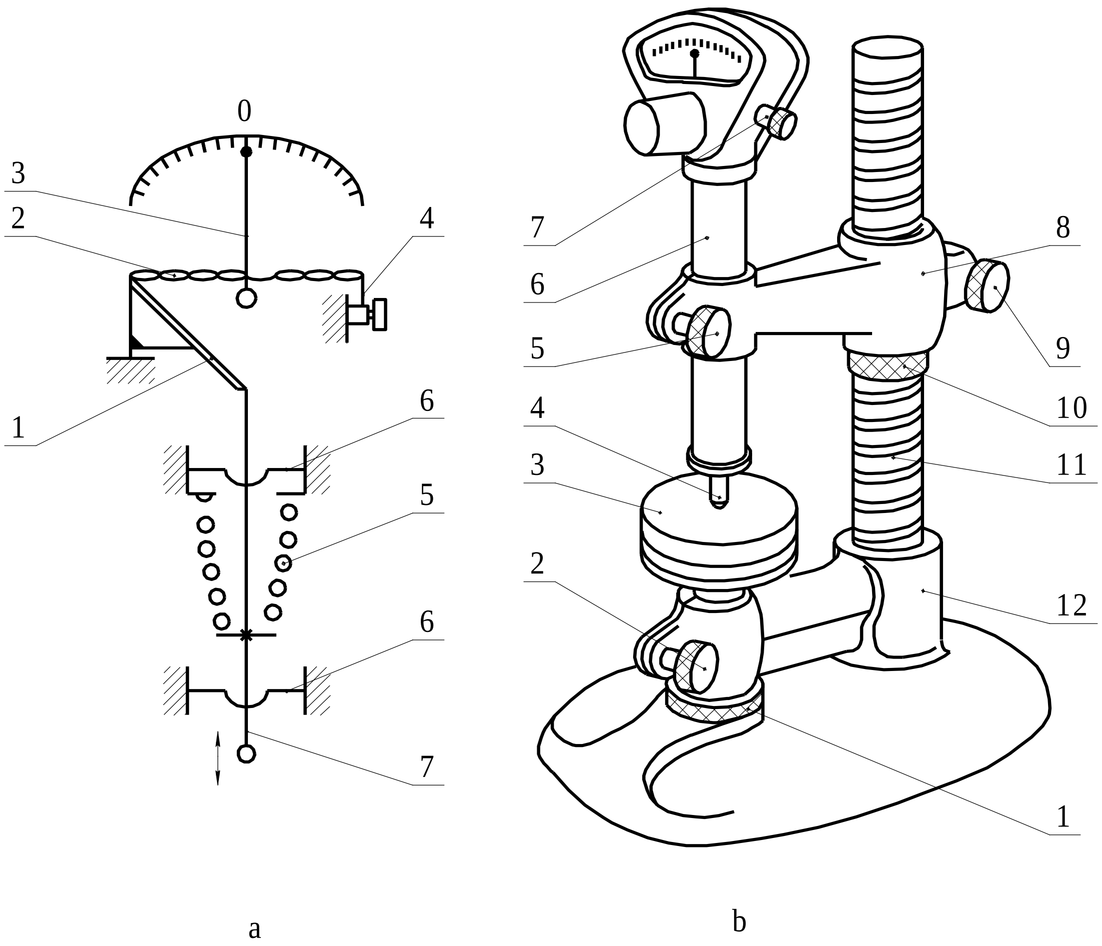


Fig.5.14 Mikrokator

8. Replace the gauge block stack with the plug gauge. The gauge block stack should be kept for zero reading check. Then gently roll the gauge under the measuring point and read the dial. The maximum deflection of the pointer to the positive or negative value represents the deviation of the actual dimension from the gauge block size. Repeat the measurement several times for repeatability. The measurements

are performed in three sections along the plug gauge axis and in two perpendicular planes (Fig. 5.15).

9. Calculate the actual sizes of the plug gauge in the specified sections by adding or subtracting the dial reading from the nomanl size of the gauge block stack. Fill in the table with the measurements results and calculations.
10. Check the zero reading of the mikrokator with the same gauge block stack. In case the deflection from zero is bigger than a half of the graduation distance, the mikrokator setup adjustment should be made. And then all the measurements should be repeated.
11. Decide about the applicability of the given plug gauge.

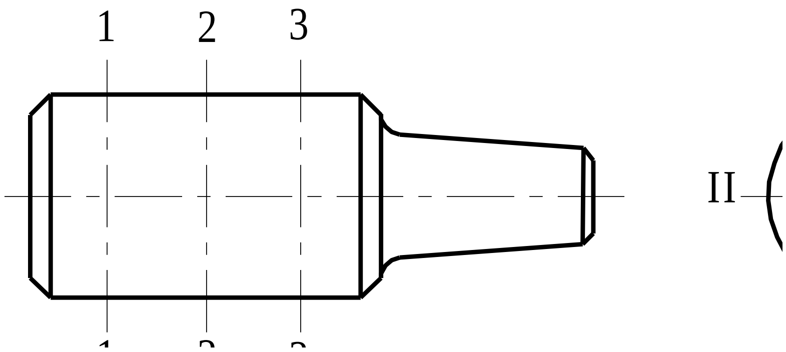


Fig.5.15 Scheme of measurements

Measurement results Table 1

Size of the gauge block stack, mm			Go gauge			Not-go gauge		
			Sections			Sections		
			1	2	3	1	2	3
Mikrokator readings, µm	Plane	I-I						
		II-II						
Actual dimensions, mm	Plane	I-I						
		II-II						

5.3.5 Roughness measurements

The objectives of the laboratory work are as follows:

1. To study the measurement principles and characteristics of the roughness measuring instruments and machines.
2. To measure the roughness parameters of the given sample with a profilometer.

The methods used for roughness measurements can be divided into two groups: qualitative and quantitative.

The qualitative method implies the use of fingers and eyes to compare surface roughness of an object to reference specimens with various surface finish. A comparison is performed by dragging a fingertip over the surfaces of the object and specimen. When the feel is the same, then the object roughness is equal to the specimen roughness value. This method is inexpensive, straightforward, but at the same time it is the least accurate and subjective. Each roughness specimen is produced according to GOST 9378-75 and has indications of the machining process type and roughness parameter Ra in μm.

Quantitative measurements are performed by either contact or non-contact method.

Non-contact methods are usually applied for an area, rather than a single section of the inspected surface and include the following techniques: optical interferometry, microscopy, electrical capacitance, electron microscopy, and stereo SEM imaging. Since there is no mechanical contact between the inspected surface and instrument, the surface is not damaged. Thus the methods can be used to inspect soft materials.

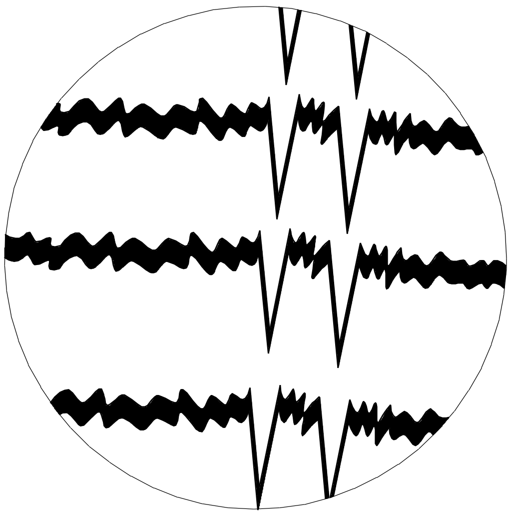


Fig.5.16 Interference image of a scratch

The optical interferometry method implies application of a flat polished glass surface which comes in contact with the object surface. Hence the roughness interferometers can be used only for measuring small irregularities, as it is impossible to get the optical interference on the rough surfaces. The magnified irregularities of the interference image (Fig. 5.16) are measured with the help of reference lines present on the microscopic eyepiece glass.

Another optical instrument involves the use of two microscopes positioned at a right angle to each other and at an angle of 45° to the surface (Fig. 5.17). The first microscope is used to illuminated the surface profile; and the second is used for the observation of the illuminated profile.

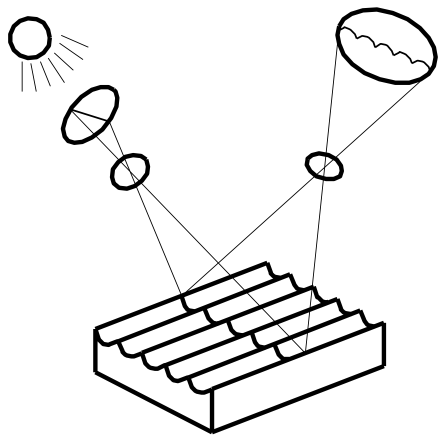


Fig.5.17 Scheme of Linnik microscope

Contact methods imply moving a measurement stylus, usually a diamond, across the surface perpendicular to the lay. The stylus is set vertically and dragged horizontally. Vertical movements are amplified and either recorded on a paper or directly processed by the machine into roughness parameters. The instruments used are called profilometers, since the texture pattern is recorded in a form of a 2D profile.

With a set of parallel profiles scanned by a profilometer over the surface, 3D topology of the surface can be obtained.

Profilometry is usually used in laboratories or for short run applications.

Assignment:

1. Sketch a surface profile in form of a profilogram.
2. Denote roughness parameters R_a and R_z on the profilogram.
3. Calculate values of the parameters R_a and R_z for the given profile.
4. Give designation of the measuring machine that was used for inspection of the surface profile.
5. Give designation of the calculated parameters R_a and R_z.

6. Standardisation

As standardisation principles were implicitly presented in the previous sections, the following discussion will concern general aspects of standardisation.

Since it is difficult to ensure accurate manufacture and measurements without standards, standardisation plays a significant role in manufacturing as well as in the worldwide economy. First steps in standardisation were taken more than 8000 years ago, and almost no activity is left unstandardised to date.

Standardisation is the activity of establishing norms, rules and specifications available for application and aimed at ensuring compatibility, interchangeability and coordination in the fields of production and products consumption activities.

The objectives of standardisation are as follows:

- Health and safety
- Consumer protection
- Products compatibility, interchangeability and interoperability
- Product competitiveness
- Repeatability of research, measurement and statistical analysis results
- Product and service quality
- Uniformity and traceability of measurements
- Resources saving production

To meet the objectives of standardization the following tasks are usually specified:

- Improvement of communication between designers, manufacturers and consumers.
- Establishment of optimal requirements to products range and quality.
- Establishment of requirements on compatibility and interchangeability of products.
- Harmonization of product characteristics.
- Provision of standard references and methods of inspection.
- Specification of requirements on manufacturing processes for resources and material consumption.
- Codification and classification of technical information.

Standardisation implies the use of special documents, such as standards, normative documents and regulations, which can be obligatory or voluntary and anyway act in a capacity of reference. These documents can be of enterprise, regional, national or international type. Among national standardisation bodies there are: BSI, AFNOR, DIN, JISC, SJS, NSF, NIST etc. International standardisation organisations include: ISO, IEC, ITU, EOQ, CEN, W3C etc.

Standardisation activity in Russian Federation is led by Federal Agency on Technical Regulating and Metrology (FATRM). FATRM sets up special technical committees that involve academic institutions and private sector to create specific standards. Apart from the technical committees, FATRM distributes its functions among FATRM Research Institutes and Centers for Standardisation and Metrology, which are positioned in economic centers of Russian Federation.

Standardisation develops along the engineering progress and forms a basis for current and future production. To keep pace with rapid technological advances the following principles of standardization are defined:

- Balanced interests of parties
- Principle of system optimization
- Advanced standardisation
- Flexible standardization
- Explicitness of standards
- Principle of harmonization
- Performance standardization
- Standardization in priority directions

In general the following methods are used for standardization: simplification, systematization, codification, unification, parametric standardization, modular standardization, and typification.

In terms of international co-operation the Russian National Standards Body (FATRM) defines its line of development as gradual convergence and collaboration with international standards organizations. Hence the conformity of Russian and International Standards is a task of vital importance, especially for advanced standards or standards with higher requirements.

7. Certification

The term “certification” was firstly introduced by ISO committee in 1982. According to this document the **certification of conformity** is the confirmation by a conformance certificate or a conformance mark that an object complies with the corresponding standard. This definition was used as a basis for a corresponding term for certification that is used in the certification system of the Russian Federation, which is defined as follows: **certification of conformity** is the third-party actions that assure compliance of a properly identified product, service or process with requirements of the corresponding standard or normative document. As compared to the definition formulated in 1982, several distinctive modifications can be found in the latter definition.

Firstly, certification is carried out by the independent third-party.

Secondly, certification involves assessment of a “properly” identified product, which implies application of strict certification system with its rules and procedures for certification.

Thirdly, the field of certification is extended to products, services and processes including quality management.

The certification quality is assured by its trustworthiness and impartiality, which are implemented in a “quality loop”. This “quality loop” represents the whole life-cycle of a product (refer to Fig. 7.1).

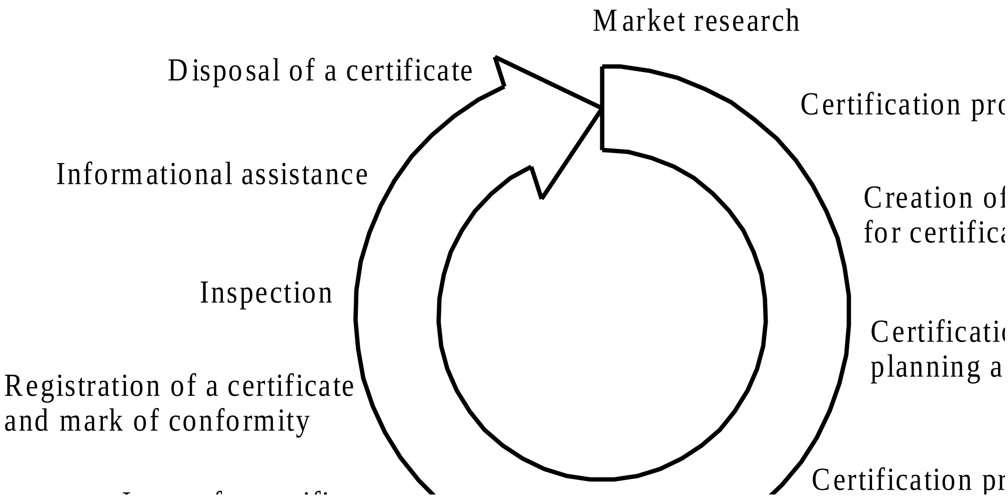


Fig.7.1 Quality loop for certification

As can be seen from this loop, the certification activity starts with market study. At this stage the certification scheme and test laboratories should be chosen correctly in order to eliminate the problems with recognition of the certificate in various countries.

At the stage of design all aspects that influence quality, such as statutory requirements, customer requirements etc., are taken into account.

At the third stage all resources needed for the certification process are collected. These resources include personnel, normative documents, inspection instruments and other equipment required for certification.

The next step is planning the certification processes. Certification processes should be planned in order to make corrections, internal and external audits feasible. At this stage the trustworthiness and impartiality of certification are assured.

The fifth step involves certification of specified product samples and the following agreement about product conformity. The process of confirmation should be performed according to specified procedures and rules of certification.

To guarantee quality of certification it is necessary to perform inspection of the assessment processes. This is usually done by means of internal and external audits. Internal audit is performed by the personnel of certification body or test laboratory. External audit is performed by accreditation bodies or independent experts. Final inspection of certification results is performed at the stage of making decision.

A declared certificate should contain information on the name of the object of certification, the name of the corresponding standard referred to by the object, the name of the certification body, the declaration date, the period of validity and mark of certification body accreditation.

The postcertification period is characterized by inspection of certified objects, keeping the register of certified objects and providing information on certification results to the interested parties.

On expiration of a certificate of conformity the manufacturer of a product can start the certification of the product again and perform all the steps mentioned above.

Certification of conformity can be of obligatory or voluntary nature.

Obligatory certification covers safety related products and services; and is carried out in accordance with the list of products given in the Decree of the Government of the Russian Federation No. 982 of December 1, 2009 and its revisions.

Voluntary certification is applied when there is no need for strict observance of the existing standards or other normative documents requirements.

Conclusion

Up-to-date production is characterized as interchangeable high-performance and precision manufacturing. To ensure safety, quality, interchangeability etc. of products the principles of metrology, standardization and certification are used as a unity in practice, and thus are described in this textbook.

Standard parameters used to describe dimensional, geometrical and roughness deviations of a product are given in the first, second and third sections of this textbook. To check the geometric parameters of the manufactured product the measuring instruments in accordance with metrological principles are used, and are discussed in the fifth section. Tolerance analysis techniques applied to products or manufacturing processes are discussed in the fourth section. Since it is impossible to perform measurements without reference to corresponding standards, the principles of standardisation are described in the sixth section. The seventh section discusses certification principles, which ensure compliance of the objects, say measurement techniques or instruments, with corresponding standards.

References

1. Сергеев А.Г. и др. Метрология, стандартизация и сертификация. – М.: Логос. 2005. 560 с.
2. Крылова Г.Д. Основы стандартизации, сертификации и метрологии. – М.:ЮНИТИ-ДАНА. 1999. 711 с.
3. Марков Н.Н. и др. Нормирование точности в машиностроении. – М.: Высшая школа. 2001. 335 с.
4. Допуски и посадки: Справочник. В 2-х частях / Под ред. В.Д. Мягкова. – Л.: Машиностроение, 1978. 817с.
5. Анухин В.И. Допуски и посадки. – СПБ.: Питер. 2004. 207 с.
6. Димов Ю.В. Метрология, стандартизация и сертификация. Учебник для вузов. 2-е изд. – СПБ.: Питер. 2004. 432 с.
7. Якушев А.И., Воронцов Л.Н., Федотов Н.М. Взаимозаменяемость, стандартизация и технические измерения. – М.: Машиностроение, 1986. 352 с.
8. Червач Ю.Б. Метрология, стандартизация и сертификация. – Томск: Изд-во Томского политехнического университета,2010. – 54с.
9. ANSI Y14.5M – 1994, Dimensioning and Tolerancing.
10. ISO 1101:2004, Geometrical Product Specifications (GPS) – Geometrical tolerancing – Tolerances of form, orientation, location and run-out.
11. ISO 8015:1985, Technical drawings – Fundamental tolerancing principles
12. Drake P. Dimensioning and tolerancing handbook. McGraw-Hill, New York, 1999.
13. Encyclopedia of production and manufacturing management / Editor Paul M. Swamidass. Kluwer Academic Publishers, 2000.
14. Foster L.W. Geometrics III: The application of geometric and tolerancing technique. Addison-Wesley, 1994.
15. Galyer J.F.W., Shotbolt C.R. “Metrology for Engineers”, Cassell, 1969.
16. Greve J.W., Wilson F.W.. “Handbook of Industrial Metrology”, Prentice Hall, 1967.
17. H. Dagnall M.A. Exploring Surface Texture. Rank Taylor Hobson, 1980
18. H. Dagnall M.A. Let’s Talk Roundness. Rank Taylor Hobson, 1976
19. Ibrahim Z. Mastering CAD/CAM. McGraw-Hill, New York, 2005.
20. Jain R. K. Engineering Metrology. Khanna Publishers, New Delhi
21. Kruliowski A. Geometric dimensioning: a self-study course. SME Publications, Effective training Inc., 1992.

22. Manufacturing Engineering and Technology. Fifth edition. Serope Kalpakjian, Steven R. Schmid, 2006
23. Meadows D.J. Geometric dimensioning and tolerancing: application and techniques for use in design, manufacturing and inspection. Marcel Dekker, New York, 1995.
24. Scarr A.J.T. “Metrology and Precision Engineering”, McGraw-Hill, 1967.
25. Wilson B.A. Dimensioning and tolerancing handbook. Genium, New York, 1995.
26. Calculation packet MITCalc // http://www.mitcalc.com/doc/tolerances/help/en/tolerances.htm
27. Electronic Design Library: Fits and Tolerances // http://www.ecs.umass.edu/mie/labs/mda/dlib/fit_tol/fitandtol.html
28. Federal Agency on Technical Regulation and Metrology // http://protect.gost.ru
29. Institute for Geometrical Product Specifications // www.ifGPS.com
30. International Organization for Standardization // www.iso.org/iso/home.html
31. Virtual Machine Shop // http://www.kanabco.com/vms/index.html

Educational Edition

КИМ Алексей Богович

МЕТРОЛОГИЯ, СТАНДАРТИЗАЦИЯ И СЕРТИФИКАЦИЯ

Учебное пособие

На английском языке

Published in author's version

Lingvistic Advisor *Doctor of Science (Philology)*
R.I. Tolbanova

Printed in the TPU Publishing House in full accordance with the quality of the given make up page

Signed for the press 00.00.2009. Format 60x84/16. Paper "Snegurochka".
Print XEROX. Arbitrary printer's sheet 000. Publisher's signature 000.
Order XXX. Size of print run XXX.

ISO 9001 Registered

Tomsk Polytechnic University
Quality management system
of Tomsk Polytechnic University was certified by
NATIONAL QUALITY ASSURANCE on BS EN ISO 9001:2008

UKAS QUALITY MANAGEMENT 015

TPU PUBLISHING HOUSE. 30, Lenina Ave, Tomsk, 634050, Russia
Tel/fax: +7 (3822) 56-35-35, www.tpu.ru