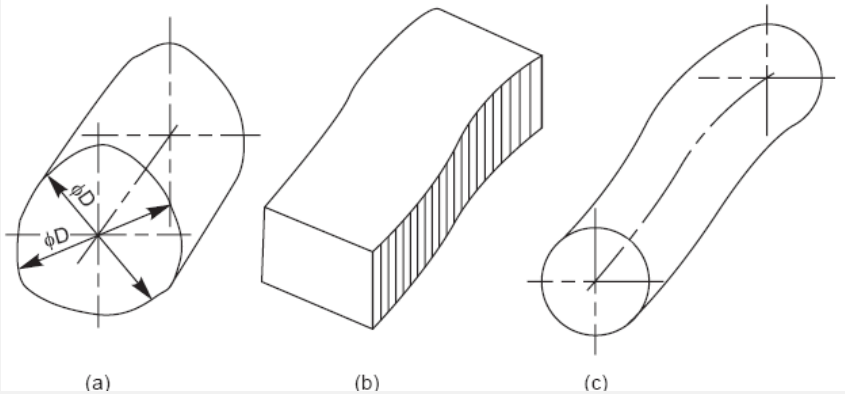
Global Engineering Solutions

Serving Globally

GEOMETRIC TOLERANCES

INTRODUCTION OF GEOMETRIC TOLERANCE

- Dimensional accuracy can be controlled by specifying tolerance of size. But this will not be sufficient to control the form of the component
- In Figure (a) the shaft has the same diameter measurement in all possible positions but is not circular
- In Figure (b), the component has the same thickness throughout but is not flat
- In Figure (c), the component is circular in all cross-sections but is not straight



INTRODUCTION OF GEOMETRIC TOLERANCE

Geometric tolerance may be;

- ***FORM TOLERANCE:*** *This is a group of geometric tolerances that limit the amount of error in the shape of the feature.*
- ***LOCATION TOLERANCE:*** *This group of geometric tolerances that limit the location or placement of features.*
- ***ORIENTATION TOLERANCE:*** *This is a group of geometric tolerances that limit the direction or orientation of a feature in relation to other features.*
- ***PROFILE TOLERANCE:*** *This is a group of powerful geometric tolerances that control the size, location, orientation and form of a feature.*
- ***RUN-OUT TOLERANCE:*** *This is a group of geometric tolerances that simultaneously limit the form, location and orientation of cylindrical parts.*
- ***BONUS TOLERANCE:*** *This is an additional tolerance that applies to a feature as its size shifts from a stated material condition.*

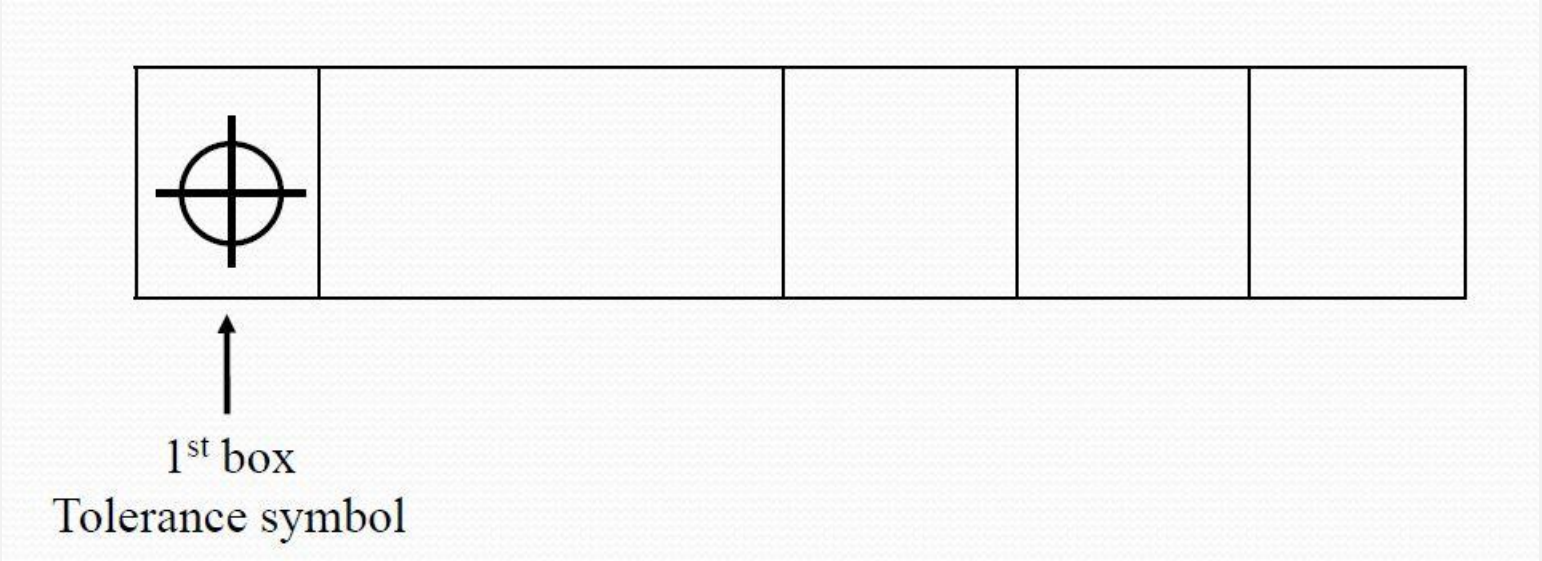
BASIC TERMINOLOGY

- **FEATURE:** *The physical feature of a machine part or component is that naturally contains variation and imperfections. Corners, edges, flat surfaces and holes are some examples of possible features.*
- **TOLERANCE ZONE:** *It is an imaginary zone in which a part feature must be completely contained for the part to pass inspection.*
- **TOLERANCE FRAME:** *A series of compartments containing symbols and values that describe the geometrical tolerance of a feature.*
- **DATUM:** *This is an imaginary, perfect geometric shape or form on the basis of which all dimensioning are placed. A perfect point, line, flat plane, circle and cylinder are examples of possible datum.*
- **MEDIAN POINT:** *This is a point that is exactly the same distance between two outer points.*

BASIC TERMINOLOGY

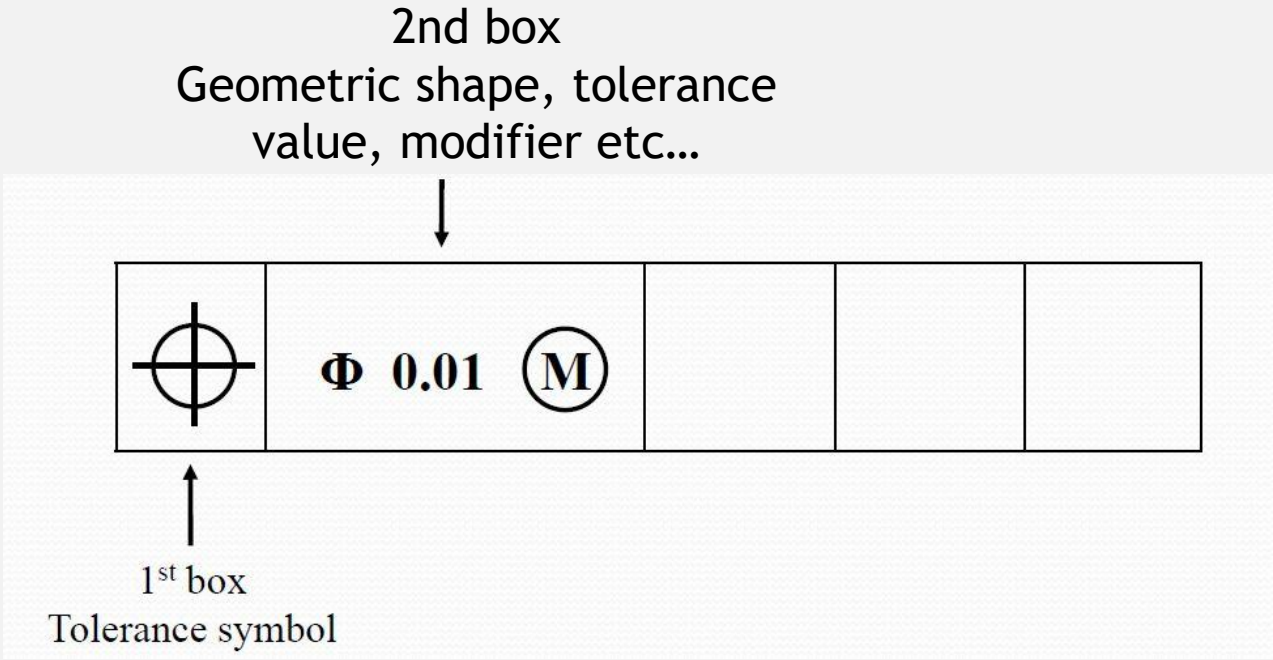
- ***SYMMETRY:*** *This is a 3D geometric tolerance that controls how much the medial point between two features may deviate from a specified axis or centre plane.*
- ***TRUE POSITION:*** *This is the imaginary, perfect position of a feature described by the design specifications.*
- ***TRUE PROFILE:*** *This is the perfect, imaginary profile described by the design specifications.*
- ***FIXTURE:*** *This is a dedicated work-holding device that is used to locate and hold a part during machining or inspection.*
- ***FUNCTIONAL GAUGE:*** *This is a gauge for a specific part that quickly checks its form and fit in a manner similar to its intended use.*
- ***GRANITE SURFACE PLATE:*** *This is a precise, flat plate made of granite that is used to establish a datum plane for inspection.*

TOLERANCE FRAME



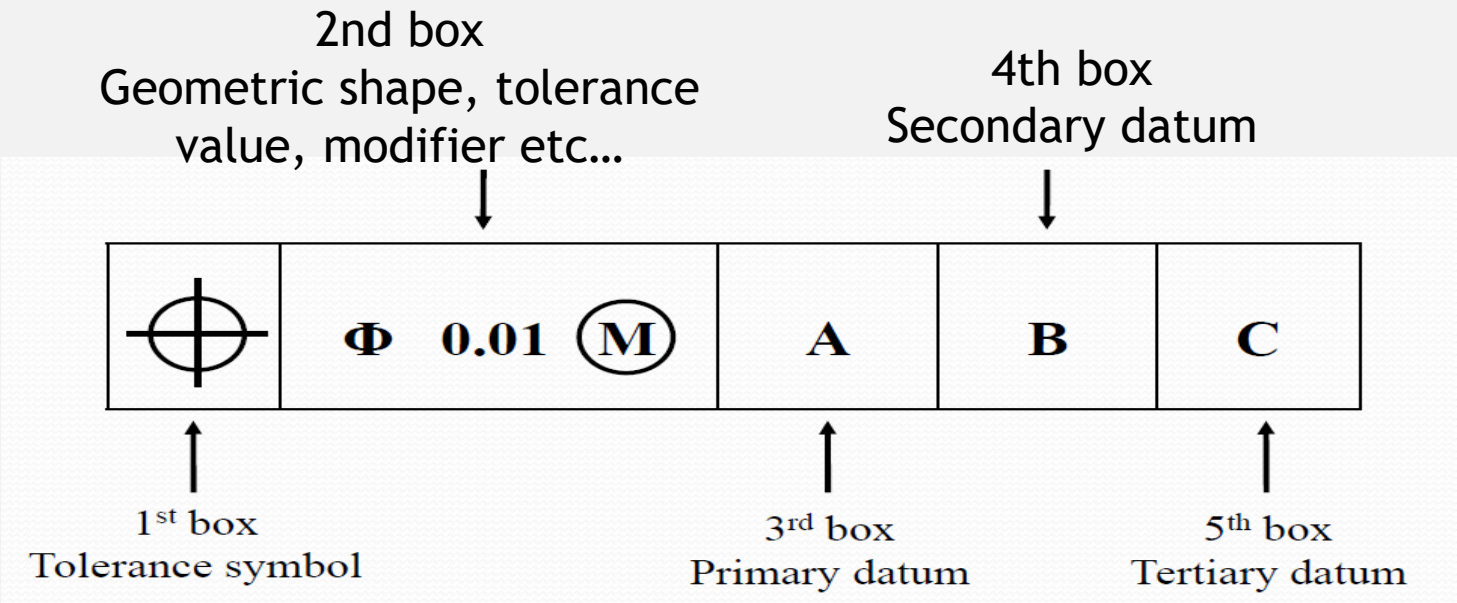
- It has several small boxes.
- The first box at extreme left side contains one of the fourteen geometric tolerance symbols such as flatness, parallelism, concentricity etc...

TOLERANCE FRAME



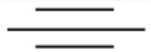
➢The next or second from left box gives the information about the nature of geometric shape, its tolerance value and modifiers that describe the tolerance conditions.

TOLERANCE FRAME



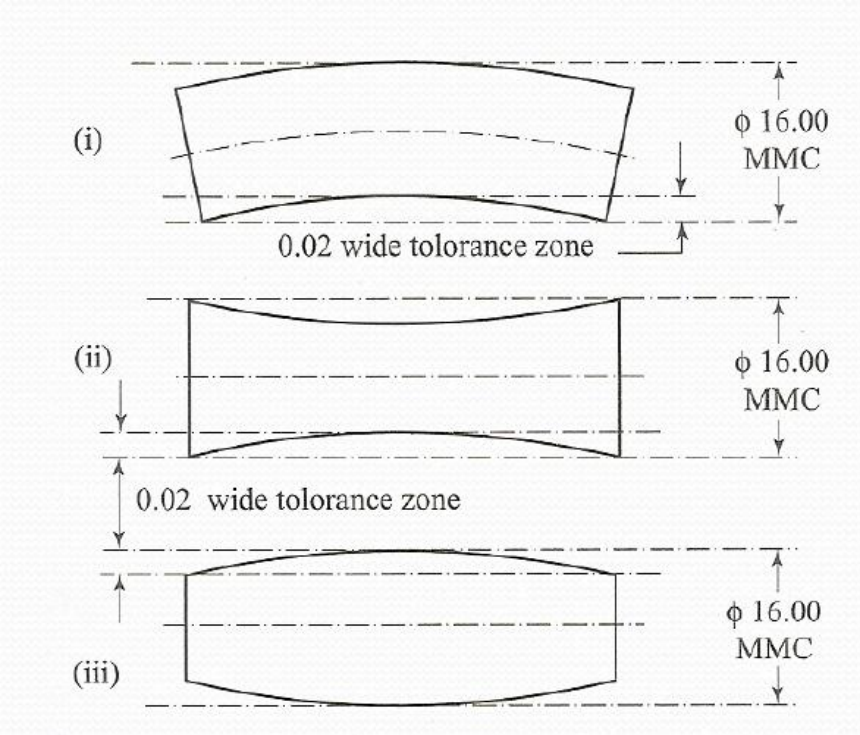
➢The third, fourth and fifth boxes provide information about datum. If it is necessary to specify more than one tolerance characteristic for a feature, the tolerance specifications are given in the tolerance frames one under the other.

TOLERANCE CHARACTERISTICS AND SYMBOLS

CATEGORY	CHARACTERISTIC	SYMBOL	USE OF DATUM
FORM	STRAIGHTNESS		NEVER
	FLATNESS		
	CIRCULARITY (ROUNDNESS)		
	CYLINDRICITY		
PROFILE	PROFILE OF A LINE		SOMETIMES
	PROFILE OF A SURFACE		
ORIENTATION	ANGULARITY		ALWAYS
	PERPENDICULARITY		
	PARALLELISM		
LOCATION	POSITION		
	CONCENTRICITY		
	SYMMETRY		
RUNOUT	CIRCULAR RUNOUT		
	TOTAL RUNOUT		

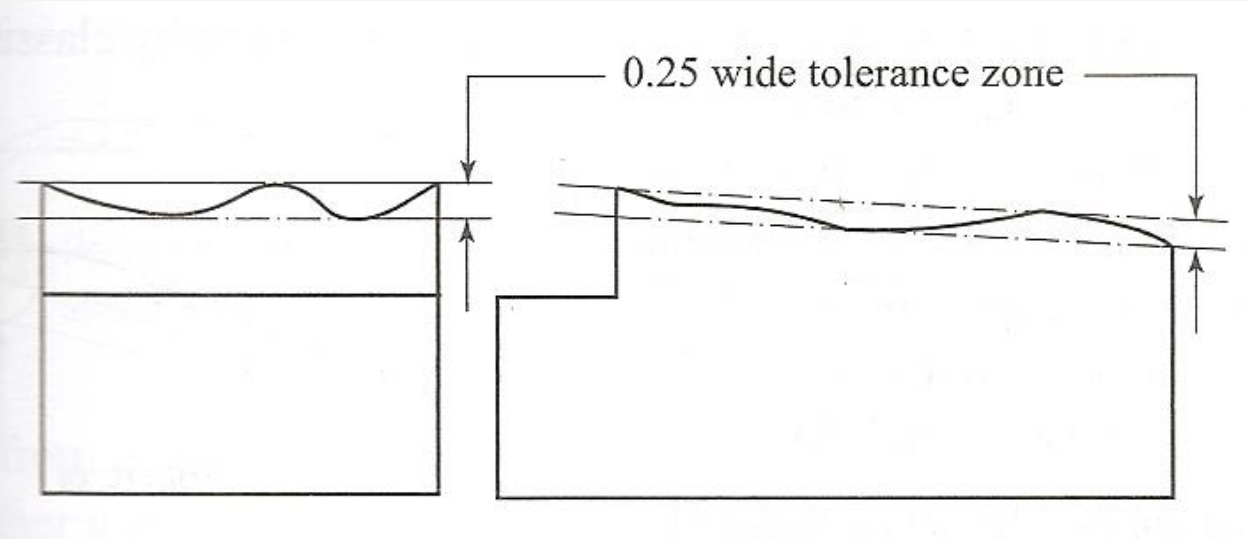
TOLERANCE CHARACTERISTICS AND SYMBOLS

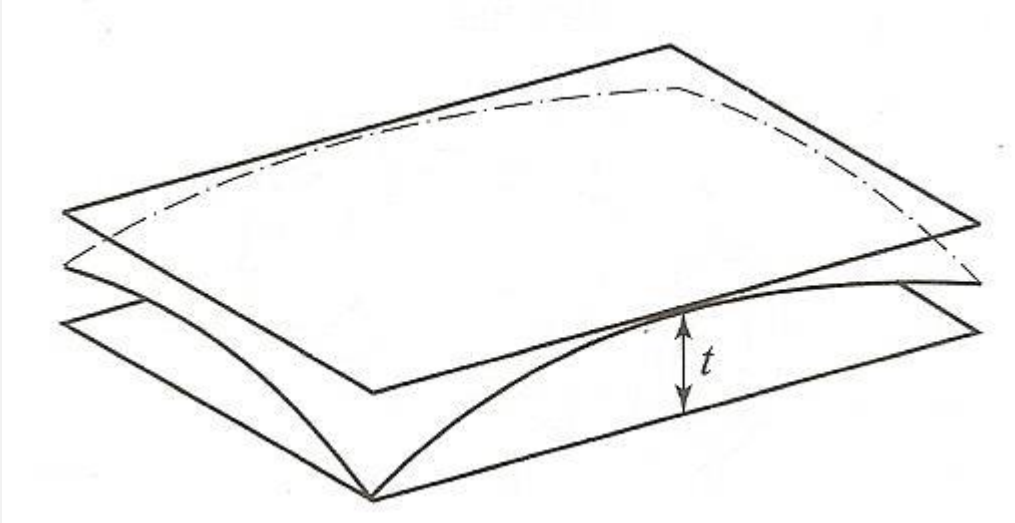
STRAIGHTNESS: *Straightness is a condition where an element of a surface or an axis is a straight line. The straightness tolerance specifies a tolerance zone within which the considered element or derived median line must lie.*



TOLERANCE CHARACTERISTICS AND SYMBOLS

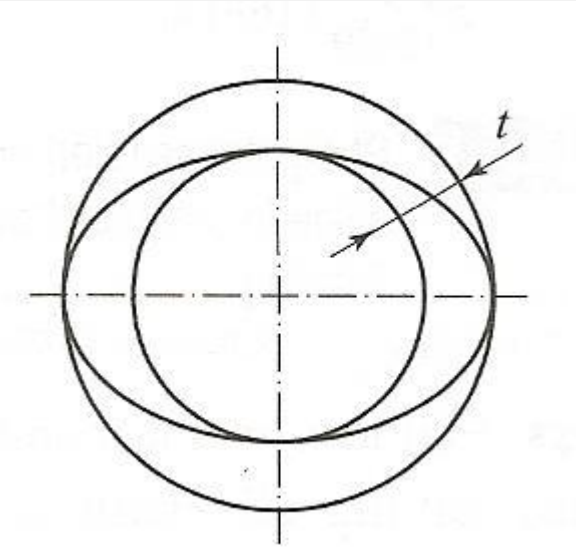
FLATNESS: Flatness is the condition of a surface having all elements in one plane. The flatness tolerance specifies a tolerance zone defined by two parallel planes at a distance t apart within which the surface must lie.

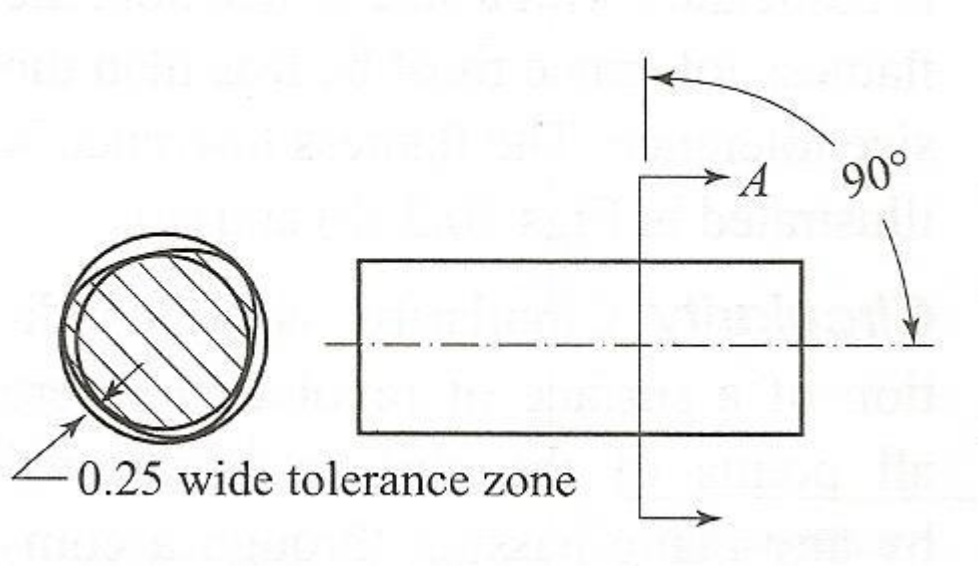




TOLERANCE CHARACTERISTICS AND SYMBOLS

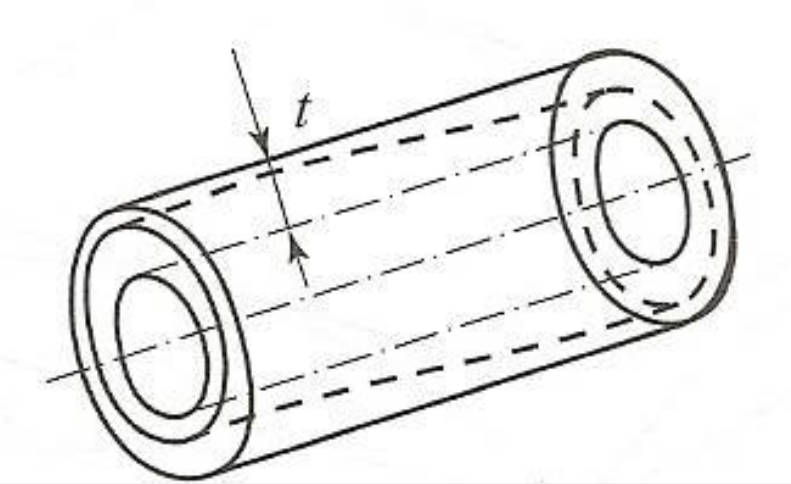
CIRCULARITY: *Circularity is a condition of a surface of revolution where all points of the surface intersected by any plane passing through a common centre are equidistant from that centre. The circularity tolerance specifies a tolerance zone bounded by two concentric circles at a distance 't' apart within which each circular element of the surface must lie.*

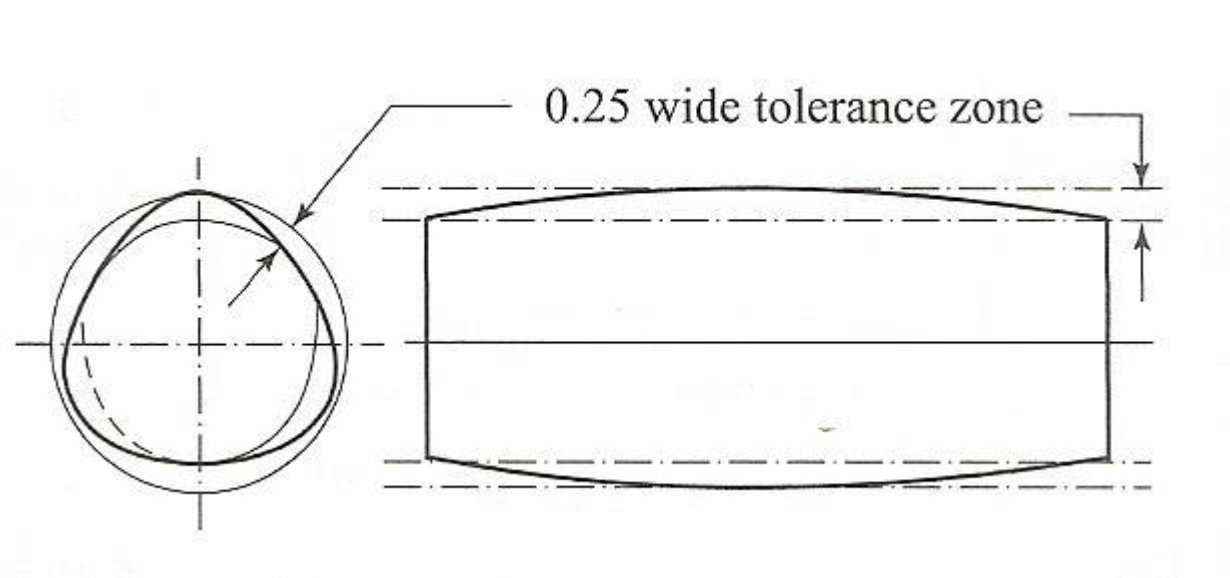




TOLERANCE CHARACTERISTICS AND SYMBOLS

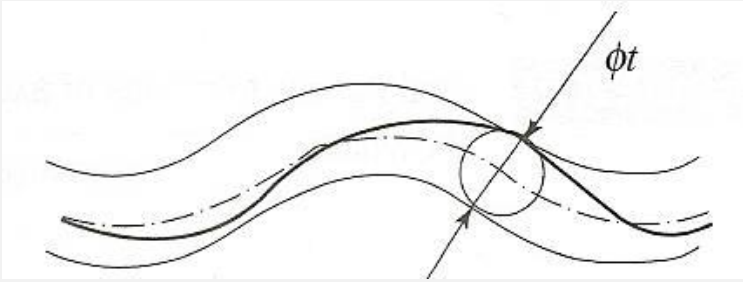
CYLINDRICITY: *Cylindricity is a 3D geometric tolerance that controls how much a feature can deviate from a perfect cylinder. Cylindricity tolerance specifies a tolerance zone bounded by two concentric cylinders at a distance 't' apart within which the surface must lie.*



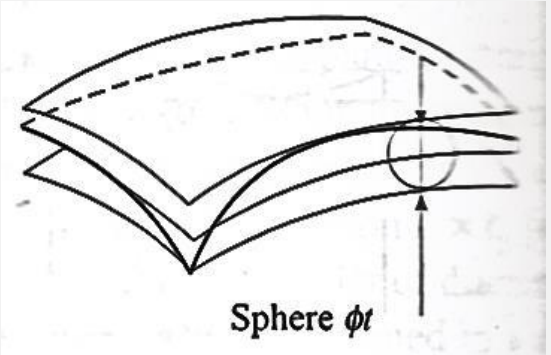


TOLERANCE CHARACTERISTICS AND SYMBOLS

PROFILE OF LINE: *The profile tolerance of a line is a 2D geometric tolerance that controls how much the outline of a feature can deviate from the true profile.*

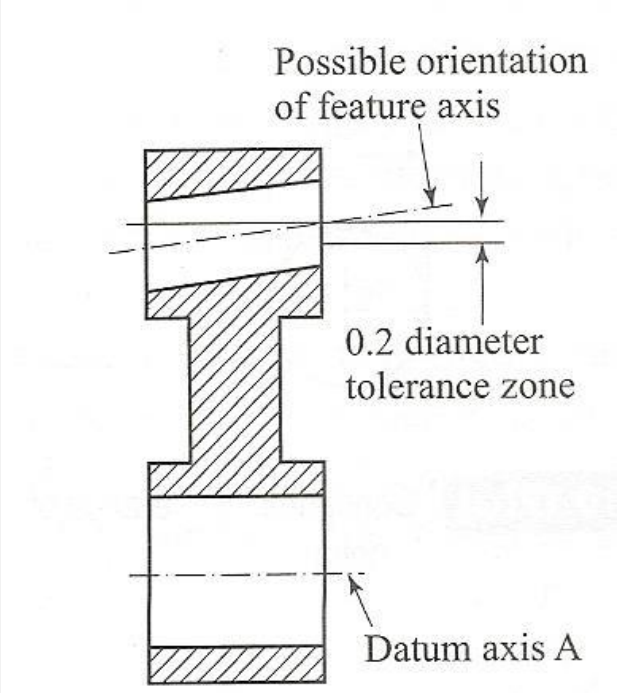


PROFILE OF SURFACE: *The profile tolerance of a line is a 3D geometric tolerance that controls how much the surface can deviate from the true profile.*



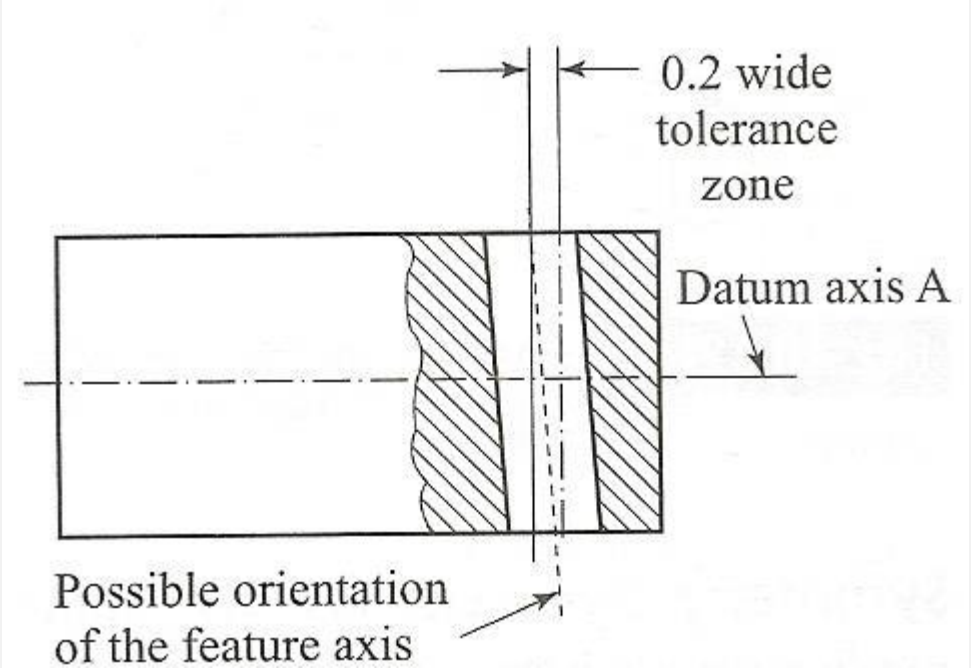
TOLERANCE CHARACTERISTICS AND SYMBOLS

PARALLELISM: *Parallelism is the condition of a surface, line or axis that is equidistant at all points from a datum plane or axis. The parallelism tolerance is a 3D geometric tolerance that controls how much a surface, axis or plane can deviate from an orientation parallel to the specified datum.*



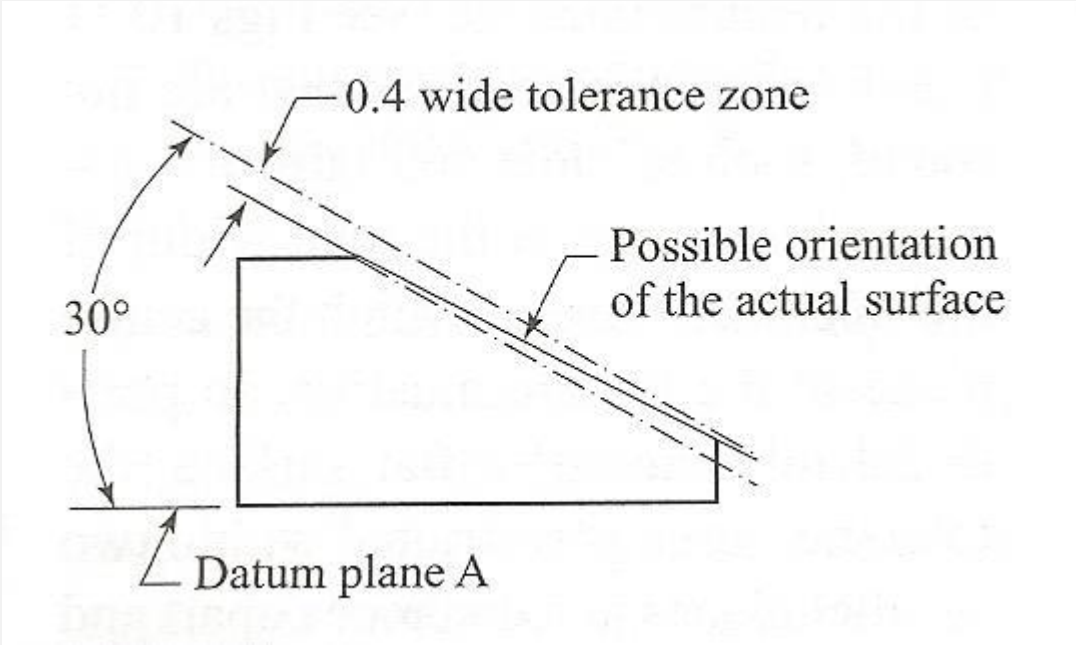
TOLERANCE CHARACTERISTICS AND SYMBOLS

PERPENDICULARITY: *Perpendicularity is a condition of a surface, axis or line that is 900 from a datum plane or a datum axis. The perpendicularity tolerance specifies either a zone define by two planes perpendicular to a datum plane or axis or a zone define by two parallel planes perpendicular to the datum axis.*



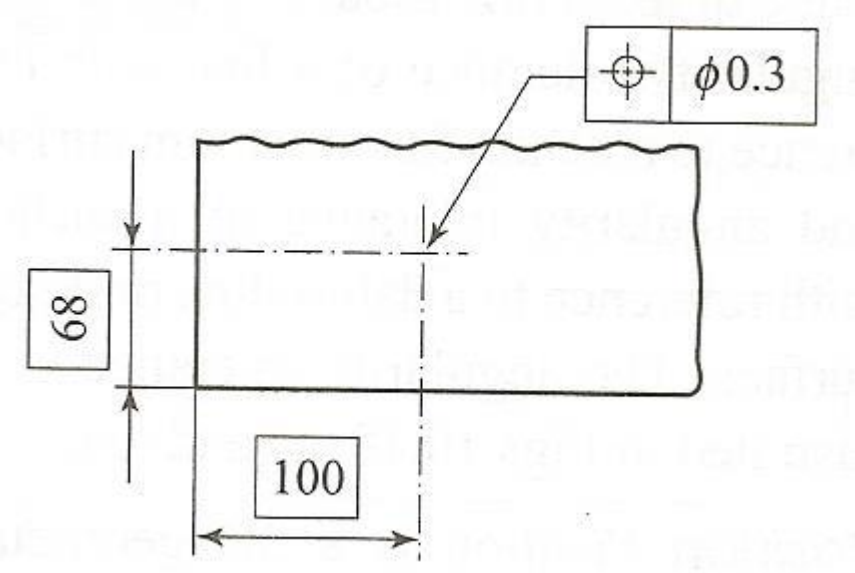
TOLERANCE CHARACTERISTICS AND SYMBOLS

ANGULARITY: *Angularity is the condition of a surface or axis at a specified angle other than 900 from a datum plane or axis. The angularity tolerance controls how much a surface, axis or plane can deviate from the angle described in the design specifications and is define by two parallel planes at the specified basis angle from a datum plane or axis.*



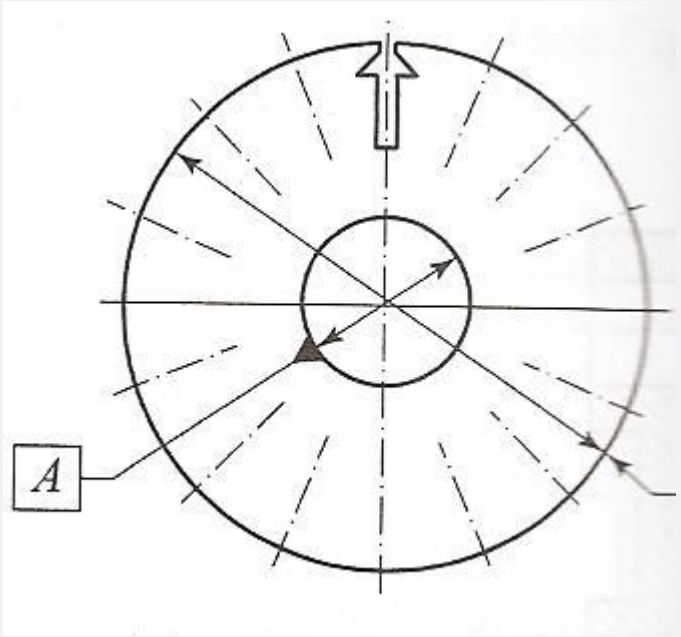
TOLERANCE CHARACTERISTICS AND SYMBOLS

POSITION: *Position is a 3D geometric tolerance controlling deviation of a feature from its true position. The positional tolerance defines a zone in which the centre axis or centre plane is permitted to vary from theoretically exact position.*



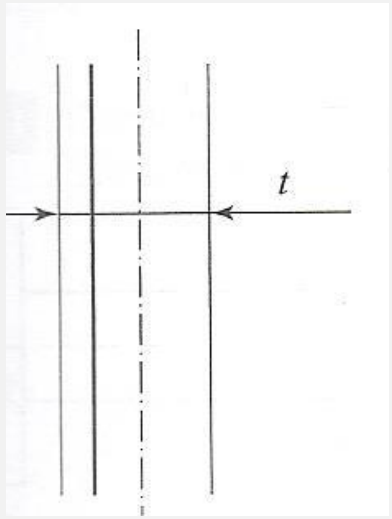
TOLERANCE CHARACTERISTICS AND SYMBOLS

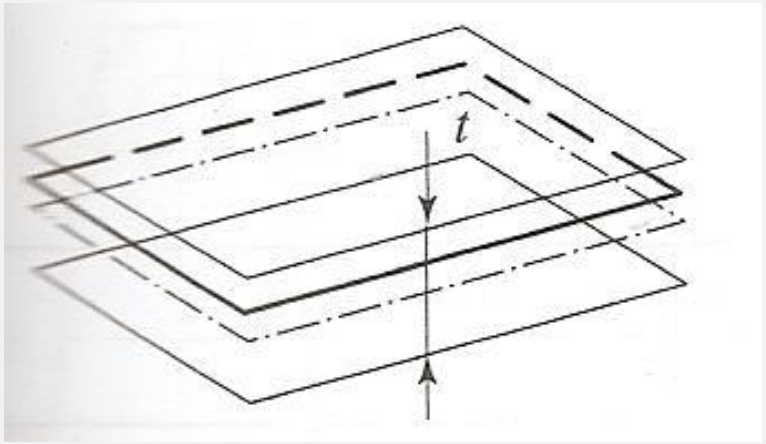
CONCENTRICITY: *Concentricity is a 3D geometric tolerance that controls how much the median points of multiple diameters may deviate from the specified datum point or datum axis.*

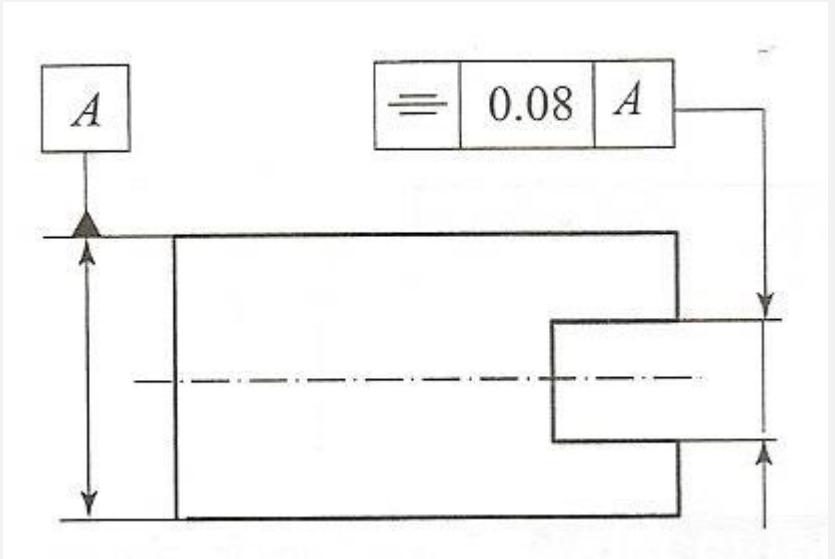


TOLERANCE CHARACTERISTICS AND SYMBOLS

SYMMETRY: *Symmetry is a 3D geometric tolerance that controls how much the median points between two features may deviate from a specified axis or centre plane. For symmetry tolerance of a median plane, the tolerance zone is limited by two parallel plane at a distance t apart and disposed symmetrically to the median plane with respect to the datum axis or datum plane.*

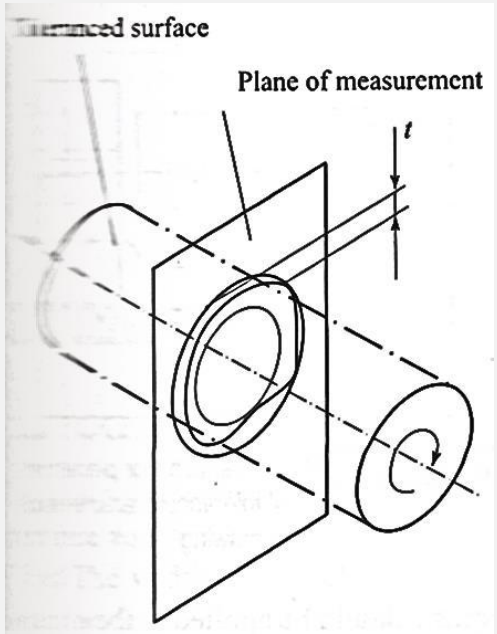


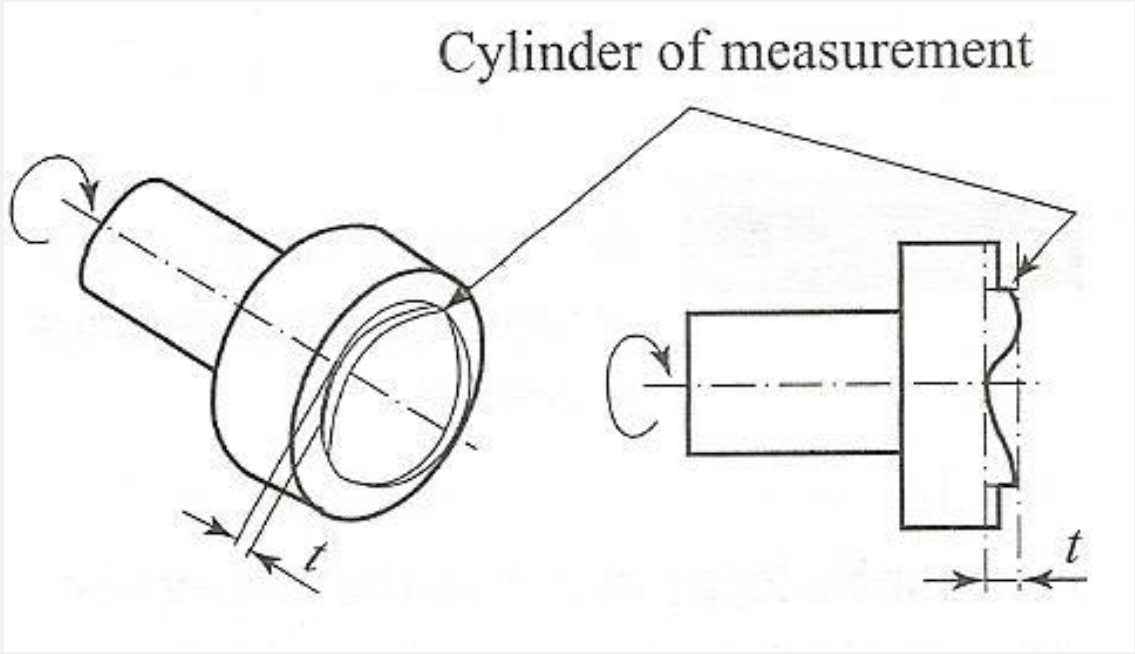




TOLERANCE CHARACTERISTICS AND SYMBOLS

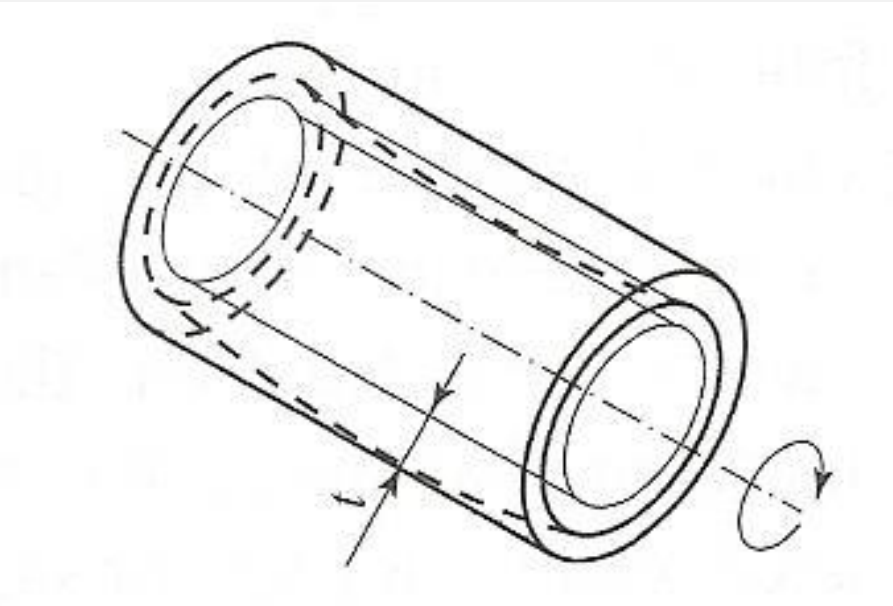
CIRCULAR RUN-OUT: *Circular run-out provides control of circular elements of a surface. The circular run-out tolerance is a 2D geometric tolerance that controls the form, orientation and location of multiple cross sections of a cylindrical part as it rotates.*





TOLERANCE CHARACTERISTICS AND SYMBOLS

TOTAL RUN-OUT: *Total run-out controls the cumulative variation of circularity, Cylindricity, straightness, angularity, concentricity, taper and profile when it is applied to surfaces constructed around a datum axis. It provides composite control of all surface elements.*



MODIFIERS

TERM	SYMBOL
AT MAXIMUM MATERIAL CONDITION	Ⓜ
AT LEAST MATERIAL CONDITION	Ⓛ
PROJECTED TOLERANCE ZONE	Ⓟ
FREE STATE	Ⓕ
TANGENT PLANE	Ⓣ
DIAMETER	Φ
SPHERODAL DIAMETER	SΦ
RADIUS	R
SPHERIAL RADIUS	SR
CONTROLLED RADIUS	CR
REFERENCE	( )
ARC LENGTH	⌒

MODIFIERS

These are parameters that communicate additional information about the drawing or tolerance of a part.

MAXIMUM METAL CONDITION (M): *MMC is related to the retention of maximum amount of metal in any machine part. When a geometric tolerance is applied on MMC basis, allowed tolerance depends on the actual mating size of the tolerance feature.*

LEAST METAL CONDITION (L): *LMC is related to the retention of minimum amount of metal in any machine part. When a geometric tolerance is applied on LMC basis, allowed tolerance depends on the actual mating size of the tolerance feature.*

MODIFIERS

PROJECTED TOLERANCE ZONE (P): When a tolerance zone is required to be projected above the part surface, it is called the projected tolerance zone. It is generally used to limit the perpendicularity of a hole to ensure proper assembly with the mating part.

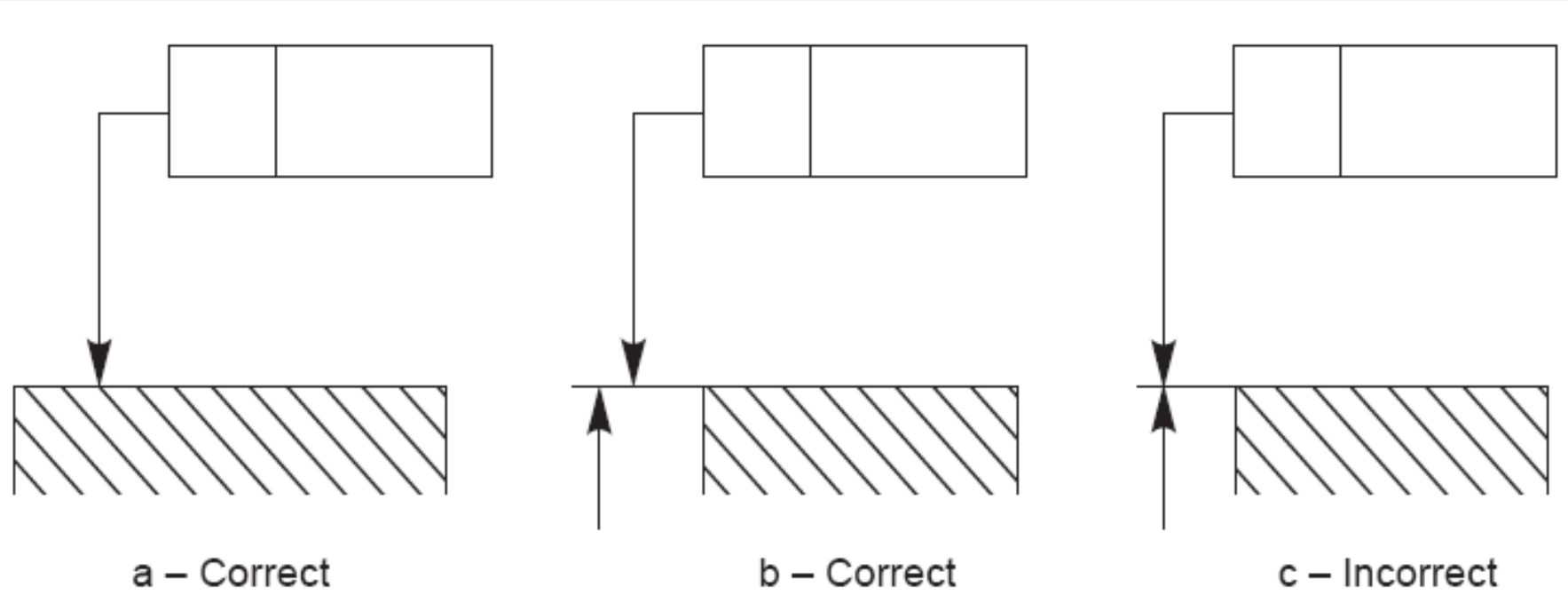
FREE STATE (F): Free state is a term that is used to describe distortion of apart after removal of applied forces during the process of manufacturing.

TANGENT PLANE (T): Tangent plane is a theoretically exact plane derived from the true geometric counterpart of the specified feature surface.

TOLERANCE CHARACTERISTICS IN DRAWING

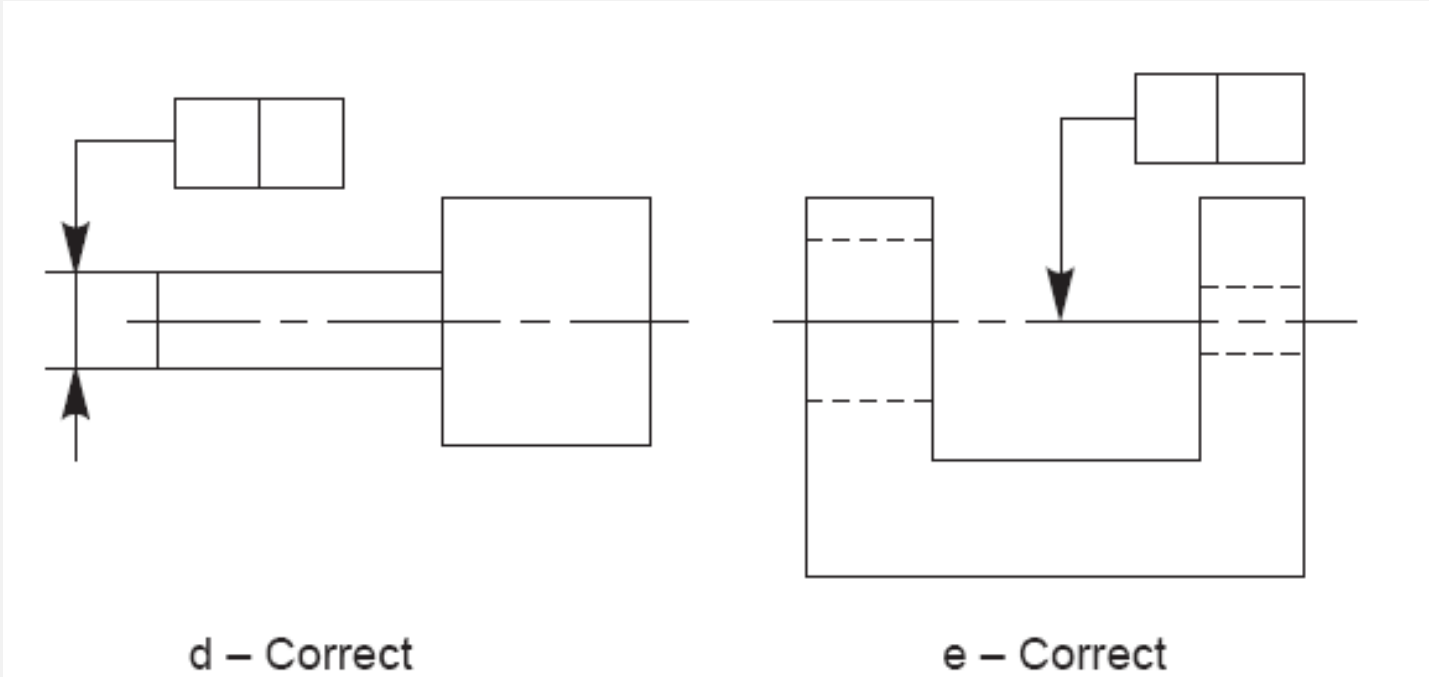
The tolerance frame is connected to the tolerance feature by a leader line terminating with an arrow in the following ways:

➢On the outline of the feature or extension of the outline, but not a dimension line, when the tolerance refers to the line or surface itself

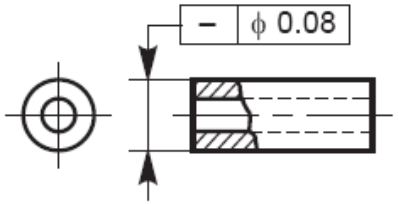
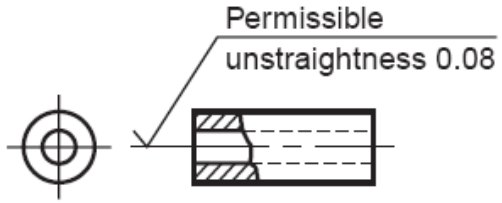
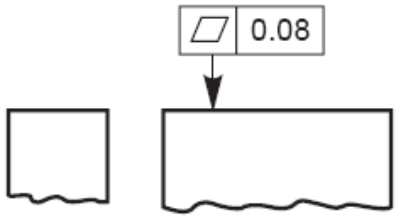
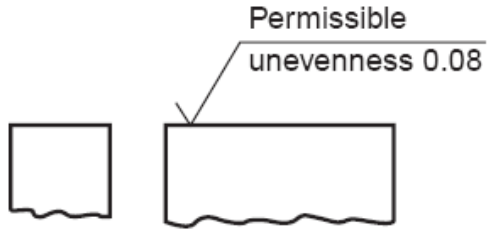
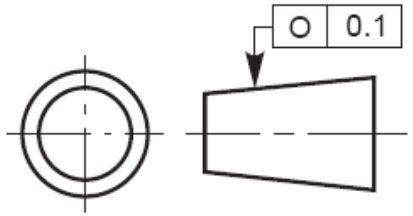
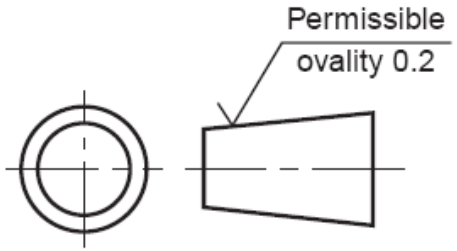


TOLERANCE CHARACTERISTICS IN DRAWING

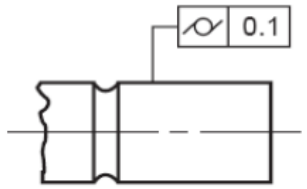
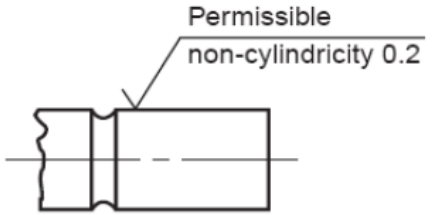
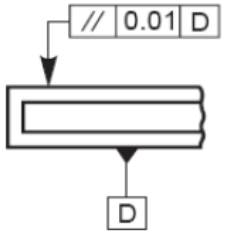
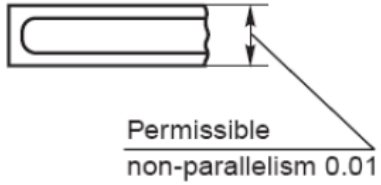
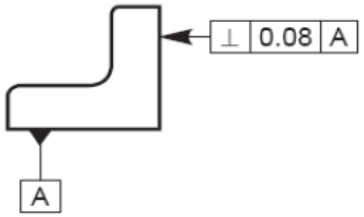
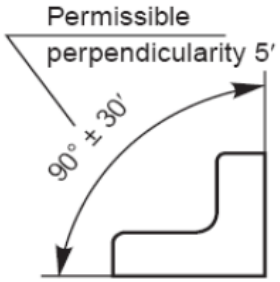
➢On the projection line, at the dimension line, when the tolerance refers to the axis or median plane of the part so dimensioned or on the axis, when the tolerance refers to the axis or median plane of all features common to that axis or median plane



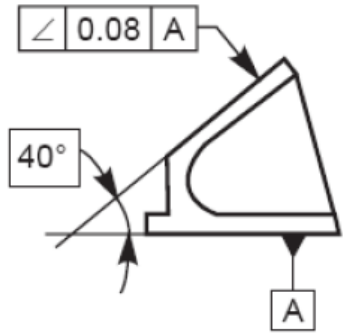
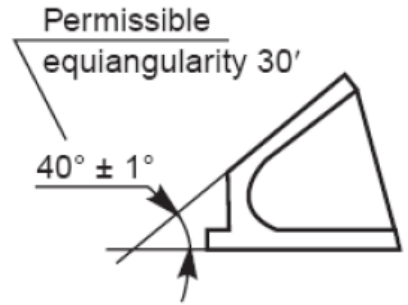
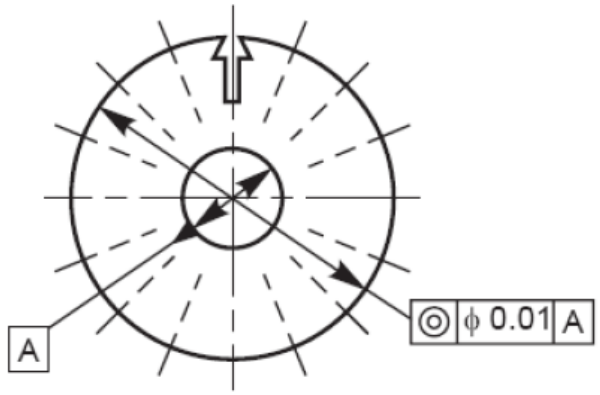
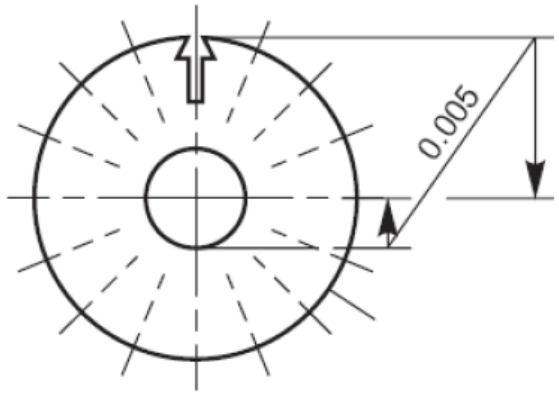
INDICATION OF GEOMETRIC TOLERANCE IN DRAWING

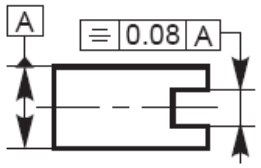
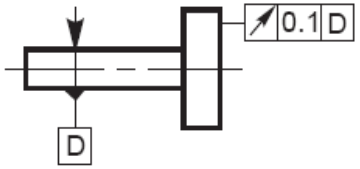
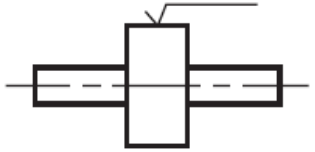
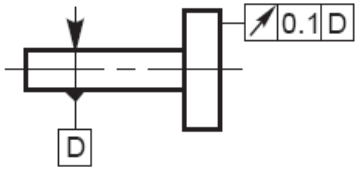
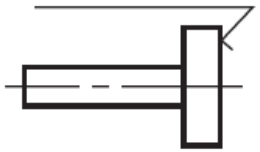
As per the standard	*As prevalent in industry*
1. Straightness tolerance	
	
2. Flatness tolerance	
	
3. Circularity tolerance	
	

INDICATION OF GEOMETRIC TOLERANCE IN DRAWING

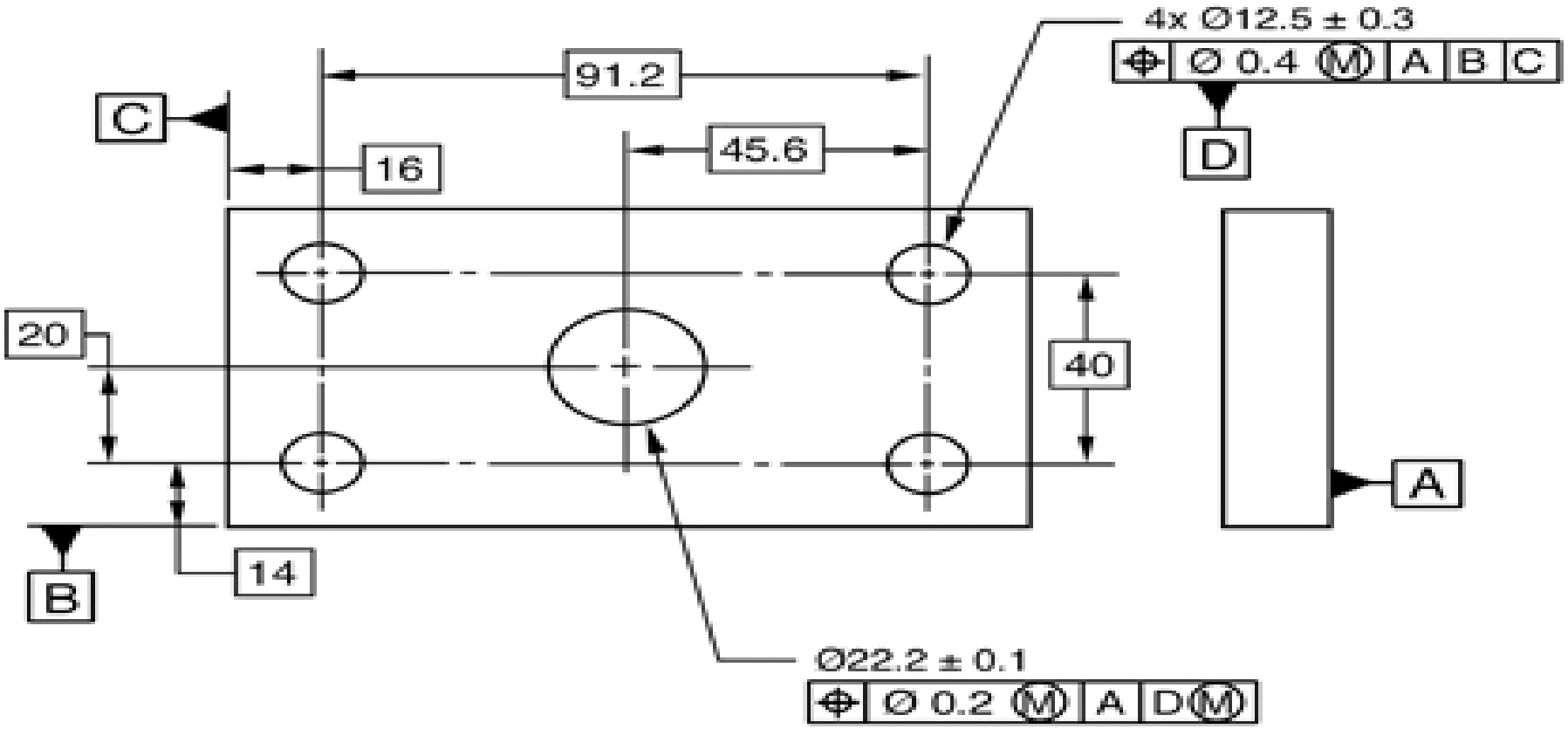
As per the standard	*As prevalent in industry*
4. Cylindricity tolerance	
	
5. Parallelism tolerance	
	
6. Perpendicularity tolerance	
	

INDICATION OF GEOMETRIC TOLERANCE IN DRAWING

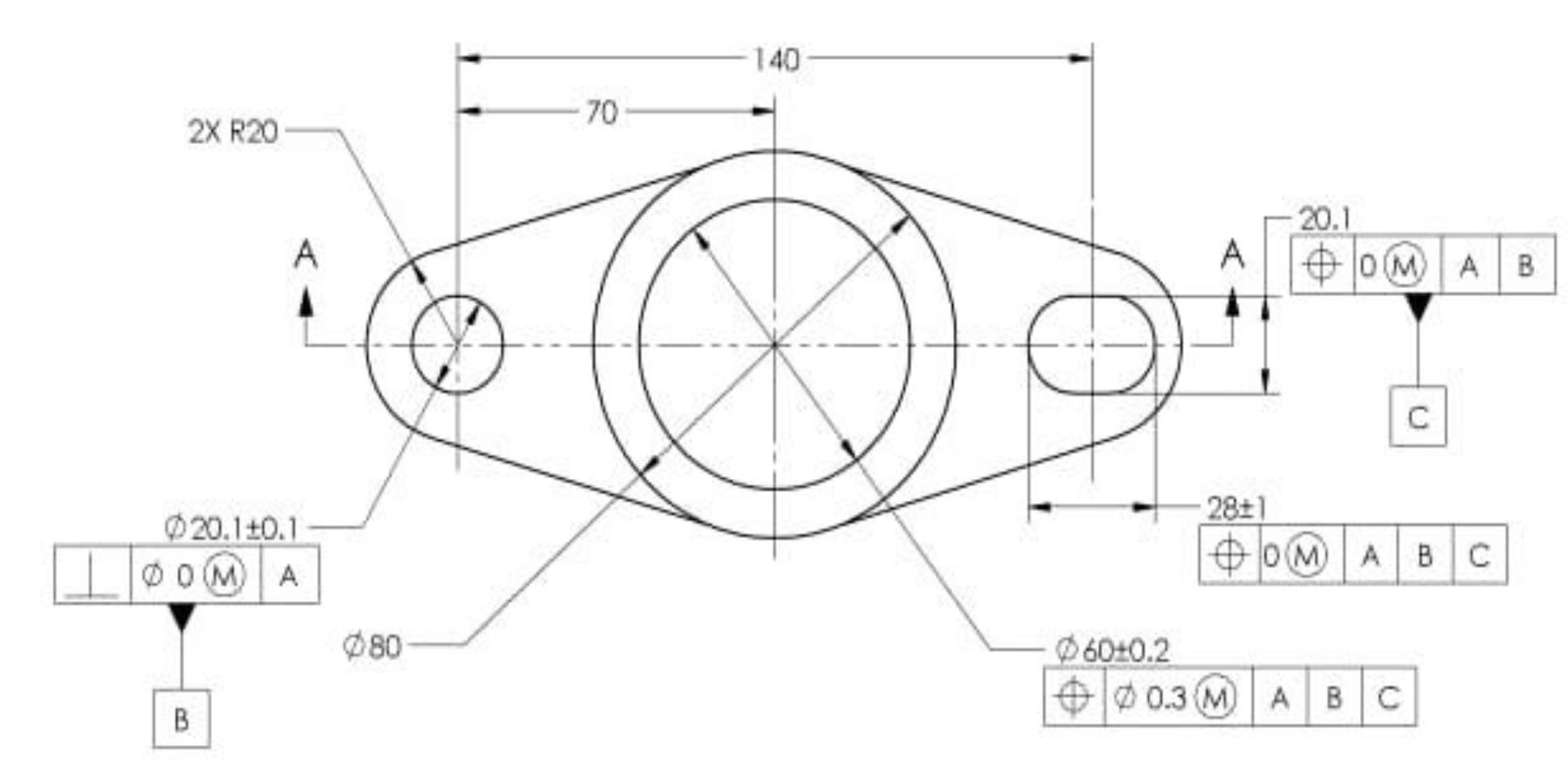
As per the standard	*As prevalent in industry*
7. Angularity tolerance	
	
8. Concentricity and coaxiality tolerance	
	

As per the standard	*As prevalent in industry*
9. Symmetry tolerance	
	
10. Radial run-out	
	Permissible cross indicator runout (Between centres) 0.1 
11. Axial run-out	
	Permissible longitudinal indicator runout (Between centres) 0.1 

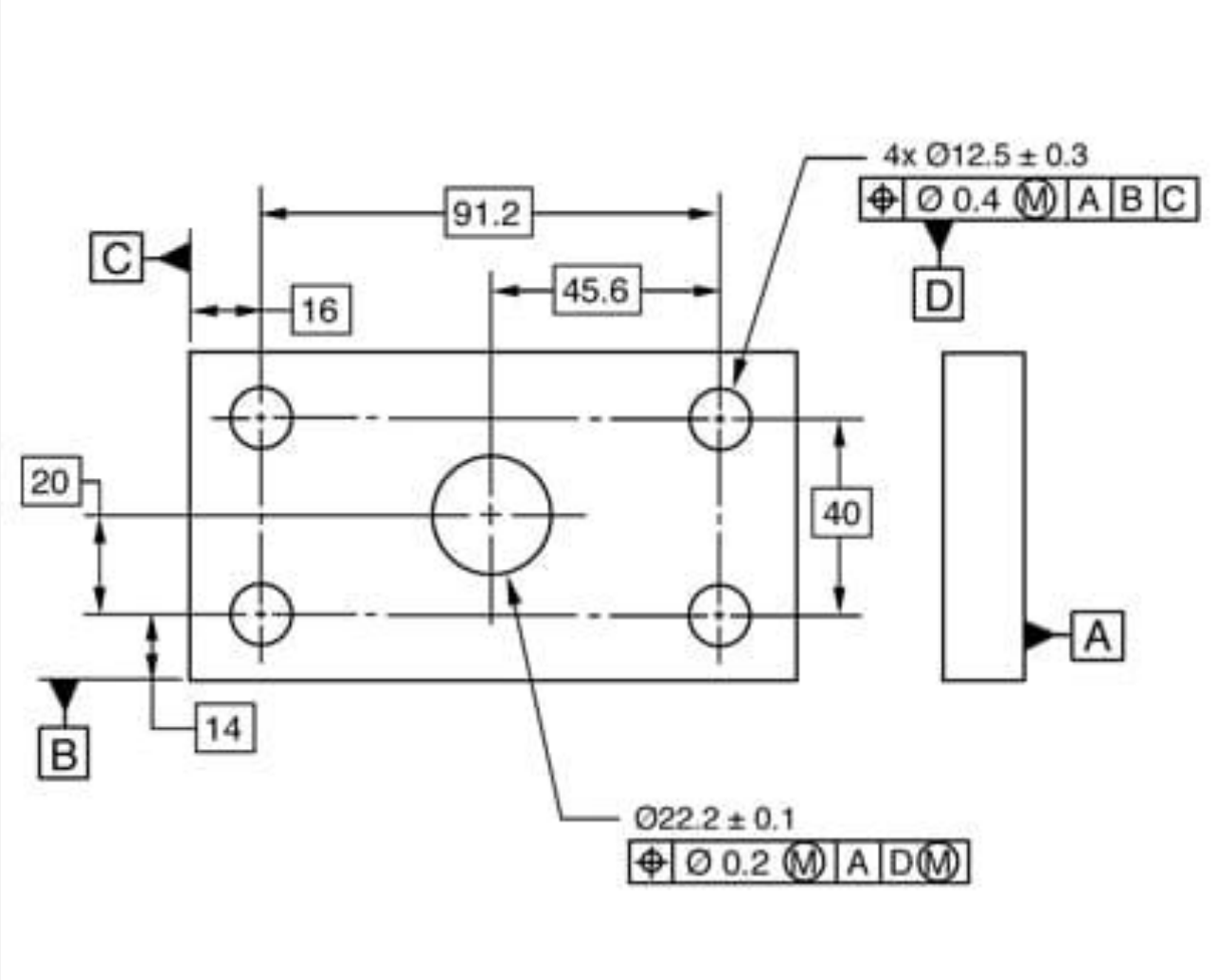
GD&T Practice Examples



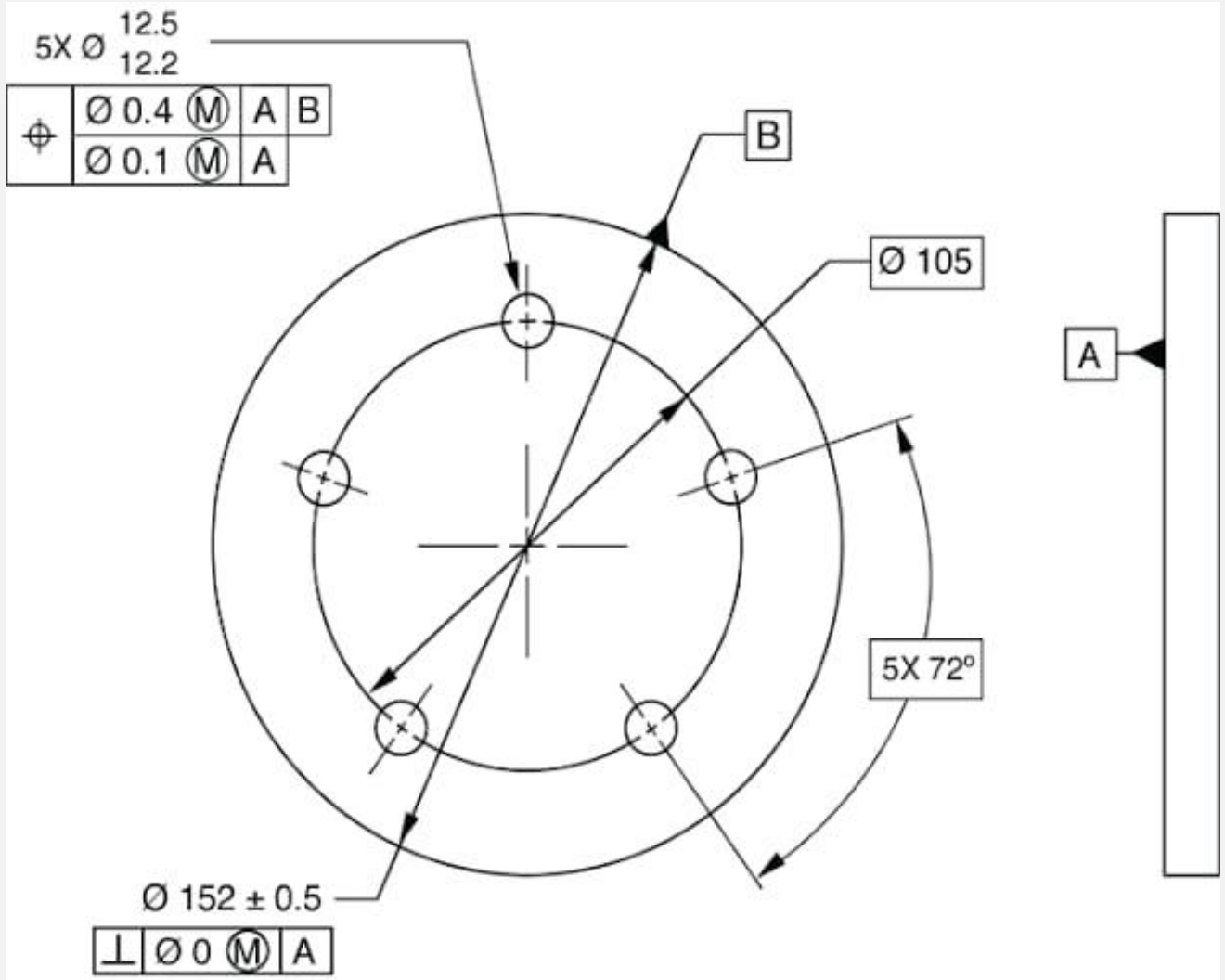
GD&T Practice Examples



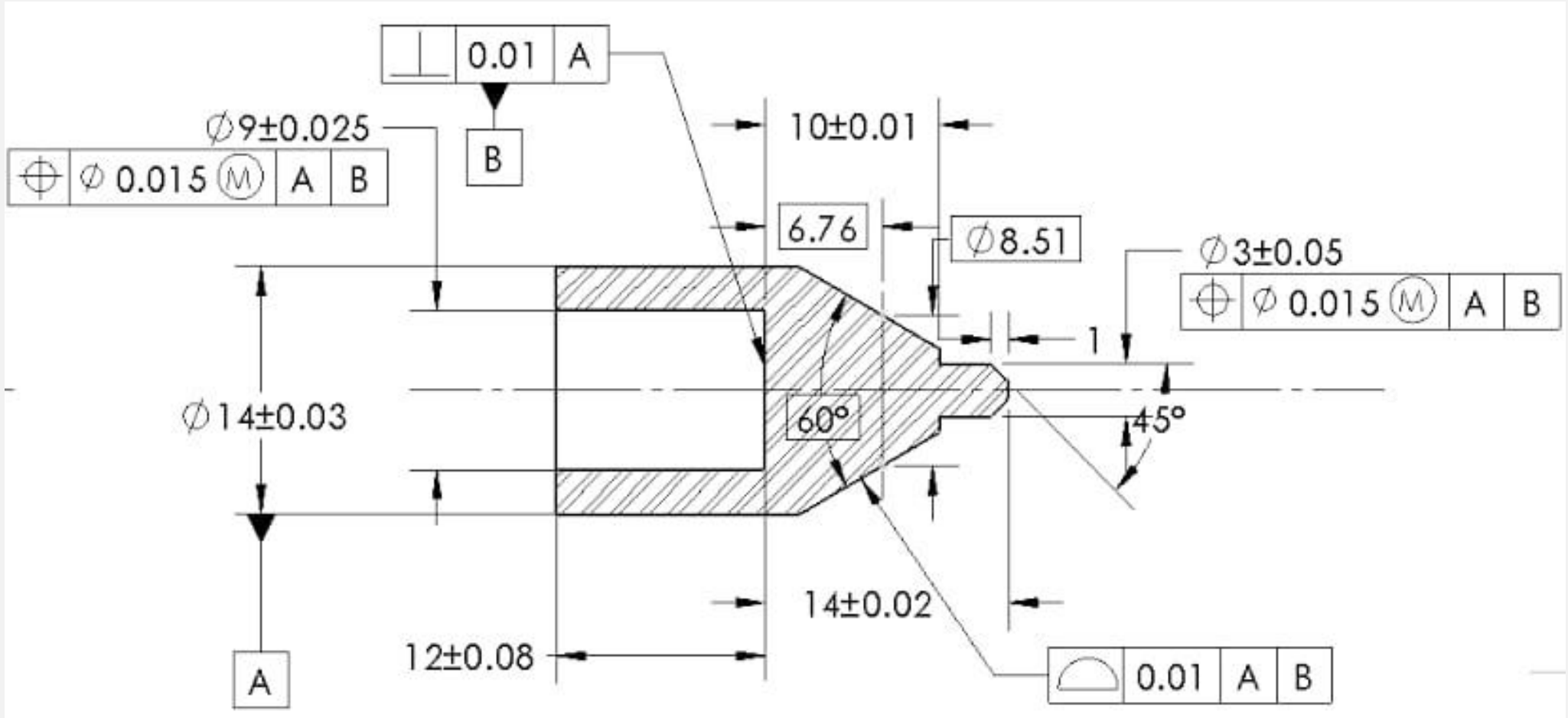
GD&T Practice Examples



GD&T Practice Examples



GD&T Practice Examples



Thank you for your attention

Global Engineering Solutions

Serving Globally