Machine Drawing

Class 4 : Geometric Tolerances

Textbooks and References

- *N. Sidheswar, P. Kanniah and V.V.S. Sastry, Machine Drawing, Tata McGraw Hill, 2001*
- *Singh, Ajeet. Machine Drawing, Tata McGraw-Hill, 2007.*
- *SP 46: 1988 Engineering Drawing Practice for School & Colleges. Bureau of Indian Standards*

INTRODUCTION

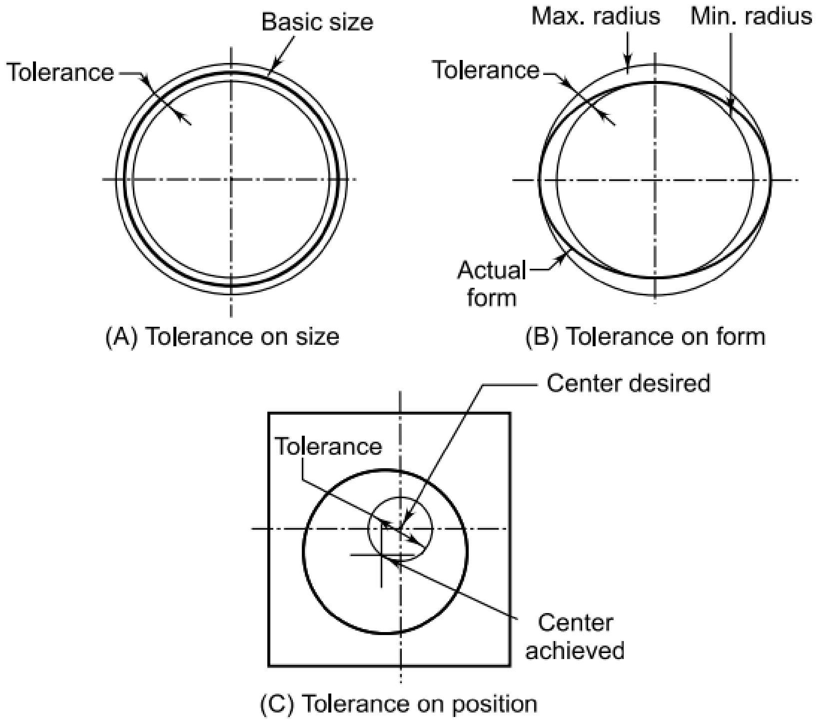


Fig. 20.2 *Types of Tolerances*

Geometric Tolerances

Permissible variations in the geometrical shapes of individual objects or orientation or position w.r.t other objects are called geometric tolerances.

Tolerance zone is an imaginary area/volume within which a component must be contained. Specific portion of a component such as a hole, surface or a profile is called a feature.

14 standardized tolerance symbols

Single feature,
Related feature
Run out

Geometric Tolerances

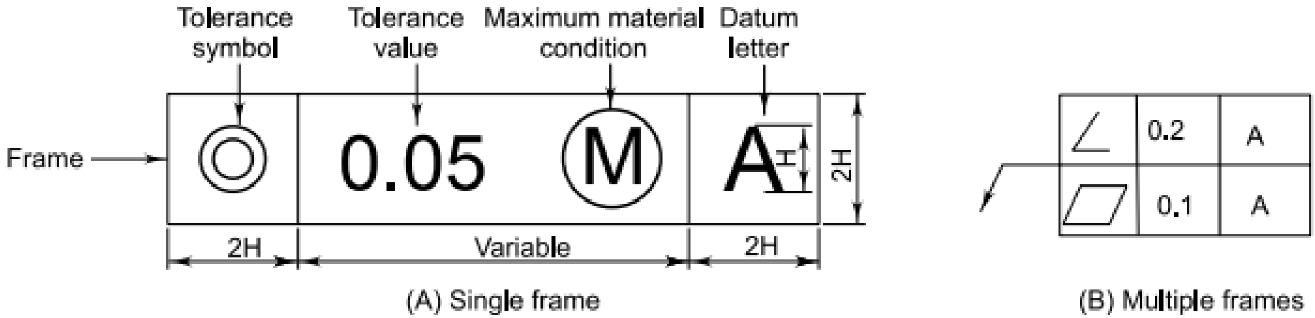


Fig. 20.4 *A Frame and Its Contents*

[(81) Understanding GD&T - YouTube]

Geometric Tolerances

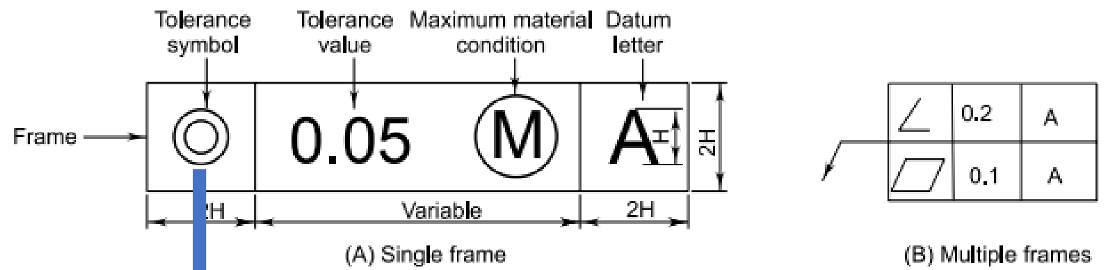


Fig. 20.4 *A Frame and Its Contents*

(81) Understanding GD&T - YouTube

INTRODUCTION

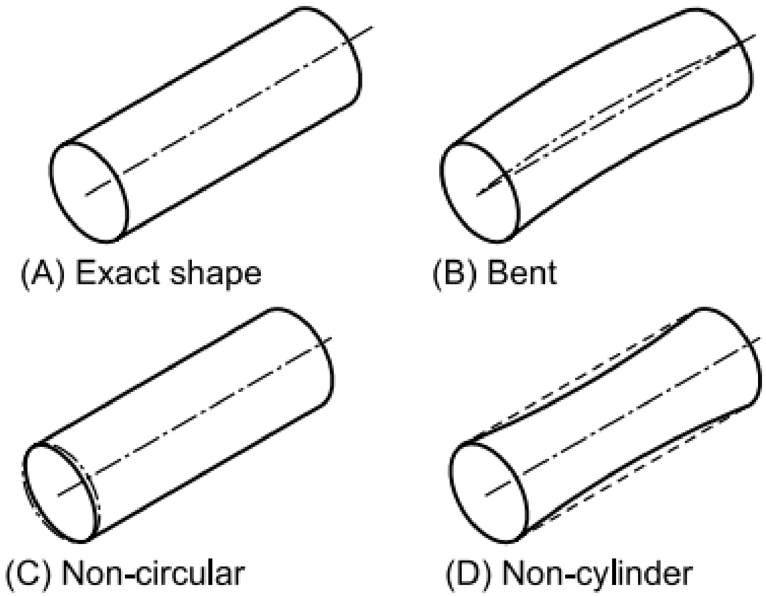
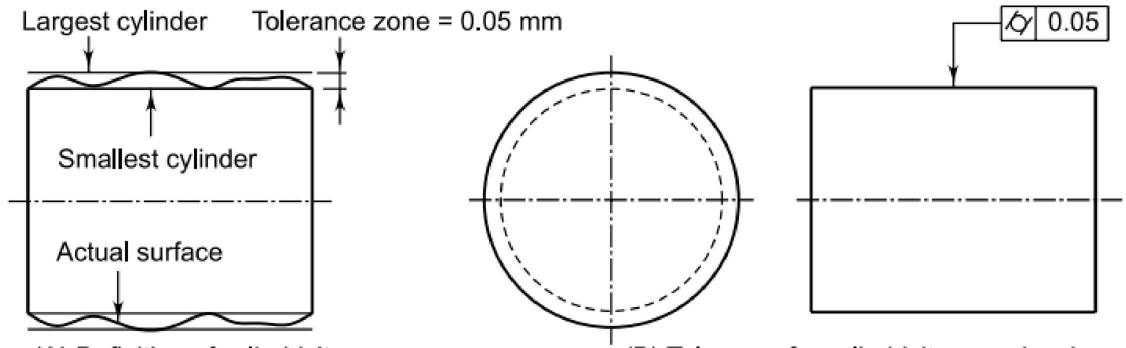


Fig. 20.1 *Geometric Shape Variations*

Cylindricity

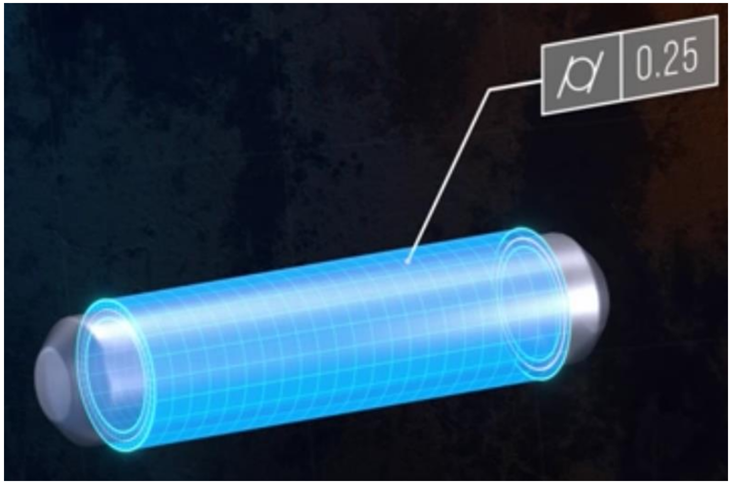
Cylindricity is the difference in value of radii between two imaginary cylinders, enveloping cylinders at outermost and innermost surfaces. Figure 20.14A shows the variation in the surface of a cylinder along its axis. The diameter at every cross-section is different and lies in a circular zone. This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



(A) Definition of cylindricity

(B) Tolerance for cylindricity on a drawing

Fig. 20.14 *Indicating Cylindricity on a Drawing*



Geometric Tolerances

Geometric Tolerance

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

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.

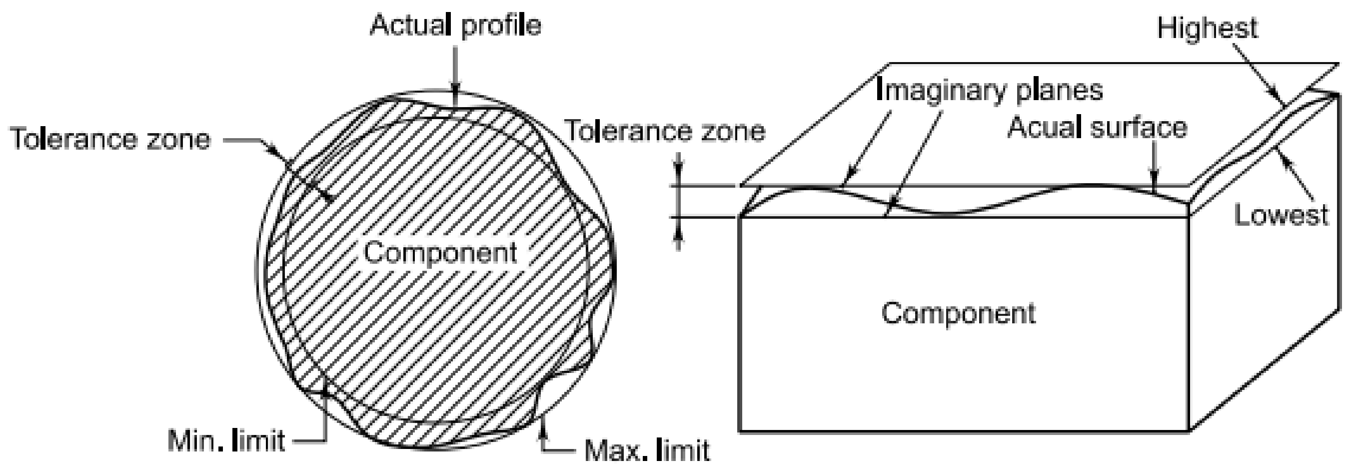


Fig. 20.3 *Tolerance Zone*

Flatness

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

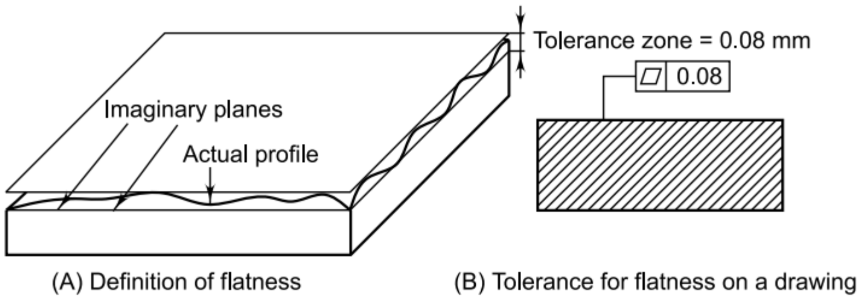
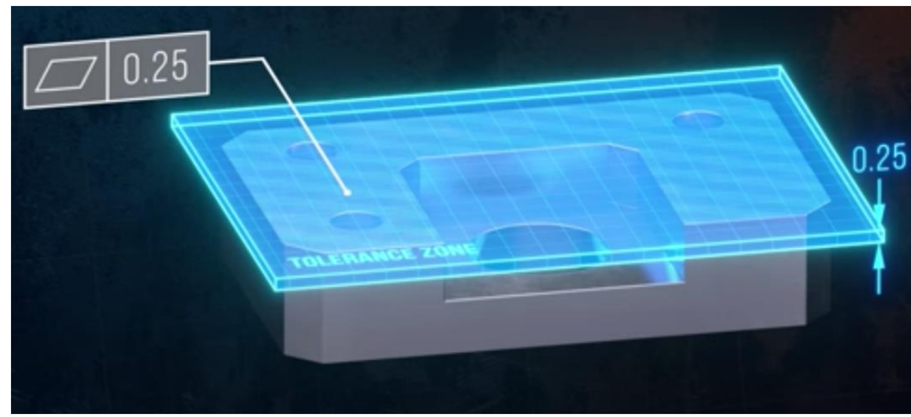
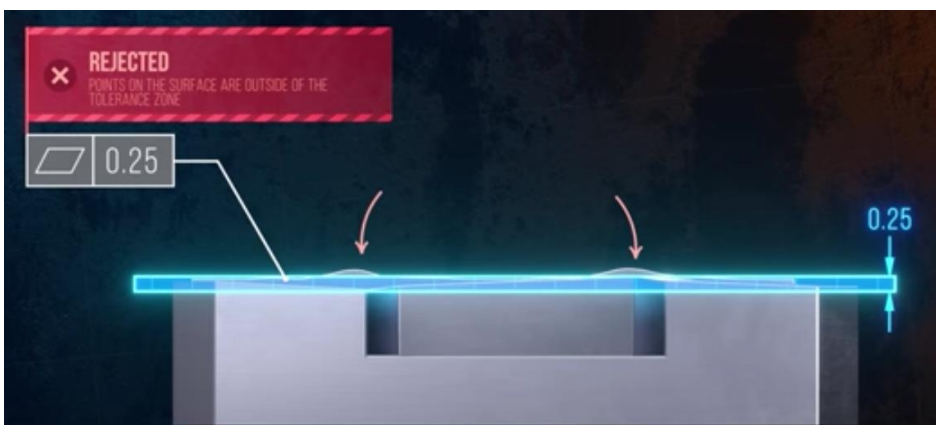


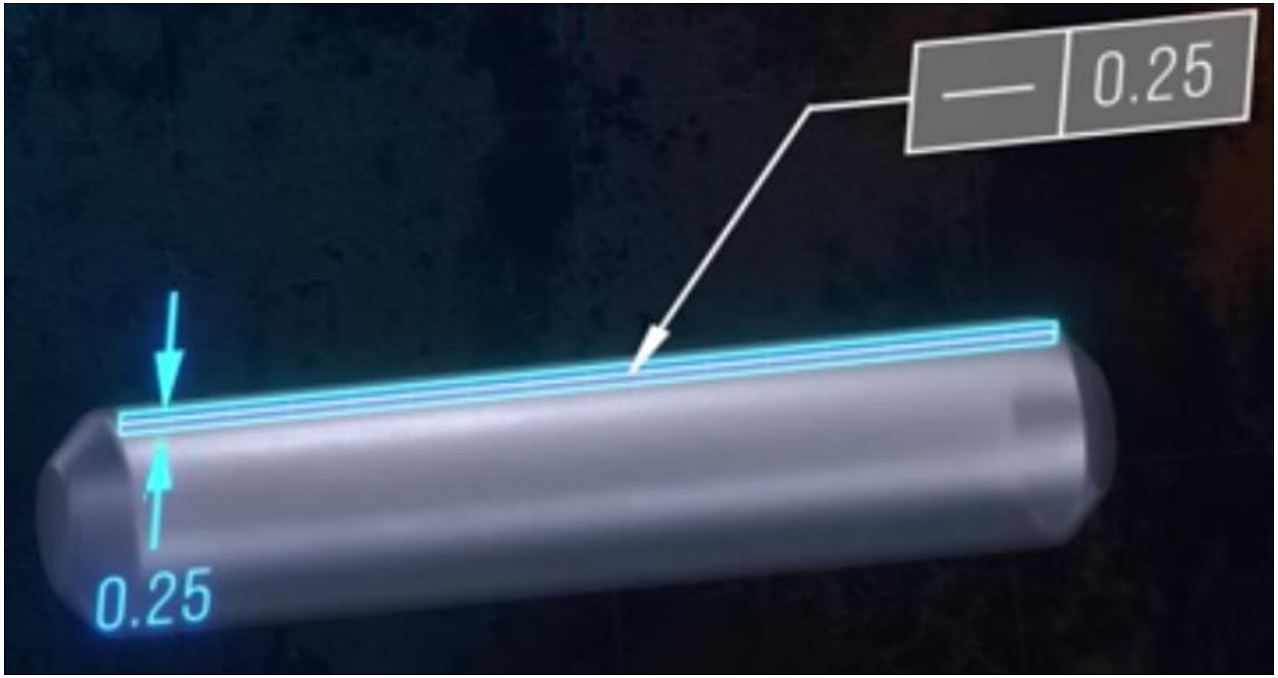
Fig. 20.12 *Indicating Flatness on a Drawing*





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.This tolerance zone is indicated in a frame as shown in Fig. 20.11B.



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.

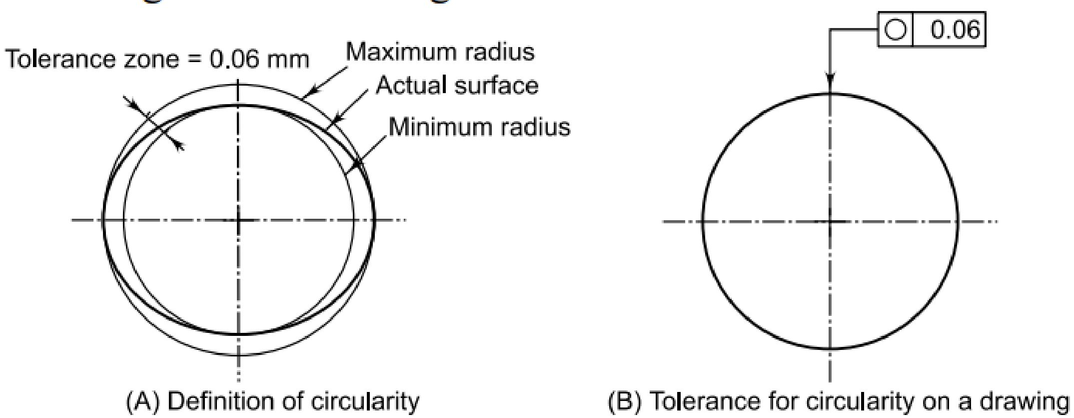


Fig. 20.13 *Indicating Circularity on a Drawing*

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.

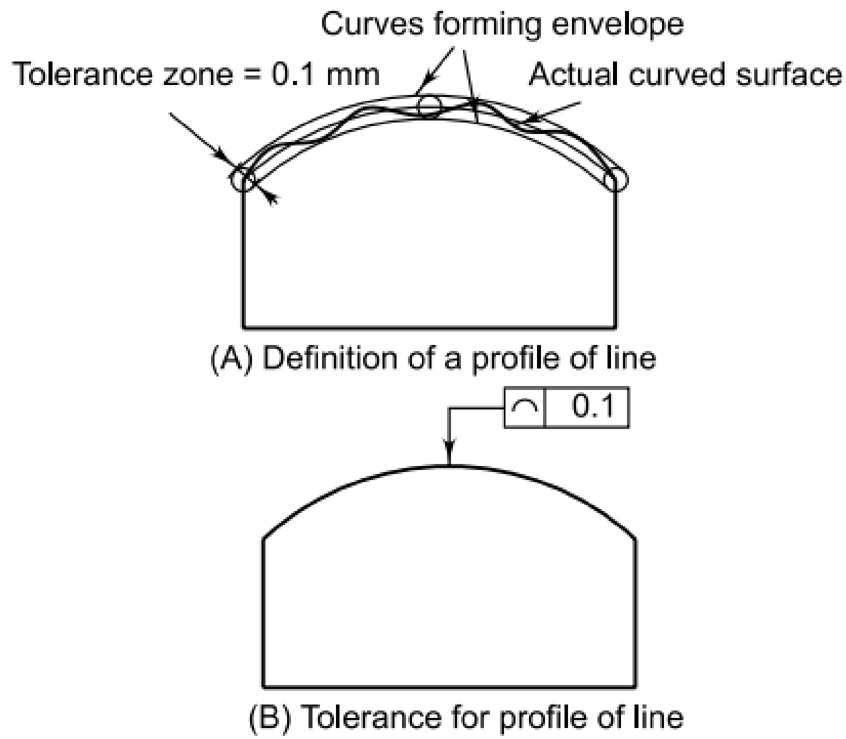


Fig. 20.15 *Indicating Profile of a Line on a Drawing*

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.

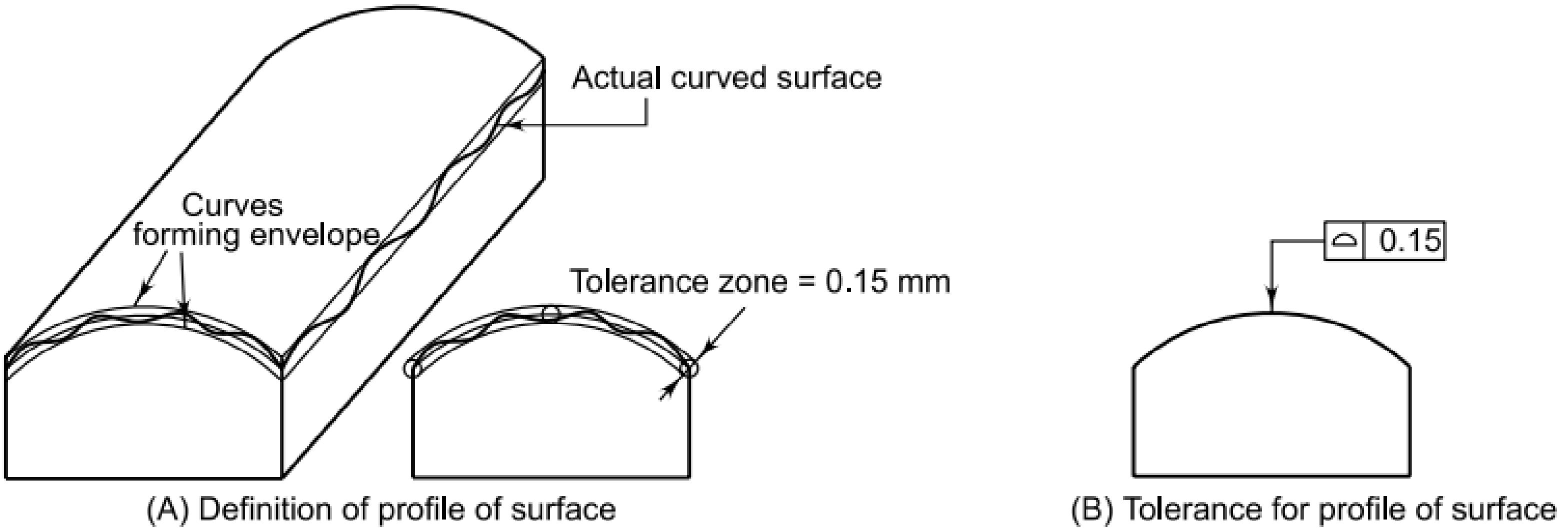


Fig. 20.16 *Indicating Profile of a Surface on a Drawing*

20.11.2 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.

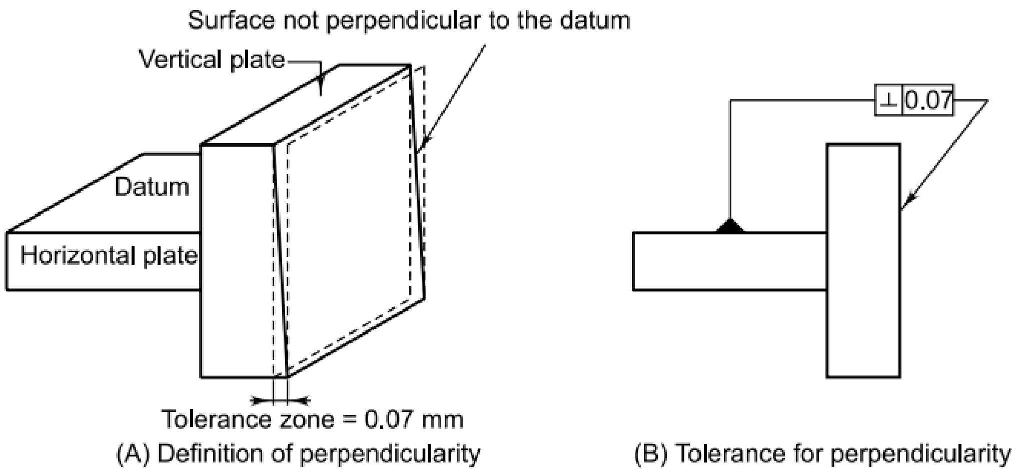


Fig. 20.18 *Indicating Perpendicularity of a Surface on Drawing*

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.

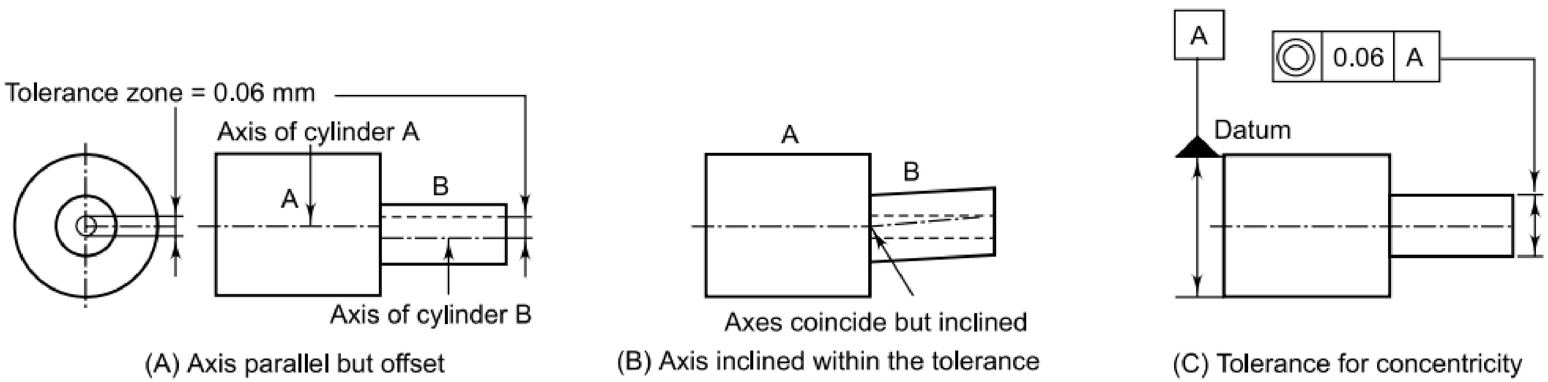


Fig. 20.20 *Indicating Tolerance for Concentricity on a Drawing*

Geometric Tolerances

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.

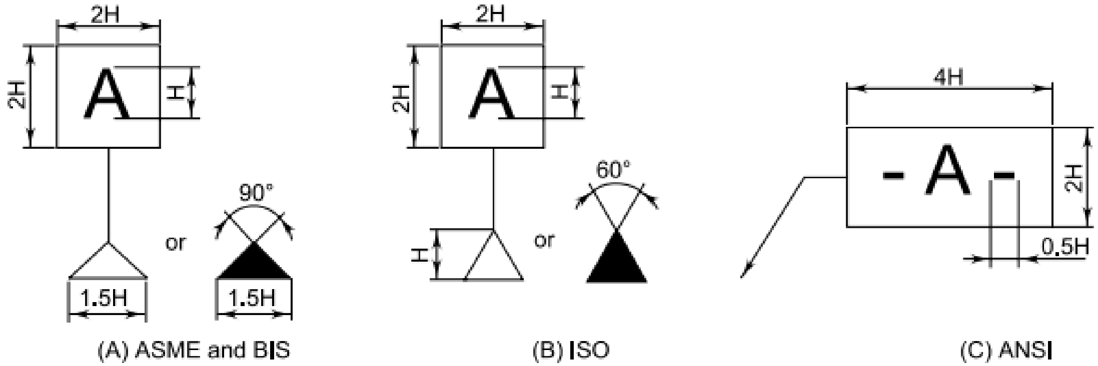


Fig. 20.6 *Datum Triangle and Datum Letters*

20.5.4 Multi-datums

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

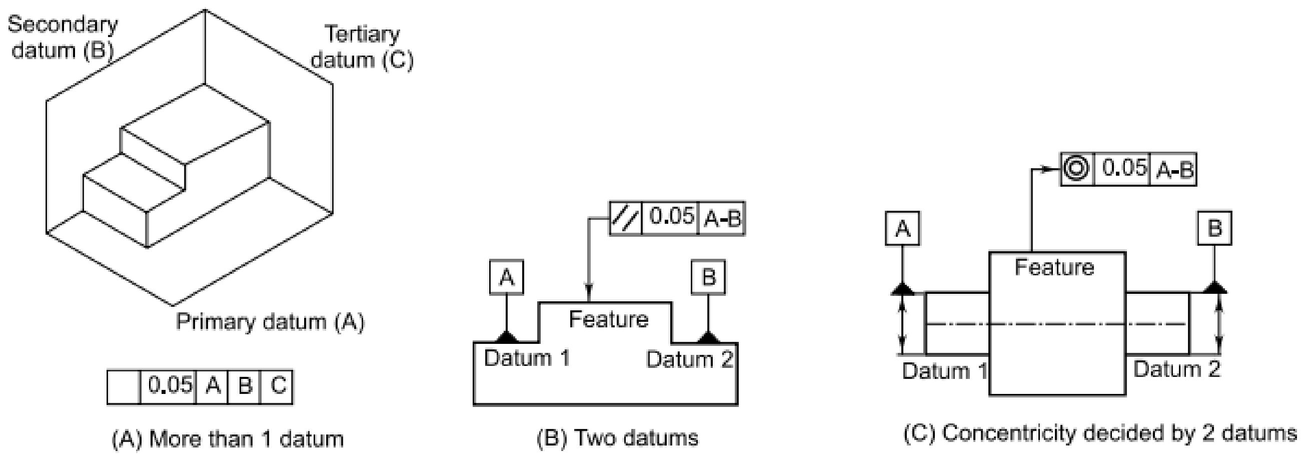
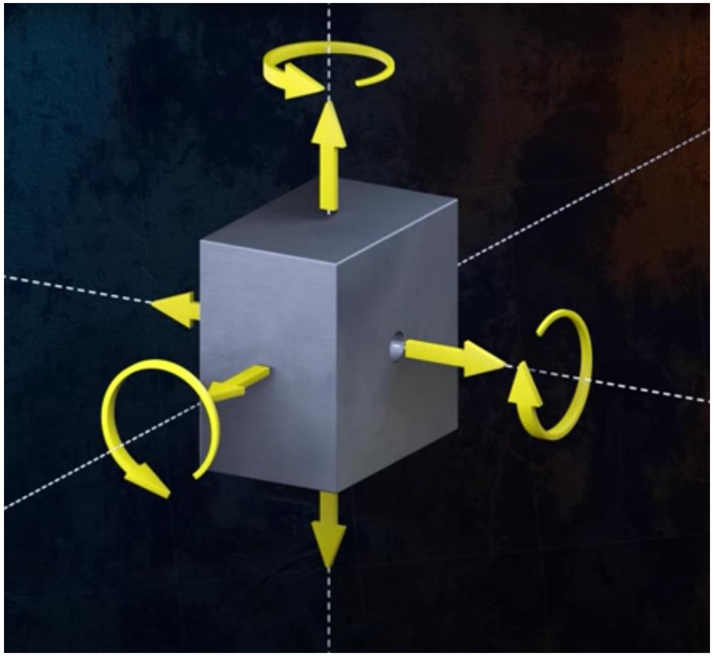
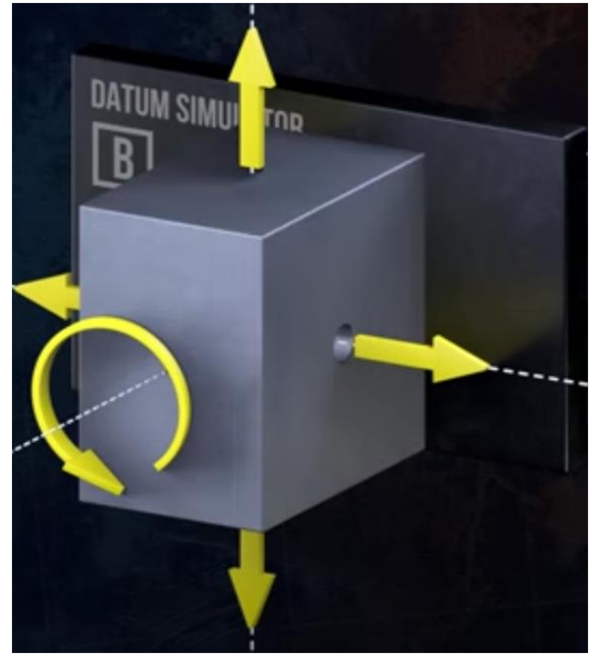


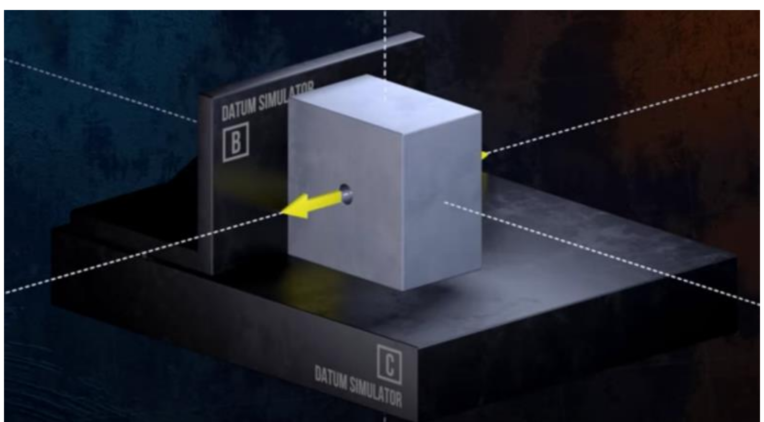
Fig. 20.7 *Multiple Datums*

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).

Datum







Next Assignment

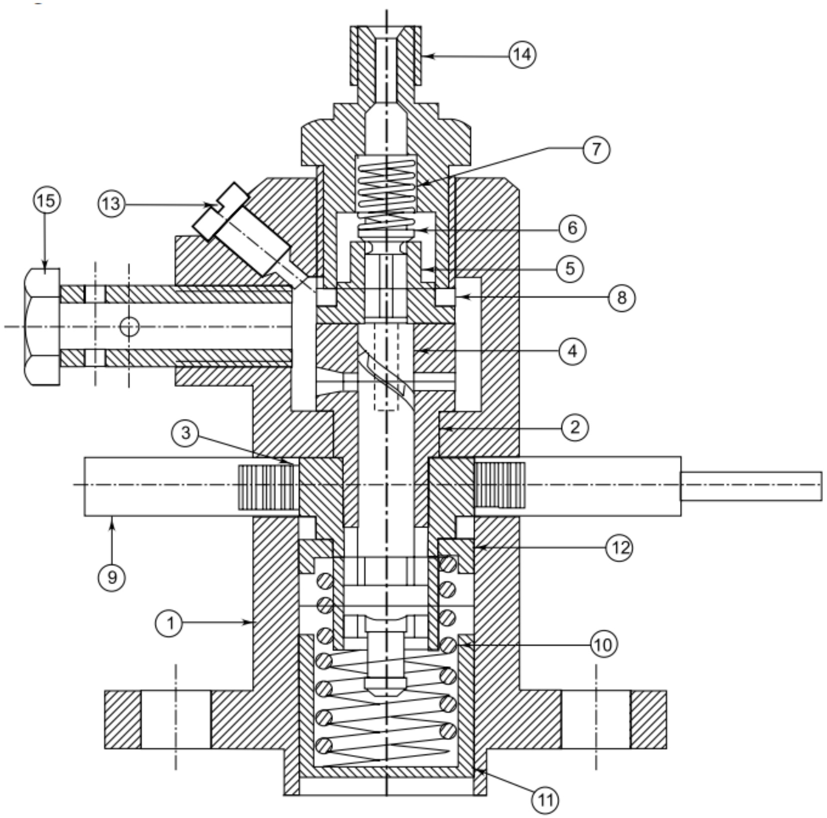
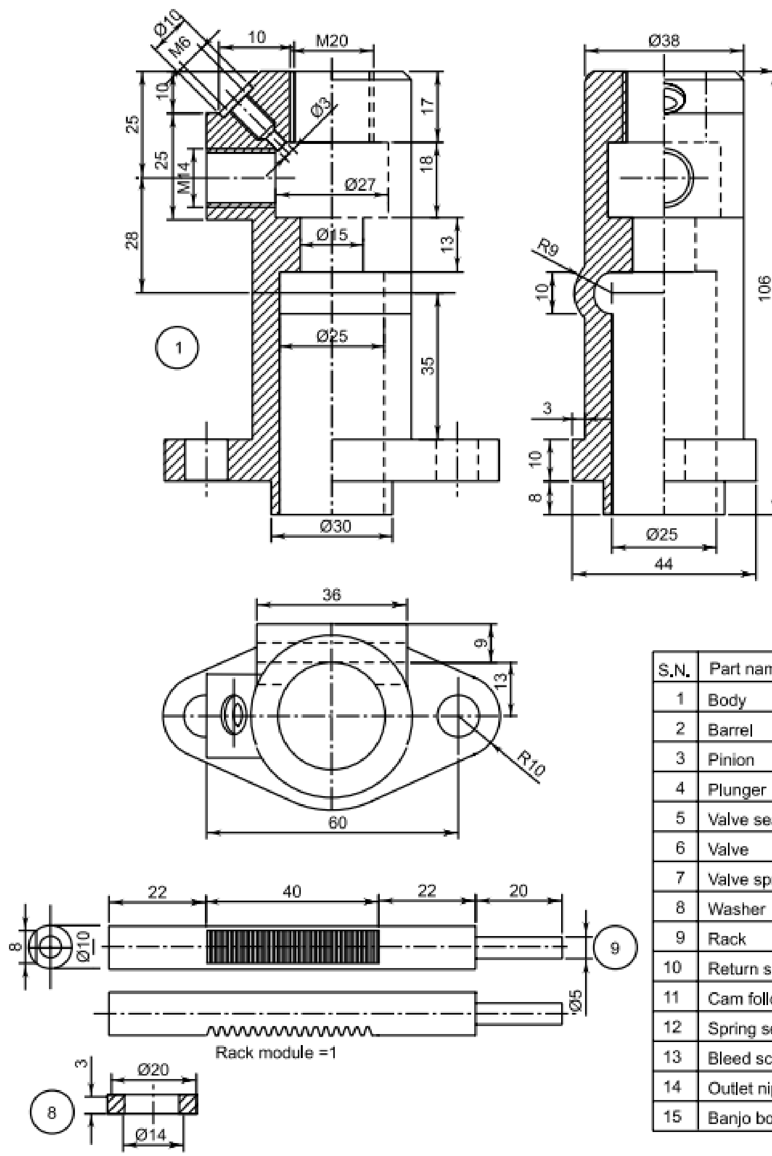


Fig. 28.19A *Assembly Drawings of a Diesel Fuel Pump*



Part list

S.N.	Part name	Material	Qty
1	Body	Cast iron	1
2	Barrel	C.Steel	1
3	Pinion	C.Steel	1
4	Plunger	C.Steel	1
5	Valve seat	M.S.	1
6	Valve	M.S.	1
7	Valve spring	Sp.S.	1
8	Washer	Copper	1
9	Rack	C.steel	1
10	Return spring	Sp.steel	1
11	Cam follower	C.steel	1
12	Spring seat	M.S.	1
13	Bleed screw	M.S.	1
14	Outlet nipple	M.S	1
15	Banjo bolt	M.S.	1

Fig. 28.19B *Part Drawings of a Diesel Fuel Pump...Contd.*

Next Assignment

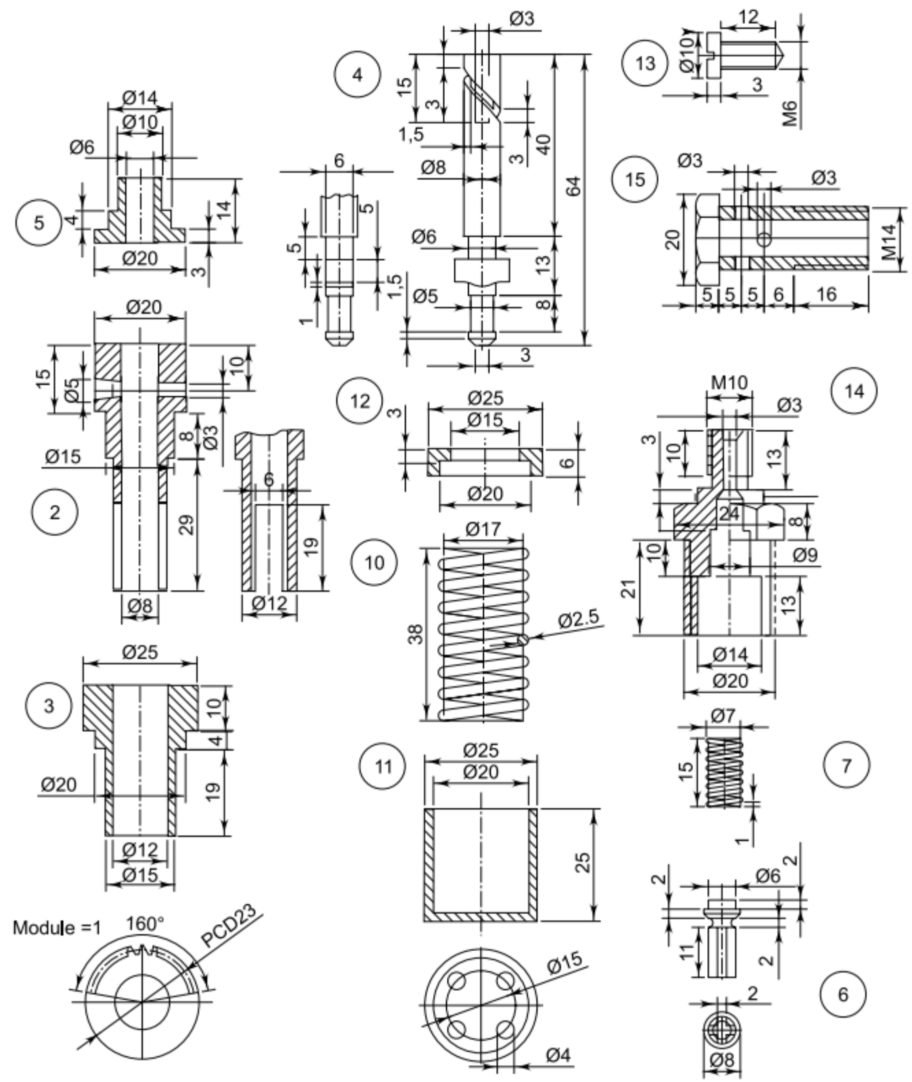


Fig. 28.19B *Part Drawings of a Diesel Fuel Pump*

Thank you !!!