20 Geometrical Tolerances and Surface Finish

20.1 INTRODUCTION

Geometric shape of a component is considered exact unless specified. For example, straight line means straightness, circle means that the profile is exactly circular, parallel lines means that these are exactly parallel, lines at right angle to each other implies perpendicularity. If there could be any variation from the exact form, it has to be specified by geometric tolerances. These tolerances should be lesser than the size tolerances.

Many times tolerances on size are not sufficient, e.g. a shaft has the size within the specified limits but even then it is not acceptable due to its deformations in geometric shape. Figure 20.1 shows an exact cylinder and some possible shapes other than exact because of which they cannot be accepted. These form variations are called geometric tolerances and these also have to be within specified limits.

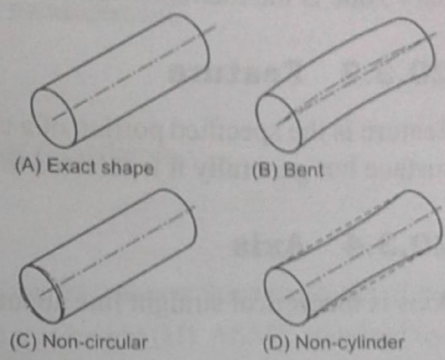


Fig. 20.1 *Geometric Shape Variations*

20.2 TYPES OF TOLERANCES

Tolerances can be of three types:

A. *Size Tolerances*

These are given for a size as described in Chapter 19 (Fig. 20.2A).

B. *Form Tolerances*

In addition to size there could be errors due to form (Fig. 20.2B). Thick line represents the actual shape. Thin lines represent the outermost and innermost radii enveloping the actual shape.

C. *Position Tolerances*

These are the errors due to position. Figure 20.2C shows two centers; one center which is ideally required and the other is the maximum error which can be tolerated while manufacturing. It is indicated by a thin circle showing the zone within which the actual center of the hole should lie.

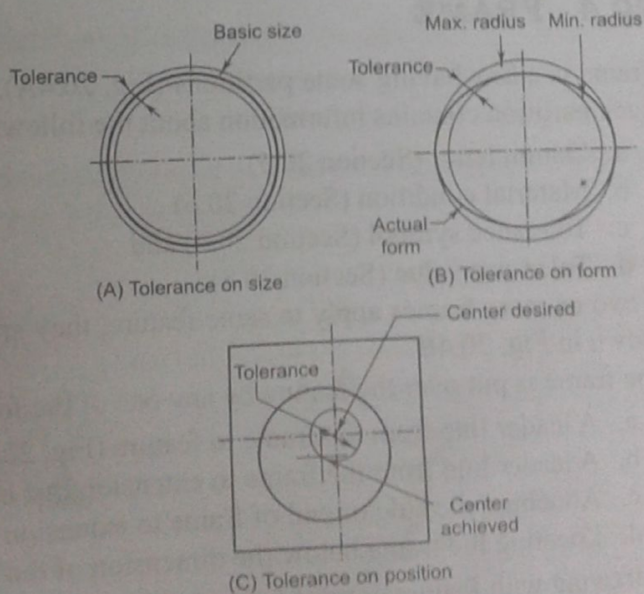


Fig. 20.2 *Types of Tolerances*

20.3 TERMINOLOGY

20.3.1 Geometric Tolerance

It is the maximum permissible variation of form, profile, orientation, location and run out specified on a production drawing.

20.3.2 Tolerance Zone

It is an imaginary zone within which a component must be contained (Fig. 20.3). The height of this imaginary zone is the tolerance value.

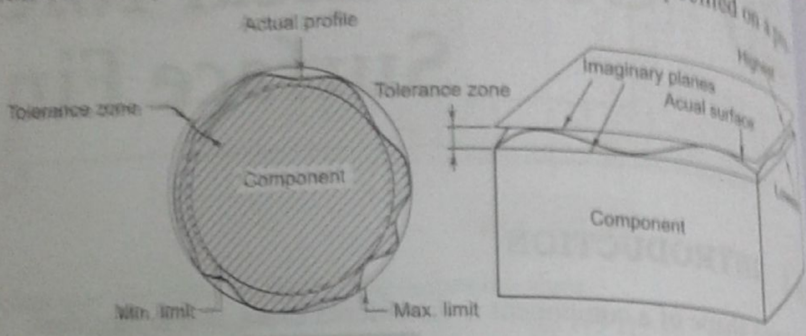


Fig. 20.3 *Tolerance Zone*

20.3.3 Feature

Feature is the specified portion of a component such as hole, slot surface or profile. It can have more than one surface but generally it is referred to a single surface.

20.3.4 Axis

Axis is theoretical straight line about which a circular feature revolves, e.g. axis of a straight cylinder.

20.3.5 Median

Median is the center line of a straight or a bent shaft.

20.3.6 Boxed Dimensions

Boxed dimensions are the dimensions subjected only to position tolerance. These dimensions are enclosed in a box as $\boxed{50}$ or $\boxed{\varnothing 80}$. Tolerances are not given along with these dimensions.

20.4 FRAME

Frame is a box having some partitions (Fig. 20.4A). The size depends upon height (H) of the datum letter. Each partition contains information about the following and is discussed in subsequent sections:

a. Datum letter (Section 20.5)
b. Material condition (Section 20.6)
c. Tolerance symbol (Section 20.7) and
d. Tolerance value (Section 20.8)

If two or more frames apply to same feature, they are drawn one over the other with a single leader line as shown in Fig. 20.4B.

The frame is put over the feature by any one of the following methods:

a. A leader line from the frame to feature (Fig. 20.9A).
b. A leader line from the frame to extension line of surface but not in line with dimension line.
c. Attaching the side or end of frame to extension line (Fig. 20.9B).
d. Locating the frame below the dimension of the feature.

A drawing with feature control frame indicating geometric tolerances is called **drawing call out**.

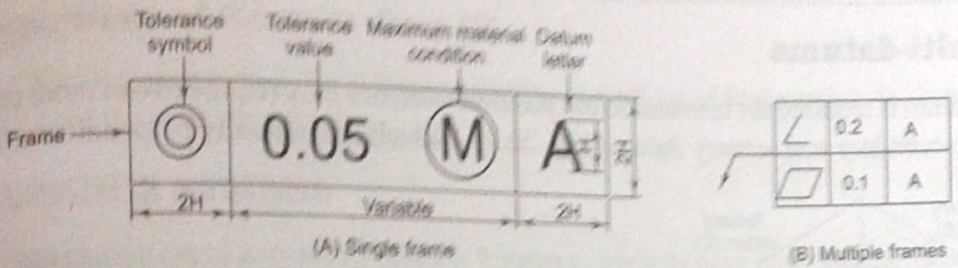


Fig. 20.4 *A Frame and Its Contents*

20.5 DATUM

Datum is a theoretical point, line, or plane from which dimensions are measured and geometric tolerances are referenced. It has an exact form, represents exact and fixed location for measurement.

20.5.1 Datum Feature

It is a feature of component like edge, surface which is taken as the basis for a datum.

20.5.2 Datum Triangle

Datum is shown by a triangle (open or filled) on the datum feature. Figure 20.5 shows three methods of representing a datum along with the proportions of the triangle in terms of text height (H). ASME standard uses a right angled triangle. BIS has also adapted the same (Fig. 20.5A). ISO method uses an equilateral triangle and (Fig. 20.5B) ANSI represents datum only by an arrow line (Fig. 20.5C).

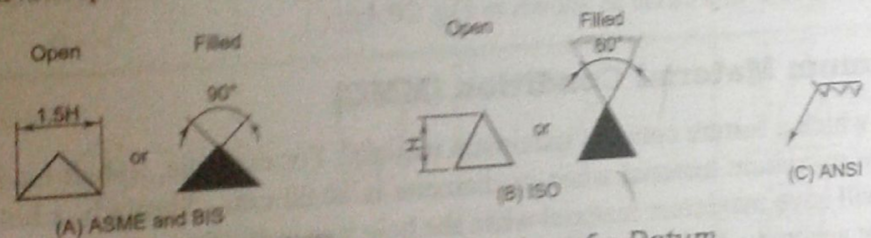


Fig. 20.5 *Representation of a Datum*

20.5.3 Datum Letter

It is an upper case letter enclosed in a box to indicate an arbitrary name of a datum (Fig. 20.6). A leader line is used to connect a frame and the datum triangle. See Fig. 20.7 for its use.

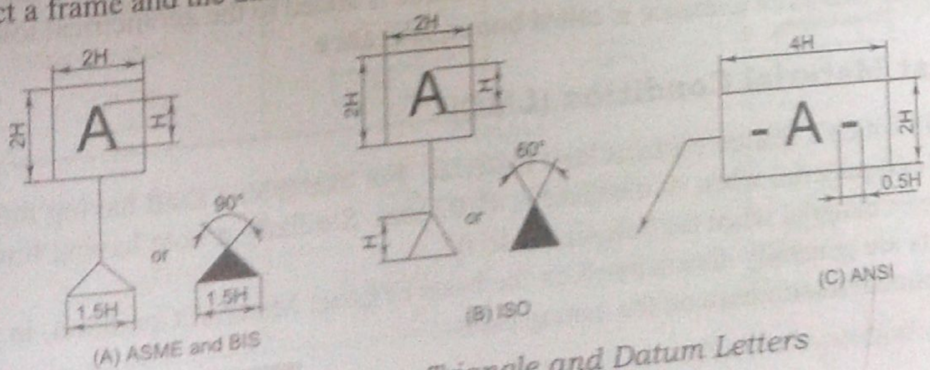


Fig. 20.6 *Datum Triangle and Datum Letters*

20.5.4 Multi-datums

Normally one datum is required for orientation but position tolerance may require two or more datum. These called primary, secondary and tertiary datum (Fig. 20.7A) which are mutually perpendicular to each other.

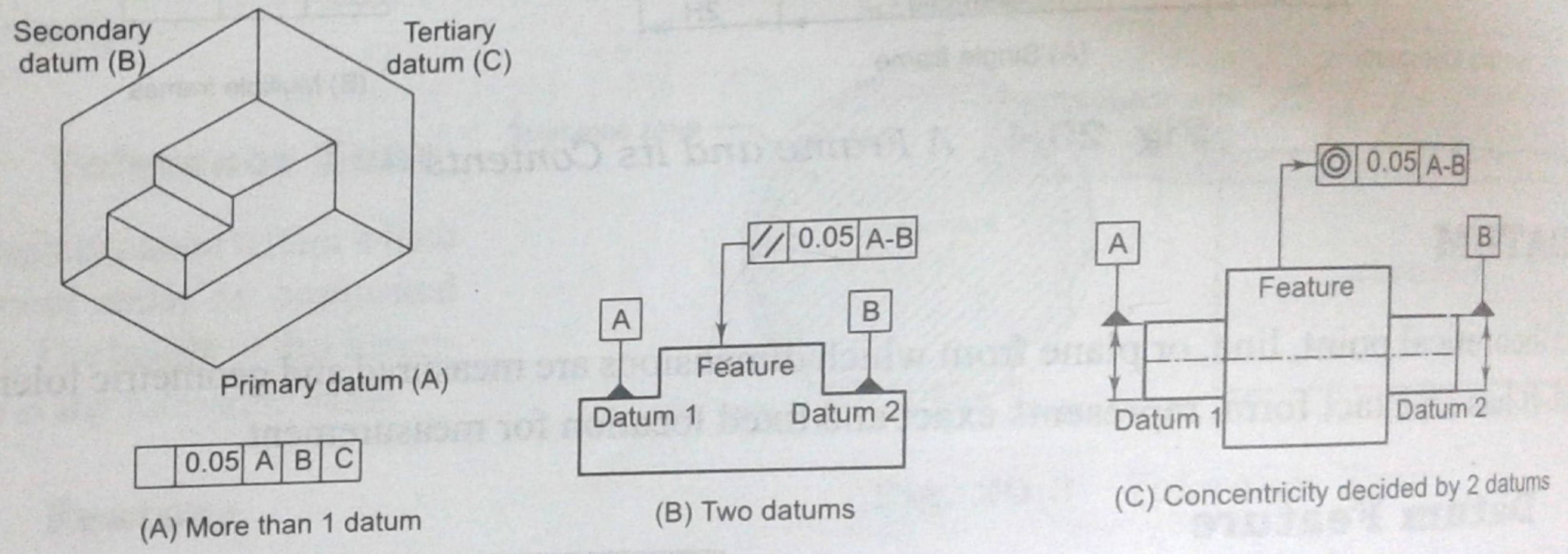


Fig. 20.7 *Multiple Datums*

Multiple datums are referred in separate partitions as shown in Fig. 20.7A. If a single datum is established by two datum features, e.g. a stepped shaft, then the datum letters are placed in the same compartment of the frame with a dash in between (Fig. 20.7B and C).

20.6 MATERIAL CONDITION

20.6.3 Regardless of Feature Size (S)

When MMC or LMC is not specified, the tolerance applies regardless of feature size. In short, it is indicated by letter 'S'. It is used for position tolerances.

20.7 TOLERANCE SYMBOL

It is a graphical representation of a tolerated feature. Fourteen symbols have been standardized and are shown in Fig. 20.8 along with their proportions in terms of size of datum letter H. Use of these symbols is described in subsequent sections.

Form tolerances are divided in the following three categories:

A. Single Feature Tolerances

Straightness, Flatness, Circularity, Cylindricity, Profile of a line and Profile of a surface (Section 20.10).

B. Related Features Tolerances

Parallelism, Perpendicularity, Angularity, Concentricity, Symmetry and Position (Section 20.11).

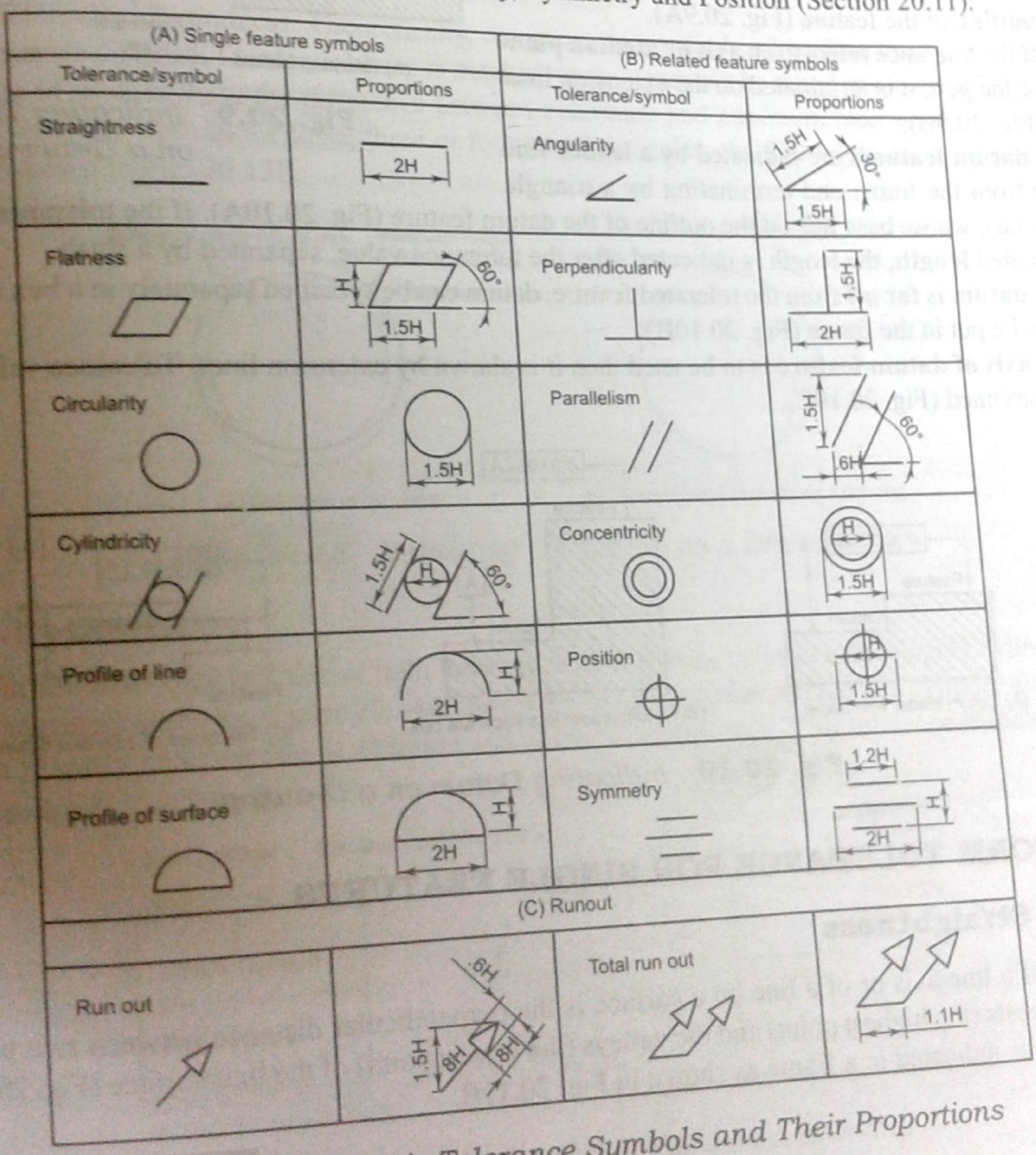


Fig. 20.8 *Geometric Tolerance Symbols and Their Proportions*

C. Runout Tolerances

Circular runout and Total runout (Section 20.12).

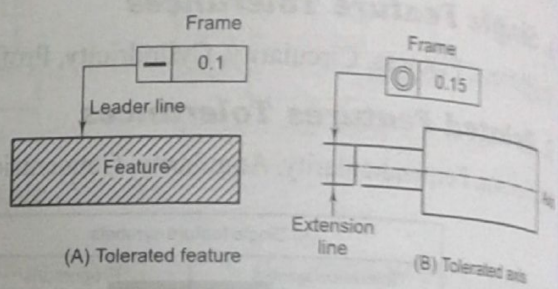
20.8 TOLERANCE VALUE

This value is indicated in the frame next to the tolerance symbol. It is given in millimeters. Its value is decided from the functional point of view of the part as to how much form variation can be tolerated. However, the value should be lesser than the size tolerances.

20.9 INDICATING GEOMETRICAL TOLERANCES ON DRAWINGS

Geometrical tolerances are put on a drawing in a frame (Fig. 20.4). The frame is connected to the tolerance feature by any of the following methods:

a. A leader line terminating with an arrow on the **outline** of the feature (Fig. 20.9A).
b. If the tolerance refers to an **axis or median plane** of the part, it is terminated on the extension lines (Fig. 20.9B).



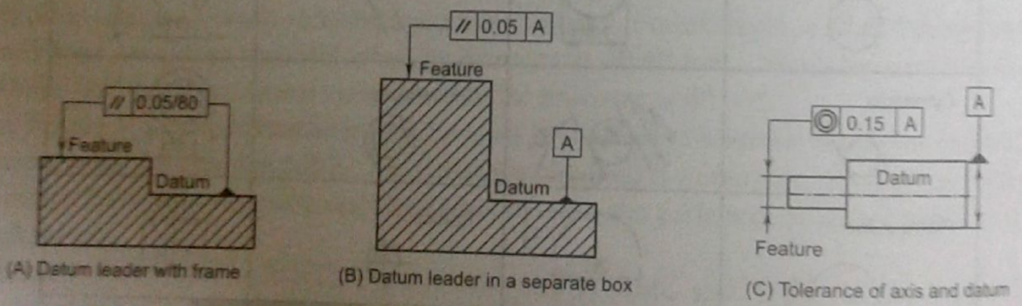
(A) Tolerated feature (B) Tolerated axis

Fig. 20.9 *Indicating Tolerances on a Drawing*

The **datum features** are indicated by a **leader line** starting from the frame and terminating by a triangle (open/solid), whose base lies on the outline of the datum feature (Fig. 20.10A). If the tolerance is applied up to a specified **length**, the length is indicated after the tolerance value, separated by a slash.

If the **datum is far off** from the tolerated feature, datum can be specified separately in a box and the datum letter can be put in the frame (Fig. 20.10B).

If the **axis of datum feature** is to be used then it is shown by extension lines. Tolerance value and datum can be combined (Fig. 20.10C).



(A) Datum leader with frame (B) Datum leader in a separate box (C) Tolerance of axis and datum

Fig. 20.10 *Indicating Datum on a Drawing*

20.10 FORM TOLERANCE FOR SINGLE FEATURES

20.10.1 Straightness

Straightness of a line/axis or of a line on a surface is the perpendicular distance between two parallel lines touching the crests (the highest point) and the valleys (the lowest point) of the line/surface (Fig. 20.11A). This tolerance zone is indicated in a frame as shown in Fig. 20.11B.

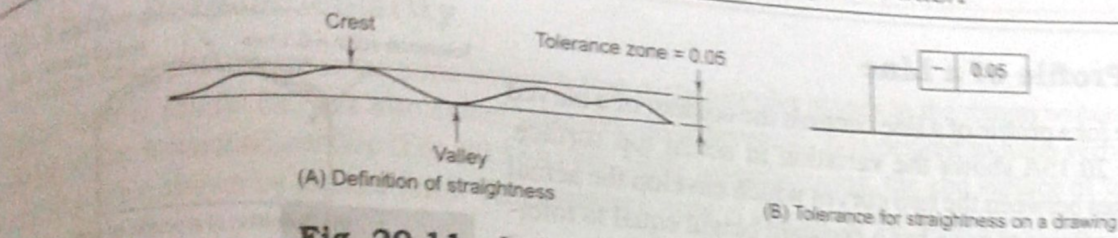


Fig. 20.11 *Indicating Straightness on a Drawing*

20.10.2 Flatness

Flatness is the distance between two imaginary planes enclosing the actual surface at the low-ermost and uppermost positions (Fig. 20.12A). Symbolically it is shown on a drawing as in Fig. 20.12B.

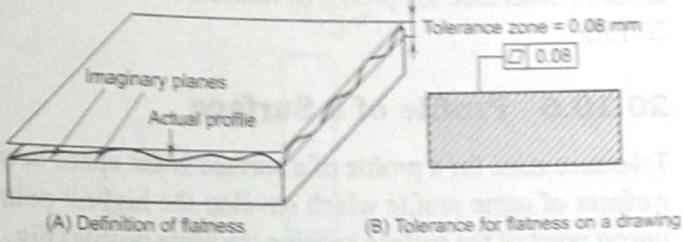


Fig. 20.12 *Indicating Flatness on a Drawing*

20.10.3 Circularity

Circularity is also called Roundness. Theoretically, any point on a cylindrical surface from the central axis should be at the same distance. Due to problems in machine tools, it may not be round as shown in Fig. 20.13A. Tolerance value of circularity is the difference between maximum and minimum radii of a cylinder at any section. It can take any form like ellipse, three or four lobed, irregular, etc. The tolerated value is indicated on drawing as shown in Fig. 20.13B.

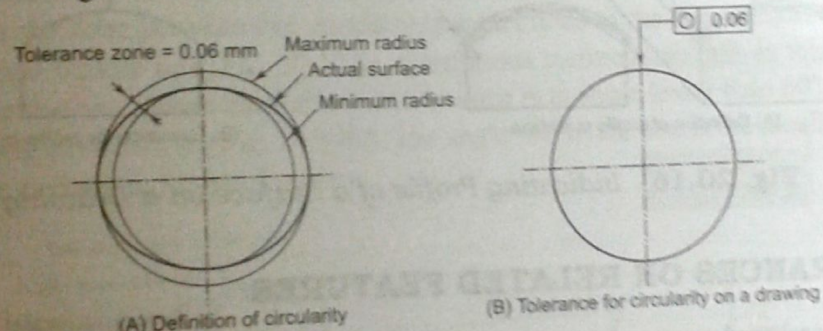


Fig. 20.13 *Indicating Circularity on a Drawing*

20.10.4 Cylindricity

Cylindricity is the difference in value of radii between two imaginary cylinders, enveloping cylinders at outermost and innermost surfaces. Figure 20.14A shows the variation in the surface of a cylinder along its axis. The diameter at every cross-section is different and lies in a circular zone. This tolerance is indicated on the drawing as shown in Fig. 20.14B.

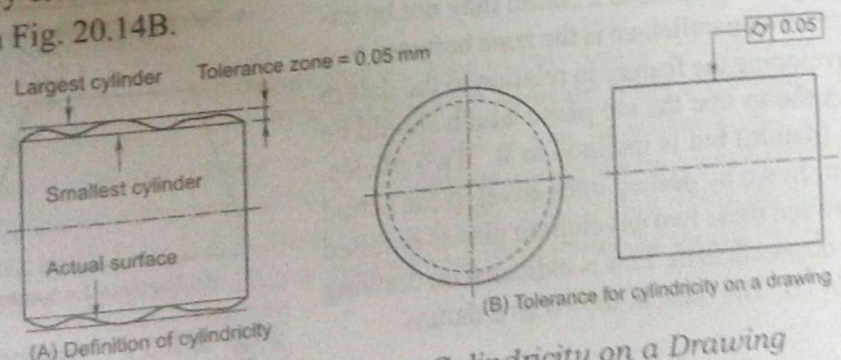


Fig. 20.14 *Indicating Cylindricity on a Drawing*

20.10.5 Profile of a Line

Tolerance zone for a profile of a line controls the contour of a curved profile. Figure 20.15A shows the variation in actual top surface. The variation lies between the two curves which envelop the actual curve. Thus the tolerance zone has a constant height equal to tolerance value normal to the theoretical profile and equally disposed about it. Tolerance for profile of line is shown on drawing in Fig. 20.15B.

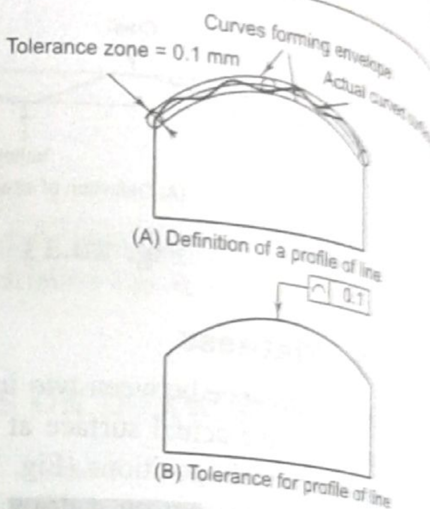


Fig. 20.15 *Indicating Profile of a Line on a Drawing*

20.10.6 Profile of a Surface

Tolerance zone for a profile of a surface is the space between two surfaces of same profile which envelop the highest point and the lowest point of the surface keeping the same profile (Fig. 20.16A). This tolerance zone is shown on the drawing in Fig. 20.16B.

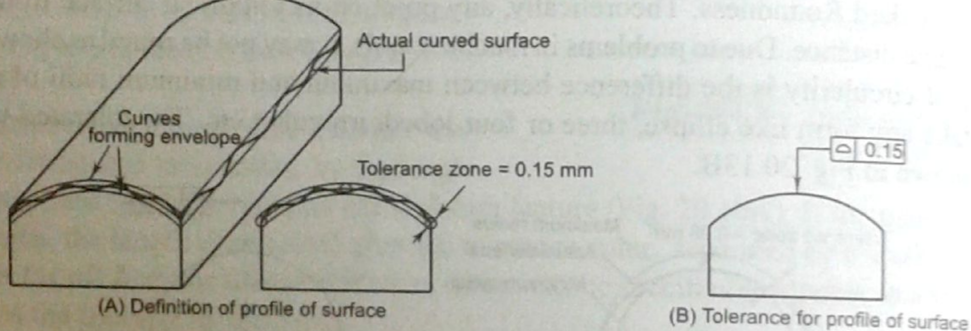


Fig. 20.16 *Indicating Profile of a Surface on a Drawing*

20.11 TOLERANCES ON RELATED FEATURES

Tolerated feature is assigned a geometric relation with respect to another datum feature for these tolerances. The relation could be parallelism, perpendicularity, angularity, concentricity, symmetry or position.

20.11.1 Parallelism

A surface which is required parallel to a datum may not be exactly parallel. Tolerance on parallelism is the zone between two parallel surfaces enveloping the feature in relation to the datum surface. Fig. 20.17A shows that the top plane which should be parallel to the base (datum) but is inclined to it. Two planes, parallel to the datum shown by dashed lines envelop the actual top surface. Gap between these two enveloping planes is called geometric tolerance on parallelism. This is indicated on drawing as shown in Fig. 20.17B. The base is shown as a datum.

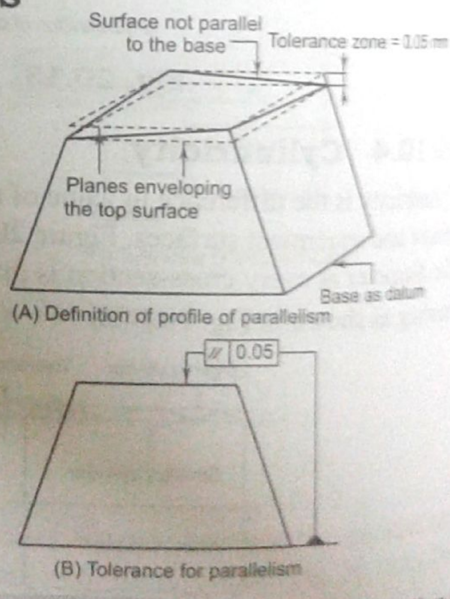


Fig. 20.17 *Indicating Parallelism of a Surface on a Drawing*

20.11.2 Perpendicularity

Perpendicularity tolerance is the zone between two perpendicular planes to the datum within which the controlled feature should lie. It is also called tolerance on squareness. It is desired to have vertical face at right angles to the horizontal surface (Fig. 20.18A) but while manufacturing, the end surface of vertical plate can vary in a zone shown by dashed lines. This tolerance zone is shown on the drawing in Fig. 20.18B.

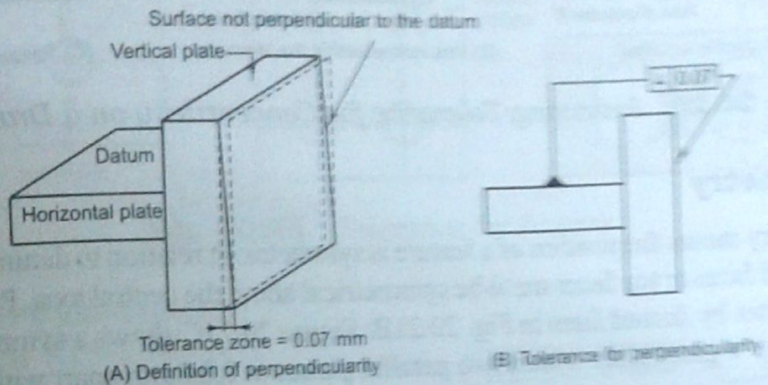


Fig. 20.18 *Indicating Perpendicularity of a Surface on Drawing*

20.11.3 Angularity

Theoretically, an inclined surface should be at a specified angle. Practically there is some deviation. Tolerance on angularity is the zone between two parallel planes inclined to the datum plane at the specified angle in which the controlled feature lies (Fig. 20.19A). Continuous inclined line falls in this zone and is at angle more than 60°, while the line shown dashed within this zone is at angle lesser than 60°. This tolerance is indicated on the drawing as shown in (Fig. 20.19B). The angle 60° in the box is the ideally desired angle. Note that angularity is not defined in terms of angles.

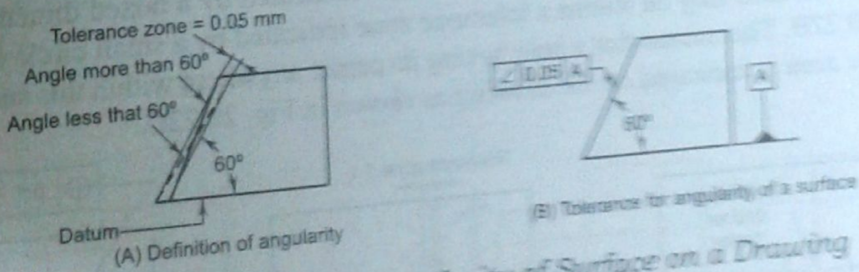


Fig. 20.19 *Indicating Angularity of Surface on a Drawing*

20.11.4 Concentricity

Theoretically, a perfect concentricity means that the axes of two coaxial cylinders are in a line and coincide. See Fig. 20.20A in which axes of cylinder A and B do not coincide. They are parallel but offset.

Even if two axes coincide, center of the cylinder B can be offset at the other end as shown in Fig. 20.20B. This maximum allowable offset in any direction is shown by a small circle showing the tolerance zone. Tolerance on concentricity is the diameter of a circular zone within which the axes of the two cylindrical features may offset from each other. This is indicated on the drawing as shown in Fig. 20.20C.

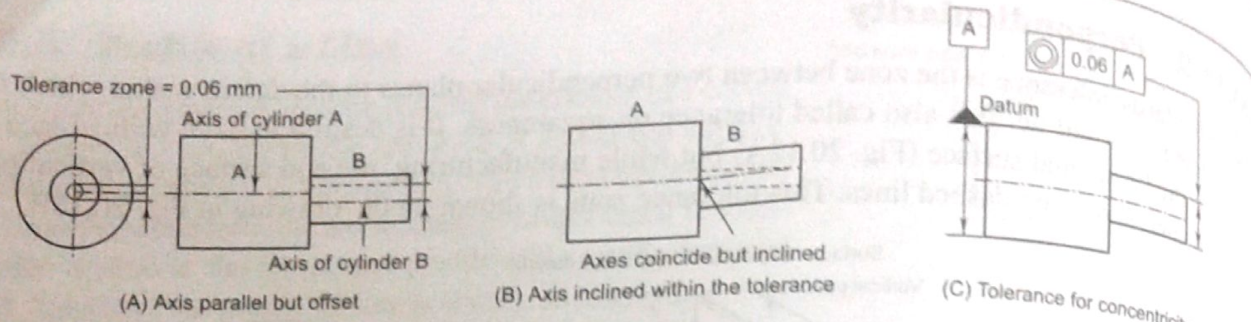


Fig. 20.20 *Indicating Tolerance for Concentricity on a Drawing*

20.11.5 Symmetry

Theoretically, symmetry means the position of a feature is symmetric in relation to datum. Figure 20.21A shows a V block. The inclined faces or top faces are to be symmetrical about the central axis. Practically they may not be symmetrical as shown by dashed lines in Fig. 20.21B. Figure 20.21C shows a symmetric tolerance so that the median plane of the V groove lies within two parallel planes at 0.04 mm apart with respect to its base.

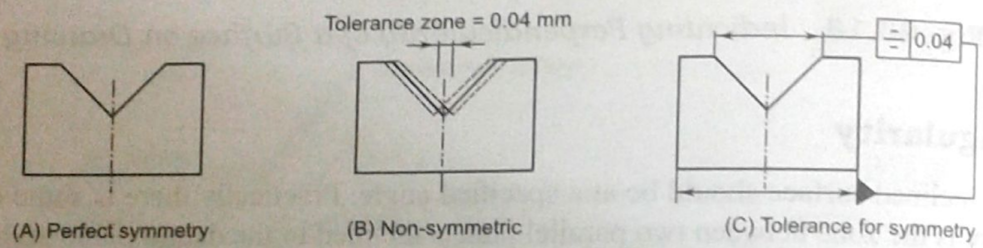


Fig. 20.21 *Indicating Tolerance for Symmetry on a Drawing*

20.11.6 Position

Theoretical true position of a feature (a hole in this case) is indicated by a boxed dimension (Fig. 20.22A). The actual center of the hole may lie within a tolerance zone indicated by a small circle of diameter 0.1 mm as shown in Fig. 20.22B. This means that a hole having its center anywhere within this small circle is acceptable. This tolerance zone is indicated on the drawing as shown in Fig. 20.22C.

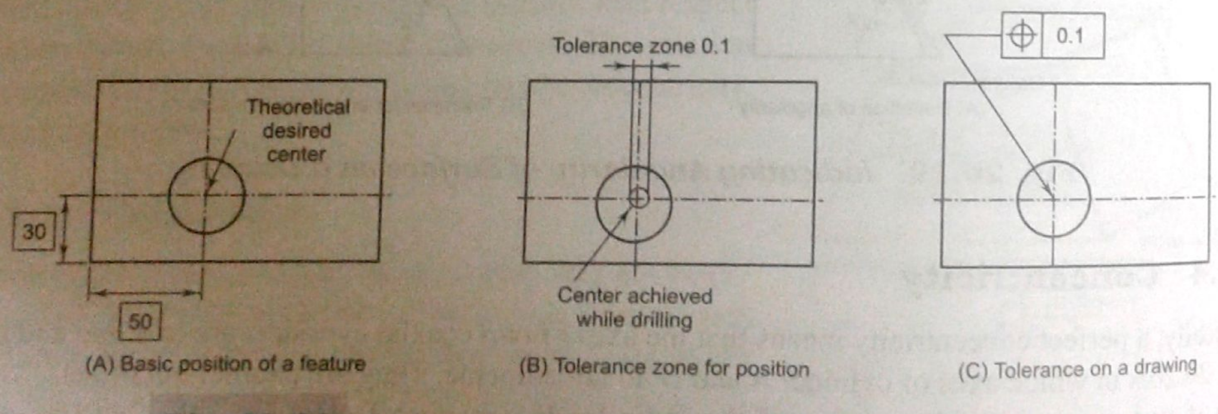
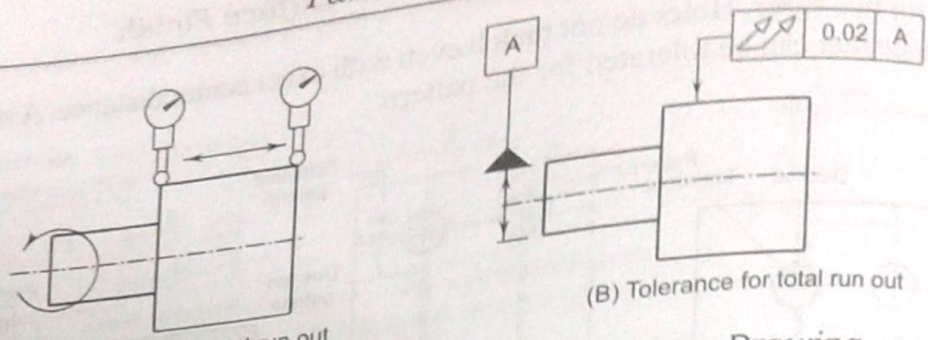


Fig. 20.22 *Indicating Tolerance for Position on a Drawing*

20.11.7 Position Tolerance for Patterns

When there are many holes arranged in a pattern (in a line, rectangle or a polar array), holes of two similar parts may not match due to distortion of the pattern even if the center distance is exact. Figure 20.23 shows



(A) Definition of total run out

(B) Tolerance for total run out

Fig. 20.26 *Indicating Total Run Out on a Drawing*

Example 1 Profile of a surface shown in Fig. 20.S1A is to be specified for geometric tolerances such that:

a. Its profile varies within 0.2 mm.
b. Its profile and its orientation is 0.3 mm.
c. Its profile and its orientation is 0.3 mm and has vertical position also in the middle as 30 mm.

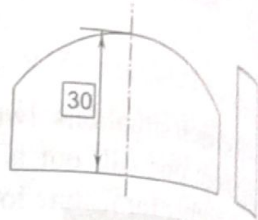


Fig. 20.S1A

Solution

a. The required geometric tolerance is shown in Fig. 20.S1B.
b. The required geometric tolerance is shown in Fig. 20.S1C.
c. The required geometric tolerance is shown in Fig. 20.S1D.

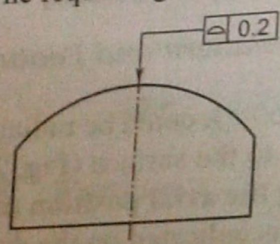


Fig. 20.S1B

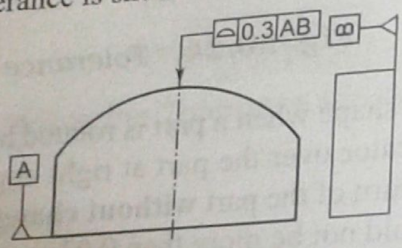


Fig. 20.S1C

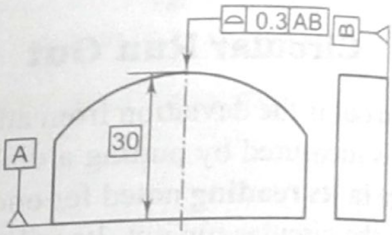


Fig. 20.S1D

Example 2 A forked support with 20 mm thick forks shown in Fig. 20.S2A has two holes 30 mm below top surface such that their concentricity is within 0.2 mm and position is within 0.05 mm both in X and Y directions. The axis should be parallel to top surface within 0.1 mm. Show the geometric tolerances.

Solution The required geometric tolerances are shown in Fig. 20.S2B.

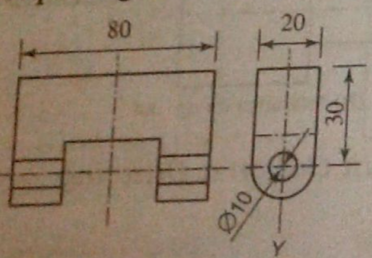


Fig. 20.S2A

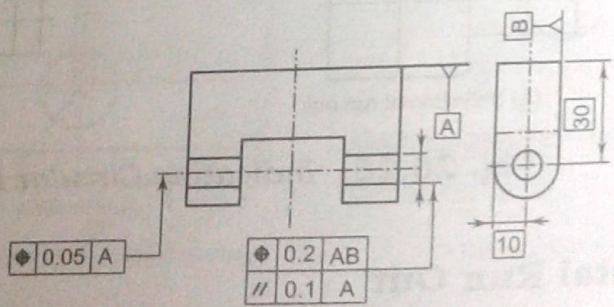


Fig. 20.S2B

Example 3 An inspection gauge is shown in Fig. 20.S3A with its dimensional and geometric tolerances for head and pin for checking coaxiality of part shown in Fig. 20.S3B. Considering feature sizes as tabulated below, complete the table for allowable distances between the two axes.

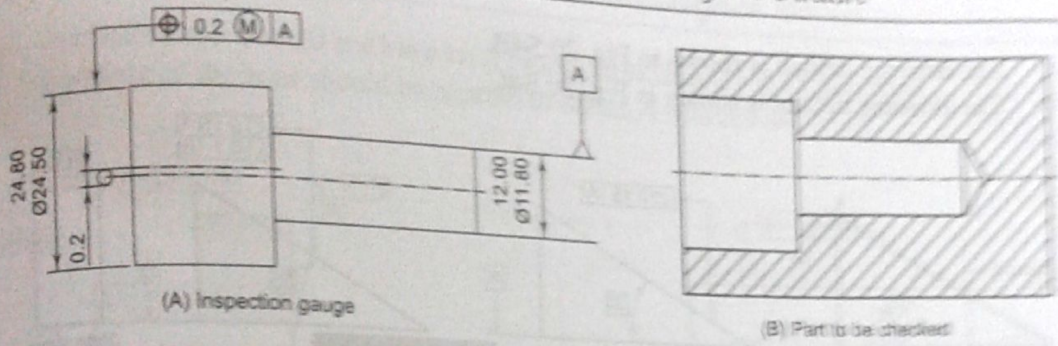


Fig. 20.S3 *An Inspection Gauge*

Head size		*Pin size*	
	12.0	11.9	11.8
24.8			
24.7			
24.6			
24.5			

Solution Ideally the axes should be coaxial, but practically there may be some variation. Coaxiality is similar to concentricity with axes arranged in a straight line. Coaxiality can be controlled by specifying any one of the following four types of geometric tolerances:

a. Concentricity tolerance
b. Positional tolerance
c. Run out tolerance
d. Profile tolerance

Selection of the method depends upon the function of the part. When surfaces of revolution are cylindrical and control of the axes can be on MMC basis, positional tolerance is recommended. Solution with positional tolerance is shown in drawing call out. The feature control frame shows maximum permissible positional tolerance as 0.2 mm, i.e. the distance between the axes at MMC should not be more than 0.1 (half of the tolerance value).

When diameters are at maximum size as 24.80 for head and 12.0 for pin, the maximum distance between the axes is 0.1. When the diameter of pin is lesser, a greater value of geometrical tolerance can be allowed for distance between the axes. The allowable values are tabulated in Table 20.S3 for different values of pin size.

Table 20.S3 Allowable values of tolerance

Head size		*Pin size*	
	12.0	11.9	11.8
24.8	0.1	0.15	0.2
24.7	0.15	0.2	0.25
24.6	0.2	0.25	0.3
24.5	0.25	0.3	0.35

Example 4 An angle block having 30 degrees angle as shown in Fig. 20.S4A has to have angularity within 0.15 mm with its base 'A'.

a. Specify its geometric tolerances.
b. If the inclined face also has to maintain squareness with its side within 0.1 mm. Specify its geometric tolerances.

Solution a. The drawing callout is shown in Fig. 20.S4B.
b. The drawing callout is shown in Fig. 20.S4C.

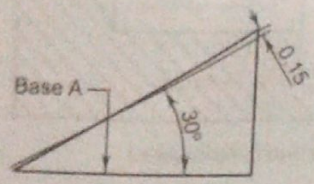


Fig. 20.S4A

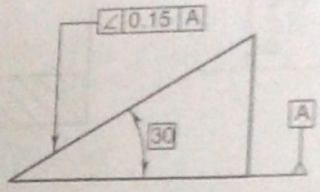


Fig. 20.S4B

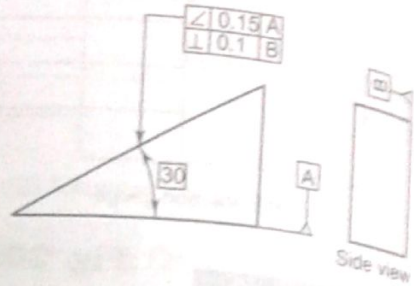


Fig. 20.S4C

Example 5 A stepped shaft is shown in Fig. 20.S5A. The middle portion of the shaft has to be concentric with the axis of ends within 0.1 mm. Show the geometric tolerances.

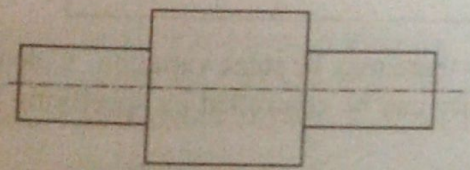


Fig. 20.S5A

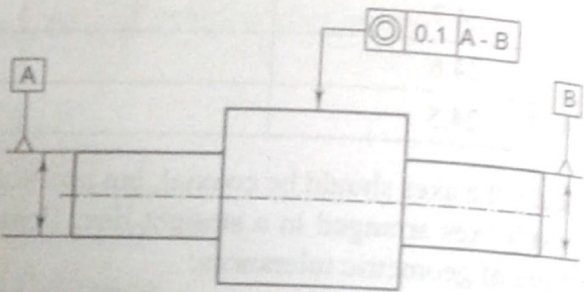


Fig. 20.S5B

Soluion The drawing callout is shown in Fig. 20.S5B.

Example 6 A cylindrical part shown in Fig. 20.S6A has a hole in it such that its concentricity with the outer surface A is within 0.1 mm at MMC. Show the geometrical tolerances on the drawing.

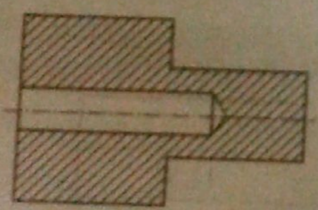


Fig. 20.S6A

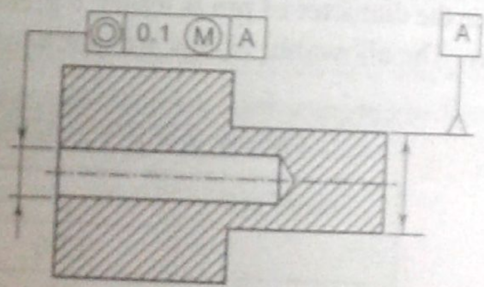


Fig. 20.S6B

Solution The drawing callout is shown in Fig. 20.S6B.

Example 7 Interpret the six geometrical tolerances shown on drawing of a flange shown in Fig. 20.S7.

Solution

Callout 1 – Right side face of the flange is datum A which should have flatness within 0.05.

Callout 2 – Hole axis should have perpendicularity to datum A within 0.02 mm and is taken as secondary datum B.

Callout 3 – Surface of the boss should be concentric within 0.1 w.r.t. hole axis.

Callout 4 – Outer surface of the flange should have run out within 0.1 w.r.t. hole axis.

Callout 5 – Surface of hole should not have total runout more than 0.1 w.r.t. hole axis.
Callout 6 – Left face of the boss should be parallel to datum A within 0.04.

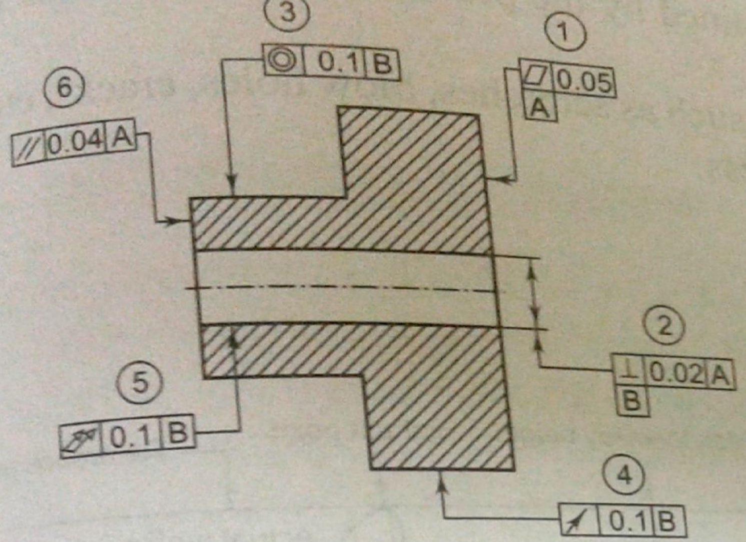


Fig. 20.S7 *Interpretaions of Drawing Callouts*

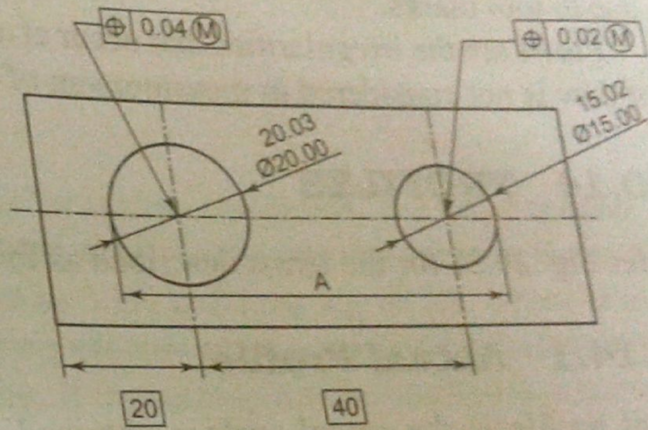


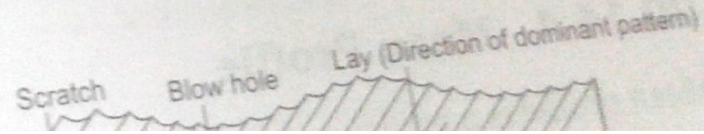
Fig. 20.S8 A Jig Plate

Example 8 A jig plate for drilling two holes is shown in Fig. 20.S8 with the position tolerances. Find the minimum and maximum center distance between the holes and distance A.

Solution Maximum distance between centers = 40 + (0.04/2) + (0.02/2) = 40 + 02 + 0.01 = 40.03 mm
Minimum distance between centers = 40 – (0.04/2) – (0.02/2) = 40 – 02 – 0.01 = 39.97 mm

Hence,
Maximum distance A will be 40.03 + (20.03/2) + (15.02/2) = 40.03 + 10.015 + 7.51 = 57.555 mm
Minimum distance A will be 39.97 + (20.00/2) + (15.00/2) = 39.97 + 10.000 + 7.500 = 57.470 mm

...ACE TEXTURE

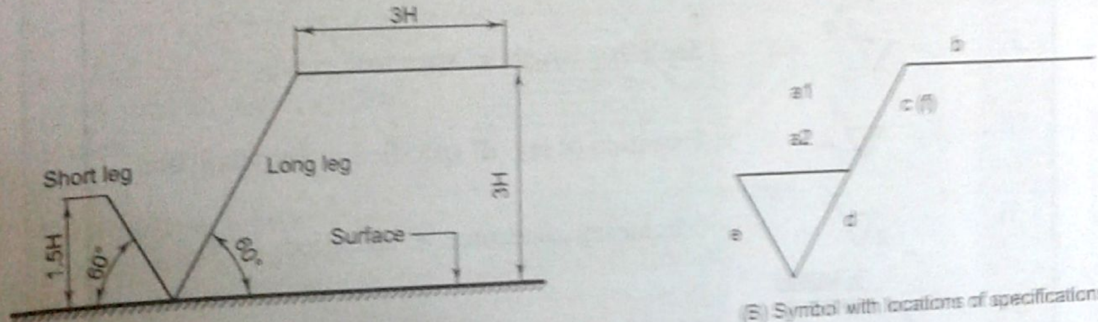


of tool marks on the surface. Surface roughness is generally expressed as Ra value in microns. The relation of Ra and heights of actual surface from the mean profile as $h_1, h_2, \ldots h_n$ etc. for n points is given as:

$$Ra = \frac{h_1 + h_2 + h_3 + \cdots + hn}{n} \tag{20.2}$$

20.16 ROUGHNESS SYMBOLS

Different types of symbols are placed over the line representing the surface roughness. The **basic symbol** for surface roughness has **two legs;** one short and other long inclined at 60° to the line representing the surface as shown in Fig. 20.29A. The height of the small leg is taken as 1.5H and long leg as 3H, where H is the height of the text written over these symbols. If a manufacturing process or treatment of the surface is to be indicated, a horizontal line of length 3H is put over the long leg. If the surface is to be machined, a horizontal line is added on RHS of the short leg of the basic symbol (Fg. 20.29B).



(A) Basic symbol with proportions

(B) Symbol with locations of specifications

Where:

a1 – Maximum permissible roughness
b – Production method
d – Direction of lay
f – Roughness criterion other than Ra

a2 – Minimum permissible roughness
c – Sampling length
e – Machining allowance

Fig. 20.29 *Roughness Symbol with Various Specifications*

Various specifications a1, a2, c, d, e and f in microns are added on this symbol as required. These are shown in Table 20.1 and each symbol is marked arbitrarily from A to M. Meaning of each is explained below:

a. **Basic** surface roughness symbol without any specifications. It does not carry any meaning if placed like this.
b. If removal of the material is not permitted; **a circle** is added between the two legs of basic symbol.
c. A horizontal line over the short leg means that this surface is required to be machined.
d. Specification 'a' is the maximum roughness in microns that can be tolerated.
e. If a limit of maximum and minimum roughness is imposed, both the values, 'a1' and 'a2', are to be put over the symbol. Maximum value is put above the minimum value.
f. If the sampling length is to be indicated, it is written adjacent to the symbol on right side of long leg near the top.
g. If required to indicate lay, it is indicated by a symbol on bottom right side of the long leg of roughness symbol. The symbol shown in table is for perpendicular lay. See Section 20.17 for lay.
h. Value of machining allowance 'e' is indicated on the left side of short leg of roughness symbol. This value is generally indicated in millimeters.

Table 20.1 Roughness symbols and their meanings

No.	*Roughness symbol*	*Meaning*
A	√	Basic symbol without any specifications
B	√	Material removal prohibited
C	√	Machining required
D	a √	Machining required. Maximum roughness is 'a'
E	a1 a2 √	Surface roughness should lie between a1 and a2.
F	√ c	Sampling length 'c' specified
G	√ ⊥	Direction of lay 'd' specified as perpendicular
H	e √	Machining allowance 'e' specified
I	a √ b Milled	Machining operation 'b' specified as milling
J	a √ Chrome plated	Treatment is indicated as chrome plating
K	a √ (Rt = 0.4)	Roughness criterion other than Ra in parentheses
L	a √ All over	A note for same roughness all over as 'a'
M	X √ = a √ b c e d	Named symbol X has same specifications as given on right hand side after equal to sign

i. If the final surface is to be produced by a specific production method, it is written above the horizontal line.

j. Treatment process like plating, oxidizing, etc. is also indicated on the top of horizontal line. Length of this can be increased if required. Unless stated, the roughness applies to the roughness after treatment or coating. If these are different before and after, both are to be indicated.

k. Generally, the roughness value which is indicated is 'Ra' value. If some other criterion of specifying roughness is taken, it is indicated below the horizontal line within parentheses.

l. If value of roughness is same for all the surfaces as 'a', a general note can be written as "All over" and this value be indicated on the surface symbol above the machining line. If it is same for most of the surfaces, except a few, a note can be written as "All over except otherwise stated". Then wherever it is different, only those surfaces may be indicated for roughness.

Table 20.3 Roughness grades and grade symbols

Roughness values—Ra in microns	*Roughness grade number*	*Roughness grade symbol*
50	N12	∼
25 12.5	N11 N10	▽
6.3 3.2 1.6	N9 N8 N7	▽▽
0.8 0.4 0.2	N6 N5 N4	▽▽▽
0.1 0.05 0.025	N3 N2 N1	▽▽▽▽

20.19 ROUGHNESS WITH MANUFACTURING PROCESSES

Surface roughness in microns is given in Table 20.4. The roughness used for average applicati is mentioned in the right column. This table helps in recommending a suitable process or machine tool on drawing for a specified roughness or specify roughness for a given process. The processes are arranged i descending order of the maximum roughness for average application range for easy selection.

Table 20.4 Range of roughness obtainable with different processes

S.No.	*Process*	*Minimum– Maximum*	*Average application range Minimum–Maximum*
1	Gas cutting	6.3-100	12.5-25
2	Sand casting	6.3-50	12.5-25
3	Hot rolling	6.3-50	12.5-25
4	Sawing	1.6-25	1.6-12.5
5	Planning	1.6-25	1.6-12.5
6	Shaping	1.6-25	1.6-12.5
7	Forging	1.6-25	3.2-12.5
8	Turning	0.32-25	0.4-6.3
9	Milling	0.32-25	0.8-6.3
10	Filing	0.25-25	3.2-6.3
11	Drilling	1.6-20	1.6-6.3
12	Boring	0.2-25	0.4-6.3
13	Chemical etching	0.8-12.5	1.6-6.3
14	Electric discharge machining	0.8-12.5	1.6-6.3
15	Electron beam	0.2- 6.3	0.8-6.3

Table 20.4 (Contd.)

S.No.	Process	Minimum– Maximum	Average application range Minimum–Maximum
16	Laser	0.2-6.3	0.8-6.3
17	Extruding	0.4-12.5	0.8-3.2
18	Reaming	0.4-3.2	0.8-3.2
19	Broaching	0.4-3.2	0.8-3.2
20	Hobbing	0.4-3.2	0.8-3.2
21	Electro-chemical machining	0.05-6.3	0.2-3.2
22	Drawing/Cold rolling	0.2-6.3	0.8-3.2
23	Die casting	0.4-3.2	0.8-1.6
24	Cylindrical grinding	0.05-12.5	0.1-1.6
25	Surface grinding	0.05-12.5	0.1-1.6
26	Burnishing	0.05-0.8	0.1-0.8
27	Honing	0.02-0.8	0.1-0.8
28	Electro polish	0.05-1.6	0.1-0.8
29	Electrolytic grinding	0.1-0.8	0.2–0.6
30	Polishing	0.05-1.6	0.1–0.8
31	Lapping	0.1-0.8	0.05–0.4
32	Super finishing	0.2-0.4	0.1–0.4
33	Plating	Roughness increases with thickness of plating	
34	Oxide black coating	Roughness does not change	

20.20 ROUGHNESS FOR TYPICAL APPLICATIONS

Table 20.5 can be used as a guideline for specifying a suitable roughness depending upon the application or a suitable manufacturing process can be recommended to achieve that roughness.

Table 20.5 Applications and suggested roughness

Roughness in microns	*Applications*
25	Low grade roughness from sand casting, flame cutting, chipping, rough forging. Suitable for un-machined surfaces and is rarely specified.
12.5	Surface resulting from heavy machine cuts, coarse feeds in turning, shaping, milling, boring.
6.3	Used for unfinished clean operations. Resulting from coarse ground surfaces, rough file, commercial turning.
3.2	Roughest surface that can be used for parts subjected to loads, vibrations, high stresses. Medium commercial machine finish on lathe, milling, shaper. Die casting, extrusion and rolled surfaces.
1.6	Good machining finish with high speed and fine feed turning. Used for close fits. Not suitable for high speed shafts having vibrations.

(Contd.)

Table 20.5 (Contd.)

Roughness in microns	*Applications*
0.8	High grade finish by machine tools like centerless, cylindrical and surface grinding machines. Suitable for parts under high stress concentration. Cost is high.
0.4	High quality surface by grinding, honing, lapping specially where smoothness is very important. Suitable for high speed shafts where lubrication is not dependable.
0.2	Fine surface produced by honing, lapping, buffing. Suitable for hydraulic cylinders, high speed shafts where lubrication is not dependable.
0.1	Costly surface produced by honing, lapping, buffing. Suitable for instrument work, chrome plated piston rods.
0.05 and 0.025	Costliest surface produced by super finishing. Used for fine and sensitive instruments, gauge blocks.

20.21 RULES FOR PUTTING ROUGHNESS SYMBOLS

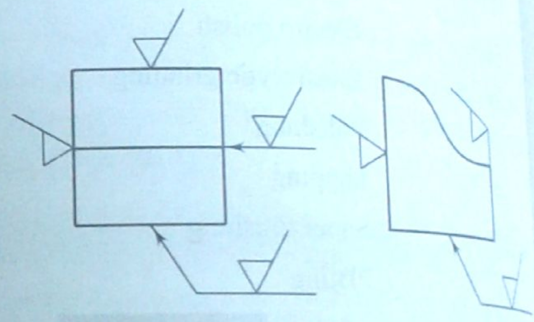


Fig. 20.30 *Roughness Symbol Orientation and Placement*

- If no surface control is specified, it is presumed that the surface produced by any manufacturing process will be acceptable. If roughness is important, it has to be indicated on a drawing.
- The symbol used is a check mark, whose conical point should touch the line indicating the surface.
- Wherever the symbol is used with dimension, it affects the entire surface defined by dimension.
- Transition areas such as fillets and chamfers normally have the same roughness as finished area next to them.
- If the roughness is indicated for a plated surface, it has to be mentioned whether it is before or after plating or both should be mentioned.
- The symbol is to be oriented such that it can be read either from bottom or from right side of the drawing (Fig. 20.30).
- The symbol can be connected to the surface by a leader line terminating with an arrow.
- It can be put over the extension lines also.
- Symbol is to be used only once for a given surface, preferable where the surface is dimensioned.
- If same roughness applies to every surface, a note can be written.
- If same roughness applies to most of the surfaces except a few, a conditional note can be written.
- To avoid clumsiness of specifications over a symbol, a symbol with similar specifications can be named.
- For symmetrical surfaces, the symbol is to be put on both the sides.
- For cylindrical surfaces, the symbol is to be put only on one side.
- All over symbol (L in Table 20.1) can also be used if same roughness aplies to all surfaces.

ANSWERS TO MULTIPLE CHOICE QUESTIONS

1. (c) 2. (d) 3. (b) 4. (b) 5. (a) 6. (b)
7. (a) 8. (a) 9. (b) 10. (c) 11. (d) 12. (a)
13. (d)

ASSIGNMENT ON GEOMETRIC TOLERANCES AND SURFACE ROUGHNESS

1. For the isometric view shown in Fig. 20.P1, draw the front view, side view and add the following geometrical tolerances on the appropriate view:
   - Bottom as primary datum A
   - Front of 200 mm length as secondary datum B
   - Right side as tertiary datum C
   - Bottom has flatness of 0.06 mm
   - Top has to be parallel with datum A within 0.04 mm
   - The hole has to be perpendicular to datum A within 0.02 mm and straightness of 0.05
2. A base plate with 3 pins is shown in Fig. 20.P2. Add the following geometrical tolerances to the drawing:
   - Top surface of the base plate as datum A
   - Datums B and C as secondary and tertiary for location of pins
   - Bottom of the base to have parallelism of 0.2 mm and flatness of 0.1 mm with datum A
   - Location of pins 1, 2 and 3 for position tolerance of 0.1 mm

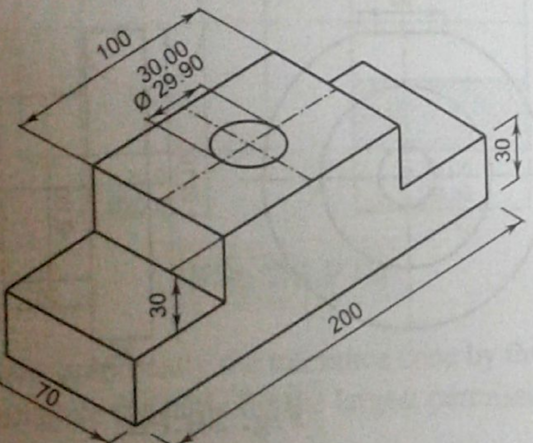


Fig. 20.P1

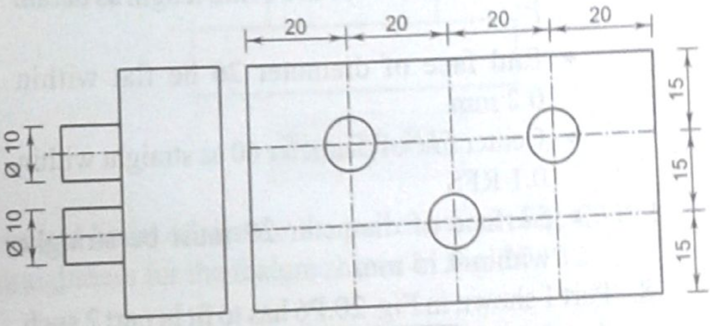


Fig. 20.P2

3. Figure 20.P3 shows a support with tolerance on dimensions of ±0.2 mm. Add the following geometrical tolerances:
   - Bottom as datum A having flatness of 0.01 mm
   - 20 mm diameter hole is to be parallel with datum A within 0.1 mm
   - 30 mm hole has to be perpendicular within 0.05 mm with datum A
   - Slot of 50 mm width is to be parallel within 0.15 mm to datum A
   - Slot has to be symmetrically in the center of length 100 mm within 0.5 mm
4. Draw orthographic views of the slide shown in Fig. 20.P4 and show the following information:
   - Vertical sides of slot should have maximum roughness of 3.2 µm
   - Top surface is to be milled having maximum roughness of 0.8 µm
   - Sides should have maximum roughness of 6.3 µm

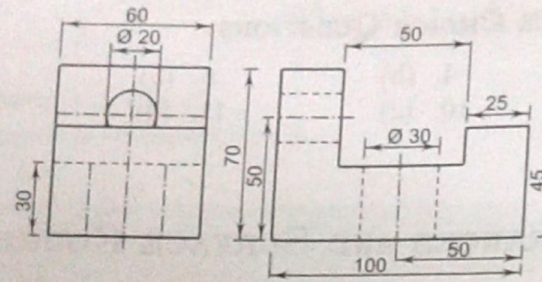


Fig. 20.P3

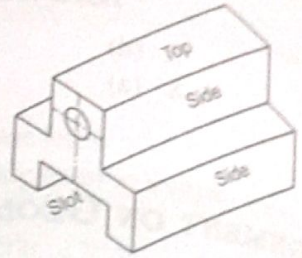


Fig. 20.P4 *A Slider*

CAD Assignment on Geometric Tolerances and Surface Roughness

5. For the problems Q 1 to Q 3 above, put geometric tolerances using AutoCAD.
6. Create a symbol for surface roughness. Use it for putting surface roughess for question 4 above.

Homework

7. Figure 20.P5 shows a stepped object with a flat portion on top of length 30 mm. Add geometrical tolerances as under:
   - Diameter 60 as datum A
   - End face of diameter 100 as datum B
   - Width of flat face of 30 mm length as datum C
   - End face of diameter 20 be flat within 0.2 mm
   - Center line of diameter 60 as straight within 0.1 RFS
   - Surface of diameter 20 must be straight within 0.15 mm

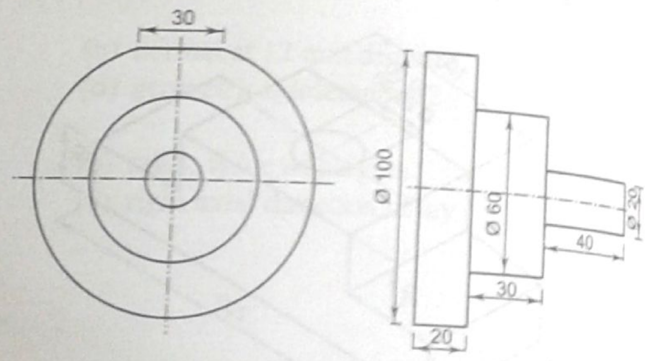


Fig. 20.P5

8. Part 1 shown in Fig. 20.P6 has to fit in part 2 such that there is no interference and maximum clearance is not more than 0.01 mm. Add maximum limit for groove of part 2. Put flatness tolerance of 0.02 mm for both the surfaces.
9. For a V block shown in Fig. 20.P7, select suitable machining processes and the roughness for that process. Then specify roughness of different face by roughness symbols.

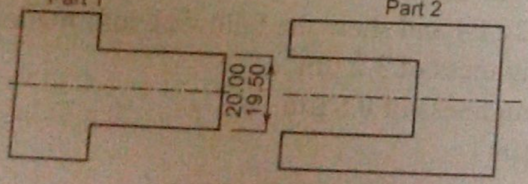


Fig. 20.P6

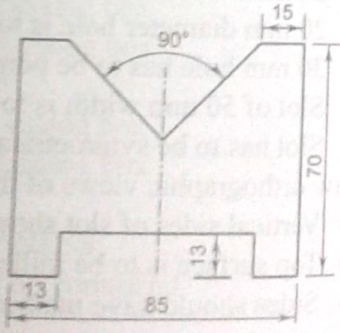


Fig. 20.P7 *A V Block*

PROBLEMS FOR PRACTICE

10. Surface A should have straightness tolerance of 0.015 mm and surface B of 0.02 mm. Add the feature control frame on the drawing shown in Fig. 20.P8.
11. Straightness is specified on a part (Fig. 20.P9) as 0.05 mm. What is the maximum permissible deviation from straightness of the center line if width of groove X = 15 ± 0.01?
12. What is the maximum possible deviation from straightness for hole shown in Fig. 20.P10, if perfect form at MMC is required?

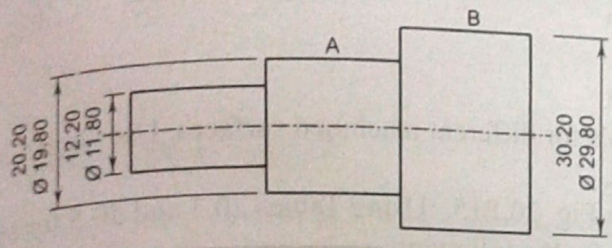


Fig. 20.P8

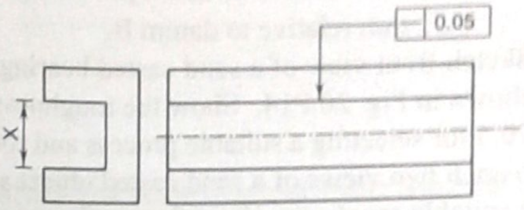


Fig. 20.P9

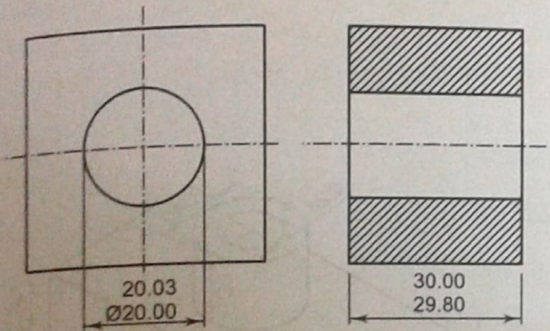


Fig. 20.P10

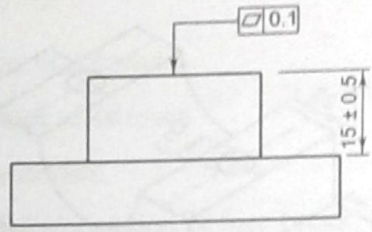


Fig. 20.P11

13. Show graphically the tolerance zone by thin lines and limits of size for the part shown in Fig. 20.P11.
14. Complete the table for the largest permissible straightness for the feature shown in Fig. 20.P12.

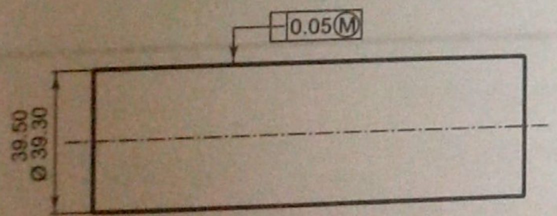


Fig. 20.P12

Feature diameter in mm	Permissible straightness tolerance
39.5	
39.4	
39.3	

15. Establish a geometric relationship between pin of diameter 35 mm stepped down to 25 mm with a ring of diameter 70 mm and length 55 mm shown in Fig. 20.P13 with following datum:

Datum for pin

Datum A – 35 mm diameter

Datum B – 25 mm diameter

Datum for ring

Datum A – 70 mm diameter

Datum B – 35.5 mm diameter

Datum C – 25.5 mm diameter

Add geometrical tolerances that will incorporate the following information:

- Position of axis of ring must be within 0.01 mm at MMC.
- Position of axis of pin must be within 0.015 mm.
- Perpendicular of left face of pin should be within 0.02 mm relative to datum B.

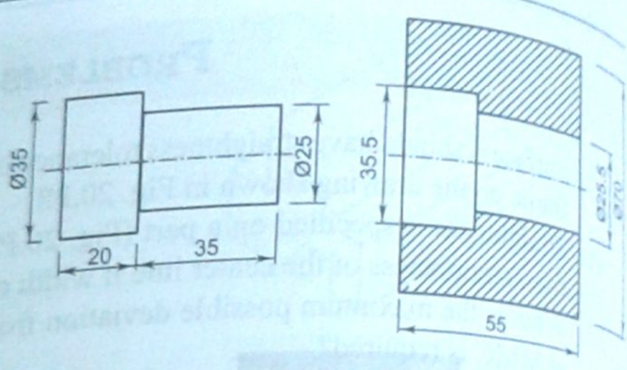


Fig. 20.P13

16. Sketch front view of a sand casted bearing support shown in Fig. 20.P14. Show the roughness symbols for different machined surfaces. Use Tables 20.3 and 20.5 for selecting a suitable process and roughness.
17. Sketch two views of a sand casted object shown in Fig. 20.P15. Using Tables 20.3 and 20.5 for selecting a suitable roughness, show the roughness numbers for the following:
    - Bottom of machine is to be milled
    - Outside of cylinder is unmachined
    - Vertical hole on a lathe
    - Sides on a shaper

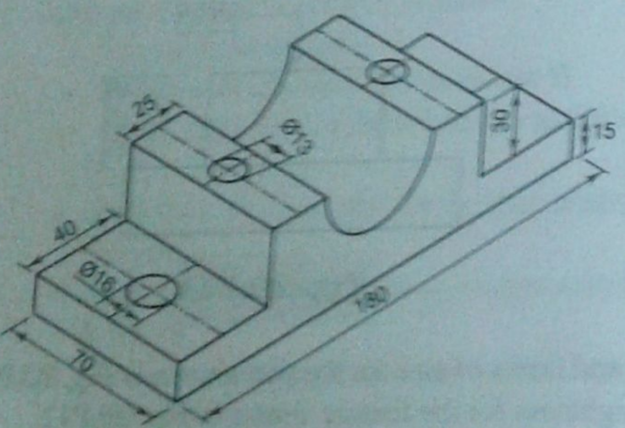


Fig. 20.P14 *A Bearing Support*

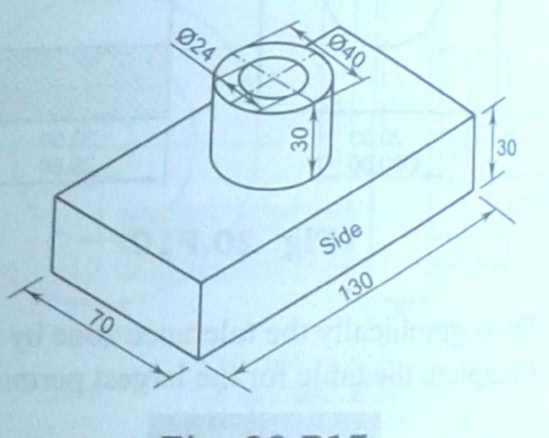


Fig. 20.P15

Callout 5 – Surface of hole should not have total runout more than 0.1 w.r.t. datum axis
Callout 6 – Left face of the boss should be parallel to datum A within 0.04

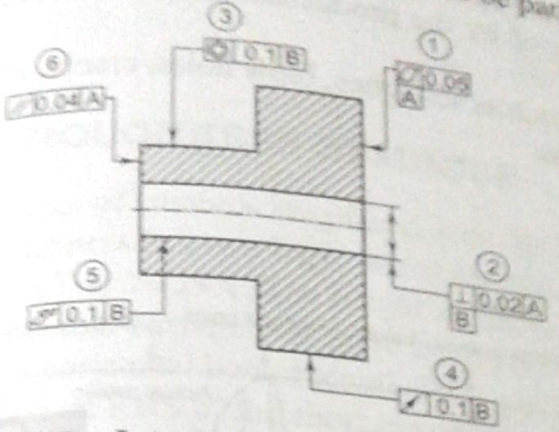


Fig. 20.S7 *Interpretaions of Drawing Callouts*

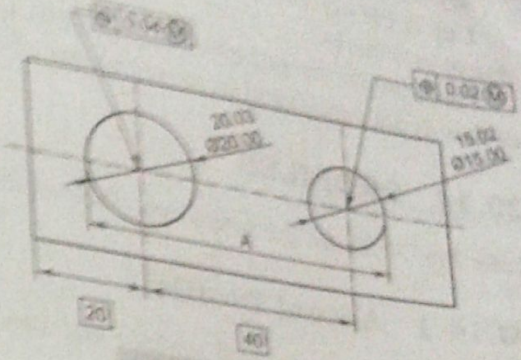


Fig. 20.S8 *A Jig Plate*

Example 8 A jig plate for drilling two holes is shown in Fig. 20.S8 with the position tolerances. Find the minimum and maximum center distance between the holes and distance A.

Solution Maximum distance between centers = 40 + (0.04/2) + (0.02/2) = 40 + 02 + 0.01 = 40.03 mm
Minimum distance between centers = 40 – (0.04/2) – (0.02/2) = 40 – 02 – 0.01 = 39.97 mm

Hence,
Maximum distance A will be 40.03 + (20.03/2) + (15.02/2) = 40.03 + 10.015 + 7.51 = 57.555 mm
Minimum distance A will be 39.97 + (20.00/2) + (15.00/2) = 39.97 + 10.000 + 7.500 = 57.470 mm

20.13 SURFACE TEXTURE

Surface texture includes surface roughness, waviness, lays, flaws, etc. No surface is smooth. Every surface has some roughness of micro-structure. Even for polished surfaces there are peaks and valleys if seen under a microscope. An actual surface, if exaggerated is as shown in Fig. 20.27. The various terms associated with this are as below:

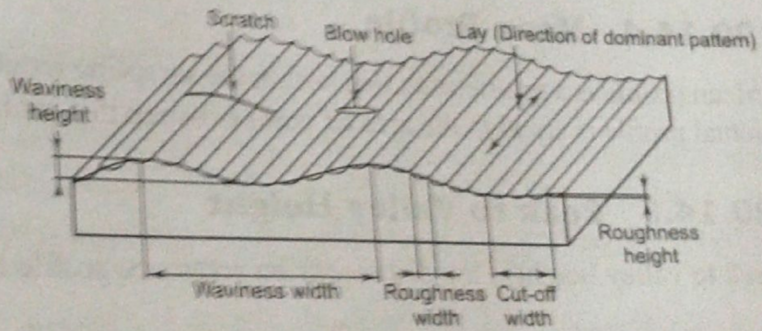


Fig. 20.27 *Surface Texture*

Roughness is the fine irregularity in the surface. For machined surfaces, roughness is caused by the cutting edge of the tool. Lesser the feed rate, lesser is the roughness.

Roughness width is the distance between two adjacent peaks or two adjacent valleys.

Roughness height is the arithmetic average deviation measured from the center line. It is indicated in microns.

Roughness cut-off is the width of the greatest spacing of repetitive surface irregularities to be included in the measurement of average roughness height. It is indicated in mm. Standard cut-off widths are 0.075, 0.25, 0.75, 2.5, 7.5 and 25 mm.

Waviness is the wider spacing than roughness cut-off width. It results from machine/work deflections vibrations, chatter, heat treatment or warping.

Waviness height is the peak to valley distance of the waviness curve.

Waviness width is the spacing in mm between two successive wave peaks or successive valleys.

Lay is the direction of dominant surface pattern determined by the production methods. This is caused due to tool marks.

Flaws are the irregularities that occur at a place or places such as scratches, blow holes, cracks, etc. Effect of flaw is not considered in measurement of surface roughness.

20.14 PROFILES

Refer Fig. 20.28 for the terms described as follows.

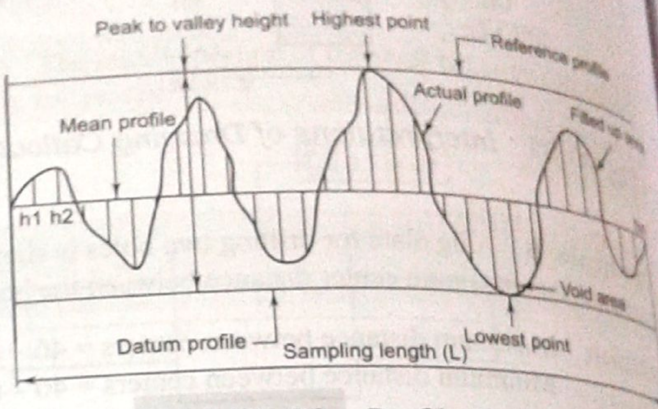


Fig. 20.28 *Profile Definitions Related to a Surface*

20.14.1 Actual Profile

Actual profile is the actual surface obtained by a manufacturing process.

20.14.2 Reference Profile

Reference profile passes through the highest point of the actual profile. All irregularities are referred to this profile.

20.14.3 Datum Profile

Datum profile passes through the lowest point of the actual profile and is parallel to reference profile.

20.14.4 Mean Profile

Mean profile is a profile such that within the sampling length (L), the filled up areas between this profile and actual profile is equal to the area of voids between this profile and actual profile.

20.14.5 Peak to Valley Height

Peak to valley height is the distance from reference profile to datum profile.

20.14.6 Mean Roughness Index (Ra)

Mean roughness index is the arithmetic mean of the absolute values of the heights $h_1 + h_2 + h_3 + \cdots\cdots h_n$ between the actual and mean profiles (Fig. 20.28). It is given by relation

$$Ra = 1/L \int_{x=0}^{x=L} |hi|\, dx$$

20.15 SURFACE ROUGHNESS NUMBER

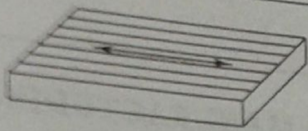
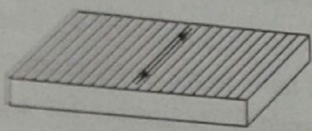
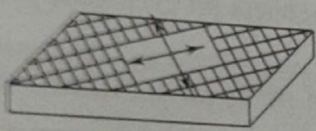
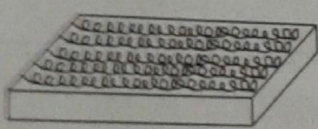
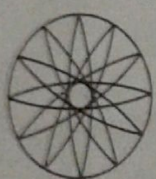
Surface roughness number is the average departure of the surface over a sampling length. This length is generally taken as 0.8 mm, i.e. 800 microns. Measurements are made along a line at right angles to the d

m. A symbol with a set of specifications can be named as X, Y, etc. and only its name can be indicated, whose meaning can be explained separately as shown in the Table 20.1.

20.17 LAY

Symbols used to indicate a lay showing the dominant tool marks are shown in Table 20.2. Tool marks of a shaper are as shown in Parallel lay. If the job is turned by 90°, for the same job the lay appears as perpendicular. If the job is set an angle, a crossed lay may appear. Vertical surface grinding machines produce a multi-directional lay. Facing of a lathe job will produce a circular lay. A suitable symbol can be used as required. No Lay symbol is taken as parallel direction.

Table 20.2 Symbols used for direction of lay

Direction of lay	*Graphical interpretation*	*Symbol*	*Indication*
Parallel to plane of projection of the view in which symbol is used		=	√ =
Perpendicular to plane of projection of the view in which symbol is used		⊥	√ ⊥
Crossed in two slant directions relative to plane of projection of the view in which symbol is used		X	√ X
Multi-directional		M	√ M
Approximately circular relative to the center of the surface to which the symbol is applied		C	√ C
Approximately radial relative to the center of the surface to which the symbol is applied		R	√ R

20.18 ROUGHNESS GRADE NUMBER AND GRADE SYMBOLS

Roughness can also be indicated by roughness grade numbers from N1 to N12 or by roughness grade symbols shown in Table 20.3.