TOLERANCES OF FORM AND POSITION (GEOMETRIC TOLERANCES)

1. Introduction

The specification of positional and form (geometrical) tolerances has become a rather complex procedure, since in many cases the geometric characteristics of a part must be given in great detail. For example, flatness straightness, roundness, cylindricity, parallelism, perpendicularity, and other such characteristics may be toleranced on a drawing by a symbol. The symbols introduced in this chapter are suggested by the Turkish Standardization Institution (TSE) and extracted from TS 1304.

The two basic types of geometric dimensions utilizing tolerances are:

a) form tolerances
b) positional tolerances.

Form tolerances can be applied to a single geometric shape. For instance, to specify flatness, only one surface has to be toleranced. Form may include the relationship between two or more features on the same part. Thus, for example, parallelism involves the relationship of one plane to another, the latter being used as a datum plane. Both planes are on the same part.

Positional tolerances are applied those dimensions that are used to locate or position geometric shapes with respect to specified datum planes.

A datum is a feature of a part that acts as a master reference used to locate other features of the part. A datum can be a point, a line , or a plane. Datums are chosen for their functional nature. Any datum should be readily available feature, such as a finished surface on a bench vise.

Symbols of geometric tolerances (both form and position) are given in Table 1. Abbreviations for modifying symbols are also shown in Table 2.

	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL
FOR INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	⏤
		FLATNESS	▱
		CIRCULARITY (ROUNDNESS)	○
		CYLINDRICITY	⌭
FOR INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A LINE	⌒
		PROFILE OF A SURFACE	⌓
FOR RELATED FEATURES	ORIENTATION	ANGULARITY	∠
		PERPENDICULARITY	⊥
		PARALLELISM	//
	LOCATION	POSITION	⌖
		CONCENTRICITY	◎
	RUNOUT	CIRCULAR RUNOUT	↗*
		TOTAL RUNOUT	⌰*

*Arrowhead(s) may be filled in.

TERM	SYMBOL
AT MAXIMUM MATERIAL CONDITION	Ⓜ
REGARDLESS OF FEATURE SIZE	Ⓢ
AT LEAST MATERIAL CONDITION	Ⓛ
PROJECTED TOLERANCE ZONE	Ⓟ
DIAMETER	Ø
SPHERICAL DIAMETER	SØ
RADIUS	R
SPHERICAL RADIUS	SR
REFERENCE	( )
ARC LENGTH	⌒

2. Form Tolerances

There are ten types of form tolerance symbols. These are:

2.1. Straightness: A surface is straight when all its elements are straight lines. Figure 1 shows the tolerance zone applied to a cylindrical component. This indicates a tolerance of 0.03 mm error in a 100 mm length. Note how the symbol is shown in the first compartment of the box, the tolerance next, followed by the testing length.

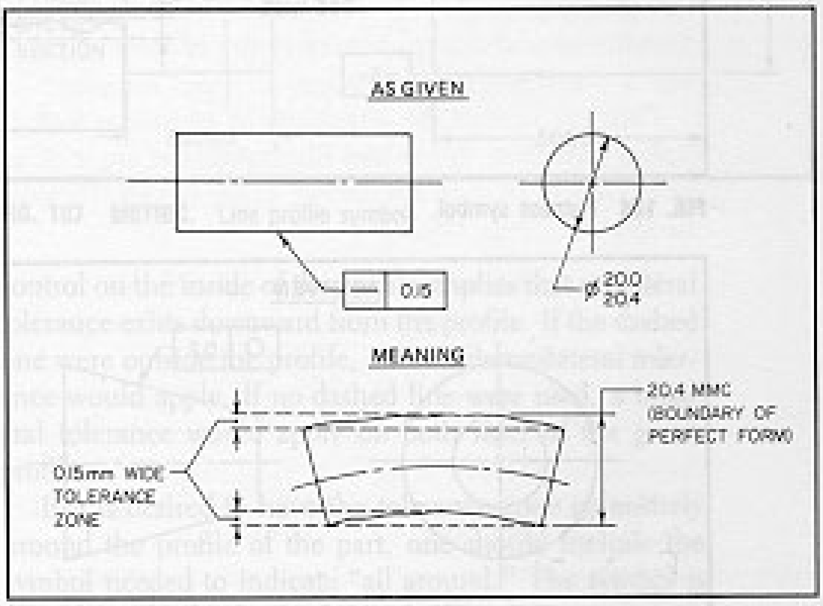


FIG. 102 METRIC. Straightness symbol controlling surface elements.

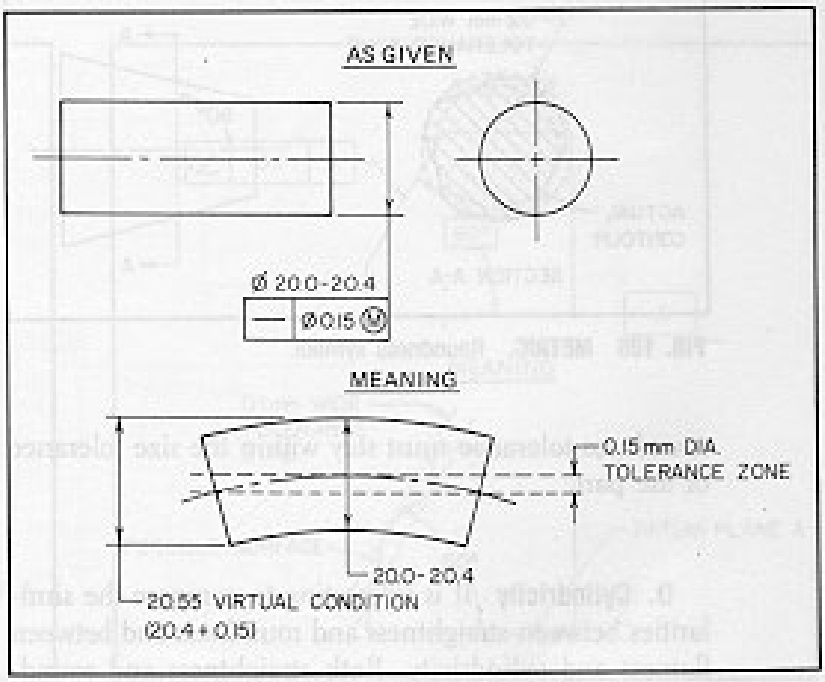
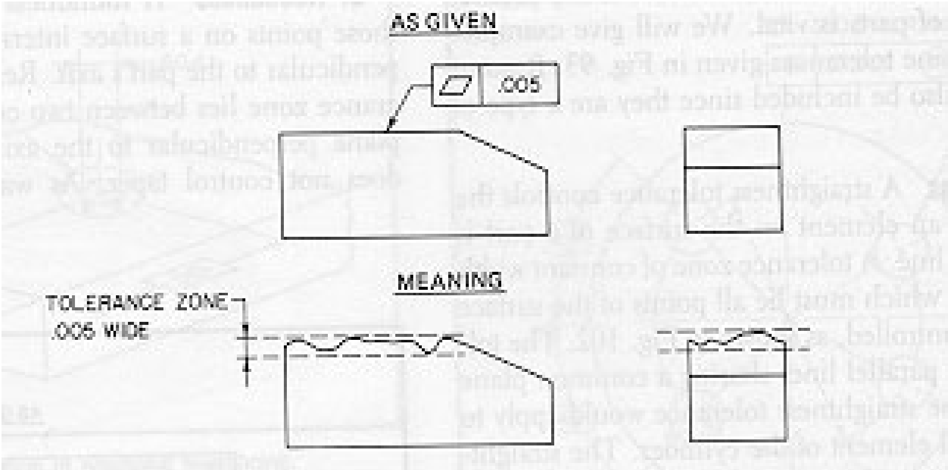
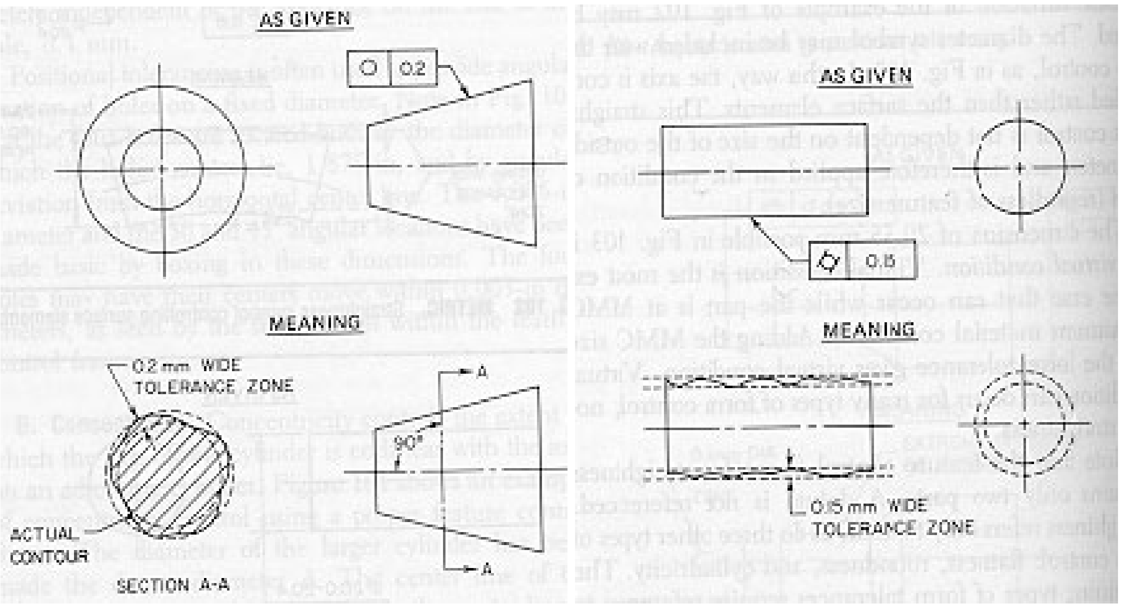


FIG. 103 METRIC. Straightness symbol controlling axis.

2.2. Flatness: A surface is flat when all its elements are in one plane. Flatness tolerance specifies a tolerance zone between two parallel planes. Note the symbol followed by the tolerance in Figure 2, length for testing is not stated at this time.

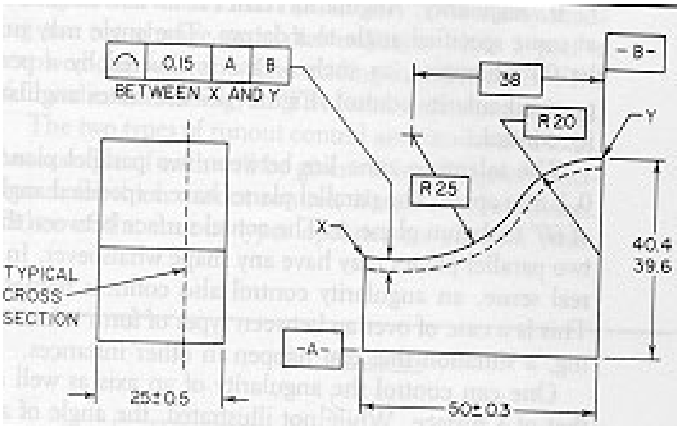


2.3. Roundness: A roundness tolerance controls only those points on a surface intersected by any plane perpendicular to the part's axis. This tolerance can be required on cylinders, cones and spheres, and the amount of ovality must lie within the tolerance zone. Note the symbol and the tolerance in the box (see Figure 3).

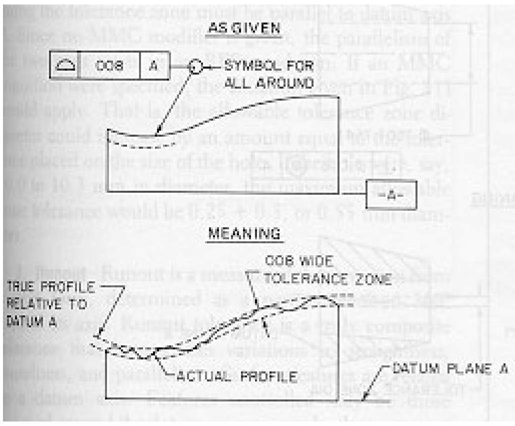


2.4. Cylindricity: It is interesting to compare the similarities between straightness and roundness and between flatness and cylindricity. Both straightness and roundness relate to a line condition. Straightness relates to a straight line, while roundness wraps the line around a surface. Flatness and cylindricity both involve areas (2-D), but the area for cylindricity wraps around a surface. As seen in Figure 4, a cylindricity tolerance defines a tolerance zone consisting of two concentric cylinders. Cylindricity tolerance is used for cylinders only, and really controls taper, roundness, and straightness.

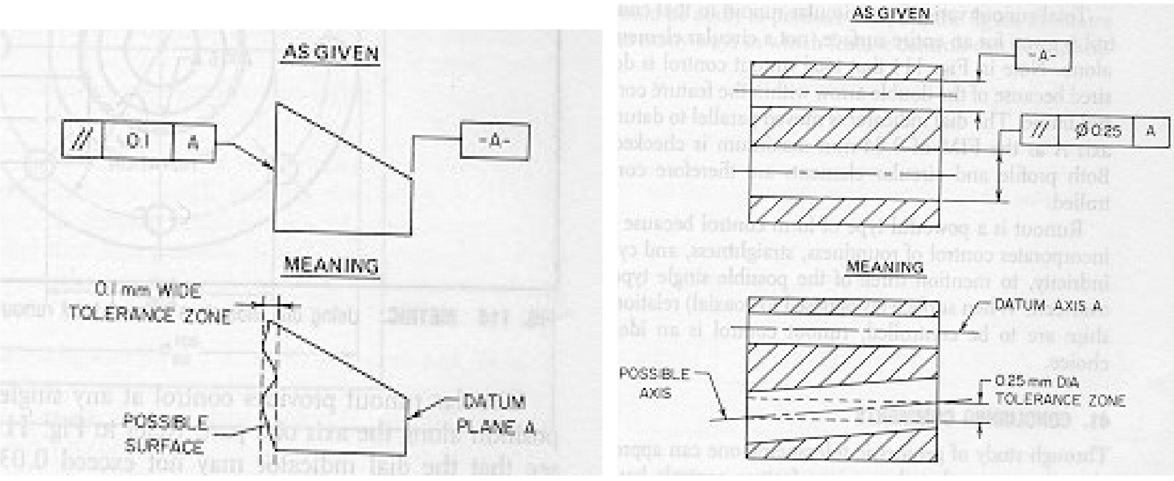
2.5. Profile of a Line: The profile of any part is outline as seen in any two dimensional view. Profiles contain combinations of lines and arcs. A true profile is described by the use of basic (boxed) dimensions without tolerance. A feature control frame contains the tolerance zone to be applied to the basic dimension. Note the symbol and the box in Figure 5.



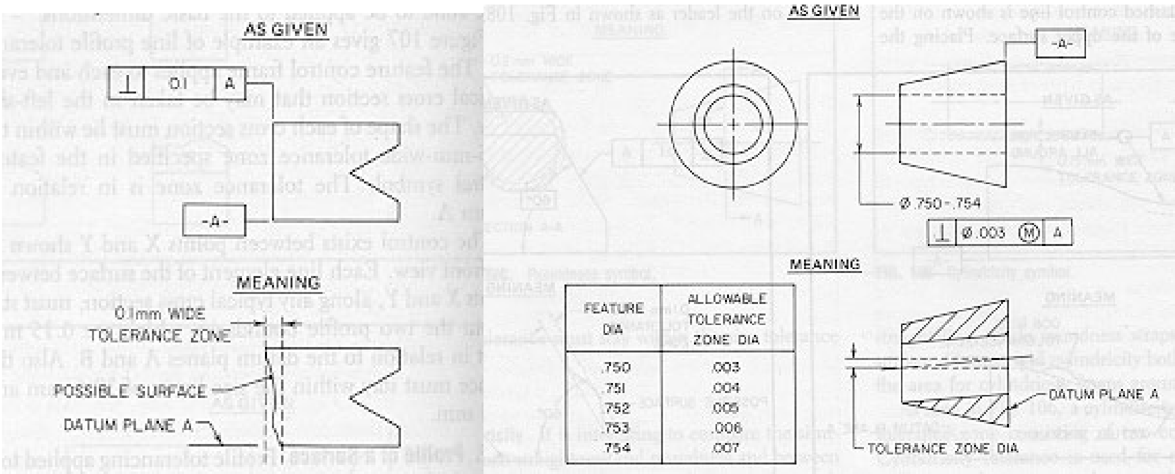
2.6. Profile of a Surface: It is applied to a surface, the tolerated error again straddles the true line, as shown in Figure 6. It is actually more common than the line profile tolerancing.



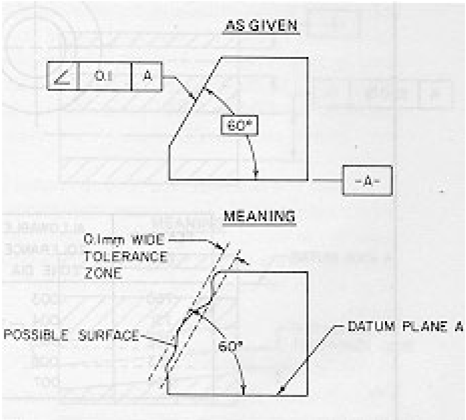
2.7. Parallelism: Parallelism exists when a feature is at a constant distance from a datum. The feature could be an axis or a plane, as could be the datum. The symbol, the tolerance and the datum letter have to be shown in the box, as in Figure7.



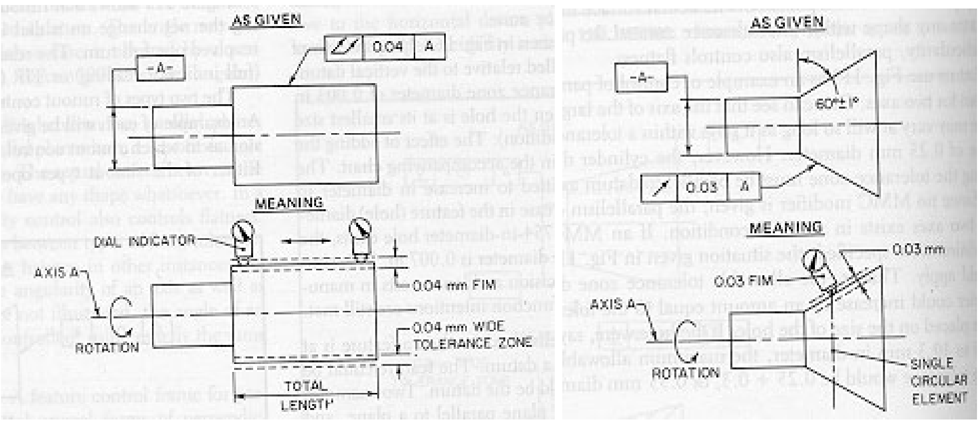
2.8. Perpendicularity: It can control perpendicularity of a plane with a plane, an axis to a plane, an axis to an axis. Figure 8 shows an example of perpendicularity tolerance.



2.9. Angularity: Angularity relates to an axis or surface at some specified angle to a datum. The angle may not be 90°, however, as such a case is covered by a perpendicularity control. (see Figure 9)

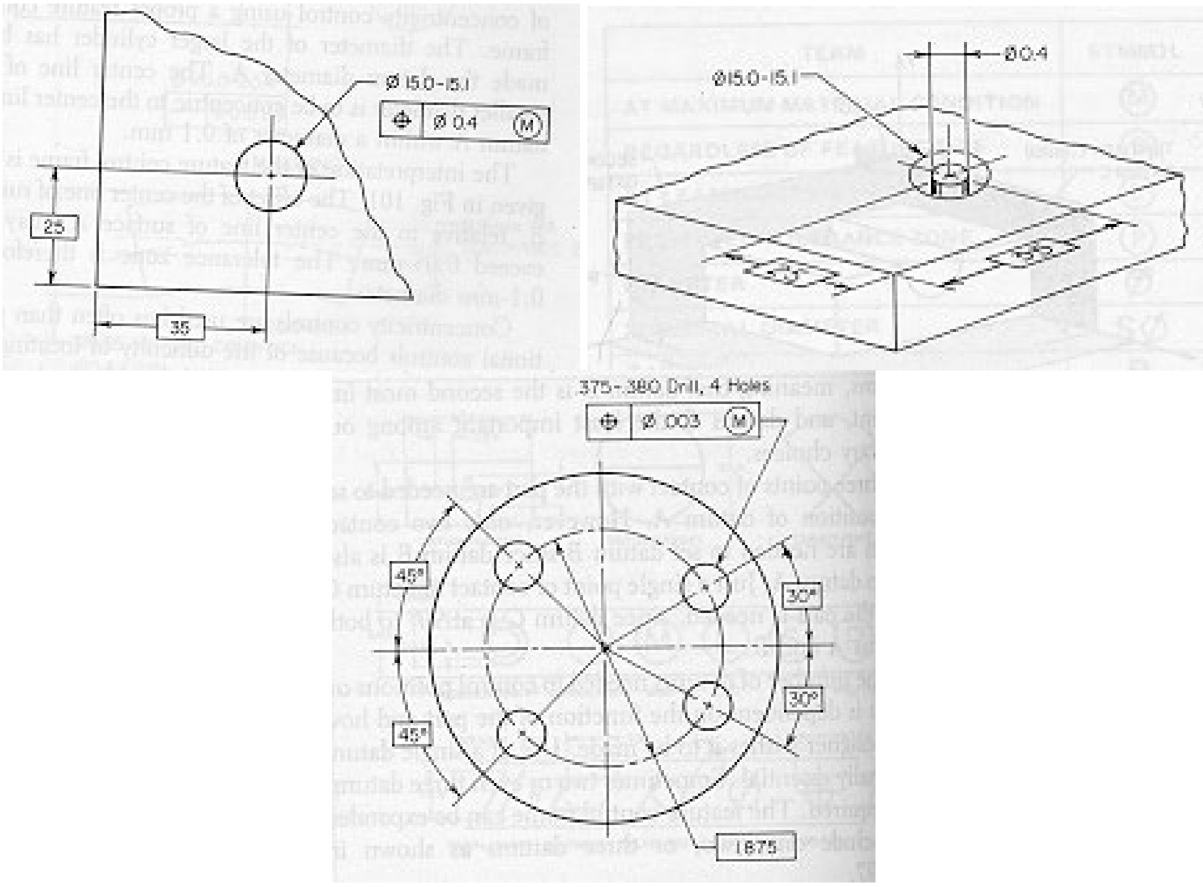


2.10. Runout: Runout is a measure of deviation from perfect form, determined as a part revolved around its axis. Runout tolerance is a truly composite tolerance that incorporates variations in straightness, roundness and parallelism. Surface features are related to a datum axis. Features controlled may be those wrapped around the datum axis, or may be those perpendicular to the datum axis. An example of runout tolerance is given in Figure 10.

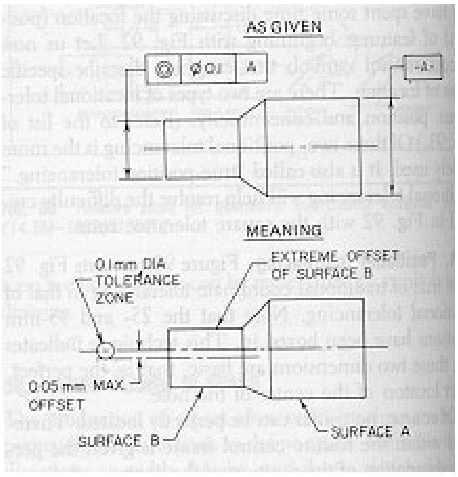


3. Position Tolerances
There are three types of position tolerance symbols. These are true position, concentricity, and symmetry.

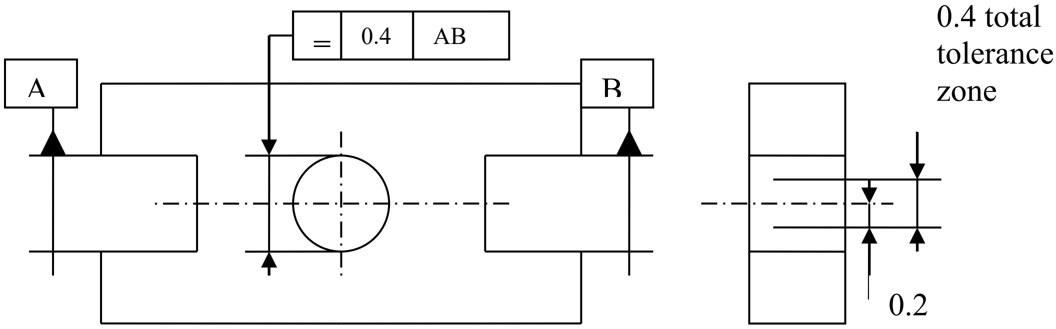
3.1. True position: As shown in Figure 11, true position is the location of holes or slots with respect to surfaces. The value 0.4 in the figure represents the tolerance zone; that means, the center of the hole must be on the theoretical interactions of two given axis and within the 0.4 mm diameter circle.



3.2. Concentricity: Concentricity controls the extent to which the axis of one cylinder is collinear with the axis on an adjoining cylinder. It is specified as shown in Figure 12.



3.3. Symmetry: A part or a feature is symmetric when it has the same contour and size on opposite sides of a control plane. In effect, a symmetry tolerance locates or positions features with respect to a datum plane. The method of specifying this tolerance by symbol illustrated in Figure 13.



Center of median plane of hole, regardless of hole size, must lie between two planes 0.4 apart equidistant from the median plane of datum, regardless of datum size.

4. APPLICATION