High Level Technical Drawing

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ABBREVIATIONS OF TERMS AND MARKS

	turning	
	developed	
R	radius	m
Ø	diameter	m
	flat taper	%
⊳	conical taper	%
sphere R	sphere radius	m
□	square	
⏤	straightness	
⏥	flatness	
○	roundness	
⌭	circularity	
⌓	profile of surface	
⌒	profile of line	
∠	angularity	
⊥	perpendicularity	
∥	parallelism	
⌯	symmetry	
⌖	position	
◎	concentricity	
↗	runout	
⌰	total runout	
A	area, cross-section	m^2
AS	actual size	m
AD	actual deviation	µm
BC	big clearance	µm
BI	big interference	µm
BL	base line	
BS	basic size	m
E	modulus of elasticity	N/m^2
ES	upper deviation (hole)	µm
EI	lower deviation (hole)	µm
es	upper deviation (shaft)	µm
ei	lower deviation (shaft)	µm
F	force, load	N
i	unit tolerance	µm
I	unit tolerance	µm
IT	international tolerance grade	
l	length	m
LD	lower deviation	µm
LL	low limit	µm
m	mass	kg
MC	mean clearance	µm
MD	mean deviation	µm
MI	mean interference	µm
MS	mean size	m
n	speed	1/min

q	factor of IT grade	
R_{eH}	yield strength	N/mm^2
R_m	tensile strength	N/mm^2
$R_{p0.2}$	yield strength	N/mm^2
SC	small clearance	µm
SI	small interference	µm
T	tolerance	µm
T_i	resultant tolerance	µm
t_1	degree of accuracy: fine	
t_2	degree of accuracy: medium	
t_3	degree of accuracy: coarse	
t_4	degree of accuracy: very coarse	
R_a	surface roughness average	µm
R_z	peak to valley height average	µm
UD	upper deviation	µm
UL	upper limit	µm
σ	strength	N/mm^2
ε	elastic elongation or compression, strain	%

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Objective

A drawing is a graphic presentation of an object, or a part of one, and is the result of the creative thinking of an engineer or technician. All machines and their parts can be presented in the form of a technical drawing and manufactured accordingly. Designing a machine part involves making sketches and technical drawings, and performing the appropriate calculations. Sometimes several alternative designs are presented in drawings, and the best one is selected for the purpose.

Machine drawing is a special "language" of engineers; it is a way of transmitting information as a graphical representation. It allows us to represent machine elements in a simplified and standardised way and to study the constructional build-up of machinery.

Machine drawing enables us to transmit all the information, which is crucial for the process of manufacturing, assembling and understanding the construction. Knowing and applying the standardized methods of presentation is necessary for the machine drawing to be understandable for everyone.

The purpose of writing and composing a book on High Level Technical Drawing is to introduce notions and presentation methods according to the Hungarian Standards.

Furthermore, it makes it possible to acquire the technical terms in connection with the given topic since it is a basic requirement for an engineer to be able to communicate with foreign engineers in English.

1. DRAWING STANDARDS

The engineering drawings are made on sheets of strictly defined sizes that are set forth in the Standard (MSZ ISO 5475). The use of standard sizes ensures convenient storage of drawings. The standard establishes five preferred sizes for drawings as tabulated in Table 1.1.

Table 1.1 Preferred sizes of drawings

Size designation	**11**	**12**	**22**	**24**	**44**
Sheet dimension, mm	297x210	297x420	594x420	594x841	1189x841
Designation of paper sheets	A4	A3	A2	A1	A0

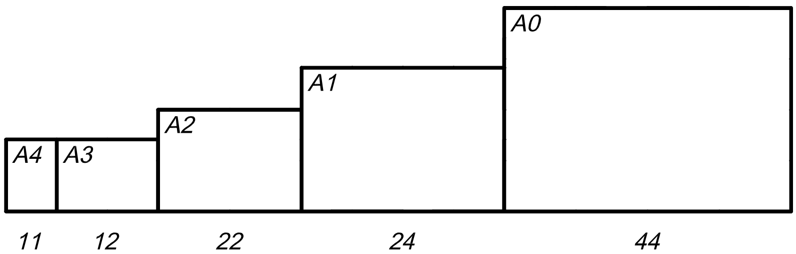


Figure 1.1 Sheet formats

Drawing size is designated by two figures in this case: the first indicates how many times one side of the drawing is longer than 297 mm, the second how many times the other side is longer than 210 mm. The basic size for drawings is S11 measuring 297 mm by 210 mm. All other sizes comprise several S11. Thus, S12 is made up of two S11 with adjoining long sides (Fig.1.1).
Depending on the form of the component other drawing sizes are applicable in order to utilize the paper sheet. Hence, in addition to the above preferred sizes, the standard provides for other sizes, which are also comprised of a whole number of S11.
(E.g.: the designation of the sheet dimension of 297x630 is S13.)

1.1 Title blocks

The title block of a drawing is located in the bottom right-hand corner of the sheet. The title block should be arranged along the short side of the sheet in the case of A4 and along the long side of the sheet in all other sizes. Sheets which are smaller than A1 can be used in the standing position with the title block located in this case along the short side too. Sizes and content of title blocks are standardised (DIN 323; MSZ 1700).
The title block may be changed somewhat to suit special requirements of different enterprises. The spaces of the title block are intended for; the name of the (assembly) drawing, name of enterprise or educational institution, description or title of drawing, dates and signatures, name of the definite part (e.g. "casing"), designation of the given drawing, quantity, material, weight, scale, etc. (Fig. 1.2).

5	*6*	*Nut*	*M12*			
4	*6*	*Bolt*	*M12x60*			
3	*2*	*Bush*	*Ø 35x50*			*MK 01.07.00*
2	*3*	*Gear*	*Ø 75x35*			*MK 01.12.00*
1	*1*	*Shaft*	*Ø 50x200*			*MK 01.01.00*
No.	*Q.ty*	*Name*	*Dimension*	*Materials*	*Designation*	*Document No.*

	Name	*Signature*	*Date*			
Designed				*Company*	*Weight*	*Scale*
Consultant				*Topic* *Gear Box*		
Drawn				*Document No.* *MK 01.00.00*		
Checked				*Sheet No.* *3.*		

Figure 1.2 Title block

In engineering drawing the standard (DIN 15; MSZ ISO 128) specifies different line thickness groups and assigns the thickness of the thick, thin and the medium thickness lines to the given thickness group (Table 1.2). Line thickness depends on the dimension of the part and the complexity, size and purpose of the drawing. In common circumstances thickness group 3a is recommended, namely thick line is 0.7 mm, medium line is 0.5 mm and thin line is 0.25 mm.

Table 1.2 Line thickness groups

Lines			**Line thickness groups**									
Desig-nation	Thickness		1		2		3		4		5	
	a	b	a	b	a	b	a	b	a	b	a	b
Thick	s		0.35		0.5		0.7		1.0		1.0	
Medium	s/2	2s/3	0.18	0.25	0.25	0.35	0.35	0.5	0.5	0.7	0.7	1.0
Thin	s/3	s/2	0.13	0.18	0.18	0.25	0.25	0.35	0.35	0.5	0.5	0.7
Display	2s		0.7		1.0		1.4		2.0		2.0	

The standard specifies various types of lines as well to enable distinction between the lines pertaining to different surfaces.

Type	Sample
Thick continuous	
Thin continuous	
Wavy continuous	
Short-dash	
Long chain (thin)	
Long chain (thin) With two dots	

Figure 1.3 Principal types of lines

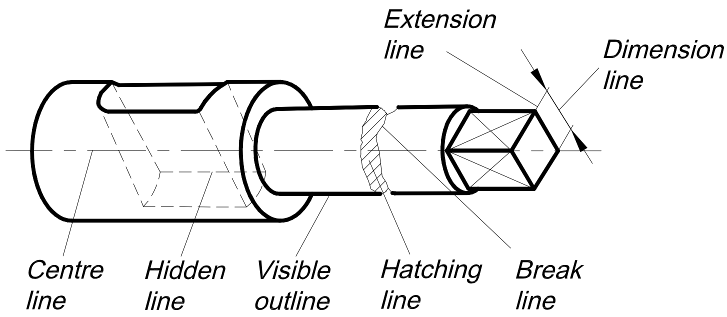


Figure 1.4 Applying different types of line

There are four principal types of lines: continuous (thick, thin and wavy); short-dash; dot-and-dash or chain (thick and thin) (see Fig. 1.3). Typical applications of the above listed types of lines are shown in Fig. 1.4.

Thick continuous lines are used for:

- drawing visible outlines which are 0.7 mm (thickness group 3a). The chosen thickness group has to be equal for all views in a given drawing representation.

Thin continuous lines used for:

- extension and dimension lines which are 0.25 mm
- hatching in sections, the hatching lines are spaced at 2 to 10 mm depending on the area to be cross-hatched
- transition lines between surfaces, gradually merging into each other as in Fig. 1.5 (a and b lines) and 1.6.

Thin wavy lines used for:

- breaks drawn by a free-hand wavy line

Short-dash lines are used for:

- hidden outlines and hidden intersections; the length of a dash ranges from 2 to 8 mm, it is medium thickness (0.35 mm), the spacing between dashes (a) is between 1/2 and 1/4 of their length (see Fig. 1.7).

Thin double-dot-dash lines with long dashes (or long chain lines) are used for:

- axes and centre lines with a thin line (0.25 mm) with dashes approximately 9 mm long spaced at about 3 mm (a1), see Fig. 1.7. Centre of circles should be marked by two intersecting dashes. For circles of diameters smaller than 12 mm the centre lines are usually drawn with thin continuous lines.

Thin dot-and-dash lines with two dots are used to show:

– a surface smoothly merging into another line indicating bending edge. In this case m2=m1 and a2=a1.

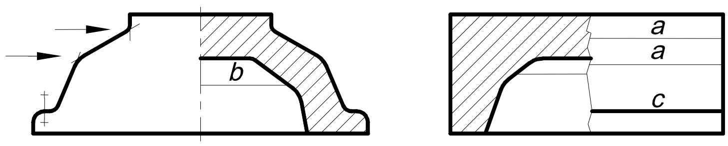


Figure 1.5 Constructing transition lines

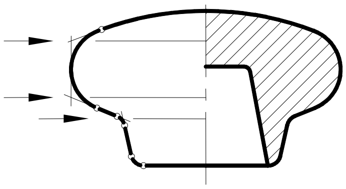


Figure 1.6 Drawing transition lines

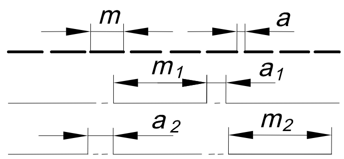


Figure 1.7 Spacing different types of lines

Drawing instruments

There are several computer programs e.g. AutoCAD that make it possible to produce technical drawing in 2 or 3D and that can be saved as a file, and modified and reconstructed without having to draw it again. The aim of attending the Technical Drawing course is to acquire the methods and rules of the simplified and standardized graphical representations of machine elements, components and machine parts and machines. The only way of learning and practicing the fundamental construction principles is to draw by hand.

Accordingly, in a Technical Drawing course the following instruments are used:

- Drawing board: with drawing paper on it, on which a drawing is made.
- Triangles: two types of triangles are used for drawing: 45^{o} -45^{o}, 60^{o}-30^{o}.
- T-square: consists of two parts, a long ruler and a crosspiece.
- Ruler, French curve, protractor, compass, pencil.

1.2 Lettering

The styles of letters and numbers used for engineering drawings are also standardized (DIN 6776; MSZ ISO 3098) (forms and slant of the capital and lowercase letters). Since all mechanical parts are manufactured according to drawings and all information is given by lettering, it has to be made very carefully.

Table 1.3 indicates the recommended heights of letters and numbers, and their line thickness. The following sizes of characters are recommended: 2.5, 3.5, 5, 7, 10, 14 and 20 mm. The size is determined by the height h (in mm) of capital letters. The letters and numbers have an inclination of 75° to the horizontal. In certain cases drawings may be lettered in vertical characters. The shape of the letters and numbers of medium width are shown in the Fig. 1.8.

Table 1.3 Recommended heights of letters

Specification	Designation	Size groups						
Height of capital letters	h	2.5	3.5	5	7	10	14	20
Height of lower-case letter	5h/7	1.75	2.5	3.5	5	7	10	14
Line thickness	h/7	0.3	0.5	0.7	1	1.5	2	3
Minimum line spacing	10h/7	3.5	5	7	10	14	20	28
Reduced line thickness	h/10	0.2	0.3	0.5	0.7	1	1.5	2

Figure 1.8 Vertical and inclined lettering

In machine drawing the 2.5 mm height capital letter is used for tolerances, the 3.5 mm height is used for dimensions, the 5 mm height is used for notice, the 7 mm is used for displaying notice and the 10 mm height is used for item number.

2. TECHNICAL DRAWING

When designing either machines or a component, the first action is often to make freehand pictorial views of the machine or its parts. This process is called technical sketching.

The aim of the technical drawing course is to acquire the methods of sketching machine parts and making and reading the working and assembly drawings. When designing a machine or construction, the following drawings are elaborated:

Schematic representation: a simplified illustration of the machine or of a system, replacing all the elements, by their respective conventional representations.

- Technical sketching: freehand pictorial views of machine or machine parts to render their shape and function.
- Assembly drawing: a drawing that shows the various parts of a machine in their correct working locations. It should furnish the overall dimensions, the linkage dimensions and the dimensions important in terms of operation.
- Shop drawing: pertaining to machine parts or components. It is presented through a number of orthographic views, so that the size and shape of the component is fully understood. It furnishes all the dimensions, tolerances and special finishing processes such as heat treatment, honing, lapping, surface finish, etc.

2.1 Presentation methods in technical drawing

We can imagine an object that is located inside a box and projected orthographically to the corresponding plane of projection. It results in six views that can exist, when all the planes of projection are brought into alignment (see Fig. 2.1).

When drawing a machine part the main view should be selected with respect to its working position (if possible) to reveal its shape and dimensions. Such parts as axles, shafts, bushes, sleeves, tubes, etc., are usually presented in the main view with the axis in the horizontal position, and not necessarily in the working position.

There are several types of views and sections that can reveal the details of a part instead of applying the bottom view, or left hand view and so on, (see chapter 3). The number of applied views and sectional views depends on the complexity of the part and the resources of the drawer. The number of views in an orthographic drawing must be as few as possible but sufficient to provide enough information on the shape and dimensions of the object.

2.1.1 Views

In technical drawings orthographic projection is used in conformity with the standards (DIN 5; MSZ 1701) when constructing views and various sections. The projection of the visible portion of the surface of an object facing the viewer is called a view.

The six views obtained of the above mentioned planes of projection are called:

1 front or main view
2 top view
3 left side view
4 right side view
5 bottom view
6 rear view

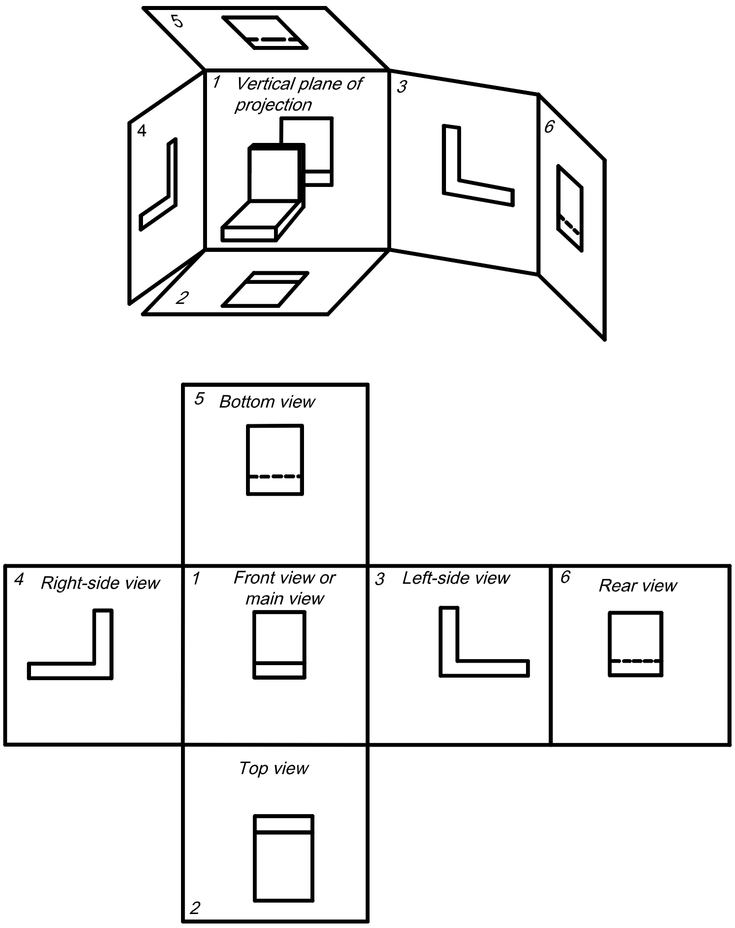


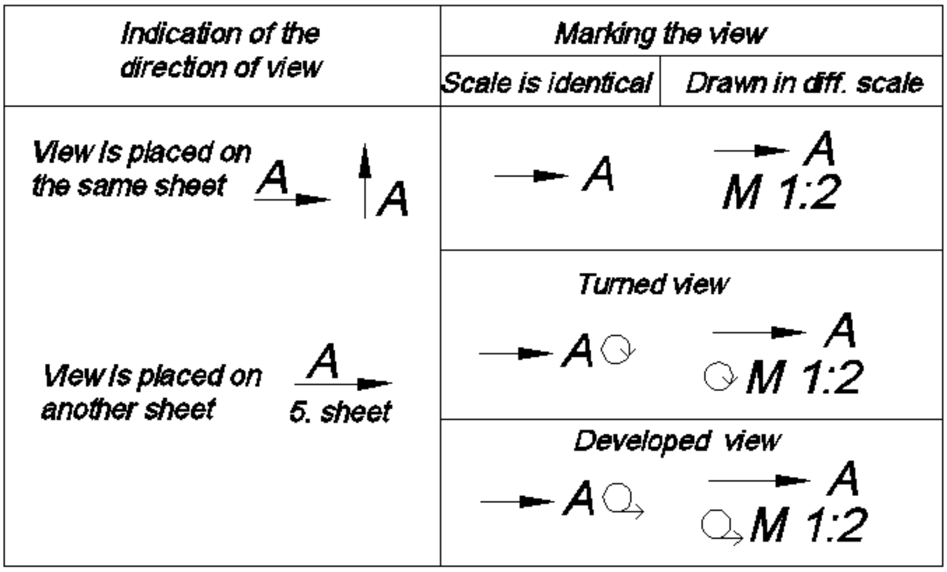
Figure 2.1 Defining principal views

The views are not indicated if they are located appropriately relating to the main view, as stipulated by the above mentioned standard. In this case views have to always be in horizontal or vertical alignment with the other views and the projection of one view to another cannot be questioned.

3. MACHINE DRAWING

3.1. Presentation methods in machine drawing

Parts are manufactured and assembled according to drawings. A drawing must contain information concerning the shape of the object, its function, dimension, material, surface finish, heat treatment and more.
When making machine drawings the views can be arranged in a specific way. Some views may be located without orthographic connection to the main view or may be incomplete in which case they are bounded by a break line or an axis of symmetry. In this case the direction of viewing is indicated by an arrow designated by the corresponding letter. For indicating the direction of views and cutting planes (see later) capital letters are used.



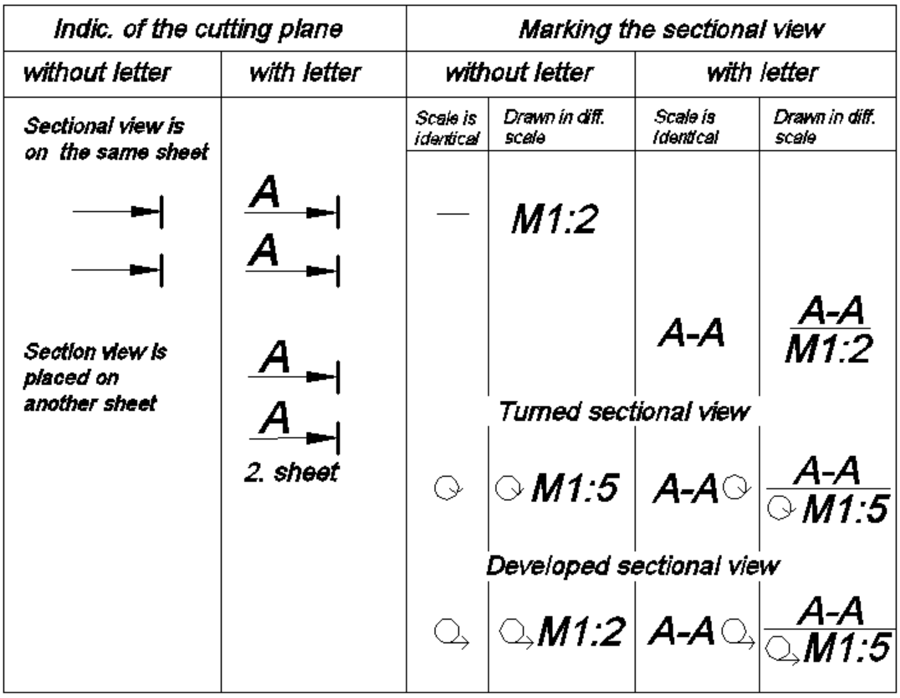


Figure 3.1 Indicating views and cutting planes

The identification of the view and the sectional view is done by using the same letters placed above or close to the view or the sectional view. If the view or sectional view is drawn in a different scale or on another sheet or it is placed in a special way, it must be indicated according to the standard (MSZ ISO 128) as in Fig. 3.1.

Besides the above considered views, machine drawings sometimes use auxiliary views facing arrows. They are resorted to when none of the six principal planes of projection gives the true shape and dimensions of a certain machine part or its elements. Therefore, any surface that is not in line with one of the three major axes needs its own projection plane which must be parallel to the object surface to show the features correctly.

3.1.1 Auxiliary view

When drawing auxiliary views distorted portions of a part are usually not shown. To avoid it the view is broken and bounded by a break line. Examples of arranging auxiliary views are given in Figs. 3.2 (a bracket) and Fig. 3.4. To avoid constructing several ellipses on the horizontal projection, the orthographic drawing should be represented as in Fig. 3.2 with the upper portion projected on an auxiliary plane of projection. Auxiliary views (say, the view facing arrow or facing arrow A) must be accompanied by the corresponding inscriptions, for instance, → A, and the appropriate projection of the part must be supplied with an arrow indicating the direction of viewing (see Fig. 3.4).

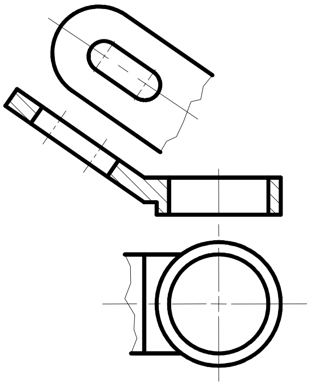


Figure 3.2 Drawing auxiliary view

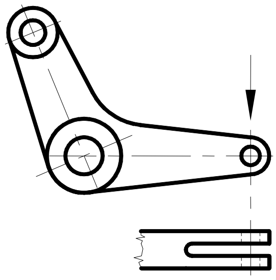


Figure 3.3 Drawing local view

When arranging views, auxiliary views may be turned through any desired angle for lucidity. In this case the mark of turning should be added, which is placed next to the marking of ⟶⟳ A, as in Fig. 3.4.

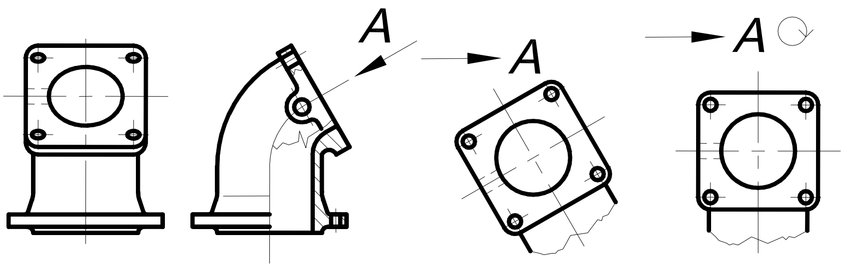


Figure 3.4 Arranging auxiliary views

3.1.2 Local view

Sometimes it is necessary to show only a certain part of the surface of a machine part. In this case it is expedient to draw only this particular part of the area which is called a local view instead of drawing all the whole orthographic views. The local view boundary is shown by a break line (as in Fig. 3.3). A local view, like an auxiliary one, may be accompanied by an inscription indicating the place where it pertains to. In simple drawing if the origin of the auxiliary view or local view is obvious the designation of the view can be omitted as in Fig. 3.2 and in Fig. 3.3.

3.1.3 Breaking

Breaking may be used to shorten certain views of long parts having similar cross sections throughout the length as in Fig. 3.5. If a view has a break, the dimension line must be drawn without breaks, and the dimension indicates the true length of the part. Fig. 3.6 illustrates a break of a part with cross sections varying in height.

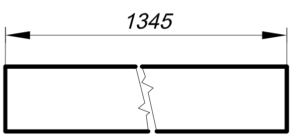


Figure 3.5 Drawing breaking

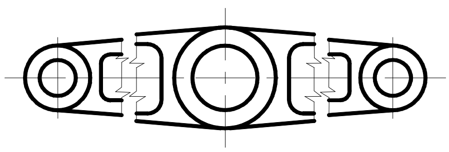


Figure 3.6 Applying breaking

3.2 Sectional views and sections

The rules of making sections are standardised (DIN 6; MSZ ISO 128). If a machine part or an element of which is cut along a plane, then a plane figure is obtained on the cutting plane. This figure is called a section. A sectional view is a view seen when a portion of the object nearest the observer is imagined to be removed by means of a cutting plane or planes, thus revealing the interior construction. However the other views are not affected in any way and always present the entire object. Some distorted elements behind the cutting plane may be omitted for simplification. The section is cross-hatched with thin lines having an inclination angle of 45°.
In assembly drawing the adjacent components should be outlined in different ways by cross-hatching in opposite directions (Fig. 3.7).

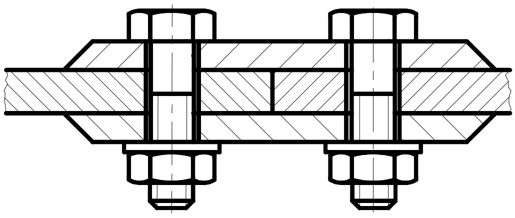
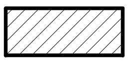
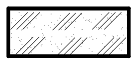
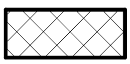
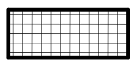
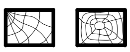
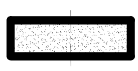
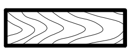
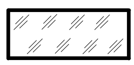
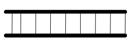
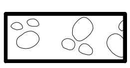
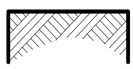


Figure 3.7 Hatching adjacent components

If a part is so shaped that 45^{o} degree sectioning runs parallel or nearly parallel to an outline or an axis, a hatching of 30° or 60° should be used.
The spacing between the hatching lines is estimated by eye but should be uniform. This spacing is usually between 2 and 10 mm depending on the hatched area. Small areas may be section lined at 1.5 mm. Long and narrow sections (less than 2 mm wide) may be made solid black independent of the material. In machine drawing the kind of material has to be indicated by using a specific type of section lining. The conventional symbols for various materials are shown in Table 3.1.

Table 3.1 Conventional representation of materials

Metals		Filled up ground	
Non-metallic material except those specified below		Electrical winding	
Wood cross grain		Grindstone	
Wood with grain		Glass and other transparent materials	
Sandwich structure		Blade pack	
Plain concrete		Earth at edges of foundations	

Depending on the number of cutting planes, sectional views are classified as follows:

simple (with one cutting plane) and
complex (with two or more cutting planes)

3.2.1 Simple sectional view

3.2.1.1 Sectional view

A sectional view is used to show the exact shape of certain interior parts which cannot be revealed in a normal two or three view drawing. A sectional view comprises the section and the view of the part behind the section plane. Only the parts actually cut by the section plane are cross-hatched (with thin continuous lines drawn at an angle of 45° to the horizontal). All sections on the sectional view of a parts are cross-hatched in the same direction and spacing and view of other parts is not affected in any way.
A part often requires cutting planes having general location to reveal its interior without distortion (section C-C). The sectional view may be arranged by and drawn in the desired position by turning. The mark of turning has to be added, as shown in Fig. 3.8.

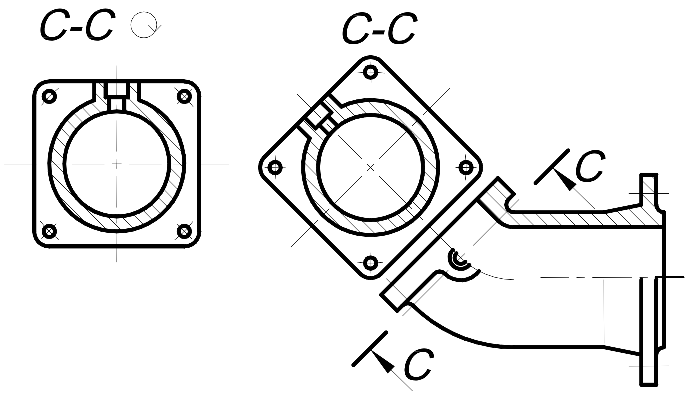


Figure 3.8 Drawing sectional view

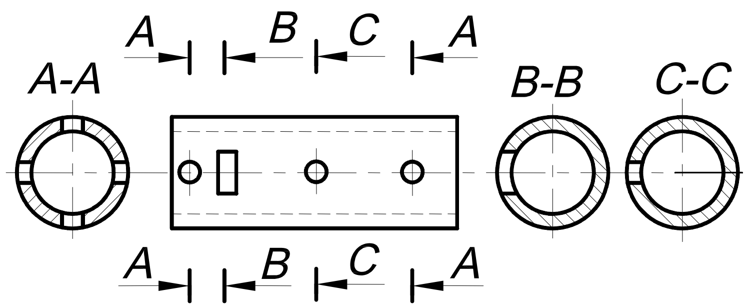


Figure 3.9 Marking cutting planes

If the interior of a part is identical in several cross sections, the identical cross sectional views may be drawn only once but the cutting planes have to be marked by the same letters, see Fig. 3.9.

3.2.1.2 Half-sectional view

In the case of a half-sectional view, only one-half of the view is shown in section and the other half is an external view. The interior part is separated from the exterior part by a centre line. It is standard practice to place the sectioned part to the right of an axis of symmetry or below it as in Fig. 3.10.

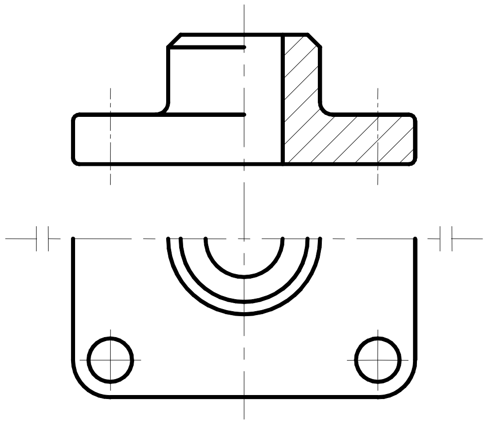


Figure 3.10 Drawing half-sectional view

3.2.1.3 Broken-out sections

The interior parts can only be represented either by section or by dashed line presenting the hidden edges. However, the interior detail of a hollow object may be shown by breaking out the particular area of the part. The break is outlined with a thin irregular wavy freehand line (see Fig. 3.11) that must not coincide with any contour or centre lines of the view. This way the presentation by dashed lines of hidden lines can be avoided.

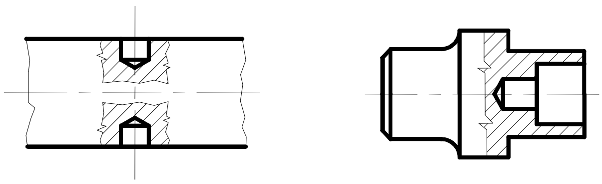


Figure 3.11 Drawing broken-out section

3.2.1.4 Removed element

If an element of a machine part requires some explanations as to its exact shape or dimensions because of the small size of its representation, then it is usually shown in an additional view drawn to a larger scale. This view is called a removed element (see Fig. 3.12).

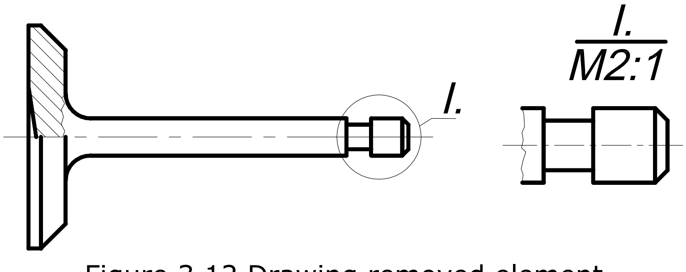


Figure 3.12 Drawing removed element

A particular place in the drawing (either on an external view, sectional view, or another section) is indicated by a circle drawn in a thin continuous line accompanied by either a Roman numeral or a capital letter with or without a number. A removed element, or detail, is usually supplied with an inscription comprising the same number and the scale. One view may have several removed elements. The removed elements may be sectioned.

3.2.1.5 Removed sections

Removed sections can be used in complicated drawings to clarify the construction of certain parts of the machine components. In this way, when applying the removed section, drawing a full or half-section can be avoided.

There are several ways to draw removed section:

- *Cutting plane is indicated by arrows.*
  The principal difference between an ordinary sectional view and a removed section is that only the section is represented and the view behind the cutting plane is ignored (Fig. 3.13 and Fig. 3.14). The section plane may be marked without capital letters if the correlation between it and the removed section is obvious. (If the removed section is located at any other places on the sheet it has to be marked by capital letters.) A removed section may be turned in this case it is indicated with the mark of turning as in Fig. 3.14.
  If it is beneficial to show many removed sections then the cutting planes are shown on the principal views by means of the usual symbol with arrows indicating the direction of viewing (Sections A-A, B-B, C-C) as in Fig. 3.15.

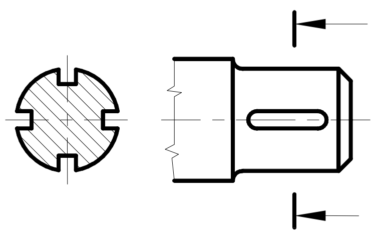


Figure 3.13 Drawing removed section by indicating the cutting plane

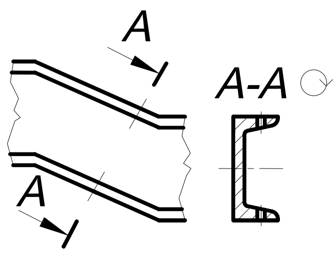


Figure 3.14 Drawing removed section turned by marking the cutting plane

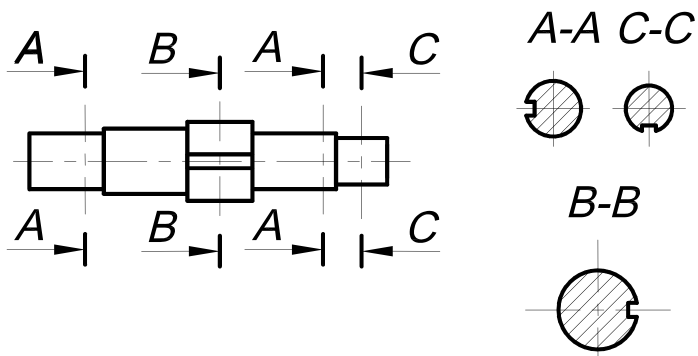


Figure 3.15 Drawing removed sections

- Cutting plane is indicated by dot-and-dash thin line.
  The removed section is drawn in thick line close to the principal view in alignment and bound to the principal view with a dot-and–dash thin line (centre line). The removed section can be placed either above or below the principal view but collinear with the section plane. In this case the section plane and the removed section are not marked as in Fig. 3.16. If it is necessary the removed section may be marked as in Fig. 3.17.

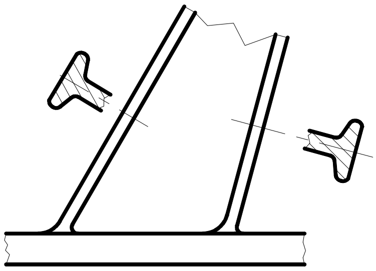


Figure 3.16 Drawing removed section indicated by its centre line

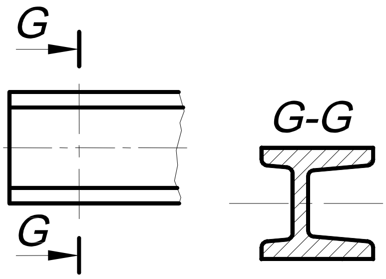


Figure 3.17 Drawing removed section by marking the cutting plane

- The principal view is broken out.
  The section plane is revolved into the drawing plane and the removed section remains in its original place. The removed section is drawn by a thick line as in Fig. 3.18.

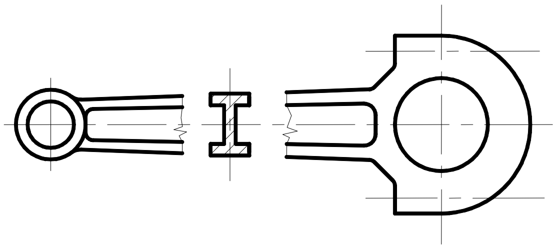


Figure 3.18 Removed section applying broken out

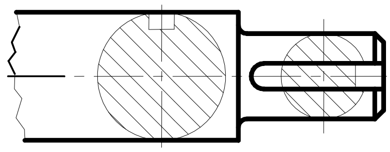


Figure 3.19 Removed section revolved into the drawing plane

- The principal view is not broken out.
  The section plane is revolved into the drawing plane and the removed section remains in its original place. The removed section is drawn by a thin line as in Fig. 3.19.

3.2.2 Complex sectional views

3.2.2.1 Offset sections with parallel cutting planes

Cutting planes nearly always pass through the axes of symmetry of the whole object or its detail. If several such axes of symmetry occur, the cutting plane can be offset (Fig. 3.20). Strictly speaking, the part is cut by two or more parallel planes. The two sectional views are comprised on one sectional view. A characteristic dashed line shows the offset cutting planes and arrows show the direction in which the view is taken on the end of the line. In the case of an offset sectional view the broken lines separate the sectional views to indicate the offsets of the cutting planes.
In the case of a rotation-symmetric component, if the cutting planes pass through the axis of rotation a revolved offset section can be applied by revolving the sectional views into the plane of the drawing as in Fig. 3.21 and Fig. 3.22. On the sectional view broken lines are used to indicate the cutting planes if the intersection lines of the cutting planes do not coincide with the axis of rotation. The cutting planes are shown on the drawing with the usual symbol.

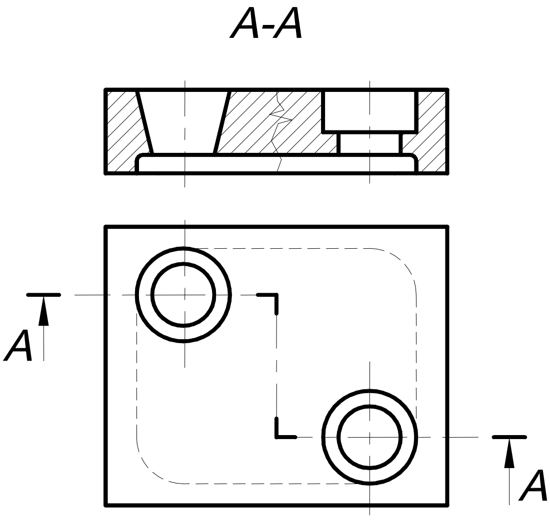


Figure 3.20 Drawing offset section

3.2.2.2 Revolved section

A revolved section may be applied if only a part of the component is rotational-symmetric as in Fig. 3.23.

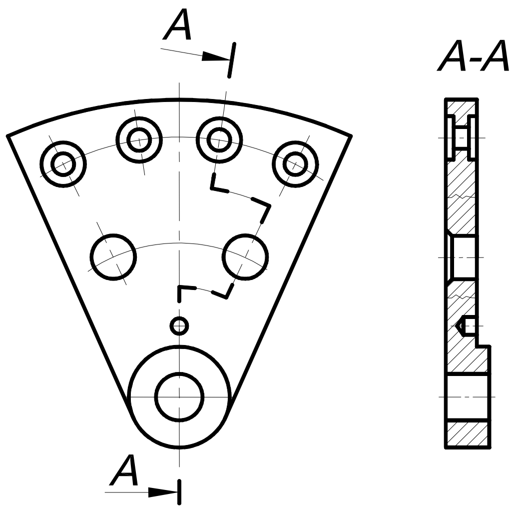


Figure 3.21 Drawing revolved offset section

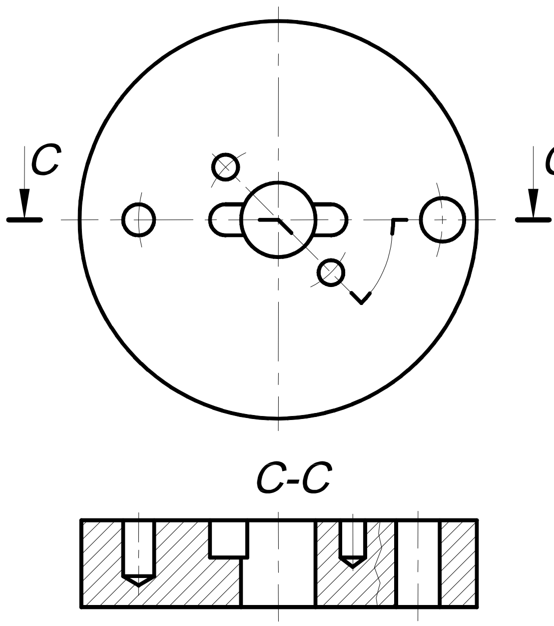


Figure 3.22 Drawing revolved offset section

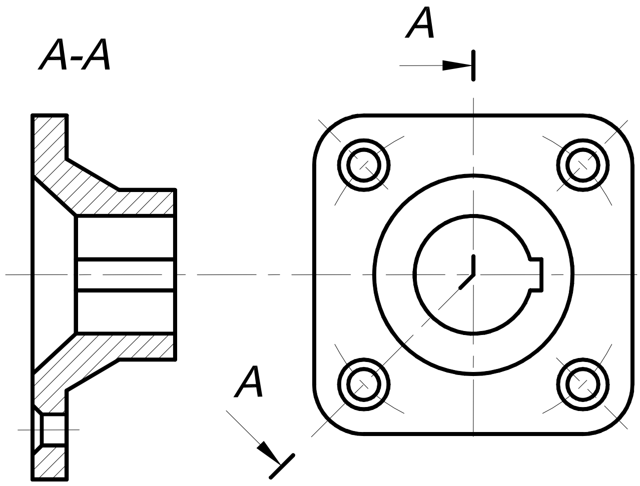


Figure 3.23 Drawing revolved section

3.2.3 Specific sectional views and sections

3.2.3.1 Developed view

If a machine part or an element thereof has a cylindrical or curved shape and its inner surface or interior construction should be represented, the developed view or developed section may be used.

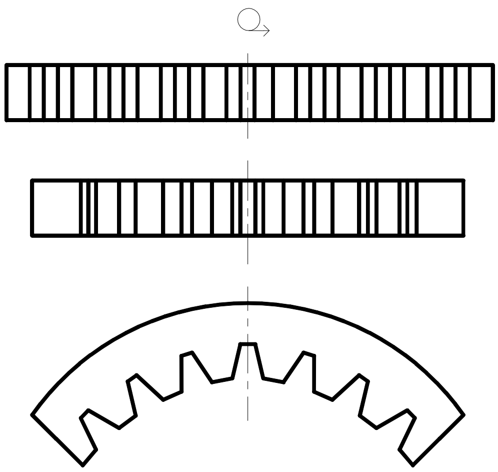


Figure 3.24 Drawing developed view

By applying a developed view, representation of distorted views can be avoided. In this case the direction of viewing may be indicated by an arrow and the view has to be indicated by the symbol of developed (Fig. 3.24).

3.2.3.2. Developed section

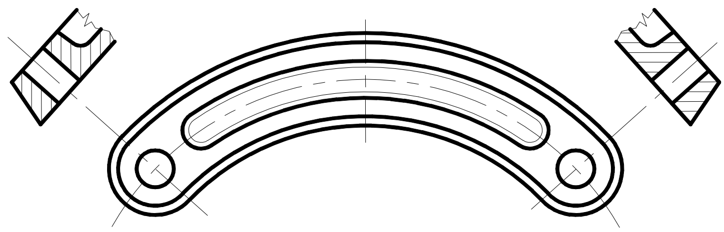


Figure 3.25 Sectioning curved shape part

To represent the interior construction of a cylindrical shaped machine part, two options are available; represent several cutting planes gaining several sections (sectional views) as in Fig. 3.25 or represent a cylindrical surface as a cutting surface gaining a developed section (Fig. 3.26).

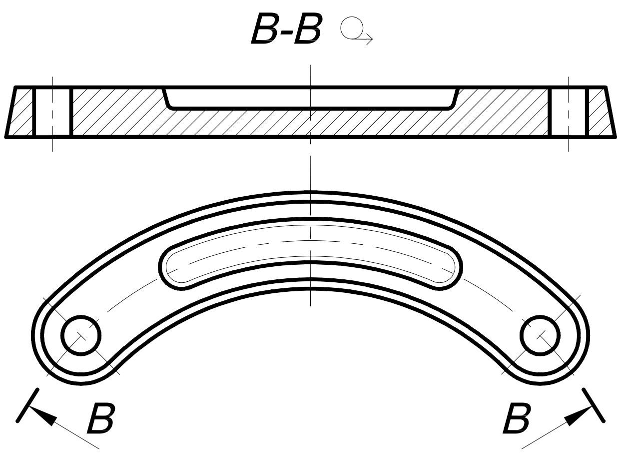


Figure 3.26 Drawing developed section

3.2.4 Conventional practice in machine drawing

The standard specifies various simplifications and conventionalities in making machine drawings.

3.2.4.1 Applying broken out section

If a visible contour line is located on the axis of symmetry (for instance an edge of a part), then the view or the section is made to be a little more than one half (as in Fig. 3.27 a, b and c) when drawing a half-sectional view.

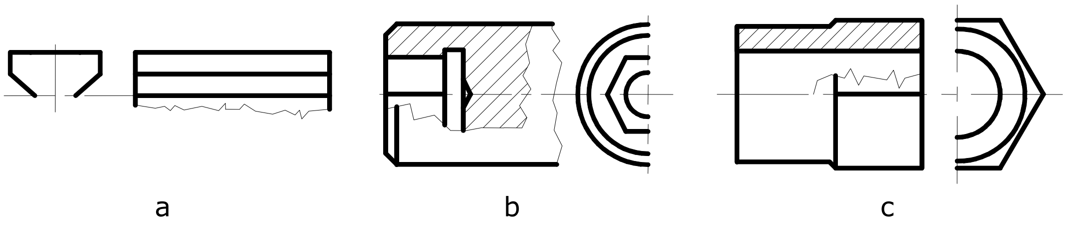


Figure 3.27 Applying broken out section on half views

If the cutting plane is grazing a surface of a part, this surface must not be sectioned. To avoid this, the broken-out section is used in the cutting plane as in Fig. 3.28. In this case the view is separated from the sectional view by a thin continuous freehand line.

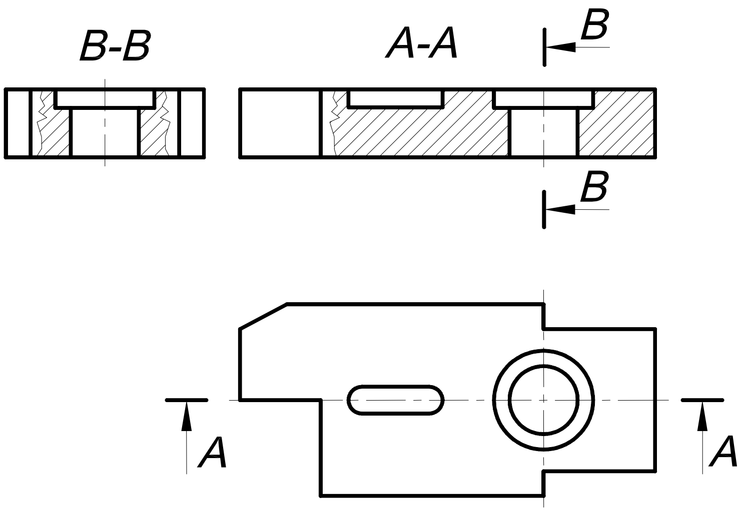


Figure 3.28 Applying broken out section when grazing a surface

3.2.4.2 Avoiding cutting specific parts

When a section plane passes through the longitudinal axes of solid cylinders such as shafts, bolts and screws, these parts may not be cut by the section plane. Nothing would be gained by showing the solid interior of such parts, and without crosshatching large areas the drawing is clearer. Elements such as thin walls, webs, stiffening ribs, lugs, spokes of pulleys and handwheels, teeth of gears and sprockets, etc., also come under this rule. Fig. 3.29 illustrates a step bearing in section. Its stiffener is not cross-hatched even though the section plane passes through the longitudinal axis of the part and the stiffener.

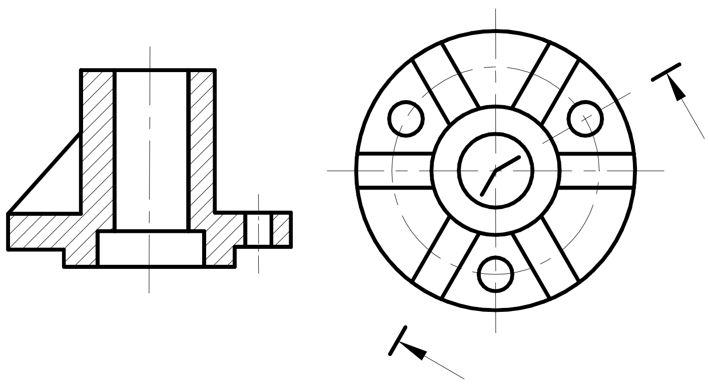


Figure 3.29 Avoiding hatching webs

If webs or stiffening ribs have recesses, blind or through holes, etc., they are shown by making broken-out sections to reveal the hidden part, as in Fig. 3.30.

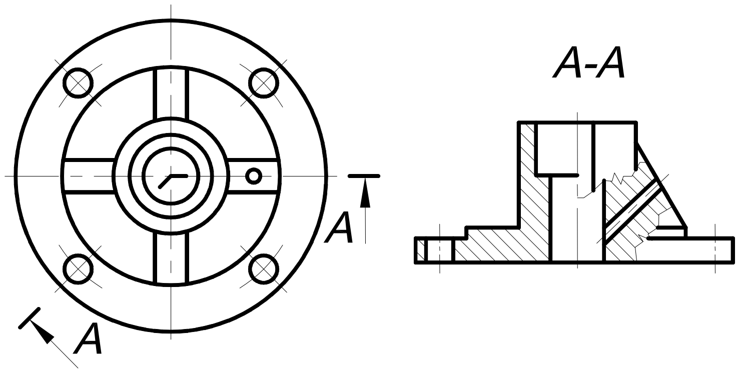


Figure 3.30 Breaking out webs

If a section plane passes through objects such as a flange (Fig. 3.31) or a flywheel (Fig. 3.32) it is conventional practice not to show the web behind the cutting plane and to represent the spoke as if turned to coincide with the section plane.

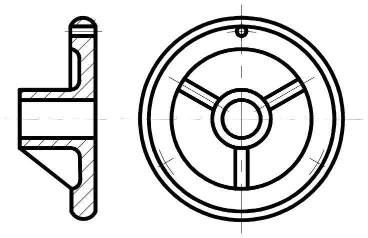


Figure 3.31 Sectioning a flange

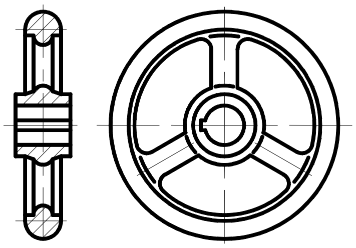


Figure 3.32 Sectioning a flywheel

3.2.4.3 Avoiding drawing transition lines

Fig. 3.33 and Fig. 3.34 indicate the conventional practice in presenting different tapers on parts. Transition lines between surfaces, gradually merging into each other, are shown conventionally or not shown at all (Fig. 3.34). This simplification can be used when the size of the drawing does not allow an accurate demonstration of the view. Transition lines are not used for dimensioning.

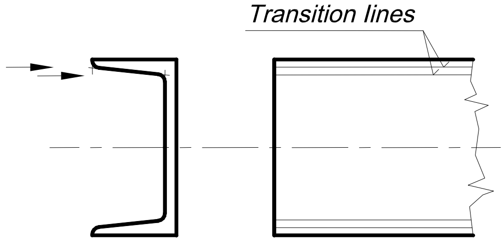


Figure 3.33 Drawing transition lines

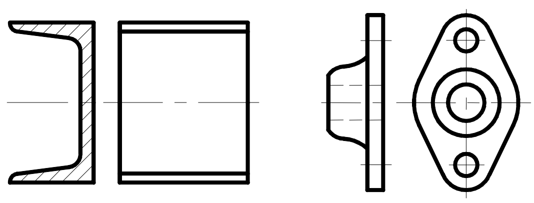


Figure 3.34 Avoiding drawing transition lines

3.2.4.4 Indicating surface texture

Representing the knurled surfaces of machine parts is standardized (DIN 82) and shown, as in Fig. 3.35. Only a certain portion of such a surface is covered with the specific symbol which is limited either by outline, centre line or by free-hand drawn wave line.

3.2.4.5 Indicating terminal position

Terminal positions of moving components are represented by thin dot-and-dash lines as in Fig. 3.36.

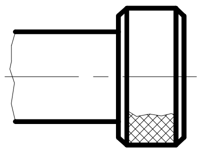


Figure 3.35 Indicating surface texture

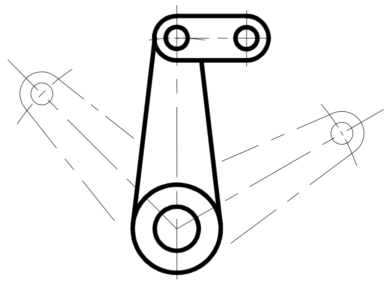


Figure 3.36 Indicating terminal position

4. SCALES AND DIMENSIONING

4.1 Scales

The scale is the relation between the dimension of the drawing of an object and the actual dimension. Scales are defined in the standard MSZ ISO 5455.
The selected scale is appropriate if the dimension of the drawing of machine parts and assemblies is suitable for presenting their true shape and dimensions and for dimensioning. Accordingly, large objects of comparatively simple shape are drawn in smaller than full size, and small objects (such as watch parts) are scaled up.
Tabulated below are the scales used in technical drawings. All drawings should be made to scale. If several scales are used on the same drawing, the most common scale used has to be stated in the title block. However, differing scales must be shown next to the view, preferably above it.

4.1.1 Scale designation

The accepted abbreviations are:

Scale 1:1	(for full size)
Scale 1:5	(for scaling down)
Scale 2:1	(for scaling up), and so on.

Reduction scales:	1:2, (1:2.5), (1:4), 1:5, 1:10, (1:15), 1:20, 1:25, 1:50, (1:75)
Full size:	1:1
Enlarged scales:	2:1, (2.5:1), 5:1, 10:1

The scales given in parentheses are permitted but not recommended.

4.1.2 Scale specification

If all drawings are made to the same scale, the scale should be indicated in the title block. If it is necessary to use more than one scale on a drawing, the main scale is given in the title block and all the other scales should be shown near the particular view or sectional view.

4.2 General instructions for dimensioning

Working drawings must indicate all necessary dimensions in a way most convenient for the workman to understand. Dimensioning methods are standardized (DIN 406; MSZ ISO 129). The size of the object or its separate parts are usually indicated in drawings by means of dimension lines, completed with numbers showing the actual measurement irrespective of the scale.

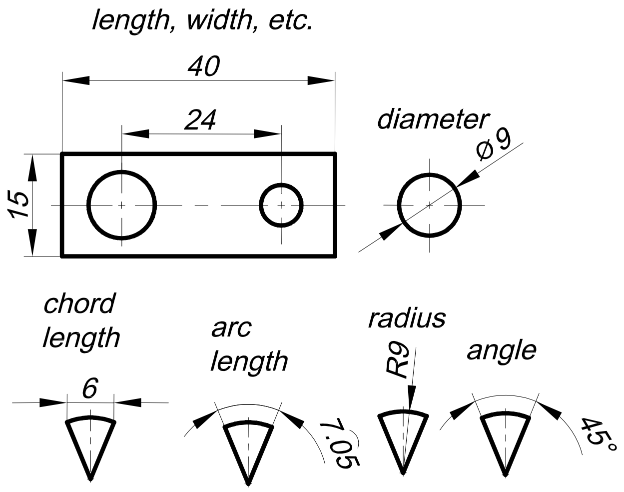


Figure 4.1 Main types of dimensioning

As a rule, dimensions in machine drawings are given in millimetres without adding the abbreviation "mm". Dimension lines are made with thin continuous lines to contrast the thick outlines of the drawing. They are drawn parallel to the surfaces whose length they indicate and are terminated by arrowheads at the ends of the dimension line. Dimension figures must be written by standard type font to avoid confusion. They have to be written above and parallel to the dimension line leaving approximately 1 mm space from it and preferably as close to its centre as possible. Fig. 4.1 shows the main types of dimensioning.
The supplementary symbols being used in dimensioning are standardised (DIN 406 (MSZ ISO 129).
In machine drawing (working drawing, assembly drawing) the dimension pertains to the completed part. When the surface of a part has a metallic coating it pertains to the coated state, but when the surface is painted or has a plastic coating, it pertains to the state before coating (Fig. 4.2).

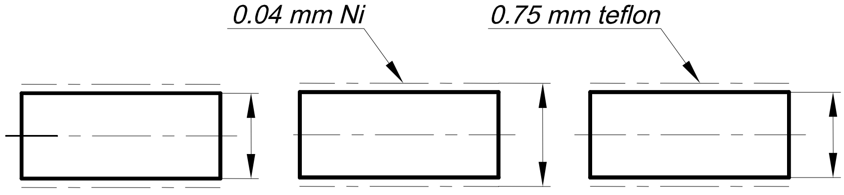


Figure 4.2 Indicating coated surfaces

Overall dimensions of a part have to be given. Dimensioned drawings can contain informative dimensions that are not used for manufacturing or dimensional inspection, but make it easier to read the drawing. Informative dimensions must be put in parentheses as in Fig. 4.3.

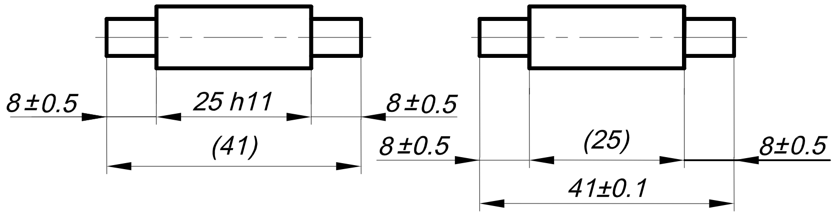


Figure 4.3 Chain dimensioning and informative dimensioning

However, if the drawing of a part has a break, the dimension line must be drawn without a gap, as in Fig. 3.6. Each dimension in a drawing must be given only once, duplicate dimensions have to be avoided. Dimension lines are preferably (but not obligatorily) drawn outside the drawing outlines.

The arrangement of dimensions should be carefully designed. Intersections of extension and dimension lines should be avoided. When a series of parallel dimension lines are in close proximity to each another, the space between them should be between 7 and 10 mm. Centre lines, cross-hatching lines and any other lines must be broken where arrowheads and figures are to be placed.

Arrowheads that terminate on the dimension lines must just touch the corresponding outlines, or centre lines, or extension lines (Fig. 4.4).

The size of arrowheads depends on the thickness of visible outlines. The arrowhead is about 2.5-3 mm in the case of the visible outline is 0.7 mm thick and must be the same for all dimension lines of a given drawing. Its vertex angle is 15^{o} (see Fig. 4.5).

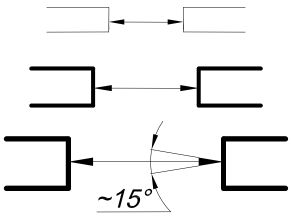


Figure 4.4 Arrowhead

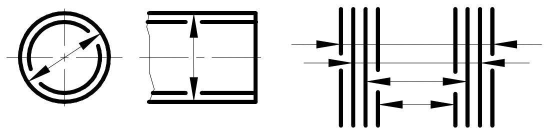


Figure 4.5 Methods of indicating dimensions

Extension lines must extend from 2 to 3 mm beyond the ends of the arrowheads. If there is not enough space for an arrowhead because two lines of the visible outline are too close to each other, the particular outline must be broken since arrowheads must not be intersected by any lines (see Fig 4.7).

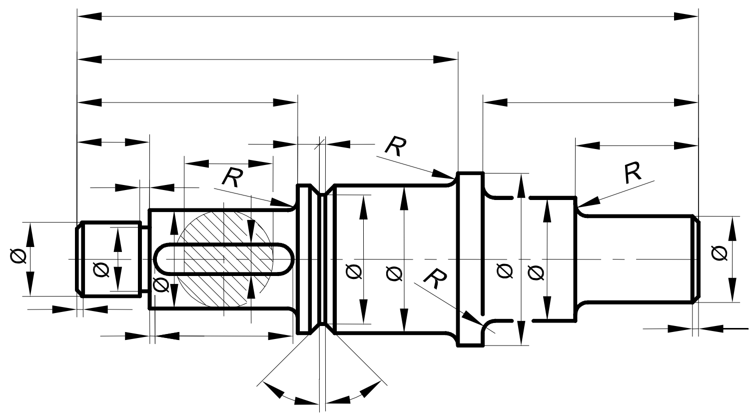


Figure 4.6 Arrangement of dimensioning

If there is no room for arrowheads at the ends of dimension lines arranged in a continuous chain, they may be replaced by an oblique stroke that is 2-3 mm long and has an inclination of 45° to the dimension line as in Fig 4.6.

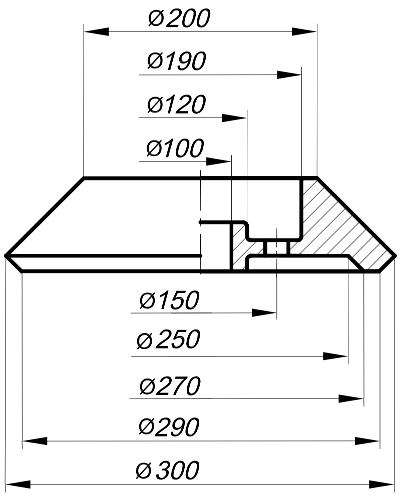


Figure 4.7 Dimensioning half sectioned view

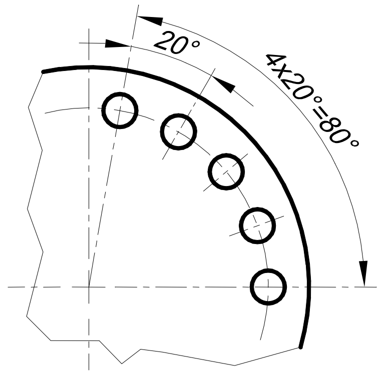


Figure 4.8 Dimensioning equiangular features

On half-sectioned views with an axis of symmetry it is permissible to use dimensions according to Fig. 4.7. In this case the dimension lines must extend somewhat beyond the axis of symmetry.

When dimensioning a number of equal spaced similar elements of a machine part, say, holes, proceed as in Fig. 4.8 and Fig 4.9 (usually only one hole is dimensioned and the number of holes and the spacing are indicated).

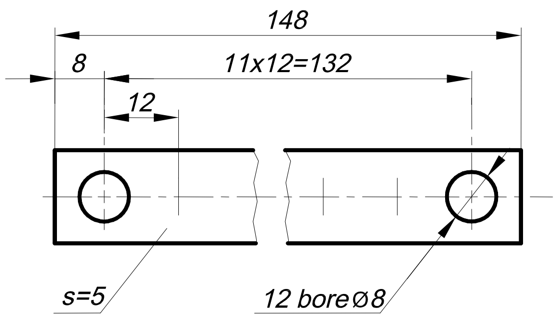


Figure 4.9 Dimensioning equidistant features

Fig. 4.10 shows how dimensions of uniform bore spacing on the same bolt circle can be given together with the diameter of the bolt circle.

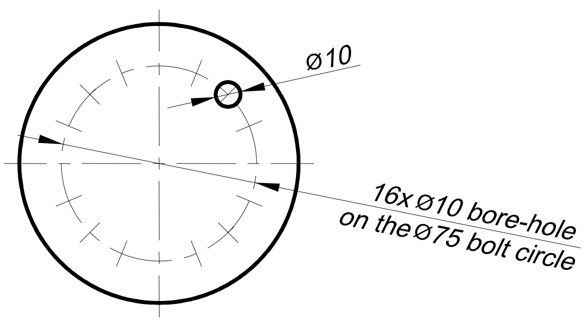


Figure 4.10 Dimensioning bolt circle and holes on it

Squares may be dimensioned according to the size of the edge as in Fig. 4.11. The radius should always be given for dimensioning a circular part consisting of different fillets. Fig. 4.12 shows how spherical surfaces are dimensioned.

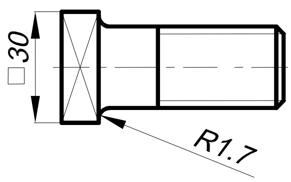


Figure 4.11 Shape identification symbol, square

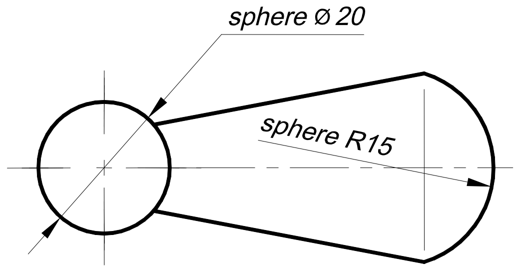


Figure 4.12 Shape identification symbol, sphere

A 45° chamfer is dimensioned using a dimension line as in Fig. 4.13 or on a leader as in Fig. 4.14.

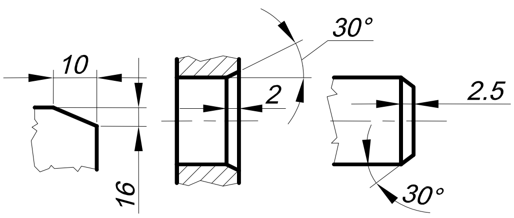


Figure 4.13 Dimensioning chamfers

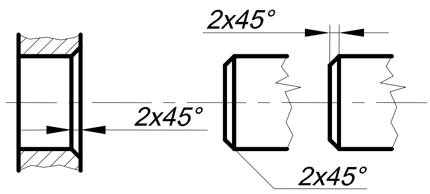


Figure 4.14 Simplified chamfer dimensioning

4.2.1 Placement of dimension figures

The placements of dimensions are shown in Fig. 4.15. The basic principle is that dimension numbers should be read from the direction of the title block and from the right. Dimension numbers pertaining to the dimension lines in the hatched area have to be placed on the horizontal leader.

Fig. 4.16 shows the recommended practice for the placing of angular dimensions. An angular dimension can be placed on a horizontal leader in all cases, but in the hatched area it is compulsory.

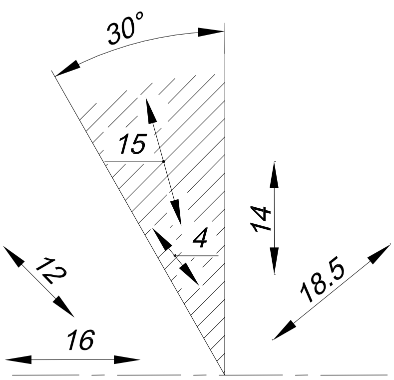


Figure 4.15 Oblique dimensioning

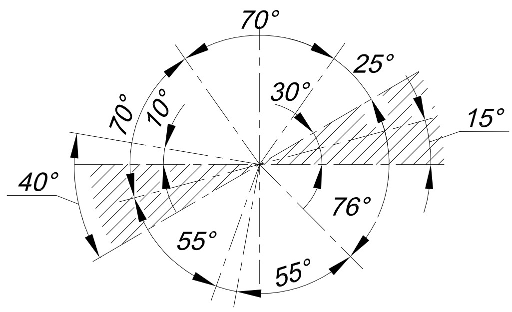


Figure 4.16 Angular dimensioning

4.2.2 Specific dimensioning

Dimension lines must not be the continuation of outlines, centre line or extension lines of any other dimensioning. Similarly, the outlines, centre lines and extension lines must not be used as dimension lines.
Although dimension lines must not be used as extension lines, they are used when dimensioning the coordinates of points on a non-circular curve (Fig. 4.17).

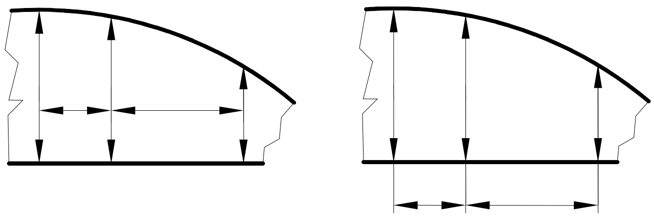


Figure 4.17 Dimensioning non-circular curve

If the dimension lines of a non-circular curve have a uniform spacing, the two conventional dimension types can be combined as in Fig. 4.18.

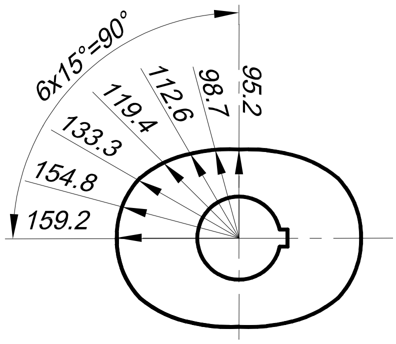


Figure 4.18 Simplified dimensioning of non-circular curve

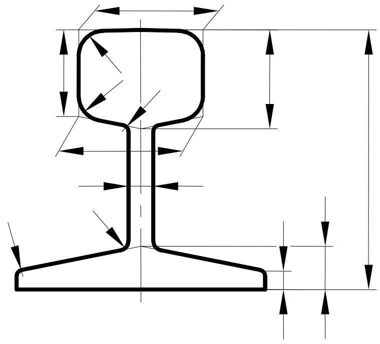


Figure 4.19 Dimensioning non-parallel surfaces

Although extension lines should be perpendicular to the dimension lines, in certain exceptional cases extension lines may be drawn not at a right angle to dimension lines as is shown in Fig. 4.19. In this case the extension lines must be parallel to each other and they are drawn from the intersection points of the continuations of the visible outlines.
Where space limitations do not permit giving a separate line for each dimension, the dimensions may be placed in one line as shown in Fig. 4.20 and Fig. 4.21.

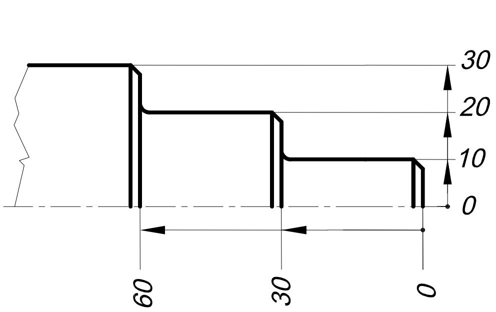


Figure 4.20 Simplified parallel dimensioning

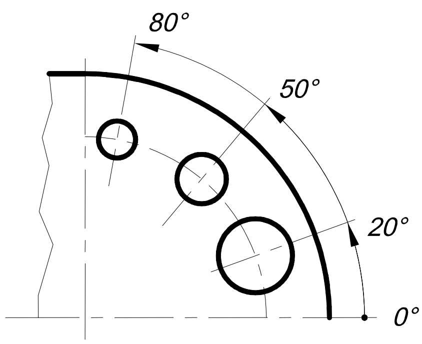
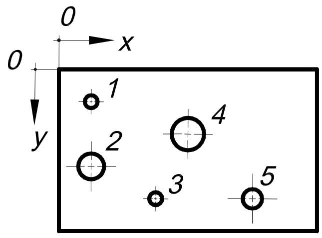


Figure 4.21 Simplified parallel angular dimensioning

In this method, called consecutive dimensioning, there is only one arrow for each dimension, thus indicating that each dimension goes back to the original base line that is e.g. zero. In this case the dimension numbers are placed along the extension lines and the common starting point is indicated by “0” (zero) dimension.
Co-ordinate dimensioning can be performed using a table containing the coordinates (Fig. 4.22) instead of a graphical dimensional specification.



	1	2	3	4	5
x	10	10	25	35	50
y	10	30	40	20	40
ø	2	4	2	5	3

Figure 4.22 Co-ordinate dimensioning

4.2.3 Choosing reference surfaces

When dimensioning a part it is expedient to choose a reference surface in the three main directions from which the position of all the details and surfaces may be dimensioned.
Reference surfaces can be:

- a significant surface in terms of working (line marked by 1 in Fig. 4.23)
- a significant axis of symmetry in terms of working (line marked by 2 in Fig. 4.23)
- boundary line/s of overall dimension/s of a stepped part (shaft), see Fig. 4.24
- boundary line of a plane or planes dimensioned from a lateral surface, Fig. 4.25

The general rule is to determine the dimensions relative to the reference surface and not to apply the closed dimension chain. However, if it is necessary derived dimensions may be put into parentheses as an informative dimension.
For clarity, the dimensions on the views should be grouped according to the dimensions of the inner surfaces, the outside surfaces, the finished surfaces and the unmachined surfaces as in Fig. 4.26.

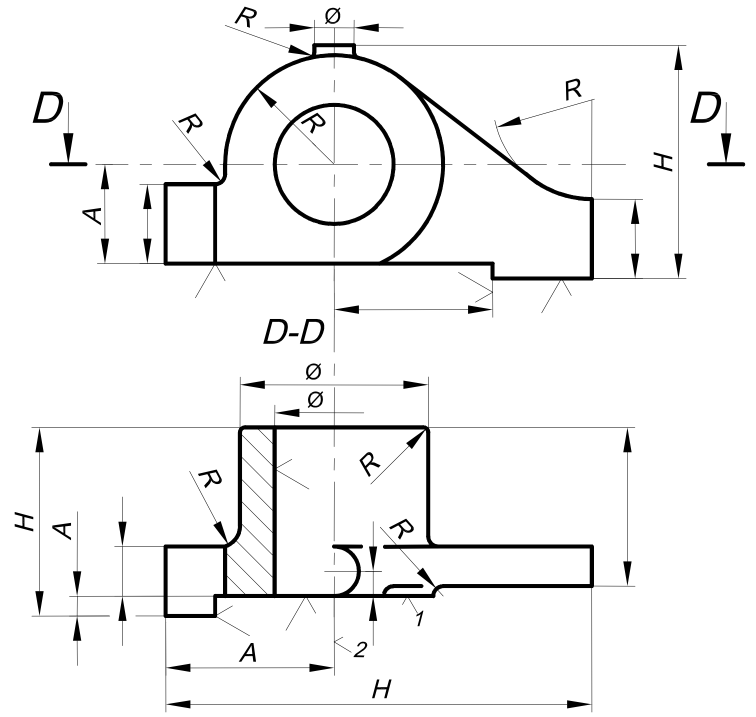


Figure 4.23 Choosing a significant surface as a reference surface

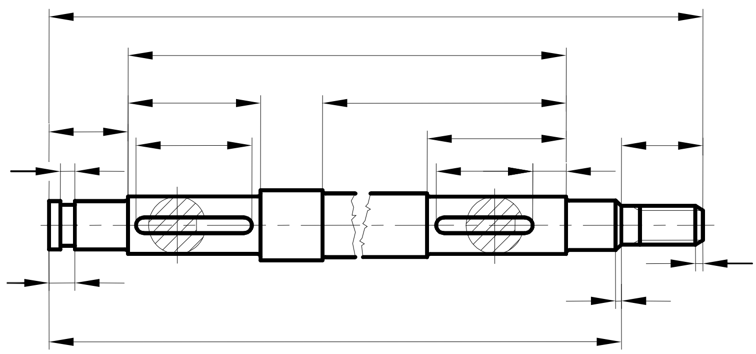


Figure 4.24 Choosing boundary lines as reference surfaces

When dimensioning details on a part having common position it is expedient to choose an appropriate reference surface as in Fig. 4.27. Placement is given relative to the main directions. The dimensions of the details can be performed according to the particular reference surface.

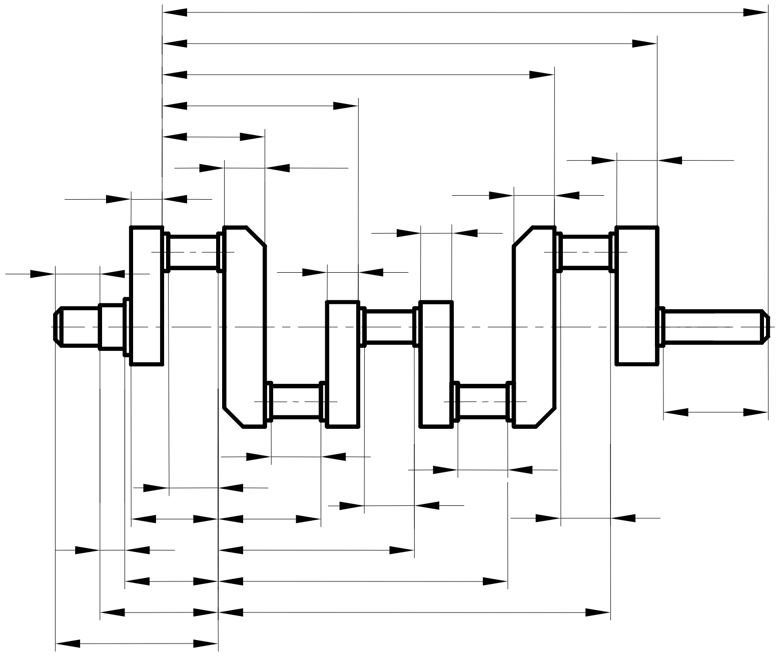


Figure 4.25 Choosing several boundary lines as reference surfaces

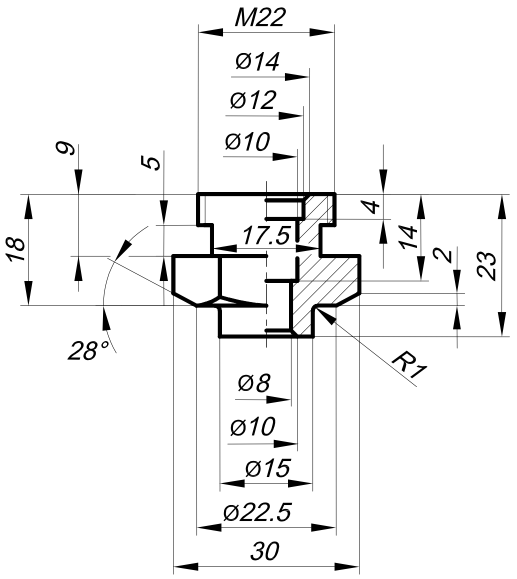


Figure 4.26 Grouping the dimensions

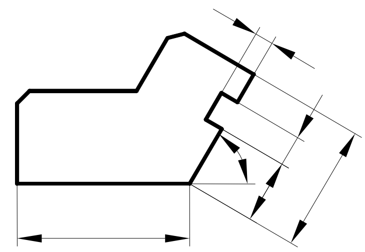


Figure 4.27 Choosing a common position surface as reference surface

Dimensioning of mating elements of connecting parts should be identical (i.e. the mirror image of each other, Fig. 4.28).

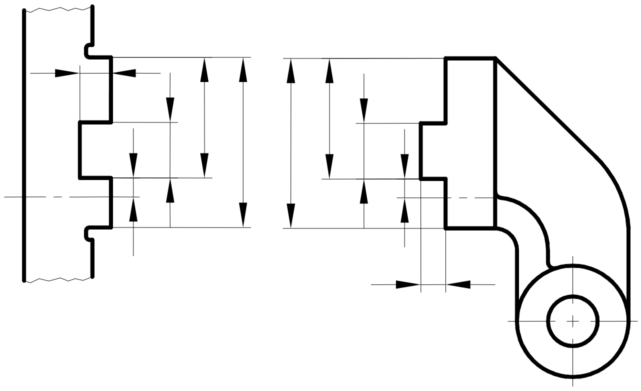


Figure 4.28 Dimensioning mating elements

4.2.4 Conventional dimensioning methods

Dimensioning of a part has to reflect the task and the significance of its surfaces in terms of working. Dimensioning is made commonly in orthogonal and rarely in the polar coordinate system.
Self evident dimensions should not be indicated in the drawing, such as:

- perpendicularity of two adjoined surfaces or edges if they are correctly drawn
- parallelism of several edges or surfaces if they are correctly drawn
- equality of dimensions when halving a dimension with a centre line
- vertex angle of a regular polygon
- open-end holes; a hole is always an open-end hole if the hole depth is not indicated

4.2.4.1 Conventional dimensioning practice of bore-holes

a. Bore-holes with and without countersinking

Bore-hole can be threaded, without thread, open-ended holes or blind holes. Since the dimensions of the bore-hole determine its geometry, the graphical representation may be simplified and drawing a bore-hole may be omitted.

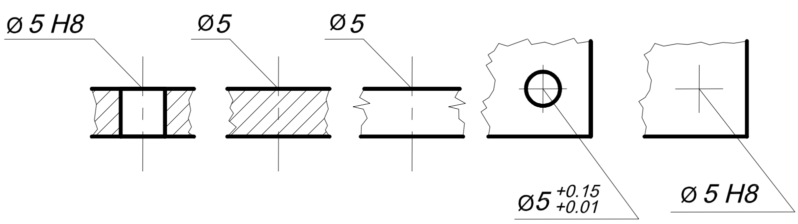


Figure 4.29 Conventional dimensioning of bore-hole

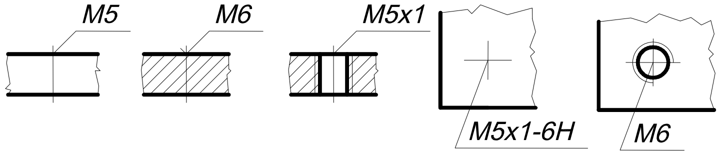


Figure 4.30 Conventional dimensioning of threaded bore-hole

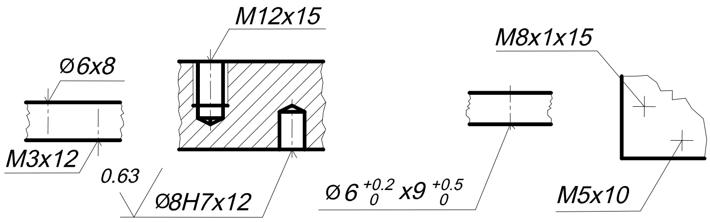


Figure 4.31 Conventional dimensioning of threaded blind bore-hole

The bore-hole, threaded bore-hole or blind hole dimensions are given on a leader indicating the place of the centre line (Fig. 4.29, 4.30 and Fig. 4.31).
The dimensioning of the bore-hole can be completed with the simplified dimensioning of the counterbore, but the bore-hole with the counterbore has to be drawn in view and in sectional view (Fig. 4.32), as well.

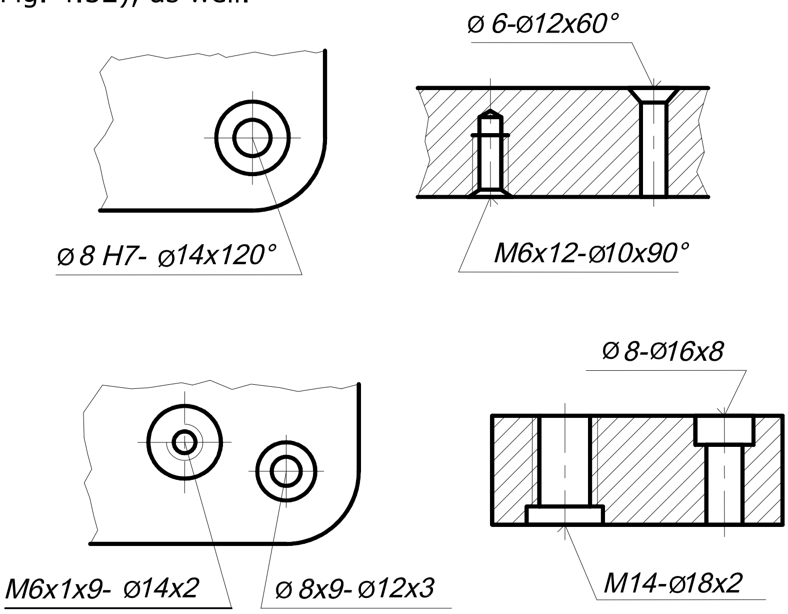
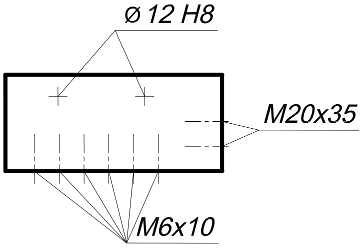


Figure 4.32 Conventional dimensioning of bore-hole and threaded bore-hole with counterbore

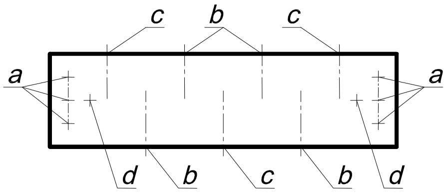
b. Repetitive bore-holes, bore-hole groups

Dimension of identical, repetitive bore-holes may be given either on the same leader, or the bore-holes can be indicated by letters with their dimensions compiled in a table, as in Fig. 4.33.



Mark	Apiece	Dim.
a	6	M6x10
b	4	M8x15
c	2	Ø10
d	1	Ø12H7

Figure 4.33 Conventional dimensioning of threaded bore-hole groups



c. Dimensioning the length of shape rolled components and of sheet gauge numbers

The length of shape rolled materials and the sheet gauge number may be given on a leader, as in Fig. 4.34 and Fig. 4.35.

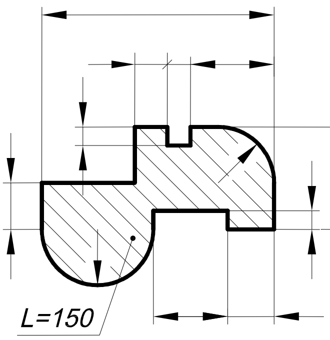


Figure 4.34 Dimensioning a shape rolled component

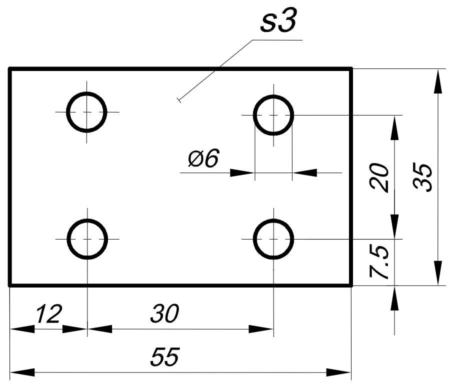


Figure 4.35 Dimensioning a sheet component

4.2.5 Dimensioning tapered parts

In the case of a cone, conical taper is defined as the ratio of the diameter difference of two diameters of the cone and the distance between the two diameters as in Fig. 4.36. Conical taper is expressed in %: D-d/L∗100. Flat taper can be defined for planes. It characterises the angle of a plane relative to another one (Fig. 4.37). Flat taper is expressed in %: A-B/L∗100.

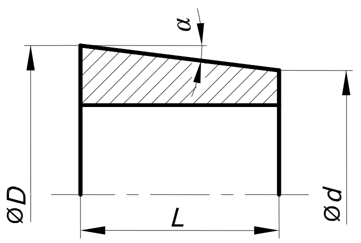


Figure 4.36 Conical taper

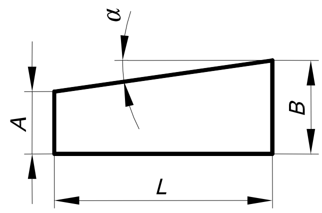


Figure 4.37 Flat taper

4.2.5.1 Dimensioning conical taper and flat taper

Indicating a taper by its symbol on a view is standardized (MSZ ISO 3040). The symbol is either on the centre line or on a leader parallel to the centre line, as in Fig. 4.38.

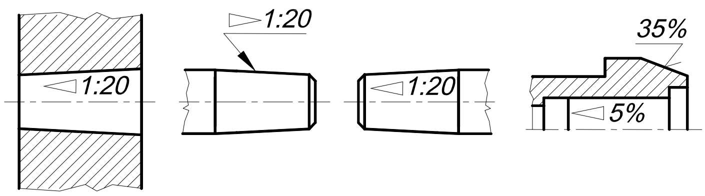


Figure 4.38 Dimensioning conical taper

Slant can be indicated by its symbol either on the slant line, or on a leader parallel to the reference plane of the slant, as in Fig. 4.39.

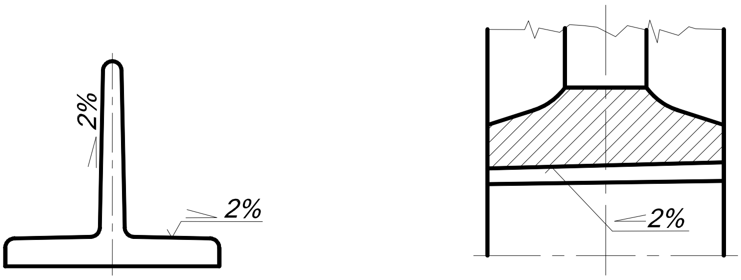


Figure 4.39 Dimensioning flat taper

The taper is given in %, but if the flat taper is bigger than 30° and the conical taper is bigger than 45°, they can be given either by the angle, or by dimensioning, as in Fig. 4.40.

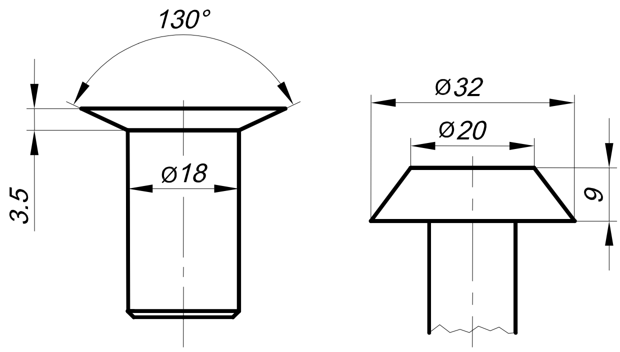


Figure 4.40 Showing the taper by dimensions and angle

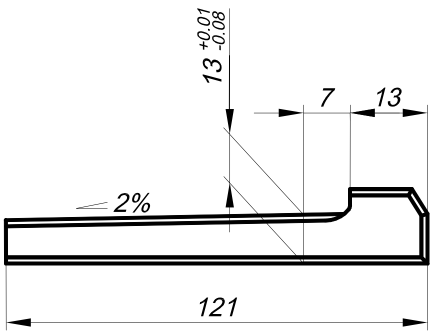


Figure 4.41 Giving of measuring place of tapered part

The typical dimension of the tapered or slant part is the biggest diameter or the biggest thickness. If it is difficult to measure these dimensions because of a fillet at the margin, the location of the measured point should be given, as in Fig. 4.41. Other dimensions of the tapered and slant parts are calculated from the typical dimension and the given taper or slant. If a dimension is not given directly, then it is a result that has to be signed by ≈ mark as in Fig. 4.42.

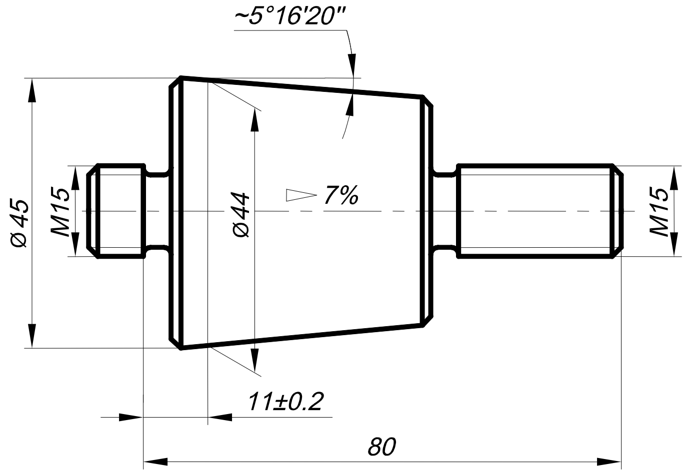


Figure 4.42 Giving informative dimension signed by mark

5. REPRESENTATION OF THREADS AND THREADED JOINTS

Rules of machine drawing include drawing standardized parts. The application of the standardized representation facilitates drawing components and making the drawing clear and understandable.

5.1 Thread forms

Threaded fasteners (screws, bolts, nuts) should clamp parts together with a force that is big enough to hold them in contact without dislocation.
The following thread forms are in common use:
Metric threads have a sharp V profile, where the thread angle is 60°. They are widely used for bolts, screws, studs, nuts, and other fasteners.
Inch threads have a 55° triangular profile. They are used mainly for manufacturing different spare parts.
Inch taper threads have a 60° angle. They are used in taper threaded joints of fuel, oil, water and air pipelines for various machines and machine tools.
Cylindrical pipe threads have a 55° angle with a pitch considerably smaller than that of inch thread.
Trapezoidal threads are used in different screws for driving. Its profile is trapezoidal with an angle of 30°.
Buttress threads have a profile composed of a series of non equilateral trapezium with an angle of 33°.
Standardised and practical notations and terms:

A_s Stress area of thread
D_{Km} Acting diameter for determining the friction torque at the fastener head or nut bearing face
F_M Tension force in the fastener after completed mounting
M_M Tightening torque for mounting
P Thread pitch
d Fastener diameter (biggest diameter of the thread)
d_2 Mean flank diameter of the thread
d_3 Smallest diameter of the thread
μ_G, μ_K Coefficient of thread friction and of the fastener head or nut bearing faces

Property classes
The designation system for threaded fasteners consists of two numbers separated by a decimal point. The first number represents ≈1/100 of the ultimate tensile strength in N/mm^2, while the second number is 10 times the ratio of the minimum yield point to the minimum tensile strength.
The property classes of standard nuts are indicated by a number which corresponds to 1/100 of the stress under proof load in N/mm^2, this stress corresponds to the minimum tensile strength of a fastener in the same property class.

5.2 Conventional representation of threads

The drawing of threads is standardised (DIN 202, DIN 406 (MSZ ISO 6410). Threads are illustrated by simplified presentations according to the enveloping surface. The conventional presentation of external threads is the following: the outline of the rod, corresponding to the major diameter of the thread is drawn in continuous straight thick lines (see Fig. 5.1). Lines corresponding to the minor diameter of the thread (or lines of the roots) are drawn by continuous fine line. Internal thread in sectional view is presented by continuous visible line (the minor diameter) and continuous fine line (the major diameter), as shown in Fig. 5.2.

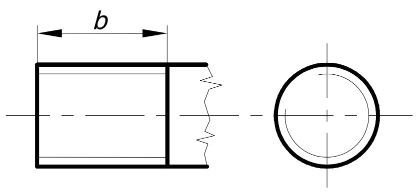


Figure 5.1 Conventional presentation of external threads

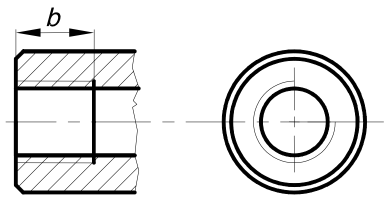


Figure 5.2 Conventional presentation of internal threads

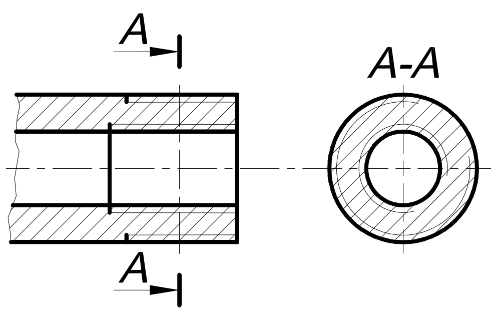


Figure 5.3 Presentation of threads in section

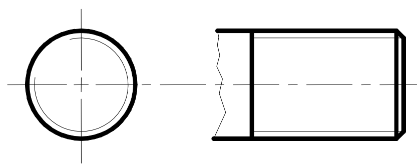


Figure 5.4 Ignoring the presentation of the chamfer

In views perpendicular to the axis, the thread is drawn by continuous fine arc of about three-fourths of a circle. The beginning and end of the thread are presented by continuous thick lines as in Fig. 5.3. The distance between the continuous visible line and the fine line presents the thread and it is equal to the depth of thread, but it should be minimum 0.8 mm. In general only the effective length of a thread is depicted and dimensioned. If it is necessary the thread runout may be shown. In the case of a threaded blind hole, the cone angle of the hole is not of importance, it is drawn at 120° without indicating it. Chamfers on threaded rods and holes are not shown when projected on a plane perpendicular to the axis of the rod or hole (Fig. 5.4*)* because they would disturb the depiction of the thread. If the chamfer of the rod is bigger than the depth of the thread, it is depicted as in Figure 5.5. The simplified presentation of intersecting threaded holes in section is shown in Fig. 5.6. In the case of dimensioning a non standardized thread profile, the measurements may be given either in a broken out section, or on a removed element (Fig. 5.7 and Fig. 5.8). When drawing connecting male thread and female thread, the basic principle is that the male thread covers the female thread in a sectional view as in Fig. 5.9.

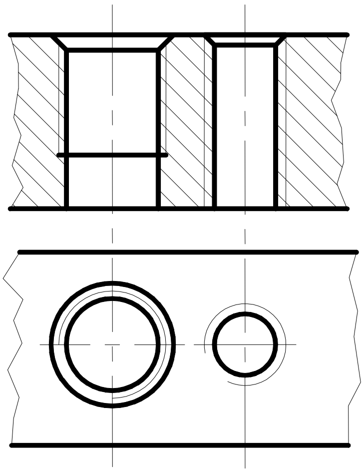


Figure 5.5 Presenting the chamfer of the thread

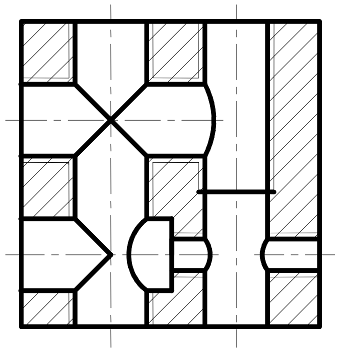


Figure 5.6 Drawing intersecting threaded holes in section

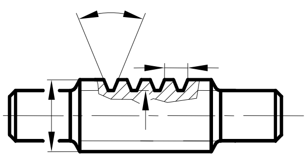


Figure 5.7 Dimensioning a thread profile

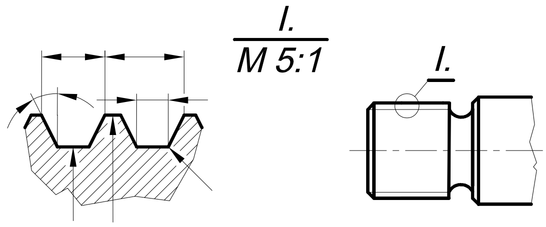


Figure 5.8 Dimensioning a thread profile in on a removed element

If the cutting plane of the threaded rod is along a groove or a keyway, the connection has to be drawn as in Fig. 5.10.

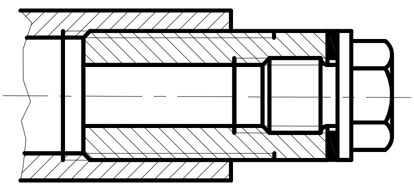


Figure 5.9 Drawing a connecting male and female thread in section

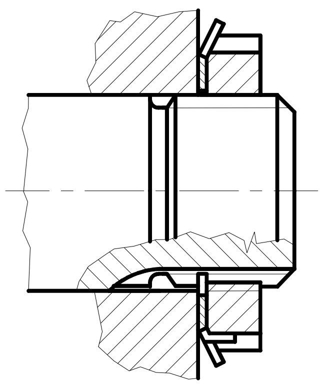


Figure 5.10 Drawing a connecting male and female thread in section having a keyway

A tapered thread is presented by three (instead of four) concentric circles, as in Fig. 5.11 and Fig. 5.12.

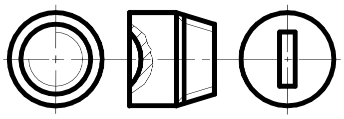


Figure 5.11 Drawing a tapered male thread

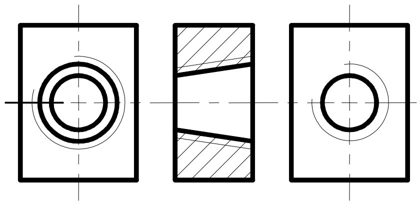
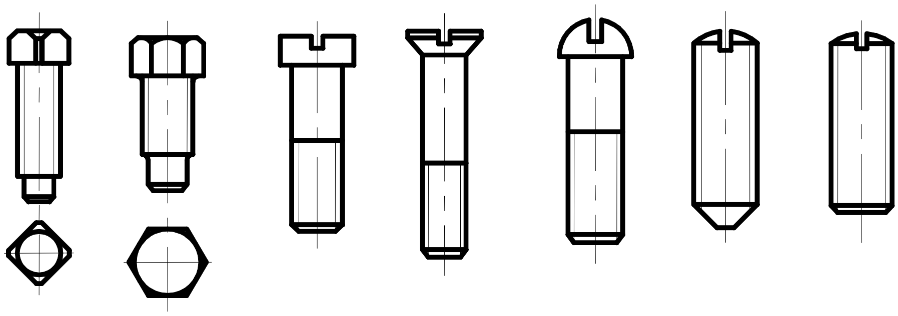


Figure 5.12 Drawing a tapered female thread

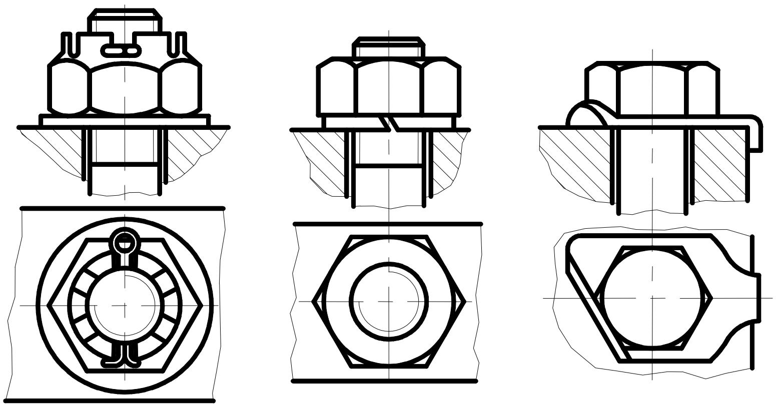
Generally, screws are manufactured with various types of heads and ends depending on the purpose. Some standard screws are illustrated in Fig. 5.13.



a. b. c. d. e. f. g.

a. set screw with a square head
b. set screw with a hexagonal head
c. screw with slotted flat-fillister head
d. slotted flat-head screw
e. slotted button-head screw
f. set screw with a cone end
g. set screw with a flat end

Figure 5.13 Different types of screws



a. b. c.

Figure 5.14 Representing locking devices

In order to prevent nuts from loosening due to vibrations, etc., various locking devices are used, such as nut locking devices, various types of washers, cotter pins, etc. (see Fig. 5.14); their dimensions and shapes are standardised.

Fig. 5.14a. is an example of locking a slotted nut with a cotter pin made of steel wire of semi-circular cross section. Spring washers are made of spring steel. They present one turn of a coil spring with a square section and sharp edges which prevent the nut from unscrewing (Fig. 5.14b.). A tongued lock washer with its tongue bent on the bolt is shown in Fig. 5.14c.
The left-hand thread in the case of a slotted head screw, is indicated with two notches parallel to the slot as in Fig. 5.15.

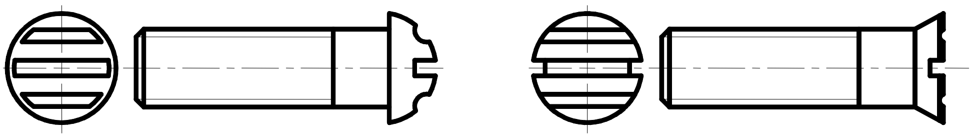


Figure 5.15 Representing left-hand threaded screw

6. TERMINOLOGY OF COMMON MACHINE PART SHAPES

Before setting forth the fundamental rules and methods of machine drawing some common machine element shapes should be defined, as can be seen in Fig. 6.1.

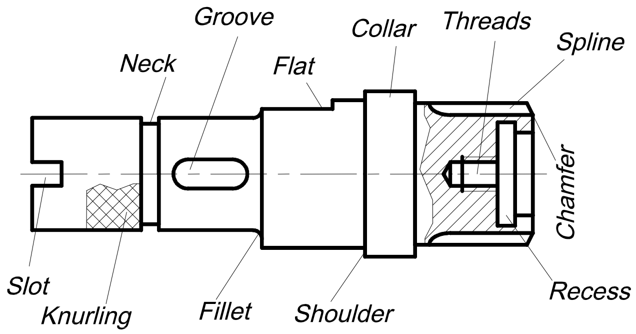


Figure 6.1 Defining common machine element shapes

Chamfer: Short bevelled surface at the ends of shafts, axles, rods, bolts, studs, etc.
Fillet: Short curved surface between two adjoining cylindrical surfaces of different diameters as in stepped shafts.
Collar: Short protruding cylindrical surfaces on shafts and axles.
Shoulder: A plane surface transient from one cross section of a shaft or axle to another.
Groove or recess: Short cylindrical or conic surfaces inside a machine part.
Keyway: Grooves for keys provided on shafts or inside wheel hubs.
Spline: Rectangular or other profiles on shafts and in hubs protruding longitudinally.
Slot: Narrow grooves for a screwdriver.
Knurled surface, or knurling: A fluted surface on a part.

7. CONVENTIONAL REPRESENTATION OF SPRINGS

According to the principle of operation springs can be grouped into compression, tension, torsion, buffer springs, etc.
According to the dominating stress the springs are subdivided as follows:

Springs for bending stress

- Rectangular cantilever spring
- Single-leaf spring
- Multi-leaf spring
- Spiral torsion spring

Springs for torsion stress

- Torsion-bar spring with rectangular or circular cross section
- Helical spring made of round or rectangular wire:
- Compression spring
- Tension spring

Standardized and usually applied notations and terms:

b	Width of spring leaf	mm
c	Spring rate (spring constant)	N/mm
d	Wire diameter	mm
D	Mean coil diameter	mm
E	Modulus of elasticity	N/mm^2
F	Spring force	N
G	Shear modulus	N/mm^2
H	Height of spring leaf	mm
n	Number of active coils	-
M	Spring torque	N mm
s	Spring deflection	mm
α	Angle of rotation	^{o}
σ_b	Bending stress	N/mm^2
τ	Torsional stress	N/mm^2

7.1 Drawing springs

Springs are drawn according to the standards (DIN ISO 2162; MSZ 531). If the number of active coils or leaves of a spring is less than four, it has to be drawn entirely as in Fig. 7.1. If the spring consists of more than four coils or leaves, then as a simplification only one or two coils or leaves should be drawn at each end (besides the supporting coils). In this case the coils or leaves ignored are marked either by centre lines (Fig. 7.2) or by thin continuous lines marking the place of the spring parts (Fig. 7.3). In the case of long springs, presented with a break, the ends can be drawn closer to each other but they have to be presented by scaled figure.

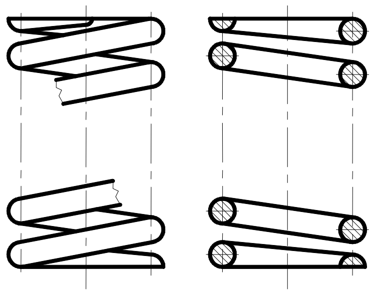


Figure 7.1 Drawing a helical spring in view and in section

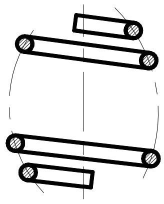


Figure 7.2 Ignoring drawing spring coils with centre lines

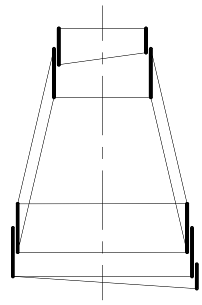


Figure 7.3 Drawing ignored spring coils with thin continuous lines

The graphical presentation of a helical spring is generally done by using straight lines instead of heliclines. If the thickness of a spring coil on a drawing is less than 2 mm, such springs should be presented by single lines. In axial sections of cylindrical and conical springs with the thickness of a coil on a drawing less than 2.5 mm the cross sections of the turns should be made solid black. When a spring is presented in a drawing, it should be drawn in sectional view, if the connecting parts are drawn in sectional view, as well. Springs comprising several leaves having a leaf thickness of 2 mm and less are usually presented in a drawing by a single solid line as in Fig. 7.3. The end turn of a compression spring is normally bent in and ground perpendicularly to the spring axis to provide a supporting surface at the ends as in Fig. 7.1.

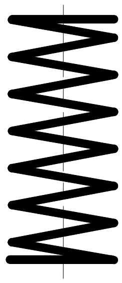


Figure 7.4 Symbolic presentation of a cylindrical helical spring

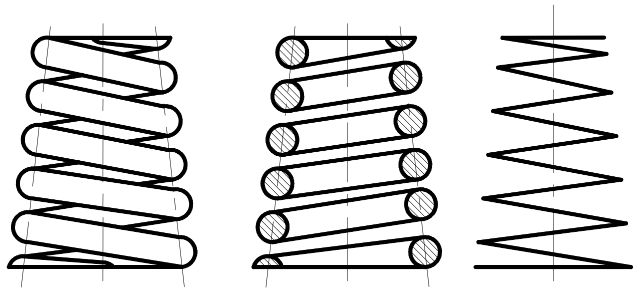


Figure 7.5 Presentation of a conical helical spring in view, in section and symbolic

The symbolic presentation of cylindrical helical spring independently on its wire section can be seen in Fig. 7.4. Fig. 7.5 presents a conical spring in view, in section and by symbolic drawing.

Fig. 7.6 presents a helical extension spring with its spring eye while Fig. 7.7 presents turning helical spring.

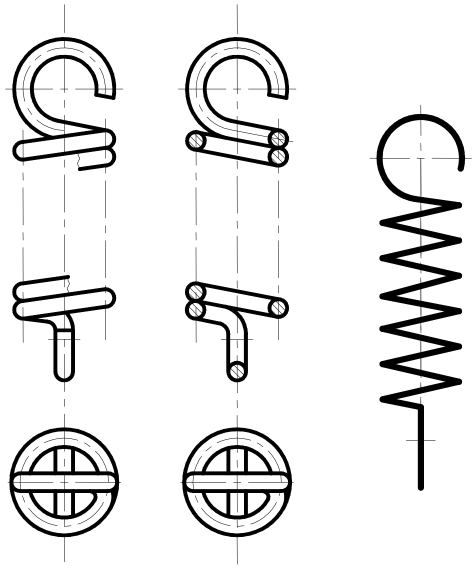


Figure 7.6 Presenting a helical extension spring

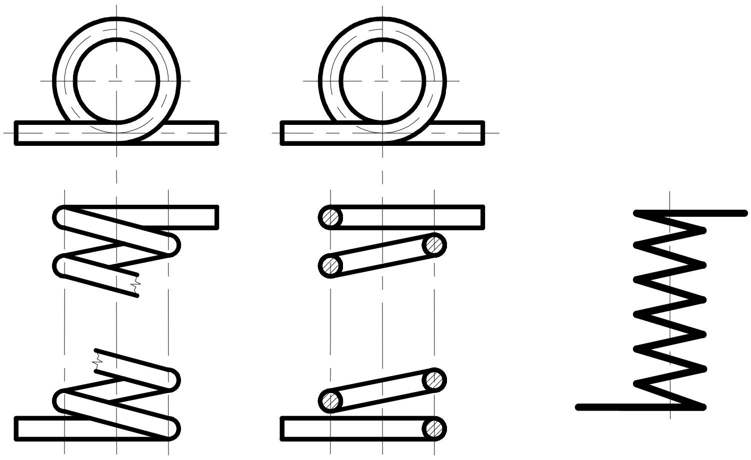


Figure 7.7 Presenting a turning helical spring

A buffer spring is formed by winding a strip into a helix and is presented in Fig. 7.8. Fig. 7.9 presents annular springs containing two types of elements: external and internal ones.

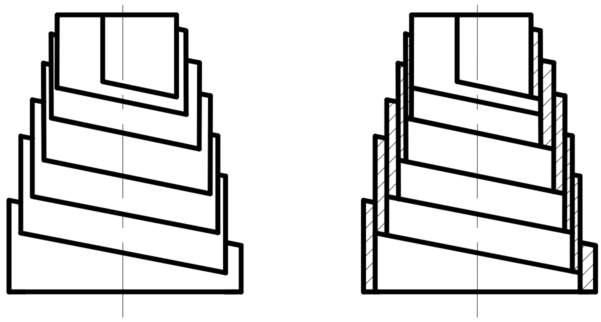


Figure 7.8 Representing a buffer spring

Belleville spring contains identical elements and is presented in Fig. 7.10.

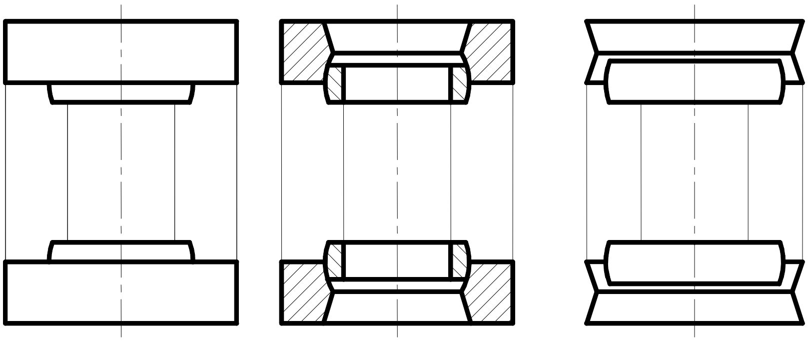


Figure 7.9 Representing an annular spring

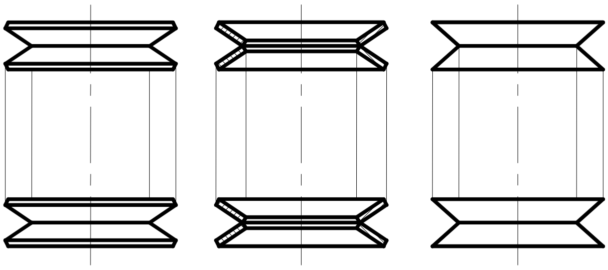


Figure 7.10 Presenting a belleville spring in view, in section and symbolic

A multi-leaf spring can be drawn simplified and symbolic as in Fig. 7.11.

Figure 7.11 Simplified and symbolic presentation of multi-leaf spring

Shop drawing of helical springs

Drawing springs has to be done according to the standard (DIN 2098). The drawing contains a specification table. In the case of a helical spring the specification table has to be completed with the following dimensions: the outside diameter of the spring D_{out}, wire diameter d, pitch P and the free length of the spring Ho.
The following information should also be given: the number of active coils n, the total number of coils n1, the inside diameter Din and the length of the wire L and the direction of the helix: Right-hand or Left-hand.
Drawings of springs have to contain load diagrams (see Fig. 7.12).

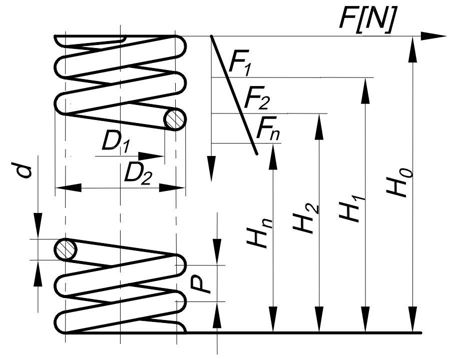


Figure 7.12 Drawing with load diagram

8. REPRESENTATION OF SEPARABLE AND PERMANENT FASTENINGS AND JOINTS

All the fastening of machine parts may be grouped into separable and permanent. Separable fastenings are threaded joints; permanent joints are riveted and welded joints.

8.1 Drawing threaded joints

The standardised representation of threads was introduced in chapter 5.
The same components are used for bolted joints, studded joints and screw fastenings.

8.1.1 Bolted joint

A bolted joint consists of three parts: a bolt, a nut and a washer. The bolt has a chamfered head (in most cases a hexagon head) and a shank with a triangular thread as in Fig. 8.1.

8.1.2 Studded joint

Generally, a stud is a cylindrical rod with threads on both ends. On one of its ends, the stud is screwed into a blind threaded hole of a part. On the other end of the stud a nut or slotted nut is screwed under which the connecting part and a washer are placed (Fig. 8.2).

8.1.3 Screw fastening

The fastening screw is driven into a threaded hole in one of the two pieces to be joined (there is no nut in this case) as in Fig. 8.3. Generally, the different screws are manufactured with various types of heads and ends depending on the purpose. Left-hand bolt and nut has to be indicated in the drawing as in Fig. 8.4.

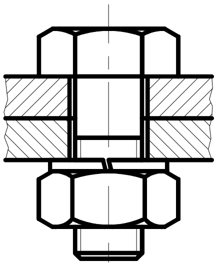


Figure 8.1
Bolted joint

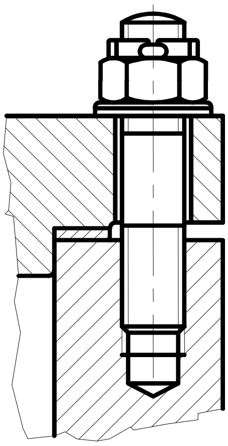


Figure 8.2
Studded joint

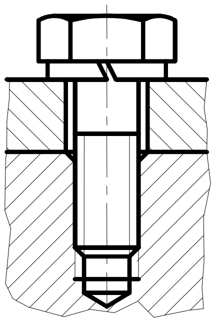


Figure 8.3 Screw fastening

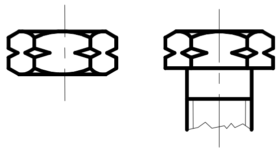
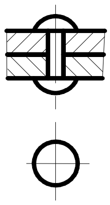
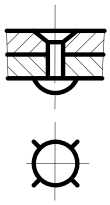
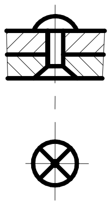
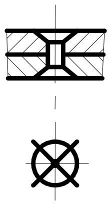


Figure 8.4 Left-hand bolt and nut

8.2 Riveted joints

Rivets are manufactured with a head at one end, and the other head is formed after the rivet is driven into the hole of connecting parts. Holes for the rivets are drilled or punched slightly larger than the rivet diameter. Therefore, it is easy to place it into the hole. The various shapes of rivet heads and their conventional presentation on drawings are shown in Fig. 8.5. The rivets are presented only by centre lines and only some typical riveted joints are shown in section in assembly drawings.

Rivet heads (shape and position)	*Button*	*Countersunk (flat top)*		
	both sides	*near side (outside)*	*far side (inside)*	*both sides*
Conventianal represen-tation				

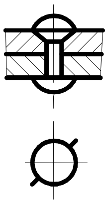
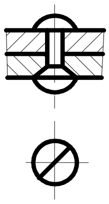
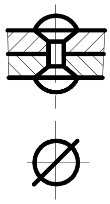
Rivet heads (shape and position)	*Countersunk (round top)*		
	near side (outside)	*far side (inside)*	*both sides*
Conventianal represen-tation			

Figure 8.5 Rivet head shapes

8.2.1 Drawing riveted joints

Fig. 8.6 shows a lap rivet joint while Fig. 8.7 shows a single strap butt joint in section.

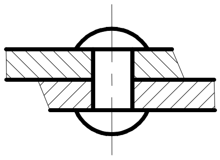


Figure 8.6 Lap rivet joint

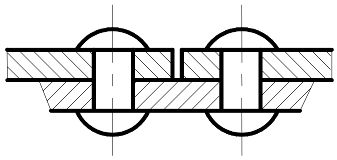


Figure 8.7 Single strap butt rivet joint

8.3 Welded joints

A welded device usually consists of several parts. These parts are fixed to each other by welding joints. These are fastened together in different relative positions. Welded joints can be single-sided or double-sided joints. The type and size of the joining edges are standardised depending on the thickness of the parts to be joined.
However, there are certain standard joints that are frequently proposed for welding operations:

butt joint (Fig. 8.8), a: plain butt joint,
b: single V-butt joint,
c: double V-butt joint

lap joint (Fig. 8.9),
tee joint (Fig. 8.10),
corner joint (Fig. 8.11).

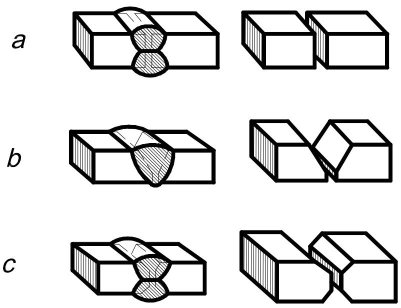


Figure 8.8 Butt joints

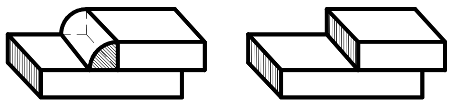


Figure 8.9 Lap joint

While butt welds are used for making butt joints, fillet welds are used for lapping, T and corner joints.

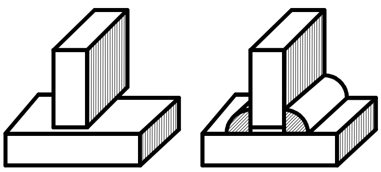


Figure 8.10 T joint

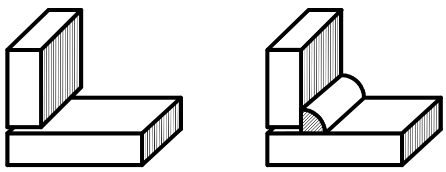


Figure 8.11 Corner joint

8.3.1 Presentation of welded joints in drawings

Presentation must be done according to the standard (DIN 1912; MSZ ISO 2553). Welding is presented in views with thick continuous lines and in section by solid black
The assembly drawing of a welded device can be detailed, as follows:

- Edges are drawn in views and their sections are presented in their state before welding. The cross sections of the welded parts are cross-hatched, as in workshop drawings. However, the welded joints are shown in solid black (Fig. 8.12).
- The exact location of the welding and that of the welded joint has to be given in a simplified way. This method is applied to different steel constructions (Fig. 8.13).
- Welded parts in a welded joint can be cross-hatched in one direction without showing the welded joint itself. It is applied when drawing an assembly drawing of a complex construction and the welded part is not of importance.

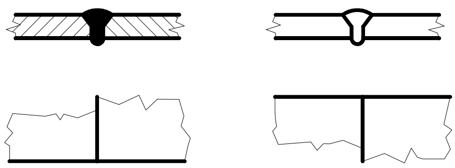


Figure 8.12 Presenting a welding joint in section and in view

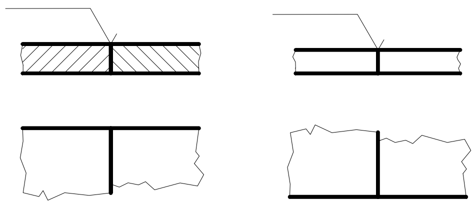


Figure 8.13 Simplified presentation of a welding joint

Welding symbols are standardised. Fig. 8.14 shows the common welded joints and symbols. The commonly used signs of welded joints include the following:

- The dimensions of the cross section of the welding.
- The length and the pitch of the welding.
- Supplementary symbols schematically show the mutual positions of the welding, and indicate the later manufacturing and so on (see Fig. 8.19)

8.3.2 Welding symbols

The welding symbol should be placed, as follows: above the leader for a visible weld, and below it for a hidden one (Fig. 8.15). The welded seams should not be presented. If each weld that is shown in a drawing is made by the same welding method, it should be specified among the manufacturing specifications in the drawing.

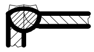
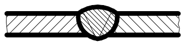
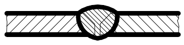
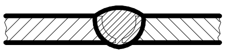
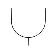
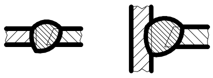
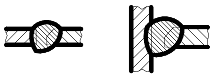
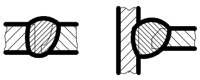
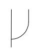
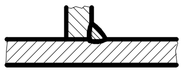
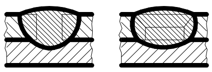
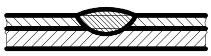
Joint		Symbol	Symbol		
Designation	Section		Angle	Width	symbol
Symmetrical end lap weld			-	*Height x1*	
Asymmetrical end lap weld			-	*x3/4*	
Square-butt weld			-	*x1/2*	
Single-V butt weld			*90°*	-	
Single-V butt weld with broad root face			*90°*	-	
Single-V butt weld with bottom plate			*30°*	*x1*	
Single-U butt weld			-	*x2/3*	
Single-bevel butt weld			*45°*	-	
Single-bevel butt weld with broad root face			*45°*	-	
Single-J butt weld			-	*x1/2*	
Fillet weld			*45°*	*x1*	
Plug weld; plug or slot weld			*30°*	*x2*	
Spot weld			*30°*	*x2*	
Optional butt weld	-		-	-	

Figure 8.14 Welding symbols

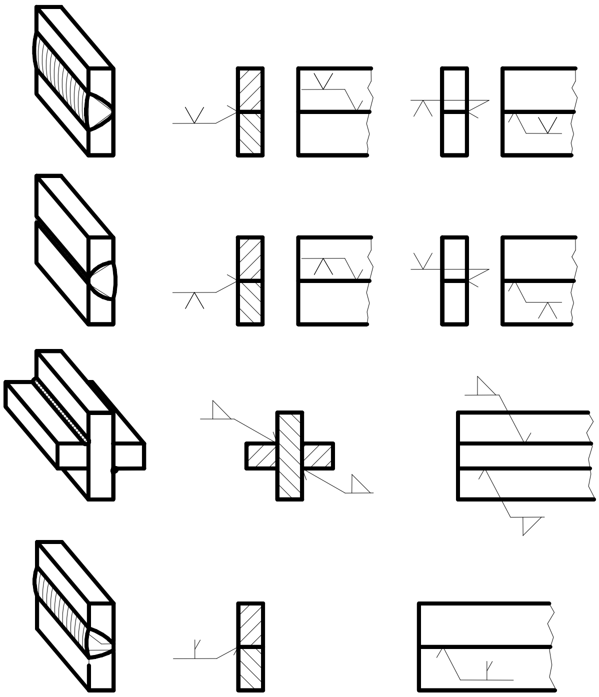


Figure 8.15 Placing welding symbols

Fig. 8.16 shows a dimensioned single V-butt and Y-butt joint and a double V-butt joint.

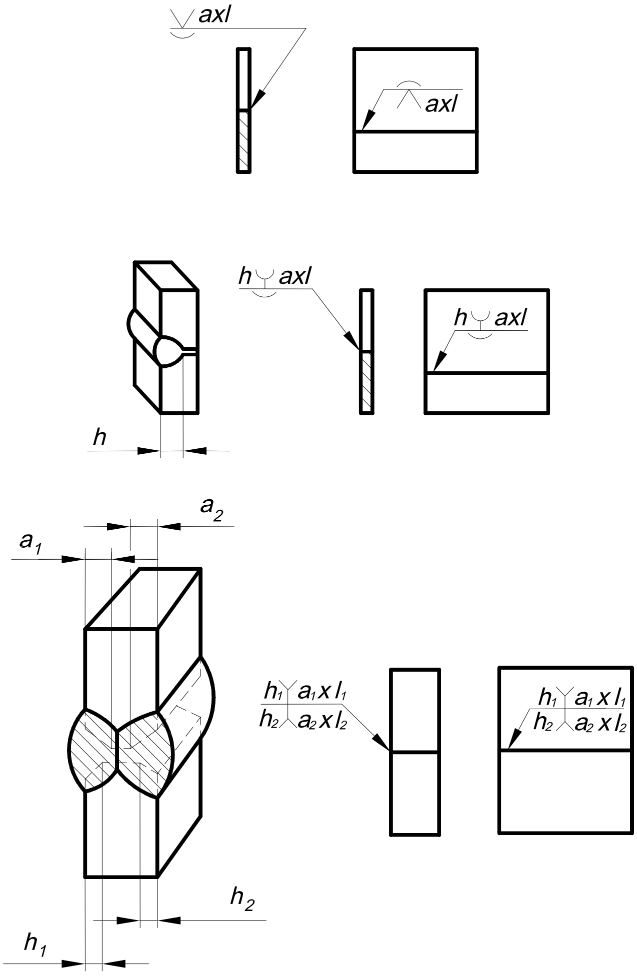


Figure 8.16 Dimensioning single V-butt, single Y-butt and double V-butt joints

Fig. 8.17 and Fig. 8.18 show examples for symmetric and offset intermittent welding, where l': length of the weld without end craters, t: spacing between the adjacent weld elements, N: number of weld elements.

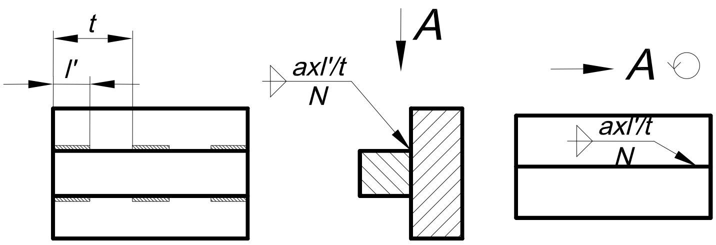


Figure 8.17 Presenting symmetric intermittent welding

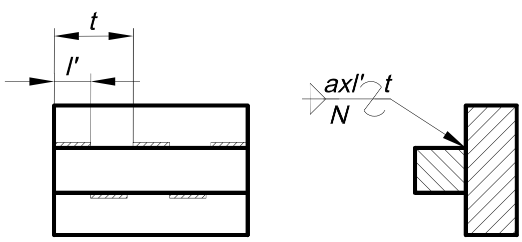
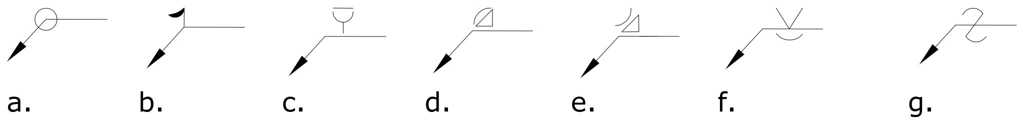


Figure 8.18 Presenting offset intermittent welding

8.3.3 Additional welding symbols

Supplementary welding symbols are used for specifying welding position and shape of weld surface and so on (see Fig. 8.19).



a. welding all around
b. welding on site
c. flat weld surface
d. convex weld surface
e. concave weld surface
f. root weld
g. intermittent welding

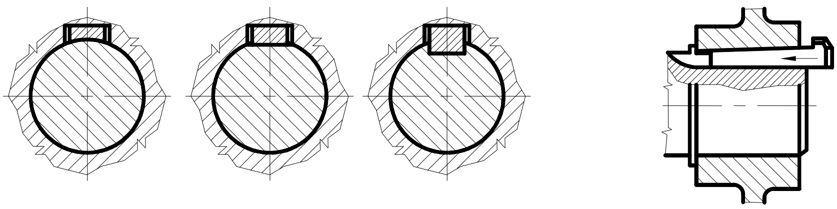
Figure 8.19 Additional welding symbols

9. CONVENTIONAL PRESENTATION OF KEYED JOINTS

Keyed joints are separable and contain three parts: a shaft having a keyway generally, a hub with a keyway and a key. The operation principle of transmitting torque by a keyed joint can be based either on positive or frictional connection. Keyed joints may be grouped as follows.

9.1 Saddle keys

These are taper keys, with uniform width but tapering in thickness on the upper side. The magnitude of the taper is 1/100. Taper keys are made in two forms: hollow and flat.



a. b. c. d.

Figure 9.1 Presenting saddle keys

9.1.1 Hollow saddle keys

The hollow saddle key (DIN 6881) and the gib-head hollow saddle key (DIN 6889) have a concave shaped bottom to suit the curved surface of the shaft, on which they are used (Fig. 9.1a). A keyway is made in the hub for mounting, with a tapered bottom surface. When a hollow saddle key is fitted into its position, the relative rotation between the shaft and the mounting is prevented due to the friction between the shaft and the key.

9.1.2 Flat saddle keys

Flat saddle keys (DIN 6883) and gib-head flat saddle keys (DIN 6884) are similar to the hollow saddle key, except that their bottom surface is flat. The keyway in the hub for mounting is tapered, a flat surface is provided on the shaft that is used to fit this key into its position (Fig. 9.1b).

9.2 Sunk keys

These are the standard forms of keys used in practice, and can be either square, or rectangular in cross-section. The end can be squared or rounded. Generally, half the thickness of the key fits into the shaft keyway and the remaining half in the hub keyway.
These keys are used for heavy duty, as the connection between the key and the shaft is positive. Sunk keys can be classified as: taper keys, parallel or feather keys and woodruff keys.

9.2.1 Taper sunk keys

Taper sunk keys (DIN 6886; MSZ 2303) are square or rectangular in cross-section, uniform in width but tapered in thickness. The bottom surface of the key is straight and the top surface is tapered, the magnitude of the taper is 1/100. Hence, the keyway in the shaft is parallel to the axis and the hub keyway is tapered (Fig. 9.1c). A tapered sunk key can be removed by its gib-head end (DIN 6886; MSZ 2303), see in Fig. 9.1d. The clearance between the side faces of a key and the groove should be shown on the drawing as in Fig. 9.1.

9.2.2 Parallel or feather keys

A parallel or feather key (DIN 6885 (MSZ 12 868) is a sunk key, uniform in width and thickness as well. These keyed joints may be fixed or movable. To achieve this, a clearance fit must exist between the key and the keyway in which it slides (either in the shaft or in the hub).
The feather key may be fixed into the keyway provided on the shaft by two or more screws or into the hub.

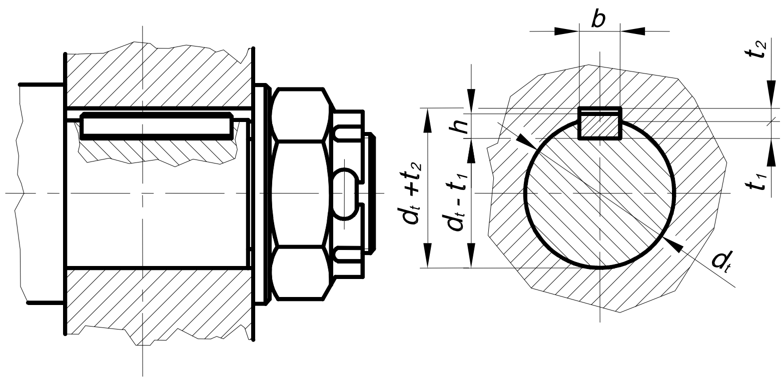


Figure 9.2 Presenting a parallel keyed joint

The parallel key is made of rectangular drawn steel bar. Parallel keys may be rounded or square. There is a small radial clearance between the key and the keyway in the hub, as can be seen in Fig. 9.2.

9.2.3 Woodruff key

A Woodruff key (DIN 6888) is a sunk key, in the form of a segment of a circular disc of uniform thickness (Fig. 9.3). As the bottom surface of the key is circular, the keyway in the shaft is in the form of a circular recess to the same curvature as the key. A keyway is made in the hub for mounting, in the usual manner. Woodruff keys are mainly used on tapered shafts of machine tools.

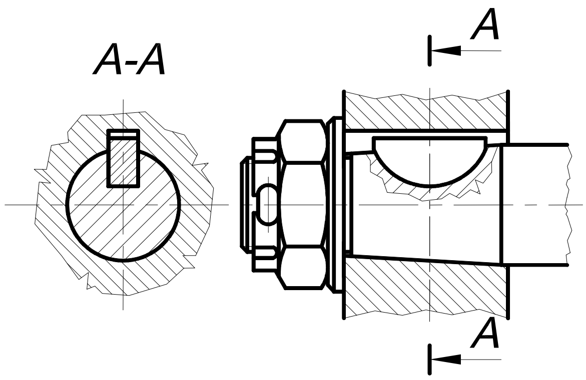


Figure 9.3 Presenting a Woodruff keyed joint

10. SPLINED SHAFT JOINTS

A splined shaft joint is formed by the splines of the shaft and slots of the same profile in the hub, as shown in Fig. 10.1.

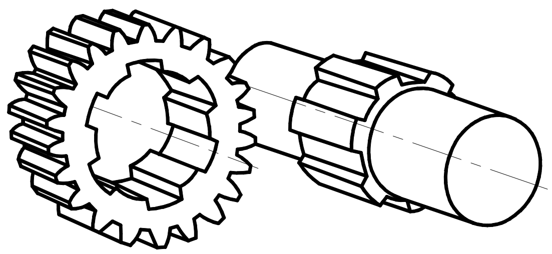


Figure 10.1 Presenting a splined shaft and hub in axonometric projection

The following spline profiles are used:

Parallel-side:	The corresponding fittings are divided into three series: light, medium, and heavy. In this case the hub can be located relative to the shaft in the following three ways: according to the major diameter, according to the minor diameter, and according to the side surfaces of the splines. The ends of shaft and hub splines are usually chamfered or rounded.
Involute spline:	This profile is similar to that of the gears. In this case the mating elements (the shaft and the hub) are located according to the side surfaces of the splines, or according to the major diameter.
Serration:	Serrations are usually small and suitable for transmitting small torques.

10.1 Presentation of splined shaft joints

The presentation of splined shaft joints is standardised (DIN ISO 6413). In longitudinal view, the outline of the splined shaft is drawn with a thick continuous line and the lines corresponding to the minor diameter of the splines are drawn in fine continuous line. The spline run-out and the end of the spline are drawn in thin line as a contrast to the end of a thread.

In end-view and in cross section at least one spline has to be drawn, the circle corresponding to the minor diameter is drawn in fine line, but it does not adjoin to the contour line (see Fig. 10.2).
The splined shaft may not be drawn in longitudinal section unless it is a hollow shaft. In this case, the lines corresponding to the minor diameter are drawn with thick continuous line as well (see Fig. 10.3).

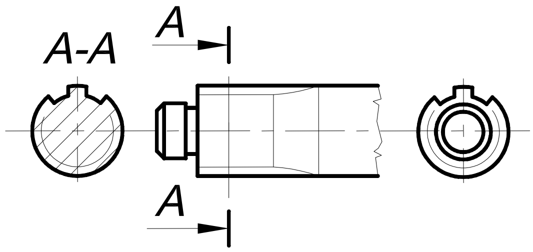


Figure 10.2 Drawing a splined shaft

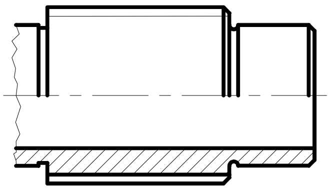


Figure 10.3 Drawing a hollow splined shaft in section

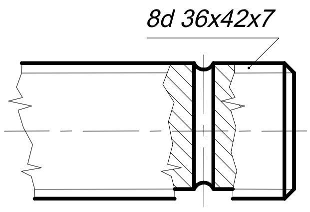


Figure 10.4 Drawing a broken-out section of a splined shaft

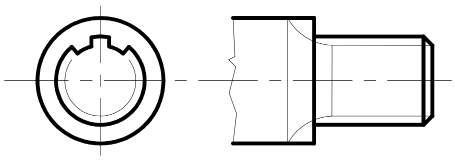


Figure 10.5 Presenting the spline run-out of the shaft

In other cases a broken-out section has to be used in the longitudinal view to show details, eg. oil passage passing through the spline, as in Fig. 10.4. On the end-view the spline run-out is not presented (see Fig. 10.5). In end-view and in cross section of the splined hub at least one spline has to be drawn. The outline of the splined hub and the line corresponding to the minor diameter are drawn by thick continuous line, the circle corresponding to the major diameter is drawn in fine line, but it does not adjoin to the counter line. In longitudinal section the lines corresponding to the minor and major diameters are drawn with thick continuous lines (see Fig. 10.6).

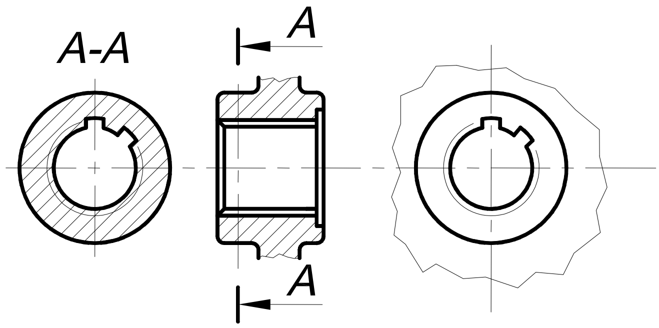


Figure 10.6 Drawing a splined hub in view and in section

If the splined shaft joint is drawn in longitudinal and cross section, the splined shaft covers (hides) the splined hub. In end-view the presentation of the splined shaft is important (see Fig. 10.7).

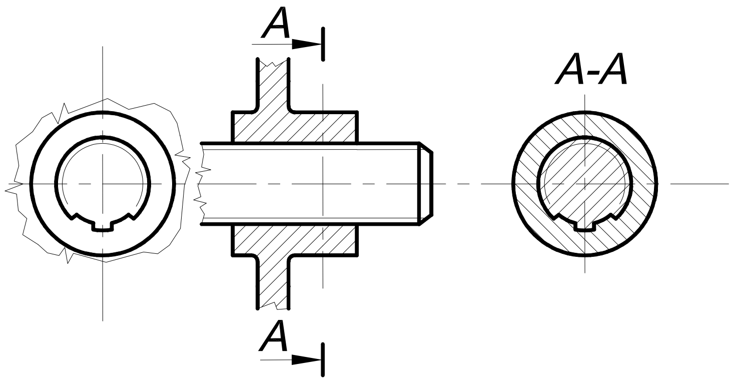


Figure 10.7 Drawing a splined joint in view and in section

11. CONVENTIONAL REPRESENTATION OF GEARS AND TOOTHED PARTS

11.1 Presenting toothed gears

Gear drives are frequently used machine parts. A gear drive is a pair of meshing gears. Gear transmission between parallel shafts is generally designed by the means of spur or helical gears. A bevel gear with straight or spiral teeth is required to transmit motion and power between intersecting shafts. A worm drive is used to transmit torque between non-intersecting shafts. It consists of a worm, which is like a screw with a trapezoidal thread, and a wheel. A rack and pinion gear serves to transform rotary into translational motion and consists of a spur or helical gear and a rack.
The parts of a gear tooth are presented in Fig.11.1.
Presentation of toothed gears is according to the standard (MSZ ISO 2203). When drawing toothed gears, racks, worms, and sprockets the following rules should be taken into account:
Circles and other lines of the top parts of the teeth of spur, bevel and other gears have to be drawn with thick continuous lines. Circles and other lines of the bottom parts have to be drawn by thick continuous lines in sections. On side views, the presentation of the bottom parts is compulsory only if it is the case that some teeth are to be presented (rack, rack circle and anchor wheel). The line or circle corresponding to the bottom parts has to be drawn in thin line. Otherwise the presentation of the bottom part should not be shown in the view. Pitch circles and lines are drawn with thin dot-and-dash lines.

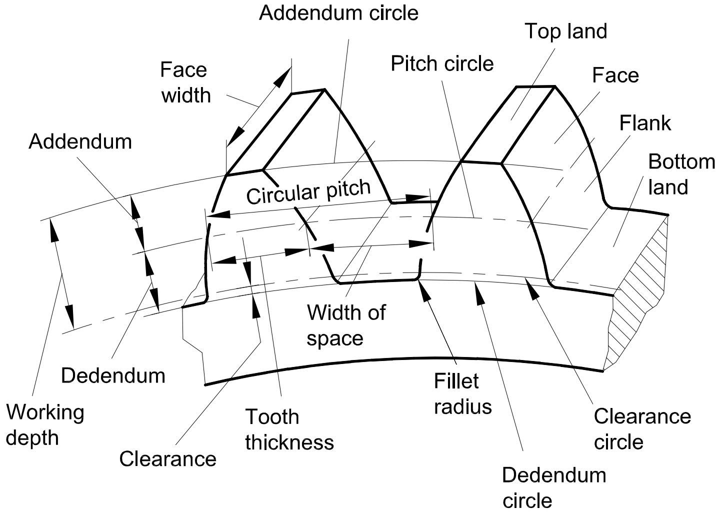


Figure 11.1 Presenting part of a gear tooth in axonometric projection

Quantities and units

a	centre distance	mm
b	face width	mm
c	bottom clearance	mm
d	reference diameter	mm
d_a	outside diameter	mm
d_b	base circle diameter	mm
d_f	root diameter	mm
h_a	addendum	mm
h_f	dedendum	mm
j_n	normal backlash	mm
m	module	mm
n	rotational speed	min^{-1}
p	pitch	mm
s	tooth thickness (circular)	mm
W	span measurement	mm
x	addendum modification coefficient	mm
z	number of teeth	
α	pressure angle	^{o}
β	helix angle	^{o}
ε	contact ratio	

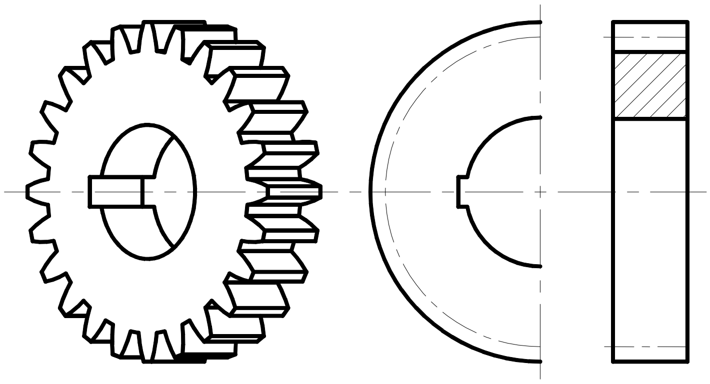


Figure 11.2 Drawing a spur gear

The pitch circle of a bevel gear in the side view pertains to the maximum circle of the pitch cone. If it is necessary to show the profile of a tooth or a worm, then it has to be drawn in a limited area, using a local section or a removed element. The view and sectional view of a toothed gear is independent on the profile and direction of the teeth and the number of the teeth. The shop drawing of a gear contains a specification table arranged in the top right corner of the sheet. Its shape and dimensions are prescribed by the standard. The drawing has to contain cutting and heat-treatment information.

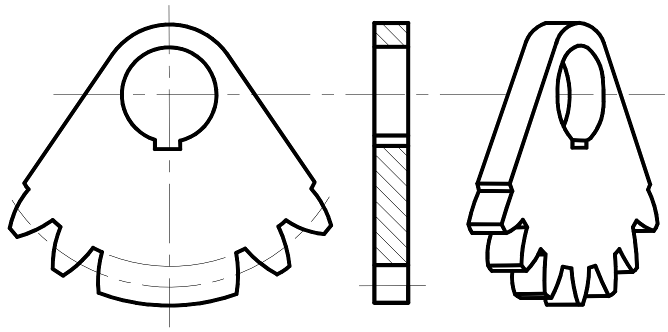


Figure 11.3 Drawing rack circle on view, in section and in axonometric view

Fig. 11.2 is a drawing of a spur gear. The main view is sectioned and presented according to the above rules. On the side view the pitch circle is drawn by dot-and–dash lines.

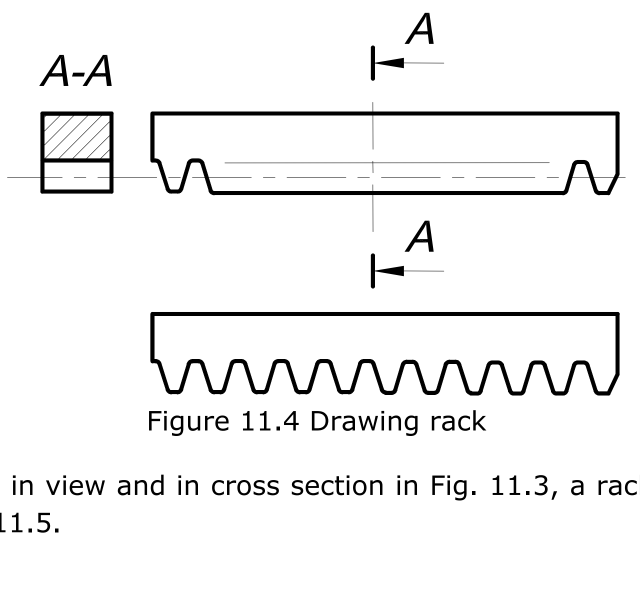


Figure 11.4 Drawing rack

Rack circle presented in view and in cross section in Fig. 11.3, a rack in Fig. 11.4 and an anchor wheel in Fig. 11.5.

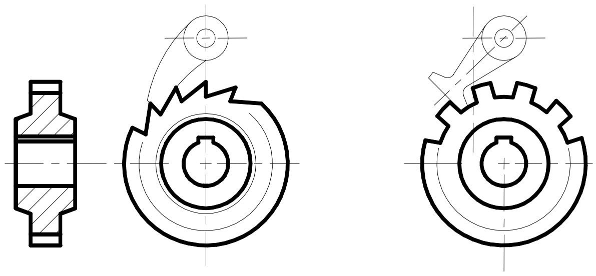


Figure 11.5 Presenting anchor wheel

A worm-wheel is presented in Fig. 11.6, and a bevel gear in Fig. 11.7.

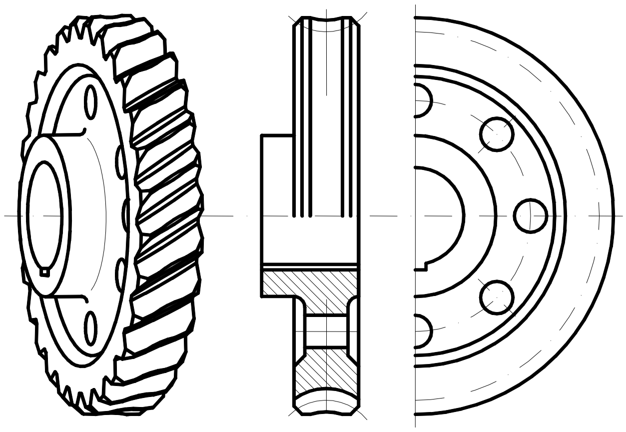


Figure 11.6 Presenting worm-wheel

When drawing meshing gears the following rules have to be applied:

- the rolling circles have to be presented instead of the pitch circles and lines
- in side view both of the addendum circles are drawn in thick line
- in section the gear teeth cover the other ones.

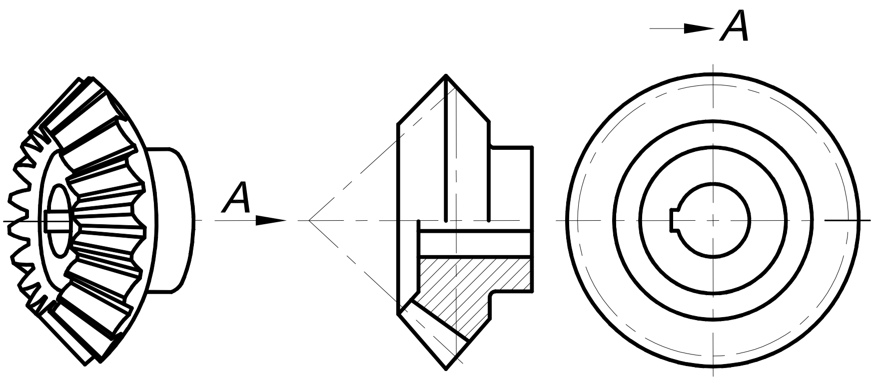


Figure 11.7 Presenting a bevel gear

Meshing spur gears are presented in Fig. 11.8 and rack and rack circle in Fig. 11.9.

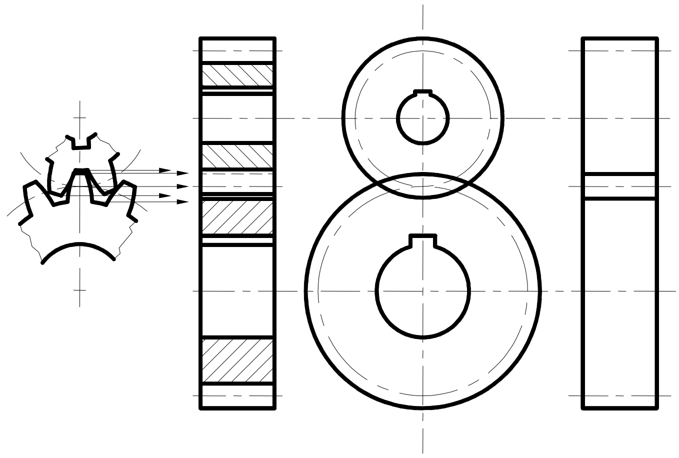


Figure 11.8 Drawing meshing spur gears

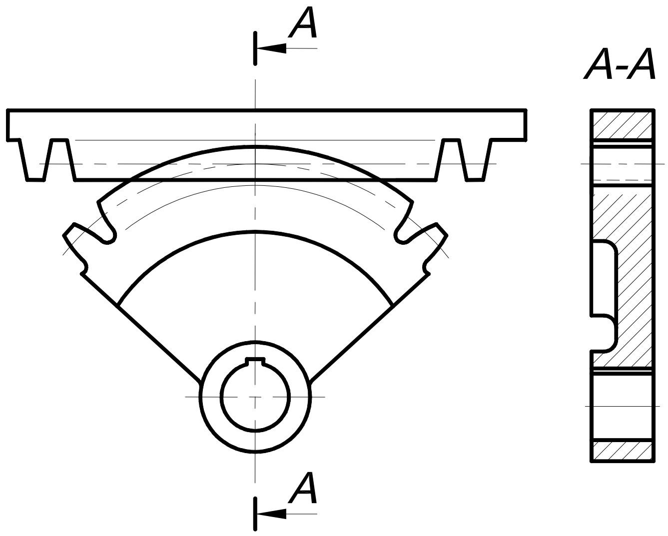


Figure 11.9 Drawing a meshing rack and rack circle

Meshing bevel gears are presented in Fig. 11.10.

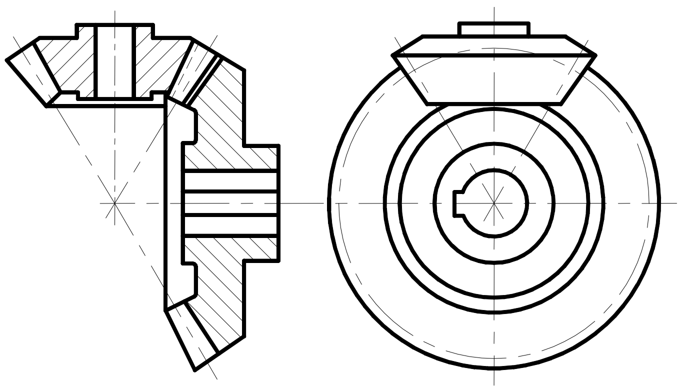


Figure 11.10 Drawing meshing bevel gears

Meshing worm and worm-wheel are presented in cross section and in view in Fig. 11.11.

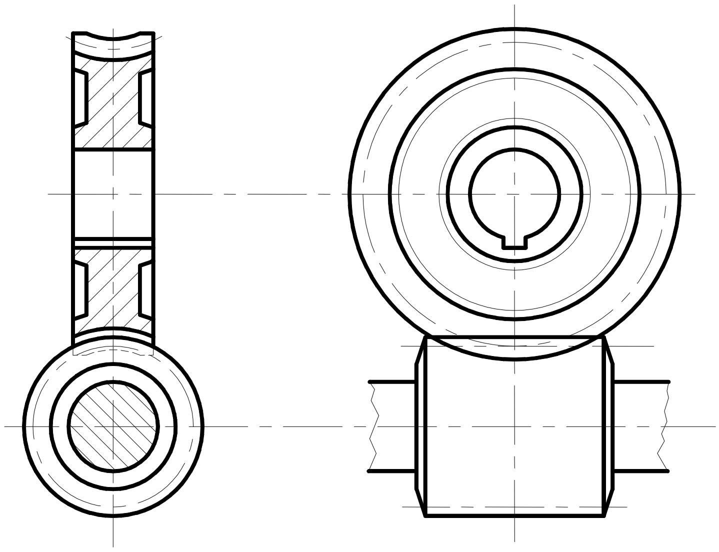


Figure 11.11 Drawing a meshing worm and worm-wheel

11.2 Presenting chain drive

Chain sprockets are drawn in the same way as spur gears, but at least one tooth has to be drawn in the side view (Fig. 11.12). The chain is presented in the drawing by a single dot-and-dash thin line independently in the chain type (Fig. 11.13).

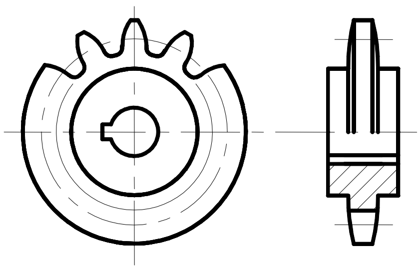


Figure 11.12 Drawing a sprocket

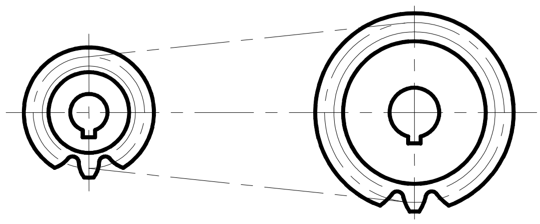
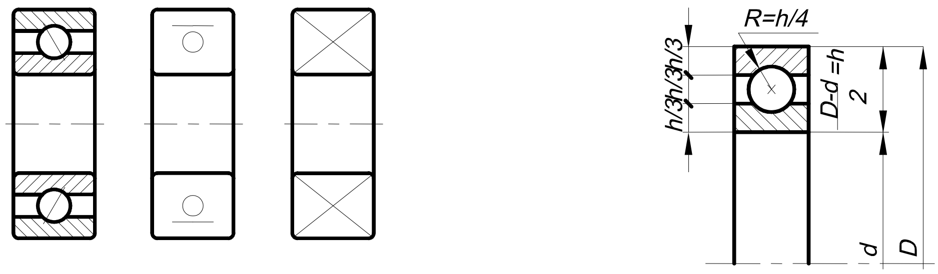


Figure 11.13 Drawing a chain drive

12. CONVENTIONAL REPRESENTATION OF ROLLING BEARINGS

A rolling bearing consists of an inside and outside ring with a raceway, balls or rollers and a bearing cage for ensuring the relative position of the balls, or rollers. Depending on the type of drawing the bearing profile can be shown in different ways.
The drawing can be, as follows:

- simplified sectional presentation (Fig. 12.1.a)
- symbolic presentation (Fig. 12.1.b)
- respective conventional presentation (Fig. 12.1.c)



a. b. c.

Figure 12.1 Drawing a single grooved ball bearing

While the symbolic presentation can be used only in a simple scheme the simplified presentation is suitable for assembly drawing (without ball or roller separator). Although the size of the bearing components generally is not known, the simplified figure of the bearing should be proportional to the size of the real one, as it is in Fig. 12.1.

12.1 Drawing rolling bearing

The simplified presentation of the most widely used rolling bearings in mechanical engineering are, as follows:
Radial rolling bearing:

single grooved ball bearing (Fig. 12.1)
self aligning ball bearing (Fig. 12.2)
single row angular contact ball bearing (Fig. 12.3)
double row angular contact ball bearing (Fig. 12.4)
double row self aligning roller bearing (Fig. 12.5)

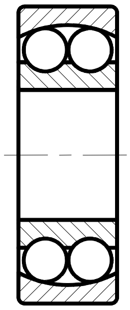


Figure 12.2 Self aligning ball bearing

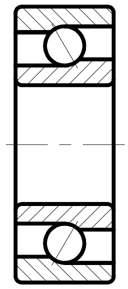


Figure 12.3 Single row angular contact ball bearing

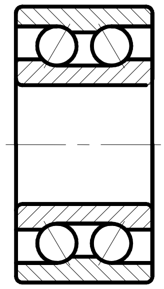


Figure 12.4 Double row angular contact ball bearing

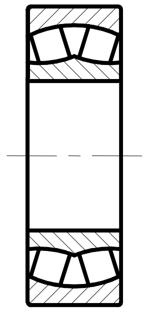


Figure 12.5 Double row self aligning roller bearing

single row cylindrical roller bearin (Fig. 12.6)
needle-roller bearing (Fig. 12.7)
tapered roller bearing (Fig. 12.8)

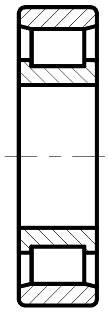


Figure 12.6 Single row cylindrical roller bearing

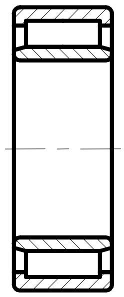


Figure 12.7 Needle-roller bearing

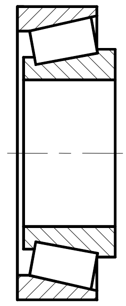


Figure 12.8 Tapered roller bearing

Axial rolling bearing:

single row ball thrust bearing (Fig. 12.9)
double row ball thrust bearing (Fig. 12.10)
self aligning roller thrust bearing (Fig. 12.11)

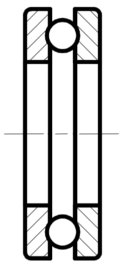


Figure 12.9 Single row ball thrust bearing

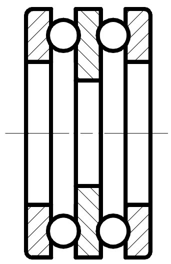


Figure 12.10 Double row ball thrust bearing

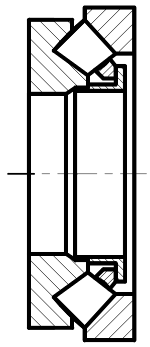


Figure 12.11 Self aligning roller thrust bearing

13. TOLERANCES AND FITS

13.1 Dimension tolerance

It is impossible to manufacture any part to an absolute accurate size. Parts can be made to very accurate dimensions, but high accuracy is extremely expensive. Generally different accuracies are needed according to the functional requirements.
When machining parts by modern precision lathing then the minimum deviation from the wanted dimensions is approximately 5 microns (0.005 mm), and about 2 microns by grinding. The value of deviation depends on the rigidity of the system comprising the part, the machine tool, and the cutting tool, the degree of wear of the machine tool and cutting tool, the metal cutting conditions used and other factors. The allowed error of the parts has to be specified with the widest tolerance zone which meets the requirements. Since higher accuracy costs more, the designer has to specify the tolerance as wide a tolerance as is possible.
With modern machines and especially with mass production the aim is to achieve strictly interchangeable parts without individual fitting, and during repairs any part or unit of the machine has to be replaced without any special adjustment. It can be ensured only by means of components produced or manufactured according to the dimensions tolerated.
To avoid this expense, part selection for assembling method may be applied. In this case relatively high tolerances are specified and parts are inspected after machining and classified into several grades according to actual sizes. The appropriate components can be matched in order to ensure the proper operation. This way satisfactory fits can be obtained at lower costs than by machining all parts to a very accurate dimension. Furthermore, measurement errors should also be taken into consideration.

Dimension tolerance can be:

- maximum or minimum dimensioning: max Ø50, min Ø75
- dimensional tolerance (plus and minus dimensioning)

  The basic size is followed by a plus and minus expression of deviations resulting in either

  a. unilateral tolerance $25^{+0.03}$, $25^{-0.03}$, or

  b. bilateral tolerance $25^{+0.05}_{-0.03}$
- applying the International Organization for Standardization (ISO) system

13.1.1 ISO system to limit allowances and fits

The dimensions of different parts are established when designing a machine, usually as a result of computations or practical evaluation and are rounded off to the nearest whole millimetre. Such a dimension is known as the *nominal size.*
The dimension obtained by machining a part is called the *actual size.* The dimensions of mating surfaces are called *mating dimensions.*
The ISO tolerances are defined by fundamental deviation indicated by letters and a number (IT tolerances grades) determining the size of the tolerance zone. The formulas with which the fundamental deviations are calculated are compiled in the standard (ISO 286).
The **letters** A to Z indicate the position of the tolerance field (basic deviation) relative to the zero line; small letters for shafts; capital letters for holes.
The **numbers** 01 to 18 indicate the size of the tolerance grade.
The ISO tolerance fields for shafts and holes can be combined as desired to provide with an appropriate fit.

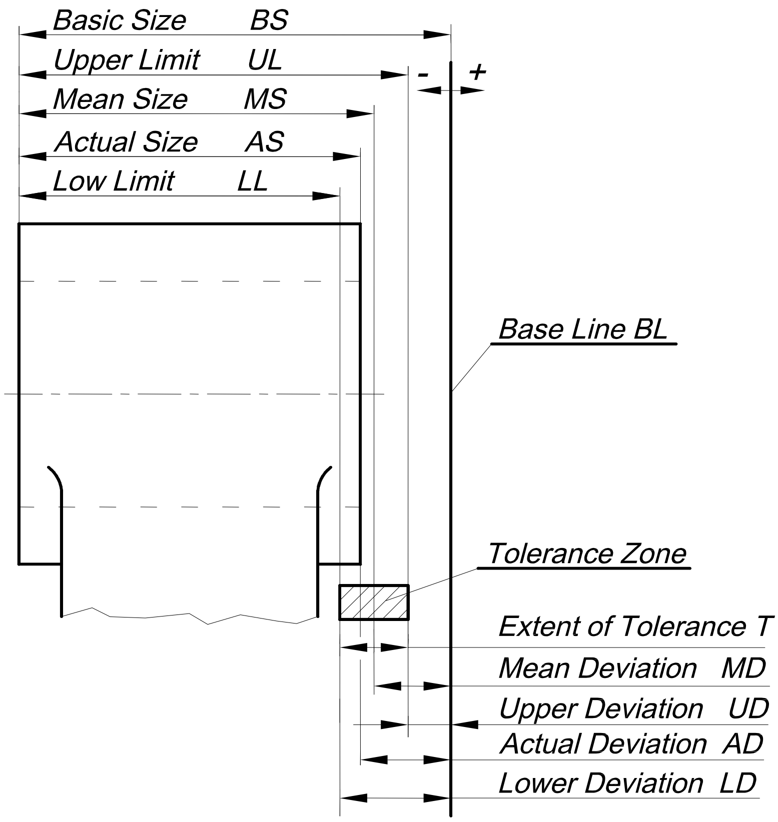


Figure 13.1 Defining the sizes

Fig. 13.1 demonstrates the sizes defined:

Basic size: the size from which deviations are assigned and it is the same for both members of the fit.
Actual size: the measured size of the finished part.
Limits (upper limit and lower limit): the maximum and minimum size indicated by the dimension tolerance.
Deviation: the difference between a size and the corresponding basic size.

Upper/lower deviation: the difference between the upper/lower limit and the corresponding basic size.

Fundamental deviation: either the upper or the lower deviation, depending on which is closest to the basic size. Designation of the fundamental deviation:

for holes: Upper deviation (E' cart superior): **ES**
Lower deviation (E' cart inferior): **EI**

for shafts: Upper deviation (E' cart superior): **es**
Lower deviation (E' cart inferior): **ei**

Tolerance: the total value by which a given dimension may vary. The tolerance is determined by its size and position.

Tolerance zone: the tolerance and its position is in relation to basic size (see Fig. 13.2 for shaft tolerances and Fig. 13.3 for hole tolerances). It is established by a combination of the basic deviation indicated by a letter and the IT grade number.

13.1.1.1 Defining the tolerance

The dimension of the tolerance zone is defined by the following forms:

$$T = qi \quad (13.1)$$

$$T = qI \quad (13.2)$$

$$i = 0.45\sqrt[3]{D} + 0.001D \qquad D \leq 500mm \quad (13.3)$$

$$I = 0.004D + 2.1 \qquad 500 < D \leq 10000mm \quad (13.4)$$

where:

i, I: unit tolerance
D: geometric mean of the nominal dimension range
q: factor depending upon the IT grade

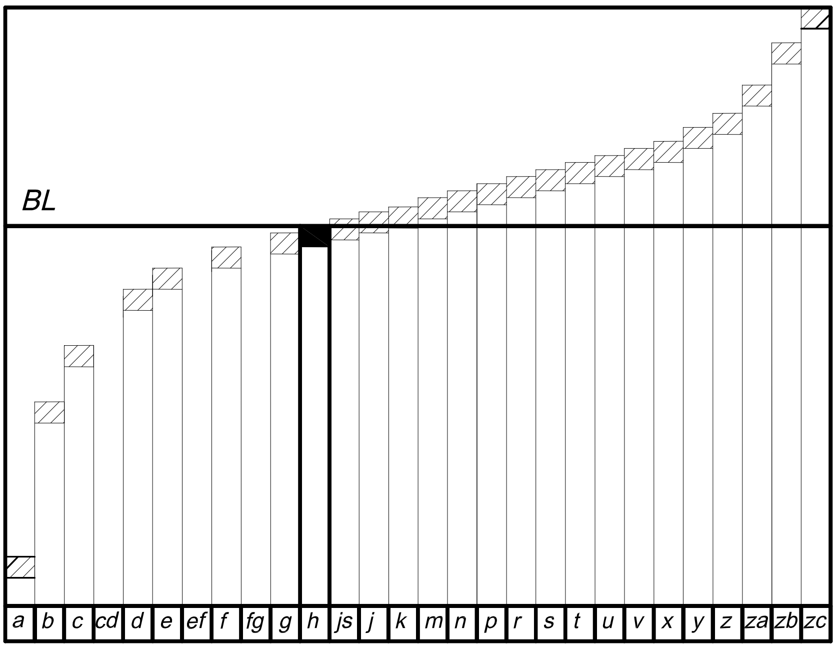


Figure 13.2 Basic deviations for shafts

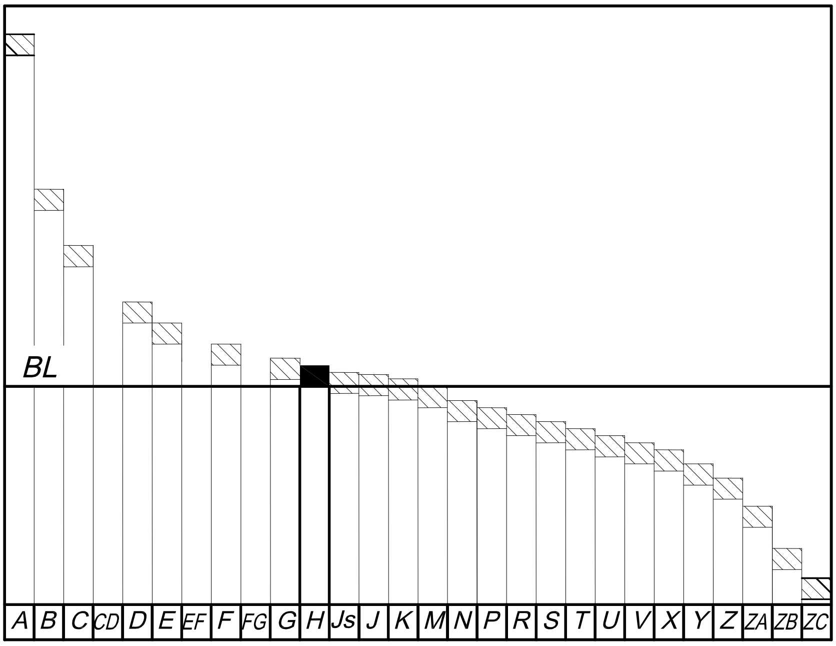


Figure 13.3 Basic deviations for holes

International tolerance grade (IT): there are 19 IT grades. The range is from IT01, IT0, IT1, … to IT17, but only grades IT6 to IT11 are needed for the preferred fits. The values of q pertaining to different IT grades are compiled in Table 13.1.

Table 13.1 The values of q factor

IT grade	5	6	7	8	9	10	11	12	13	14	15	16
q	7	10	16	25	40	64	100	160	250	400	640	1000

The positions of the tolerance ranges relative to the nominal dimension (zero line) are determined by the fundamental deviations. These are calculated by forms defined by the standard. The measures of the fundamental deviations depending on the geometric mean of the nominal dimension range and the IT grade are calculated and compiled in the standard

The positions of the tolerance ranges relative to the nominal dimension (zero line) are determined by the fundamental deviations. These are calculated by forms defined by the standard. The measures of the fundamental deviations depending on the geometric mean of the nominal dimension range and the IT grade are calculated and compiled in the standard. Table 13.2 contains the tolerance values, Table 13.3, 13.4, 13.5, and 13.6 contains the fundamental deviations for holes and shafts.

Table 13.2 Tolerance values

Diameter steps in mm	Tolerance grades																	
	01	**0**	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14***	**15***	**16***
To and inc 3	0.3	0.5	0.8	1.2	2	3	4	6	10	14	25	40	60	100	140	250	400	600
Over 3 To and inc 6	0.4	0.6	1	1.5	2.5	4	5	8	12	18	30	48	75	120	180	300	480	750
Over 6 To and inc 10	0.4	0.6	1	1.5	2.5	4	6	9	15	22	36	58	90	150	220	360	580	900
Over 10 To and inc 18	0.5	0.8	1.2	2	3	5	8	11	18	27	43	70	110	180	270	430	700	1100
Over 18 To and inc 30	0.6	1	1.5	2.5	4	6	9	13	21	33	52	84	130	210	330	520	840	1300
Over 30 To and inc 50	0.6	1	1.5	2.5	4	7	11	16	25	39	62	100	160	250	390	620	1000	1600
Over 50 To and inc 80	0.8	1.2	2	3	5	8	13	19	30	46	74	120	190	300	460	740	1200	1900
Over 80 To and inc 120	1	1.5	2.5	4	6	10	15	22	35	54	87	140	220	350	540	870	1400	2200

Table 13.3 Fundamental deviations for shafts

<table>
<tr><th colspan="16">Fundamental deviation in microns (1 micron = 0.001 mm)</th></tr>
<tr><th colspan="2" rowspan="2">Diameter steps in mm</th><th colspan="8">Upper deviation (es)</th><th rowspan="2">js+</th><th colspan="5">Lower deviation (ei)</th></tr>
<tr><th>a</th><th>b</th><th>c</th><th>d</th><th>e</th><th>f</th><th>g</th><th>h</th><th colspan="3">j</th><th colspan="2">k</th></tr>
<tr><th>over</th><th>up to</th><th colspan="8">All grades</th><td rowspan="14">±IT/2</td><th>5, 6</th><th>7</th><th>8</th><th>4 to 7</th><th>≤ 3 > 7</th></tr>
<tr><td>—</td><td>*3</td><td>− 270</td><td>− 140</td><td>− 60</td><td>− 20</td><td>− 14</td><td>− 6</td><td>− 2</td><td>0</td><td>− 2</td><td>− 4</td><td>− 6</td><td>− 0</td><td>− 0</td></tr>
<tr><td>3</td><td>6</td><td>− 270</td><td>− 140</td><td>− 70</td><td>− 30</td><td>− 20</td><td>− 10</td><td>− 4</td><td>0</td><td>− 2</td><td>− 4</td><td>—</td><td>+ 1</td><td>0</td></tr>
<tr><td>6</td><td>10</td><td>− 280</td><td>− 150</td><td>− 80</td><td>− 40</td><td>− 25</td><td>− 13</td><td>− 5</td><td>0</td><td>− 2</td><td>− 5</td><td>—</td><td>+ 1</td><td>0</td></tr>
<tr><td>10</td><td>14</td><td rowspan="2">− 290</td><td rowspan="2">− 150</td><td rowspan="2">− 95</td><td rowspan="2">− 50</td><td rowspan="2">− 32</td><td rowspan="2">− 16</td><td rowspan="2">− 6</td><td rowspan="2">0</td><td rowspan="2">− 3</td><td rowspan="2">− 6</td><td rowspan="2">—</td><td rowspan="2">+ 1</td><td rowspan="2">0</td></tr>
<tr><td>14</td><td>18</td></tr>
<tr><td>18</td><td>24</td><td rowspan="2">− 300</td><td rowspan="2">− 160</td><td rowspan="2">− 110</td><td rowspan="2">− 65</td><td rowspan="2">− 40</td><td rowspan="2">− 20</td><td rowspan="2">− 7</td><td rowspan="2">0</td><td rowspan="2">− 4</td><td rowspan="2">− 8</td><td rowspan="2">—</td><td rowspan="2">+ 2</td><td rowspan="2">0</td></tr>
<tr><td>24</td><td>30</td></tr>
<tr><td>30</td><td>40</td><td>− 310</td><td>− 170</td><td>− 120</td><td rowspan="2">− 80</td><td rowspan="2">− 50</td><td rowspan="2">− 25</td><td rowspan="2">− 9</td><td rowspan="2">0</td><td rowspan="2">− 5</td><td rowspan="2">− 10</td><td rowspan="2">—</td><td rowspan="2">+ 2</td><td rowspan="2">0</td></tr>
<tr><td>40</td><td>50</td><td>− 320</td><td>− 180</td><td>− 130</td></tr>
<tr><td>50</td><td>65</td><td>− 340</td><td>− 190</td><td>− 140</td><td rowspan="2">− 100</td><td rowspan="2">− 60</td><td rowspan="2">− 30</td><td rowspan="2">− 10</td><td rowspan="2">0</td><td rowspan="2">− 7</td><td rowspan="2">− 12</td><td rowspan="2">—</td><td rowspan="2">+ 2</td><td rowspan="2">0</td></tr>
<tr><td>65</td><td>80</td><td>− 360</td><td>− 200</td><td>− 150</td></tr>
<tr><td>80</td><td>100</td><td>− 380</td><td>− 220</td><td>− 170</td><td rowspan="2">− 120</td><td rowspan="2">− 72</td><td rowspan="2">− 36</td><td rowspan="2">− 12</td><td rowspan="2">0</td><td rowspan="2">− 9</td><td rowspan="2">− 15</td><td rowspan="2">—</td><td rowspan="2">+ 3</td><td rowspan="2">0</td></tr>
<tr><td>100</td><td>120</td><td>− 410</td><td>− 240</td><td>− 180</td></tr>
</table>

Table 13.4 Fundamental deviations for shafts

Fundamental deviation in microns									**(1 micron = 0.001 mm)**						
Diameter steps in mm		**Lower deviations (ei)**													
		m	**n**	**p**	**r**	**s**	**t**	**u**	**v**	**x**	**y**	**z**	**za**	**zb**	**zc**
over	**up to**	**All grades**													
—	**3**	+ 2	+ 4	+ 6	+ 10	+ 14	—	+ 18	—	+ 20	—	+ 26	+ 32	+ 40	+ 60
3	**6**	+ 4	+ 8	+ 12	+ 15	+ 19	—	+ 23	—	+ 28	—	+ 35	+ 42	+ 50	+ 80
6	**10**	+ 6	+ 10	+ 15	+ 19	+ 23	—	+ 28	—	+ 34	—	+ 42	+ 52	+ 67	+ 97
10	**14**	+ 7	+ 12	+ 18	+ 23	+ 28	—	+ 33	—	+ 40	—	+ 50	+ 64	+ 90	+ 130
14	**18**								+ 39	+ 45	—	+ 60	+ 77	+ 108	+ 150
18	**24**	+ 8	+ 15	+ 22	+ 28	+ 35	—	+ 41	+ 47	+ 54	+ 63	+ 73	+ 98	+ 136	+ 188
24	**30**						+ 41	+ 48	+ 55	+ 64	+ 75	+ 88	+ 118	+ 160	+ 218
30	**40**	+ 9	+ 17	+ 26	+ 34	+ 43	+ 48	+ 60	+ 68	+ 80	+ 94	+ 112	+ 148	+ 200	+ 274
40	**50**						+ 54	+ 70	+ 81	+ 97	+ 114	+ 136	+ 180	+ 242	+ 325
50	**65**	+ 11	+ 20	+ 32	+ 41	+ 53	+ 66	+ 87	+ 102	+ 122	+ 144	+ 172	+ 226	+ 300	+ 405
65	**80**				+ 43	+ 59	+ 75	+ 102	+ 120	+ 146	+ 174	+ 210	+ 274	+ 360	+ 480
80	**100**	+ 13	+ 23	+ 37	+ 51	+ 71	+ 91	+ 124	+ 146	+ 178	+ 214	+ 258	+ 335	+ 445	+ 585
100	**120**				+ 54	+ 79	+ 104	+ 144	+ 172	+ 210	+ 254	+ 310	+ 400	+ 525	+ 690

Table 13.5 Fundamental deviations for holes

Fundamental deviation in microns													**(1 micron = 0.001 mm)**							
Diameter steps in mm		**Lower deviations (EI)**														**Upper deviations (ES)**				
		A*	***B**	**C**	**D**	**E**	**F**	**G**	**H**	**Js+**	**J**			**K**		**M**		**N**		
Over	**Up to**	**All grades**									**6**	**7**	**8**	**≤ 8**	**> 8**	**≤ 8 ‡**	**> 8**	**≤ 8**	**> 8***	**≤ 7**
—	**3***	+ 270	+ 140	+ 60	+ 20	+ 14	+ 6	+ 2	0	**± IT/2**	+ 2	+ 4	+ 6	0	0	− 2	− 2	− 4	− 4	**Same dev. as for grades > 7+δ**
3	**6**	+ 270	+ 140	+ 70	+ 30	+ 20	+ 10	+ 4	0		+ 5	+ 6	+ 10	− 1 + δ	—	− 4 + δ	− 4+ δ	− 8 + δ	0	
6	**10**	+ 280	+ 150	+ 80	+ 40	+ 25	+ 13	+ 5	0		+ 5	+ 8	+ 12	− 1 + δ	—	− 6 + δ	− 6 + δ	− 10 + δ	0	
10	**14**	+ 290	+ 150	+ 95	+ 50	+ 32	+ 16	+ 6	0		+ 6	+ 10	+ 15	− 1+ δ	—	− 7 + δ	− 7	− 12 + δ	0	
14	**18**																			
18	**24**	+ 300	+ 160	+ 110	+ 65	+ 40	+ 20	+ 7	0		+ 8	+ 12	+ 20	− 2 + δ	—	− 8 + δ	− 8	− 15 + δ	0	
24	**30**																			
30	**40**	+ 310	+ 170	+ 120	+ 80	+ 50	+ 25	+ 9	0		+ 10	+ 14	+ 24	− 2 + δ	—	− 9 + δ	− 9	− 17 + δ	0	
40	**50**	+ 320	+ 180	+ 130																
50	**65**	+ 340	+ 190	+ 140	+ 100	+ 60	+ 30	+ 10	0		+ 13	+ 18	+ 28	− 2 + δ	—	− 11+ δ	− 11	− 20 + δ	0	
65	**80**	+ 360	+ 200	+ 150																
80	**100**	+ 380	+ 220	+ 170	+ 120	+ 72	+ 36	+ 12	0		+ 16	+ 22	+ 34	− 3 + δ	—	− 13 + δ	− 13	− 23 + δ	0	
100	**120**	+ 410	+ 240	+ 180																

Table 13.6 Fundamental deviations for holes

Fundamental deviation in microns (1 micron = 0.001 mm)																			
Diameter steps in mm		**Upper deviations (ES)**																	
		P	**R**	**S**	**T**	**U**	**V**	**X**	**Y**	**Z**	**ZA**	**ZB**	**ZC**	**δ in microns***					
Over	**Up to**	**> 7**												**3**	**4**	**5**	**6**	**7**	**8**
—	**3**	– 6	– 10	– 14	—	– 18	—	– 20	—	– 26	– 32	– 40	– 60	**δ= 0**					
3	**6**	– 12	– 15	– 19	—	– 23	—	– 28	—	– 35	– 42	– 50	– 80	1	1.5	1	3	4	6
6	**10**	– 15	– 19	– 23	—	– 28	—	– 34	—	– 42	– 52	– 67	– 97	1	1.5	2	3	6	7
10	**14**	– 18	– 23	– 28	—	– 33	—	– 40	—	– 50	– 64	– 90	– 130	1	2	3	3	7	9
14	**18**						– 39	– 45	—	– 60	– 77	– 109	– 150						
18	**24**	– 22	– 28	– 35	—	– 41	– 47	– 54	– 63	– 73	– 93	– 136	– 188	1.5	2	3	4	8	12
24	**30**				– 41	– 48	– 55	– 64	– 75	– 88	– 118	– 160	– 218						
30	**40**	– 26	– 34	– 43	– 48	– 60	– 68	– 80	– 94	– 112	– 148	– 200	– 274	1.5	3	4	5	9	14
40	**50**				– 54	– 70	– 81	– 97	– 114	– 136	– 180	– 242	– 325						
50	**65**	– 32	– 41	– 53	– 65	– 87	– 102	– 122	– 144	– 172	– 226	– 300	– 405	2	3	5	6	11	16
65	**80**		– 43	– 59	– 75	– 102	– 120	– 146	– 174	– 210	– 274	– 360	– 480						
80	**100**	– 37	– 51	– 71	– 91	– 124	– 146	– 178	– 214	– 258	– 335	– 445	– 585	2	4	5	7	13	19
100	**120**		– 54	– 79	– 104	– 144	– 172	– 210	– 254	– 310	– 400	– 525	– 690						

13.1.1.2 Free dimension tolerances

If dimensions are stated without a tolerance, Free Dimensional Tolerances come into consideration.
Methods of prescribing the Free Dimensional Tolerances are as follows:

1. by IT grades, from IT12 to IT17
2. by degrees of accuracy:
   - t_1: fine, approximately: IT12
   - t_2: medium IT14
   - t_3: coarse IT16
   - t_4: very coarse IT17

The standard defines 4 alternatives compiled in Table 13.7.

Table 13.7 Alternatives of free dimension tolerances

	shaft dimension		**hole dimension**		**neither shaft, nor hole**
alternative	**circular**	**else**	**circular**	**else**	
1	- IT		+ IT		± t/2
2	- t		+ t		± t/2
3	± t/2				
4	- IT	± t/2	+ IT	± t/2	± t/2

Where:

- IT single-sided tolerance, corresponds to the “h”, according to the IT grade
+ IT single-sided tolerance, corresponds to the “H”, according to the IT grade
- t single-sided tolerance, defined in negative direction, according to the degree of accuracy
+ t single-sided tolerance, defined in positive direction, according to the degree of accuracy
± t symmetrical tolerance, according to the degree of accuracy

Specifying the Free Dimension Tolerances in the drawing:

e.g.: 1 – 14 MSZ 302

where: 1 alternative

14 IT grade, or degree of accuracy (t_3)

If there is no order in the drawing relating to the Free Dimension Tolerances then the alternative 3 and the medium degree of accuracy comes into consideration as defined in the standard.

13.2 Fits

These arise from the relationship of the tolerances of pairs of components having the same basic size, and which guarantee a specific function (e.g. sliding and guiding requirements or friction contact in shrink joints).

The position of the tolerance zones determines the type of fit (clearance, transition, interference) standardised (MSZ EN 20 286; ISO 286) and the measure of clearance or interference, as shown in Fig. 13.4.

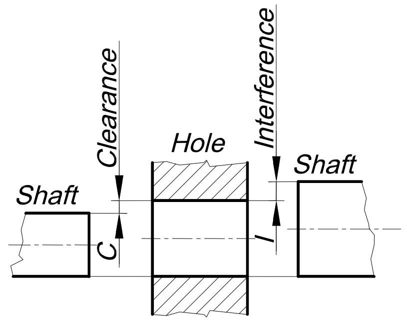


Figure 13.4 Presenting the clearance and interference fit

The ISO tolerance fields for shafts and holes can be combined as desired to give fits, however the hole base and shaft base systems of fits are preferred.
H8/f7: fit; H8: hole tolerance; f7: shaft tolerance
H8: H: basic deviation; 8: IT grade;

13.2.1 Basic hole system

The basic deviation for all holes is identical and the different fits are obtained by appropriately choosing the basic deviations for the shafts. It is applied when using standard tools as reamers, broaches to produce holes and standard plug gages are used to check the actual size. For the generally preferred hole-base system, the fundamental deviation is specified by the uppercase letter H.

13.2.2 Basic shaft system

The basic deviation for shafts is identical, and the different fits are obtained by appropriately choosing the basic deviations for the holes. It is applied when a cold-finished steel bar is used as a shaft. The fundamental deviation is given by the lowercase letter h.

The fits formed between mating parts can be: clearance, transition and interference

Clearance fit: an internal part fits into an external part and always leaves a clearance between the parts (see Fig. 13.5.a) such as loose running fit, free running fit, or sliding fit.
Transition fit: the fit could result in either clearance or interference fit (see Fig. 13.6) such as locational clearance fit or locational transition fit.
Interference fit: the internal part is bigger than the external part there is always an actual interference (see Fig. 13.5.b), such as locational interference fit, medium drive fit, or force fit.

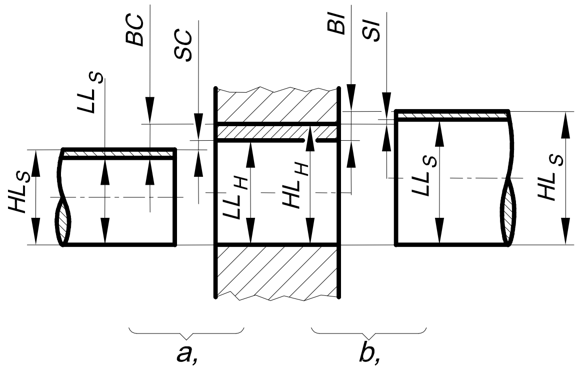


Figure 13.5 Dimensions of clearance and interference fits

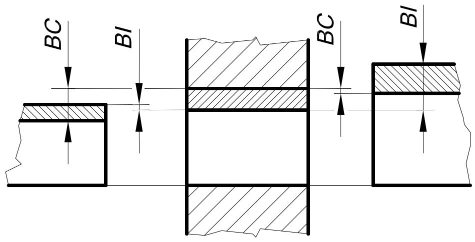


Figure 13.6 Dimensions of a transition fit

The nature of the fit can be either **mean clearance or mean interference** that means the difference between the mean hole size and the mean shaft size (see Fig. 13.7).

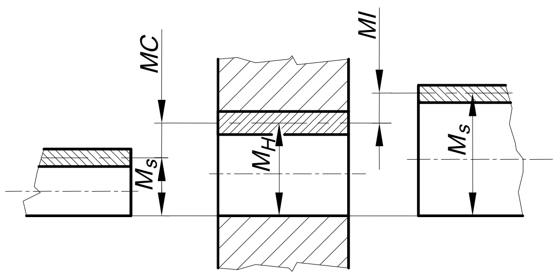
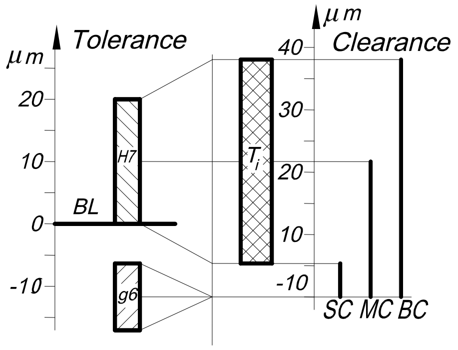


Figure 13.7 Nature of fit

Resultant tolerance of the fit: sum of the tolerances of parts fitted as presented in Fig. 13.8.



$T_i=T_H+T_S$

The resultant tolerance of fit may be calculated on the basis of Fig. 13.5 and Fig. 13.6:

$T_i=BC - SC$
$T_i=BI - SI$
$T_i=BC + BI$

Figure 13.8 Resultant tolerance of the fit

13.2.3 Preferred fits

Preferred fits in basic hole and basic shaft systems are compiled in Table 13.8.

Table 13.8 Preferred fits

Type	Hole basis	Shaft basis	Name and application
Clearance	**H11/c11**	**C11/h11**	***Loose running fit*** for wide commercial tolerances or allowances on external members.
	H9/d9	**D9/h9**	***Free running fit*** not for use where accuracy is essential, but suitable for large temperature variations, high running speeds, or heavy journal pressure.
	H8/f7	**F8/h7**	***Close running fit*** for running on accurate machines and for accurate location at moderate speeds and journal pressures.
	H7/g6	**G7/h6**	***Sliding fit*** not intended to run freely, but to move and turn freely and locate accurately.
	H7/h6	**H7/h6**	***Locational-clearance fit*** provides snug fit for locating stationary parts, but can be freely assembled and disassembled.
Transitio n	**H7/k6**	**K7/h6**	***Locational-transition fit*** for accurate location, a compromise between clearance and interference.
	H7/n6	**N7/h6**	***Locational-transition fit*** for more accurate location where grater interference is permissible.
Interference	**H7/p6**	**P7/h6**	***Locational-interference fit*** for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements.
	H7/s6	**S7/h6**	***Medium-drive fit*** for ordinary steel parts or shrink fits on light sections, the tightest fit usable with cast iron.
	H7/u6	**U7/h6**	***Force fit*** suitable for parts which can be highly stressed or for shrink fits where the heavy pressing forces required are impractical.

13.2.3.1 Application fields of selected fits

(preferred fits are in bold):

Clearance fits:
H11/a11, H11/c11, H11/c9, H11/d11, A11/h11, C11/h11, D11/h11
Fits with big clearances proposed for parts having big tolerances.
Uses: Pivots, latches, fits of parts exposed to corrosive effects, contamination with dust and thermal or mechanical deformations.
H9/C9, H9/d10, H9/d9, H8/d9, H8/d8, D10/h9, D9/h9, D9/h8
Running fits with greater clearances without any special requirements for accuracy of guiding shafts.
Uses: Multiple fits of shafts of production and piston machines, parts rotating very rarely or only swinging.
H9/e9, H8/e8, H7/e7, E9/h9, E8/h8, E8/h7
Running fits with greater clearances without any special requirements for fit accuracy.
Uses: Fits of long shafts, e.g. in agricultural machines, bearings of pumps, fans and piston-based machines.
H9/f8, H8/f8, H8/f7, H7/f7, F8/h7, F8/h6
Running fits with smaller clearances with general requirements for fit accuracy.
Uses: Main fits of machine tools. General fits of shafts, regulator bearings, machine tool spindles, sliding rods.
H8/g7, H7/g6, G7/h6
Running fits with very small clearances for accurate guiding of shafts without any noticeable clearance after assembly.
Uses: Parts of machine tools, sliding gears and clutch disks, crankshaft journals, pistons of hydraulic machines, rods sliding in bearings, grinding machine spindles.
H11/h11, H11/h9
Slipping fits of parts with great tolerances. The parts can easily be slid one into the other and turned.
Uses: Easily demountable parts, distance rings, parts of machines fixed to shafts using pins, bolts, rivets or welds.
H8/h9, H8/h8, H8/h7, H7/h6
Sliding fits with very small clearances for precise guiding and centering parts. Mounting by sliding on without use of any great force, after lubrication the parts can be turned and slid by hand.
Uses: Precise guiding of machines, exchangeable wheels, roller guides.
Transition fits:
H8/j7, H7/js6, H7/j6, J7/h6
Tight fits with small clearances or negligible interference. The parts can be assembled or disassembled manually.
Uses: Easily dismountable fits of hubs of gears, pulleys and bushings, retaining rings, frequently removed bearing bushings.
H8/k7, H7/k6, K8/h7, K7/h
Fits having small clearances or small interferences. The parts can be assembled or disassembled without great force using a rubber mallet.
Uses: Demountable fits of hubs of gears and pulleys, manual wheels, clutches, brake disks.
H8/p7, H8/m7, H8/n7, H7/m6, H7/n6, M8/h6, N8/h7, N7/h6

Fixed fits with negligible clearances or small interferences. Mounting fits by pressing and by small force.
Uses: Fixed plugs, driven bushings, armatures of electric motors on shafts, gear rims, flushed bolts.

Interference fits:

H8/r7, H7/p6, H7/r6, P7/h6, R7/h6

Pressed fits with guaranteed interference. Assembly of the parts can be carried out using cold pressing.
Uses: Hubs of clutch disks, bearing bushings.

H8/s7, H8/t7, H7/s6, H7/t6, S7/h6, T7/h6

Pressed fits with medium interference. Assembling parts by hot pressing. Assembling parts by cold pressing only with large forces.
Uses: Permanent coupling of gears with shafts, bearing bushings.

H8/u8, H8/u7, H8/x8, H7/u6, U8/h7, U7/h6

Pressed fits with big interferences. Assembling parts by pressing and by great forces under different temperatures of the parts.
Uses: permanent couplings of gears with shafts, flanges.

13.2.4 Presentation of tolerances and fits

It is a basic rule that dimension tolerance is given by limit deviations (see Fig. 13.9 – Fig. 3.14) and ISO tolerance is given by its letter symbol (see Fig. 13.15, Fig. 13.18). The limit deviations of ISO tolerance have to be assembled in the tolerance table next to the title block of the drawing. The limit deviation of angles is given in angular degree, angular minute and angular second (Fig. 13.12).

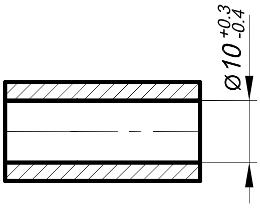


Figure 13.9

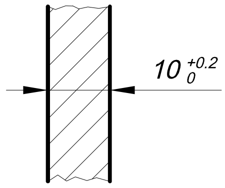


Figure 13.10

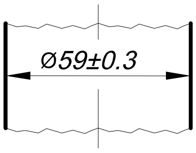


Figure 13.11

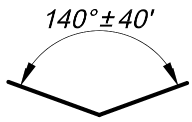


Figure 13.12

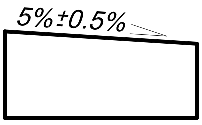


Figure 13.13

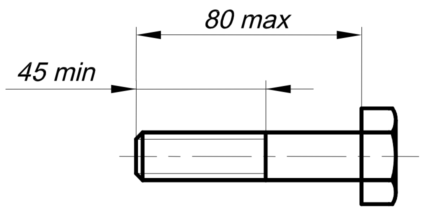


Figure 13.14

Figure 13.9–Figure 13.14. Examples for giving limit tolerances

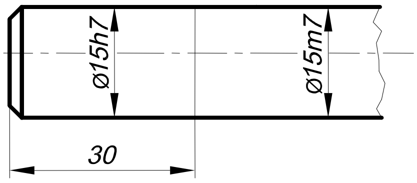


Figure 13.15

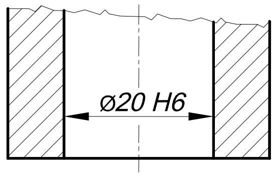


Figure 13.16

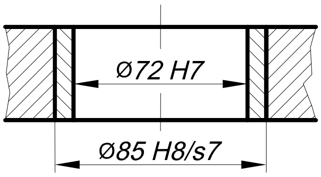


Figure 13.17

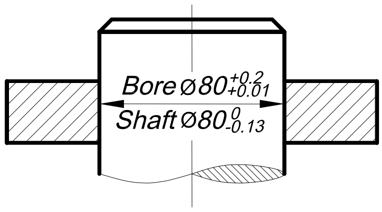


Figure 13.18

Figure 13.15–Figure 13.18 Examples for giving ISO tolerances

Fig. 13.15 shows a ø15 shaft having the same nominal size along its length but different tolerances. These parts of the shaft are indicated with a dividing thin line. Fig. 13.17 shows an example of giving fits by ISO letter symbols and Fig. 13.18 by limit deviations.

13.3 Form tolerance, position tolerance

The inaccurate size of a part in different directions results in a discrepancy from the ideal form and its ideal position from another surface. Form and position tolerance should be prescribed when necessary (e.g. as a result of functional requirements, interchangeability) according to the standard (MSZ ISO 1101).
In order to achieve the desired form and position accuracy of a part, appropriate dimension tolerances have to be defined.

Table 13.9 Form and position tolerance symbols

	Type of Tolerance	**Characteristic**	**Symbol**
For individual features	Form	Straightness	⏤
		Flatness	⏥
		Roundness (Circularity)	○
		Cylindricity	⌭
For individual or related features	Profile	Profile of a line	⌒
		Profile of a surface	⌓
For related features	Orientation	Angularity	∠
		Perpendicularity	⟂
		Parallelism	∥
		Symmetry	⌯
	Location	Position	⌖
		Concentricity	◎
		Symmetry	⌯
	Runout	Circular Runout	↗
		Total Runout	⌰

The form or position tolerance of the part applies to the entire length or surface of the definite part in any direction.

The tolerances are entered into the drawing by means of symbols (Table 13.9).

Form tolerance: straightness, flatness, roundness, cylindricity, profile of any line, profile of any surface.

Position tolerance: parallelism, perpendicularity, slope, position, concentricity, coaxiality, symmetry.

Forms tolerance for individual features:

Straightness, flatness, roundness, cylindricity and profiles (in some instances).

Form and location tolerance for related features:

Angularity, perpendicularity, parallelism, and profile (in some instances) are form or location tolerances applicable to related features. The tolerance zone is defined as before, but it is placed relatively to the datum plane or axis.

Depending on the type of the form tolerance, a tolerance zone is bounded by e.g. two lines or two parallel planes or two concentric circles or two concentric cylinders, within which all points of the considered elements or surface must lie.

13.3.1 Defining form and position tolerance

Form tolerance: it is given as a frame containing the form characteristic symbol and the allowable tolerance. A vertical line separates the symbol and the tolerance.
Position tolerance: the frame contains the position characteristic symbol, the tolerance and the datum reference letter. It can be a point, line, plane, cylinder, or other from which the location or geometric relationship of the feature of a part can be established. The tolerance frame is linked to the part or extension line by an arrow headed line which the tolerance applies to.

13.3.2 Representing form tolerances

Fig. 13.19 and Fig. 13.20 show examples for giving a straightness tolerance. The leader line of the tolerance frame is terminated by an arrowhead and touches either the surface or the extension line connecting to the surface. The tolerance can be applied to the entire surface (Fig. 13.19) or to a given length (Fig. 13.20).

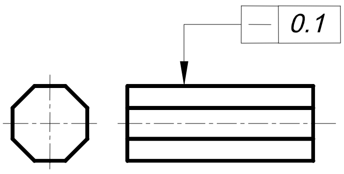


Figure 13.19

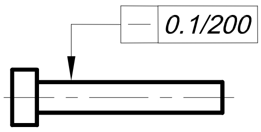


Figure 13.20

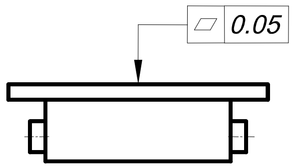


Figure 13.21

A flatness tolerance is shown in Fig. 13.21. Instead of flatness tolerance sometimes straightness tolerances are given in two directions perpendicular to each other as shown in Fig. 13.22.

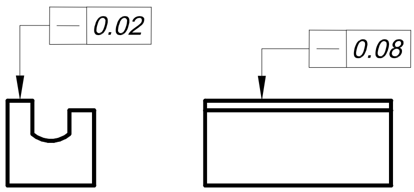


Figure 13.22

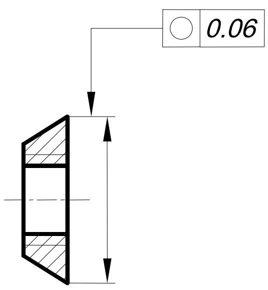


Figure 13.23

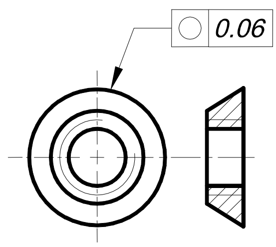


Figure 13.24

Fig. 13.23 – Fig. 13.25 show examples for giving a circularity tolerance and Fig. 13.26 for giving a cylindricity tolerance.

It has to be noted that the leader line must not coincide with the dimension line in Fig. 13.23 otherwise the tolerance would apply not to the surface but to the centre line of it

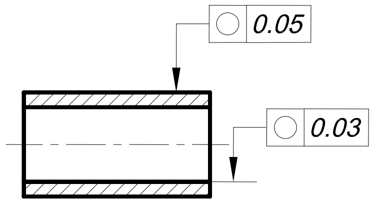


Figure 13.25

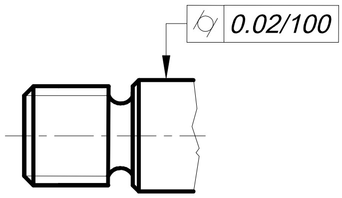


Figure 13.26

Figure 13.19–Figure 13.26 Examples for giving form tolerances

13.3.3 Presenting position tolerances

When giving a position tolerance (related features), one of the two surfaces, lines, centre lines etc., can be indicated as the basis. The leader is terminated in this case in a black triangle (see Fig. 13.27).

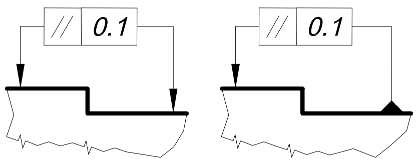


Figure 13.27

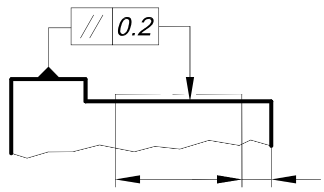


Figure 13.28

If the tolerance concerns only a limited part of the component, then this part has to be indicated by a thin dot-and-dash line and its placement has to be dimensioned as in Fig. 13.28. If the basis is far from the tolerance frame, it may be identified by a capital letter as a datum reference letter. Then the datum reference letter is placed in the tolerance frame as in Fig. 13.29 indicated by letter A. In Fig. 13.29 the surface indicated by letter B, is not the basis.

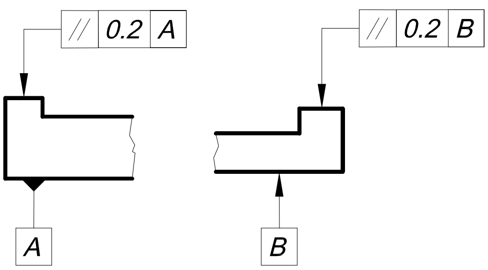


Figure 13.29

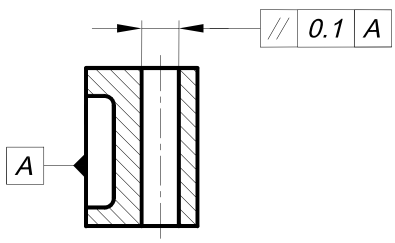


Figure 13.30

The following figures show further examples for giving parallelism tolerance pertaining to the basis (Fig. 13.30) and not to the basis (Fig. 13.31).

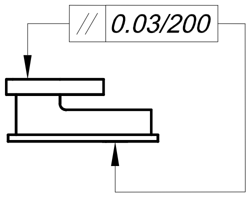


Figure 13.31

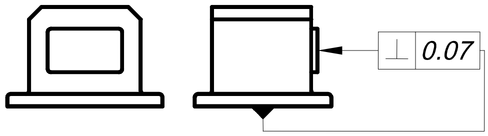


Figure 13.32

Fig. 13.32, 13.33. and Fig. 13.34 present examples for giving perpendicularity tolerance.

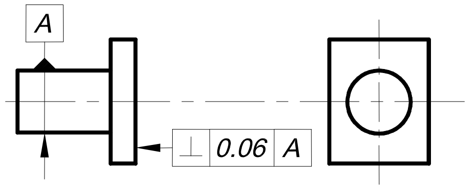


Figure 13.33

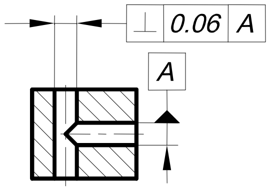


Figure 13.34

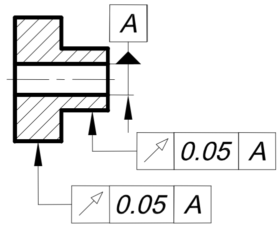


Figure 13.35

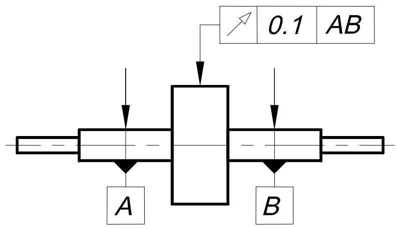


Figure 13.36

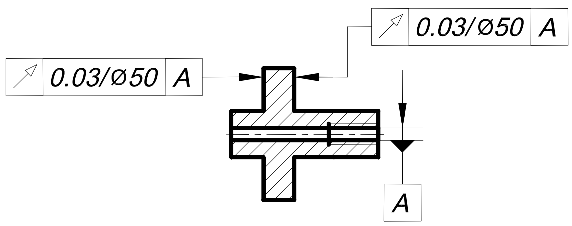


Figure 13.37

Fig. 13.35 and Fig. 13.36 show examples for circular runout tolerances while Fig. 13.37 shows examples for axial runout tolerances.

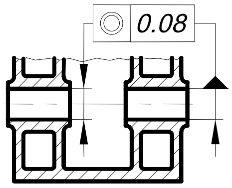


Figure 13.38

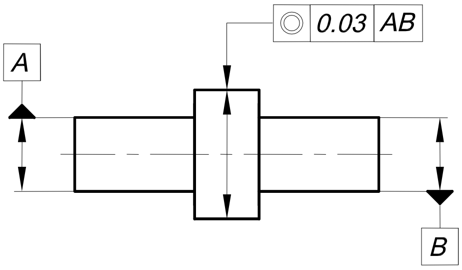


Figure 13.39

Fig. 13.38 and Fig. 13.39 show examples for concentricity tolerances. It can be seen that the basis is the center line of a hole and the tolerance applies to the center line of another hole. Fig. 13.40 shows an example for a symmetry tolerance and Fig. 13.41 shows an example for a position tolerance.

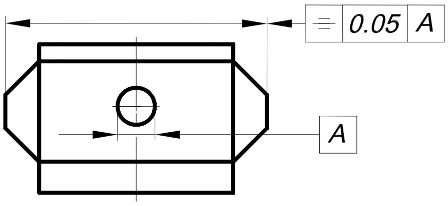


Figure 13.40

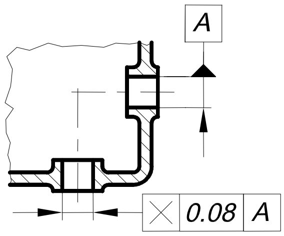


Figure 13.41

Figure 13.27–Figure 13.41 Examples for giving position tolerances

14. SURFACE ROUGHNESS

Depending on operating conditions, the surface of parts must have a certain class of finish. The magnitude of the micro-irregularities, their form and distribution are determined in the standard of surface micro geometry (MSZ ISO 1302).
Classes of surface finish are determined by the average height R_a of the micro-irregularities measured in microns.
Roughness average represented in Fig. 14.1 is defined as the arithmetic average of the absolute values of all profile measurements within the entire measurement section l.

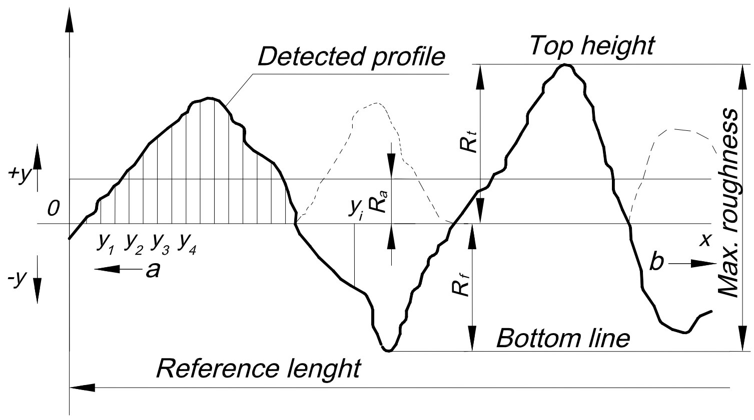


Figure 14.1 The micro-irregularities of the surface

$$R_a = \frac{1}{l}\int_a^b |y| \quad dx, \quad or \quad R_a = \frac{1}{n}\sum_{i=1}^{n}|y_i| \qquad (14.1)$$

Preferred Series Roughness Average Values (R_a) in µm:

0.10 0.20 0.40 0.80 1.60 3.2 6.3 12.5 25

Since the height of irregularities at various places of a given surface is not uniform, R_z the average peak-to-valley height is defined for the micro-irregularities more precisely. Average Peak-to-Valley Height is defined in Fig. 14.2.

Average peak-to-valley height is taken as the arithmetic average of the five highest points of the peaks and the five lowest points of the valleys measured by profilometers from a line parallel to the mean line of peaks and valleys.

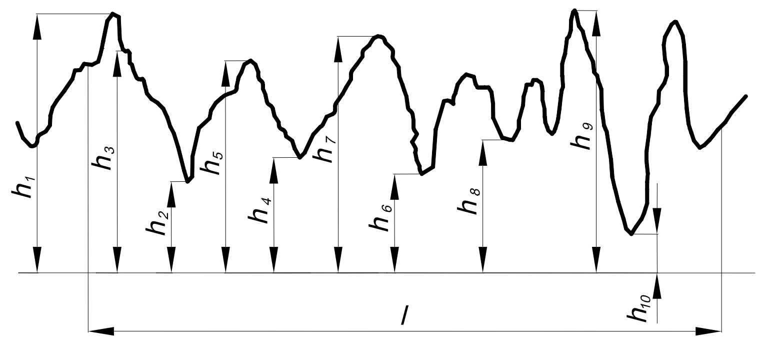


Figure 14.2 Defining the mean Peak-to-Valley Height

$$R_z = \frac{(h_1 + h_3 + h_5 + h_7 + h_9) - (h_2 + h_4 + h_6 + h_8 + h_{10})}{5} \quad (14.2)$$

Preferred Series Mean Peak-to-Valley Height Values (R_z) in μm:

0.10 0.20 0.40 0.80 1.60 3.2 6.3 12.5 25

Measurement for roughness and waviness, unless otherwise specified, has to be performed in the direction that gives the maximum reading that is usually perpendicular to the surface.

The surface roughness should not be confused with the accuracy of dimensional tolerance. For instance, a handle can be polished to a high surface quality class but has a low accuracy of dimensional tolerance, as in this particular case the accuracy of these dimensions is not of importance. However, dimensional accuracy requires a certain class of surface roughness. Since the magnitude of the surface micro-irregularities has to be smaller than the tolerance zone of the dimension applied to the same surface. The required average roughness values depend on the IT grade of the tolerance pertaining to the surface and the dimension range are compiled in Table 14.1.

Generally, the ideal surface finish is the roughest one that will do the job satisfactorily.

Table 14.1 Required average surface roughness

<table>
<tr><th colspan="2">Diameter steps in mm</th><th colspan="16">ISO Tolerance grades</th></tr>
<tr><th>Over</th><th>Up to</th><th>IT1</th><th>IT2</th><th>IT3</th><th>IT4</th><th>IT5</th><th>IT6</th><th>IT7</th><th>IT8</th><th>IT9</th><th>IT10</th><th>IT11</th><th>IT12</th><th>IT13</th><th>IT14</th><th>IT15</th><th>IT16</th></tr>
<tr><td>-</td><td>3</td><td rowspan="5">0.005</td><td rowspan="5">0.01</td><td rowspan="7">0.20</td><td rowspan="4">0.20</td><td rowspan="2">0.20</td><td rowspan="5">0.40</td><td rowspan="2">0.40</td><td rowspan="4">0.80</td><td rowspan="2">0.80</td><td rowspan="4">1.60</td><td rowspan="2">1.60</td><td rowspan="4">3.2</td><td rowspan="2">3.2</td><td rowspan="4">6.3</td><td>6.3</td><td rowspan="3">12.5</td></tr>
<tr><td>3</td><td>6</td><td rowspan="5">12.5</td></tr>
<tr><td>6</td><td>10</td><td rowspan="5">0.40</td><td rowspan="5">0.80</td><td rowspan="5">1.60</td><td rowspan="4">3.2</td><td rowspan="5">6.3</td></tr>
<tr><td>10</td><td>18</td><td rowspan="5">25</td></tr>
<tr><td>18</td><td>30</td><td rowspan="5">0.40</td><td rowspan="6">1.60</td><td rowspan="5">3.2</td><td rowspan="6">6.3</td><td rowspan="5">12.5</td></tr>
<tr><td>30</td><td>50</td><td rowspan="2">0.10</td><td rowspan="4">0.20</td><td rowspan="5">0.80</td></tr>
<tr><td>50</td><td>80</td><td rowspan="7">6.3</td><td rowspan="6">25</td></tr>
<tr><td>80</td><td>120</td><td rowspan="3">0.20</td><td rowspan="5">0.40</td><td rowspan="6">0.80</td><td rowspan="6">1.60</td><td rowspan="6">3.2</td><td rowspan="6">12.5</td></tr>
<tr><td>120</td><td>180</td><td rowspan="5">50</td></tr>
<tr><td>180</td><td>250</td><td rowspan="4">0.40</td><td rowspan="4">0.80</td><td rowspan="4">6.3</td><td rowspan="4">25</td></tr>
<tr><td>250</td><td>315</td><td rowspan="3">0.40</td><td rowspan="3">1.60</td><td rowspan="3">3.2</td><td rowspan="3">12.5</td></tr>
<tr><td>315</td><td>400</td></tr>
<tr><td>400</td><td>500</td><td>0.80</td><td>50</td></tr>
</table>

The attainable accuracy (IT grade) that can be obtained from various production methods is shown in Table 14.2.

Table 14.2 Required tolerance grades

<table>
<tr><th colspan="10">Tolerance Grades</th></tr>
<tr><th>4</th><th>5</th><th>6</th><th>7</th><th>8</th><th>9</th><th>10</th><th>11</th><th>12</th><th>13</th></tr>
<tr><td colspan="2">Lapping and Honing</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td colspan="3">Cylindrical Grinding</td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td colspan="4">Surface Grinding</td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td colspan="3">Diamond Turning</td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td colspan="3">Diamond Boring</td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td colspan="4">Broaching</td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td colspan="5">Reaming</td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td colspan="7">Turning</td></tr>
<tr><td></td><td></td><td></td><td></td><td colspan="6">Boring</td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td colspan="4">Milling</td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td colspan="4">Planing and Shaping</td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td colspan="4">Drilling</td></tr>
</table>

Generally, the ideal surface finish is the roughest one that will do the job satisfactorily. Surface finish is related to the function of a surface, and proper specification of finish of such surfaces as bearings and seals is necessary. Typical range of surface roughness values for different processing methods is shown in Table 14.3, where: ○ average achievable roughness, □ finer, ◇ coarser.

Table 14.3 General guidelines for feasible roughness Ra for different processing methods

General guidelines for feasible roughness Ra for different processing methods

Material removing or separating operations		*roughnes Ra in um*												
		0.012	*0.025*	*0.05*	*0.1*	*0.2*	*0.4*	*0.8*	*1.6*	*3.2*	*6.4*	*13*	*26*	*52*
flame cutting												□	○	◇
sawing									□	○	○	○	◇	
planing								□	○	○	○	◇		
punching							□	○	◇					
chemical treatment								□	○	○	◇			
spark erosion machining								□	○	○	◇			
drilling								□	○	○	◇			
boring								□	○	○	◇			
milling							□	○	○	○	◇	◇		
burning							□	○	○	○	◇			
broaching	√						□	○	○	◇				
reaming							□	○	○	◇				
filing							□	○	○	◇				
grinding						○	○	○	○	◇				
barreling				□	□	○	○	◇	◇					
brushing					□	○	◇							
electrolytic grinding					□	○	◇							
honing				○	○	○	◇							
polishing					○	○	◇							
lapping				○	○	○	◇							
superfinishing				○	○		◇							
Non material removing operation														
sandcasting											□	○	◇	
hot rolling											□	○	◇	
die forging								□	□	○	○	◇		
gravity die casting	√							□	○	◇				
investment casting							□	○	○	◇				
extruding							□	○	○	◇				
cold rolling						□	○	○	○	◇				
die casting							□	○	◇					

14.1 Surface roughness symbols

Surface quality is specified in drawings by a triangular symbol V (triangle with a *h* height of at least 3.5 mm) shown in Fig. 14.3.

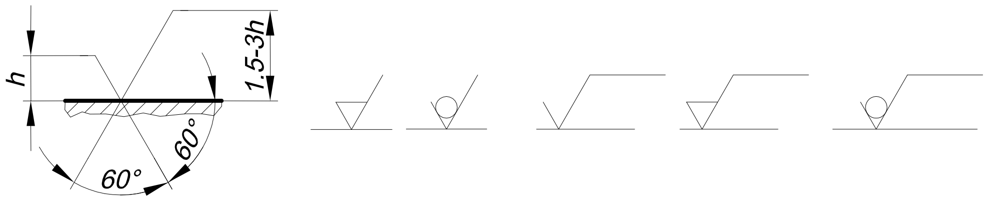
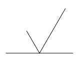
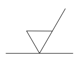
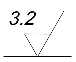
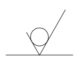
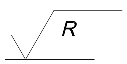


Figure 14.3 Surface roughness symbols

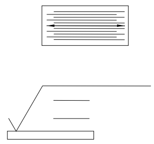
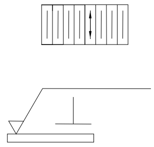
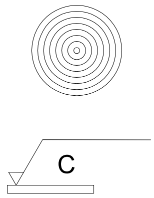
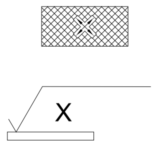
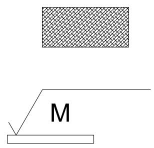
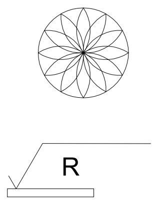
The meaning of different surface texture symbols are compiled in Table 14.4.

Table 14.4 Meaning of surface roughness symbols

Symbol	Meaning
	Basic Surface Texture Symbol. The surface may be produced by any method.
	Material Removal by Machining is required.
	Material Removal Allowance. The number indicates the amount of stock to be removed by machining in millimetres.
	Material Removal Prohibited. The surface must be produced by processes such as casting, forging, hot finishing, cold finishing, die casting, powder metallurgy or injection moulding without subsequent removal of material.
	Surface Texture Symbol. To be used when any surface characteristic are specified above the horizontal line or to the line of the symbol. Surface may be produced by any method except when the bar or circle is specified.

The lower vertex an equilateral of the symbol should touch the line representing the surface to be machined. The symbols may be completed with a leader line as in Fig 14.3. The numerical value in the symbol is written with 2.5 mm letters.
The roughness of the surfaces sliding on each other has a significant influence on decreasing the friction (the surfaces can store lubricant). The symbols specify the direction of lay (Table 14.5).

Table 14.5 Symbols specifying the direction of lay

Symbol	Designation	Example
=	The lay is parallel to the surface to which the symbol is applied.	
⊥	The lay is perpendicular to the surface to which the symbol is applied.	
C	The lay is approximately circular to the center of the surface to which the symbol is applied.	
X	The lay is angular in both directions to the surface to which the symbol is applied	
M	The lay is multidirectional	
R	The lay is approximately radial relative to the center of the surface to which the symbol is applied.	

14.2 Prescribing surface roughness in drawings

When giving R_a and R_z in drawings, the value without denoted R_a or R_z applies to the average roughness. In the case of specifying the Average Peak-to-Valley Height, the R_z sign has to be indicated with the value. If necessary, the lower limit may be given, as in Fig. 14.4. The location of the roughness symbol is shown in Fig. 14.5.

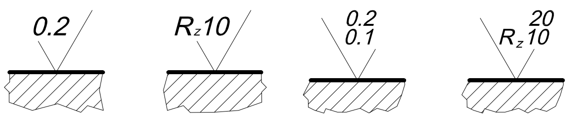


Figure 14.4 Giving average surface roughness and peak to valley height

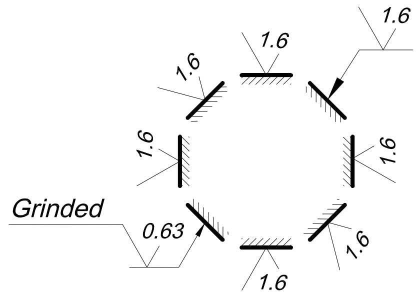


Figure 14.5 Placement of the surface roughness symbol

If the same roughness specification is applied to all or to the majority of the surfaces, the surface roughness symbol with the value should be placed at the top right-hand corner of the sheet.
The auxiliary roughness symbol in parentheses without a value indicates that there are surfaces to which other specifications apply to, as in Fig. 14.6.

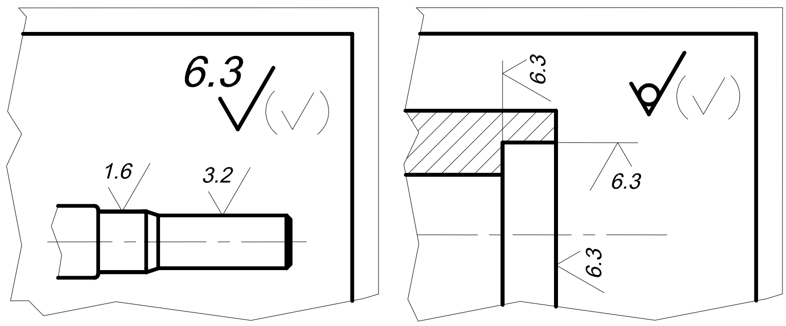


Figure 14.6 Placement of the general average surface roughness symbol

In the case of surfaces, connected to each other and having the same roughness specification, the surface roughness which applies to the whole part can be given on just one surface marked that it applies to all round the part (see Fig. 14.7.a). In the case of dimensioning a chamfer the surface roughness can be given as in Fig. 14.7.b Fig. 14.7.c shows an example for simplified dimensioning of a bore with the roughness specification placed on the leader line.

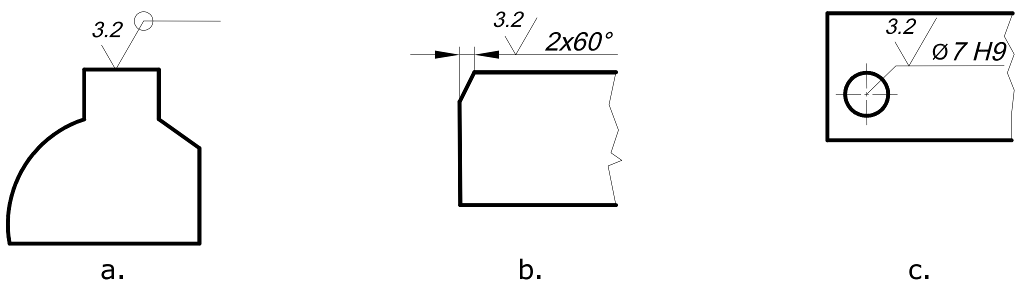
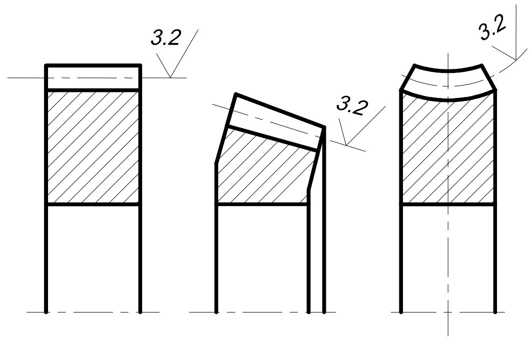


Figure 14.7 Examples for giving average surface roughness

The roughness of a tooth surface has to be given on the pitch circle (Fig. 14.8). The roughness of the base of a gear tooth can be given on the root-circle or root-line (Fig. 14.9).



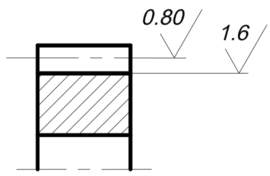
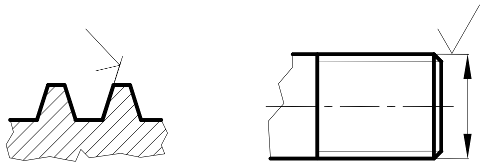


Figure 14.8 Giving average surface roughness on the-pitch circle

Figure 14.9 Giving average surface roughness on the root-line

The roughness of screw threads is not given in general. However, if it is necessary, it can be given either on the profile, or together with the dimensioning of the thread, as in Fig. 14.10 and Fig. 14.11. In machine drawing the surface roughness pertains to the completed part. When the surface of a part has a metallic coating it pertains to the coated state, but when the surface is painted or has a plastic coating, it pertains to the state before coating (Fig. 4.2).



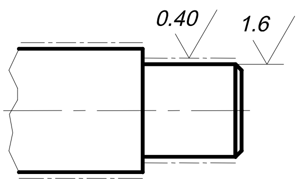


Figure 14.10 Giving surface roughness of the thread profile

Figure 14.11 Giving surface roughness of a coated surface

Fig. 14.12 shows a shaft having the same nominal size along its length but different surface roughness. The parts of the shaft are indicated with a thin dividing line in the view. The different surface roughness can be given in the view or in the section to show that the surfaces differ from each other.

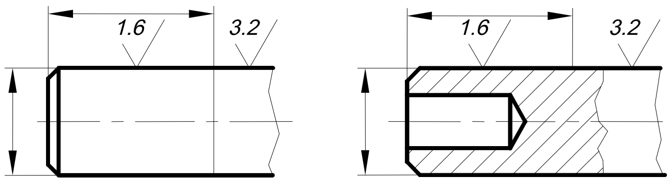


Figure 14.12 Giving different surface roughness of the same surface

15. FUNDAMENTAL MATERIAL PROPERTIES AND MECHANICAL RELATIONS

The principal material properties can be determined from the standard tensile test. It is used to obtain a variety of material characteristics and strengths to be used in design. The standardised tension test specimen and its dimensions can be seen in Fig. 15.1.

The original diameter d_0 and the gauge length l_0 are used to determine the deformations. After fitting the specimen into the test machine, it is slowly loaded in tension while the load F and elongation are monitored. The load can be converted to stress by calculation:

$$\sigma = \frac{F}{A_0} \qquad \text{where: } A_0 = \frac{d_0^2 \pi}{4} \qquad (15.1)$$

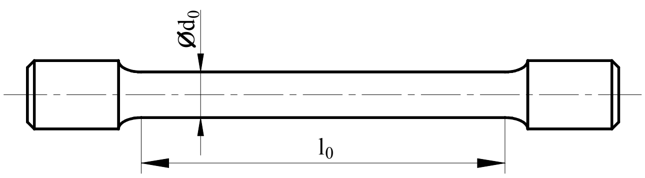


Figure 15.1 Standardized tension test specimen

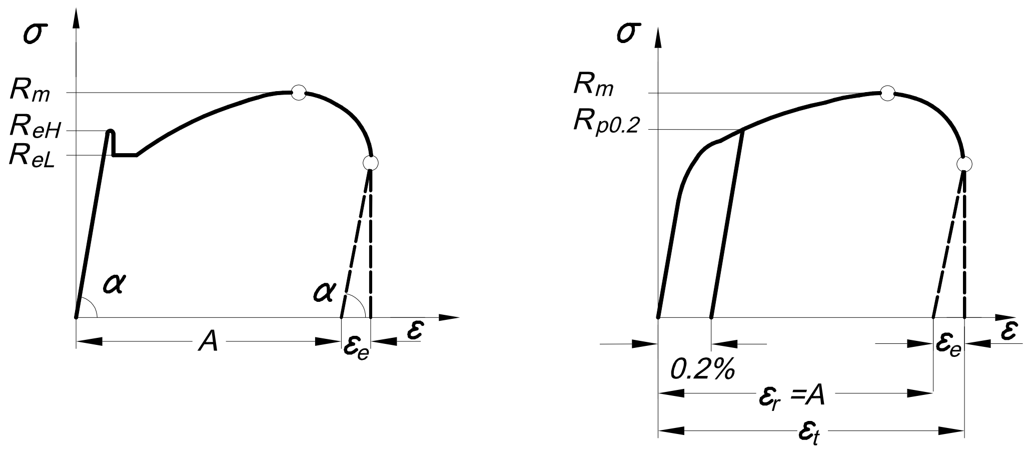
The elongation of the gauge is given by $l - l_0$ where l is the gauge length corresponding to the load F. The strain is calculated from:

$$\varepsilon = \frac{l - l_0}{l_0} \qquad \text{where: } \varepsilon \text{ is strain for elastic deformation} \qquad (15.2)$$

The result of the test is a stress-strain diagram shown in Fig. 15.2 for typical ductile and brittle materials.

Ferriferous materials that have tensile strength (Rm)

- for lower than 500 MPa the yield strength is ReH
- for higher than 500 MPa the proportional limit is $R_{p0,2}$.



a. Ductile material b. Brittle material

Figure 15.2 Stress-strain diagram obtained from the standard tensile test

15.1 Estimation of the tensile strength

Hardness is a property of solid materials and defined as the material's resistance to deformation. In metallic materials the hardness is also used to assess mechanical properties such as strength, plasticity, or wear resistance. In accordance with DIN 50150 standard the measured hardness can be converted to tensile strength. In order to obtain a characteristic value, indentation created by deformation with a defined test indenter under a defined load is measured.
A distinction is made between static and dynamic testing. In static testing, the permanent deformation caused by the indenter is measured. Standard or widely-used hardness tests are the Rockwell, Vickers, Brinell test methods. In dynamic testing, the rebound of an indenter from the flat surface of the test specimen is measured.
A further possibility is deformation by scratching with a hard indenter and measurement of the score width as the basis for calculating hardness.

15.1.1 Rockwell hardness

This method is particularly well suited to large-scale testing of metallic workpieces. The steel or diamond indenter is positioned vertically to the surface of the workpiece to be tested and a definite small load is applied. A definite high load is then applied for at least 30 seconds. After that the load is removed and the depth of permanent penetration e in mm gives the Rockwell hardness value (see Table 15.1).

Table 15.1 Rockwell test methods

Abbreviation	Intender	Small load, N	High load, N	Hardness number (e intender penetration depth)	
HRC	Diamond pyramid	98 ± 2	1471 ± 9	100-e/0.002	20…70
HRA			588 ± 5		60…881
HRB	Steel ball		980 ± 6.5	130-e/0.002	35…100
HRF			588 ± 5		60…100
HR15N HR30N HR45N	Diamond pyramid	29.4 ± 0.6	147 ± 1 294 ± 2 441 ± 3	100-e/0.001	66…92 39…84 17…75
HR15T HR30T HR45T	Steel ball	3 ± 0.06	15 ± 0.1 30 ± 0.2 45 ± 0.3		50…94 10…84 0…75

15.1.2 Brinell hardness

The method is used for metallic materials of low to medium hardness. A hard metal or hardened steel ball is used for testing with varying load F for at least 15 seconds. The diameter of the trace remaining after the ball is removed is measured and used as the basis for calculation of Brinell hardness which can be calculated, as follows:

$$\text{Brinell hardness} = \frac{0.204F}{\pi D\left(D - \sqrt{D^2 - d^2}\right)} \text{ HB} \tag{15.3}$$

Where: F applied load, N
D ball diameter, mm
d diameter of indenter, mm

15.1.3 Vickers hardness

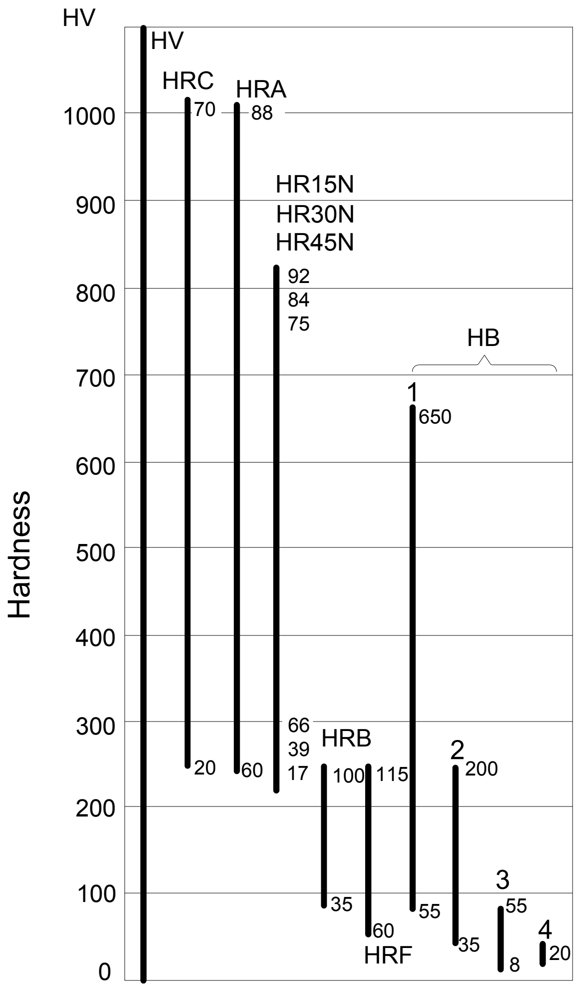
This test method can be used for all metallic materials, regardless of hardness, for usually small and thin parts and in particular for workpieces of a hardened surface. A square-based diamond pyramid with an angle of 136° is used as the indenter, and is applied to the surface of the test specimen with varying load F. The diagonal, d, of the rhombic trace left after removing the intender is measured and Vickers hardness can be calculated, as follows:

$$\text{Vickers hardness} = 0.189\frac{F}{d^2} \text{ HV} \tag{15.4}$$

Where: F applied load, N
d diagonal of trace, mm

Comparison of the hardness ranges covered by the various methods can be read from Table 15.2.

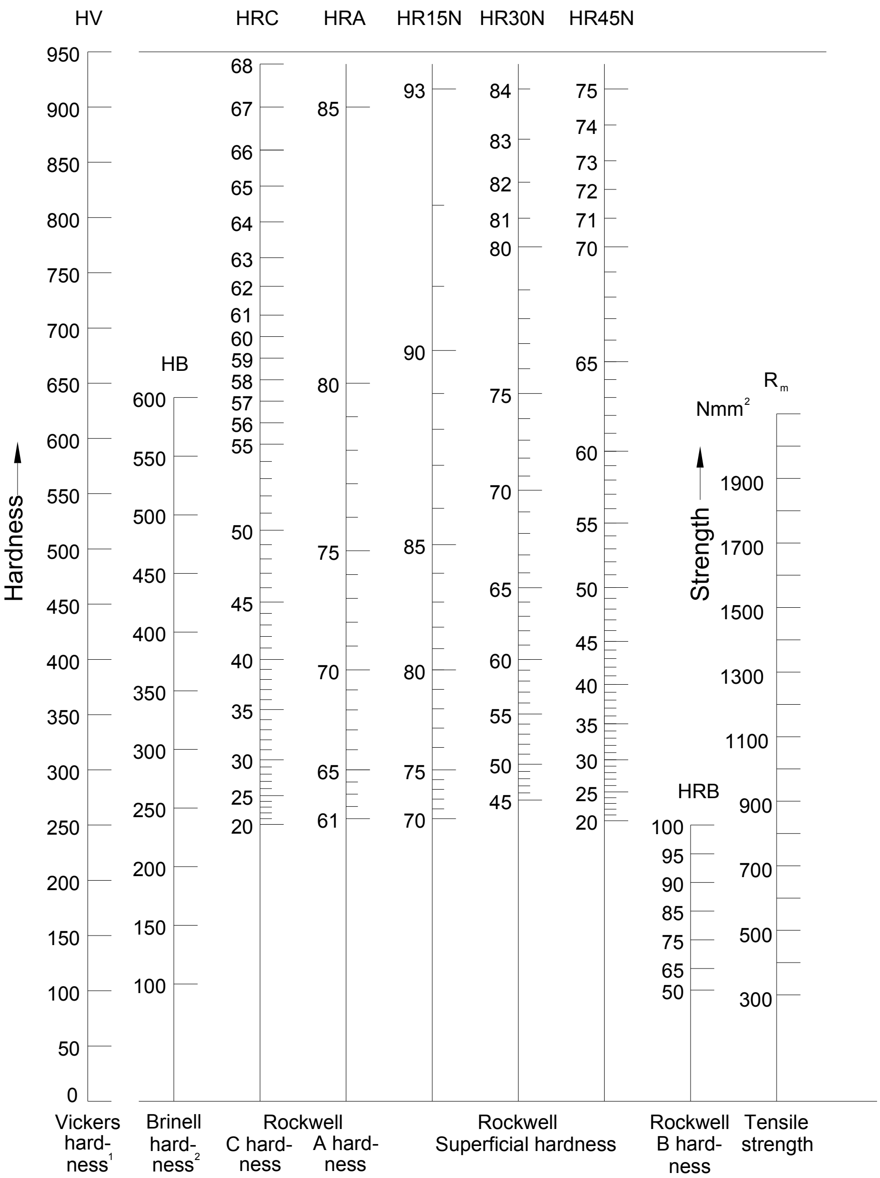
Table 15.2 Comparison of the hardness ranges
Source: [7]



15.2 Correlation between hardness and strength

Correlation between hardness and strength in accordance with DIN 50150 can be read from Table 15.3.

Table 15.3 Correlation between hardness and strength
Source: [7]



REFERENCES

[1] Ansel UGURAL: *Mechanical Design: An Integrated Approach*. 1st Edition Hardcover with access card©2004, ISBN-13 9780072921854.
[2] Joseph SHIGLEY, Charles MISCHKE, Richard BUDYNAS: *Mechanical Engineering Design*. 7th Edition Hardcover with access card, 1056 pages©2004, ISBN-13 9780072921939.
[3] Standard Collection 16: *Technical Drawing I-II.*. Budapest: Szabványkiadó, 1984. ISBN: 963 402 35107, ISBN: 963 402 356 8 (in Hungarian).
[4] István HERCZEG: *Machine Drawing*. Budapest: Tankönyvkiadó, 1980 (in Hungarian).
[5] István HERCZEG: *Technical Standards and Guidelines*. Budapest: Műszaki Könyvkiadó, 1980. ISBN: 963 10 0790 1 (in Hungarian).
[6] Zsolt TIBA: *Machine Drawing*. Debrecen: University Press, 2010, ISBN 978-963-318-066-2.
[7] Ulrich ADLER et. al.: *Automotive Handbook*. Stuttgart: Robert Bosch GmbH, 1993, ISBN: 3-1-419115-X.