

# Geometric Dimensioning and Tolerancing

1



Prof. S. S. Pande

Mechanical Engineering Department  
Indian Institute of Technology, Bombay

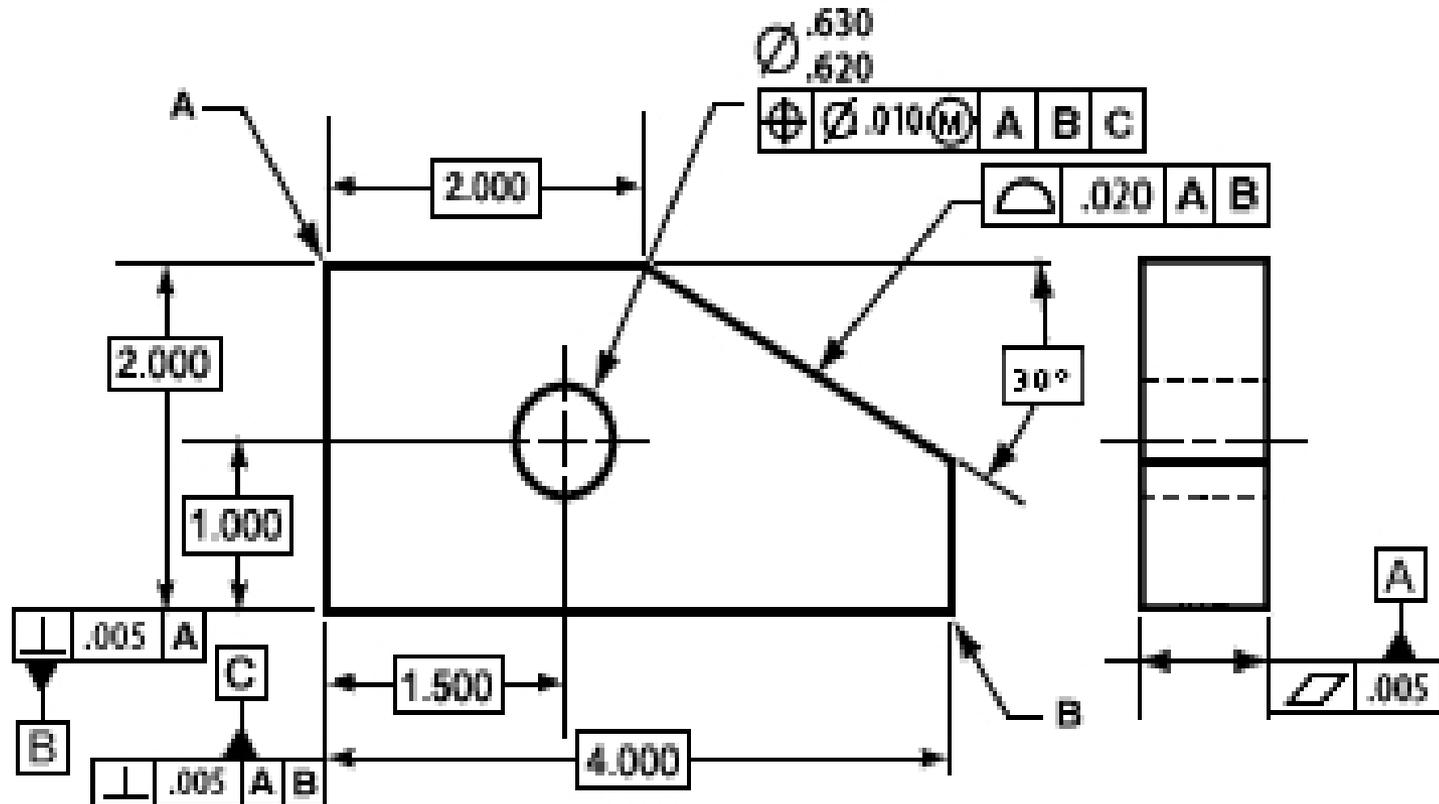
# Outline

- Importance of Geometric Dimensioning & Tolerancing
- Tolerance Symbols
- Tolerance Specification and Interpretation
  - *Form, Orientation, Location, Position, Runout*

# What is GD & T ?

- International standard to specify *Form*, *Fit* and *Function* of parts
  - ANSI Y 14.5, ISO 1101
- Significant improvement over traditional tolerancing
- Focuses on 3D part geometry features in addition to 2D drawings

# A Typical Part Drawing



# When to use GD & T ?

- Part features are critical to function or interchangeability
- Datum references are desirable
- Functional gaging techniques are preferred
- Standard interpretation of tolerance is not implied

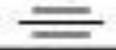
# Why to use GD & T ?

- Ensures Specification of Design Tolerance requirements
- Ensures *interchangeability* of parts during assembly
- Provides standardization and uniformity
  - Adaptation to 3D CAD modeling
- Leads to high productivity, less rework and reduced cost.

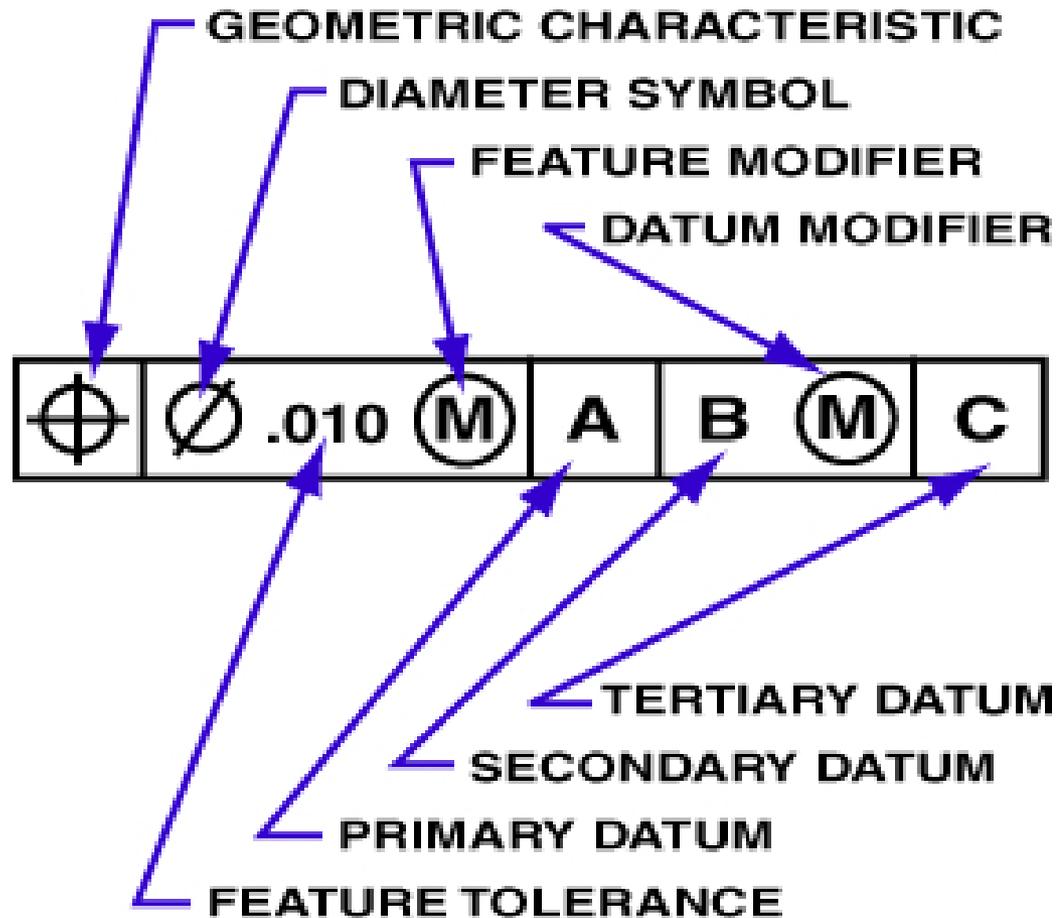
# Part Geometry Features to be controlled

- Form
- Orientation
- Location
- Position
- Runout

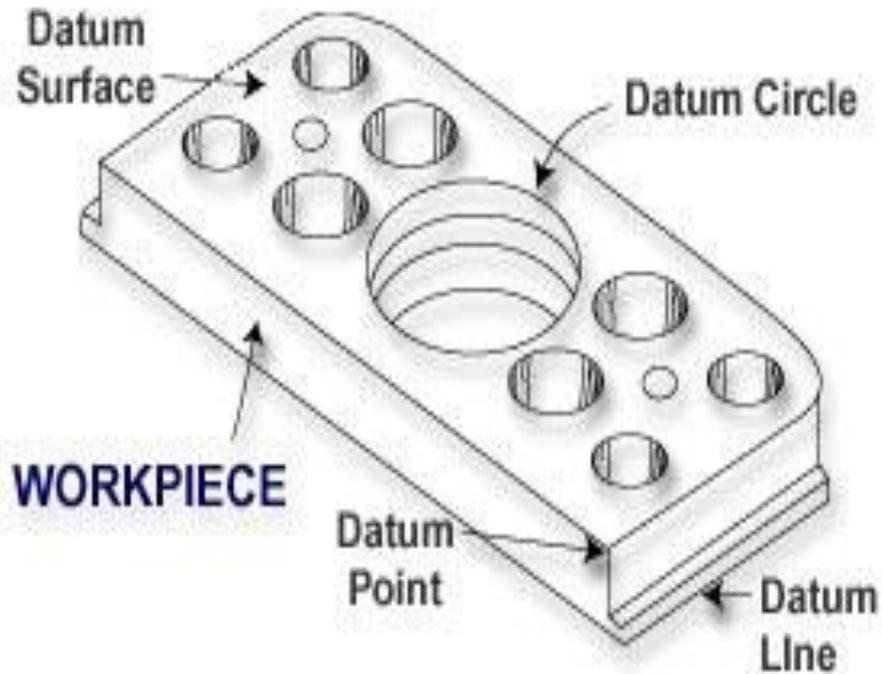
# Geometric Tolerance Symbols

TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	DATUM REFERENCE
FORM	STRAIGHTNESS		INDIVIDUAL
	FLATNESS		
	CIRCULARITY (ROUNDNESS)		
	CYLINDRICITY		
PROFILE	PROFILE OF A LINE		INDIVIDUAL OR RELATED
	PROFILE OF A SURFACE		
ORIENTATION	ANGULARITY		RELATED
	PERPENDICULARITY		
	PARALLELISM		
LOCATION	POSITION		
	CONCENTRICITY		
	SYMMETRY		
RUNOUT	CIRCULAR RUNOUT		
	TOTAL RUNOUT		

# Tolerance Feature Frame



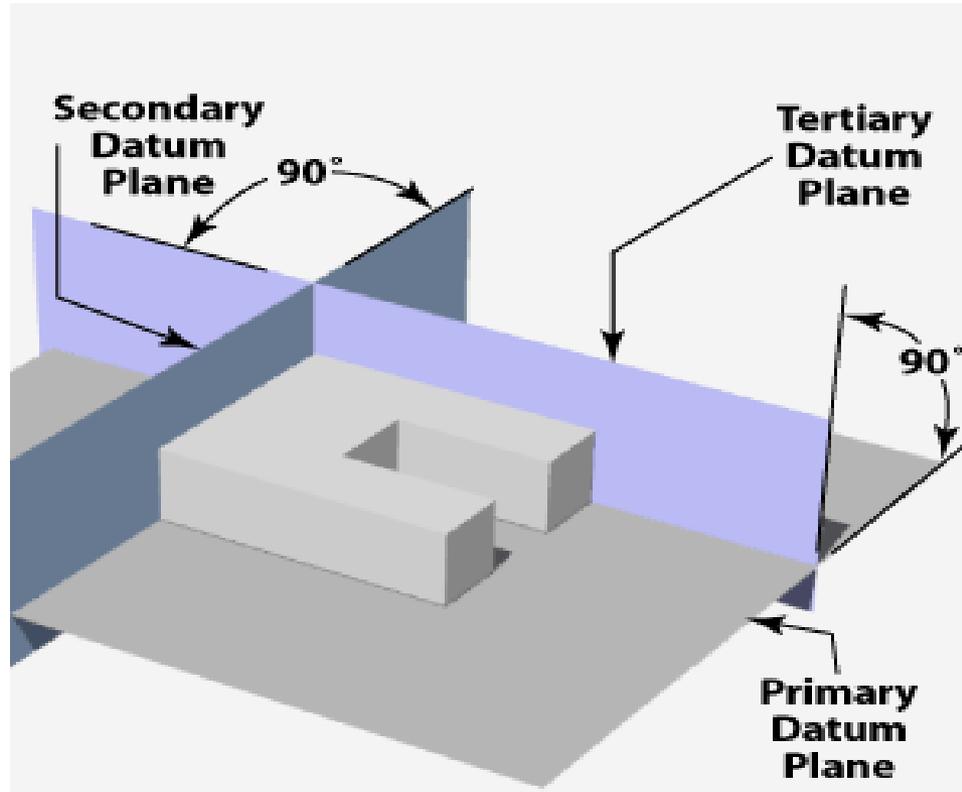
# Part Datums



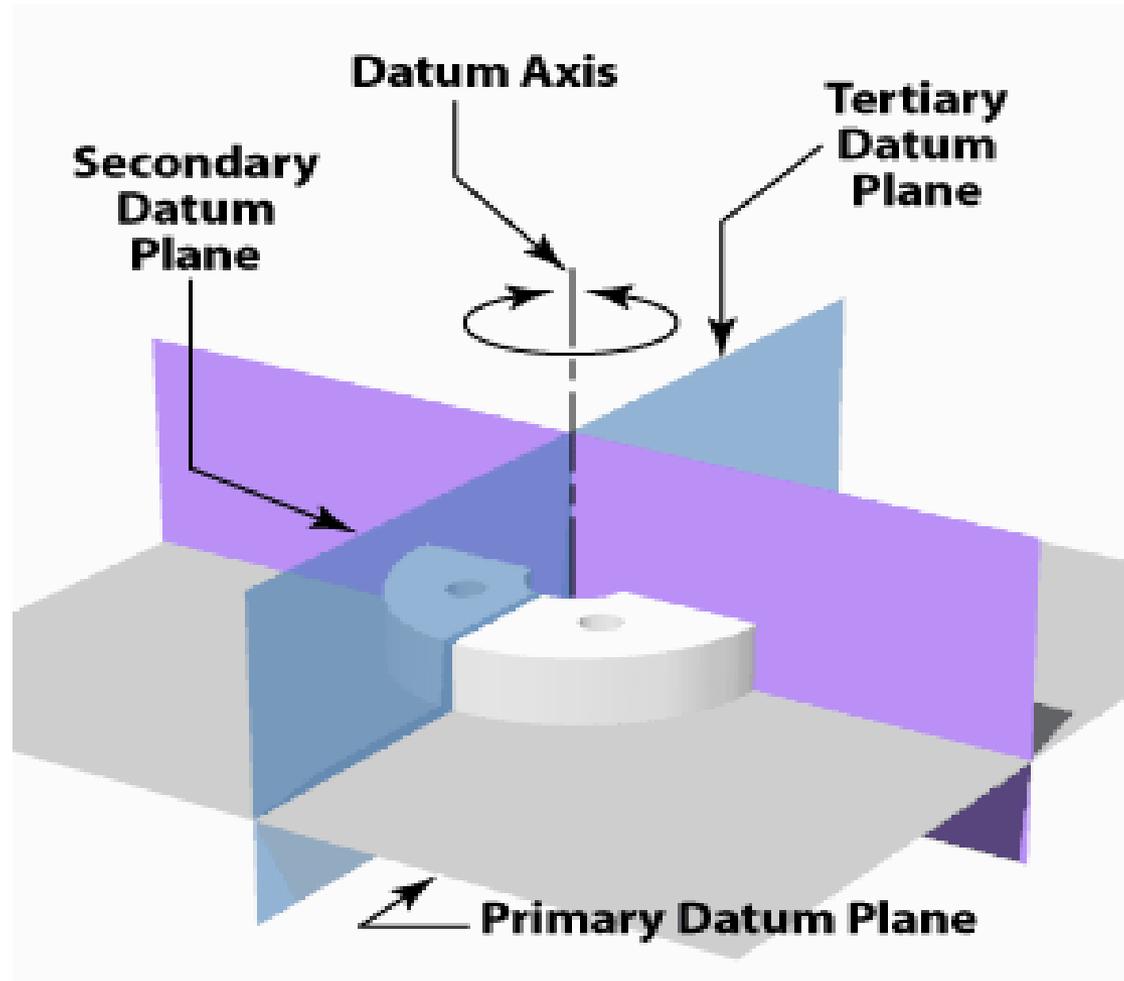
# Part Datum for Inspection



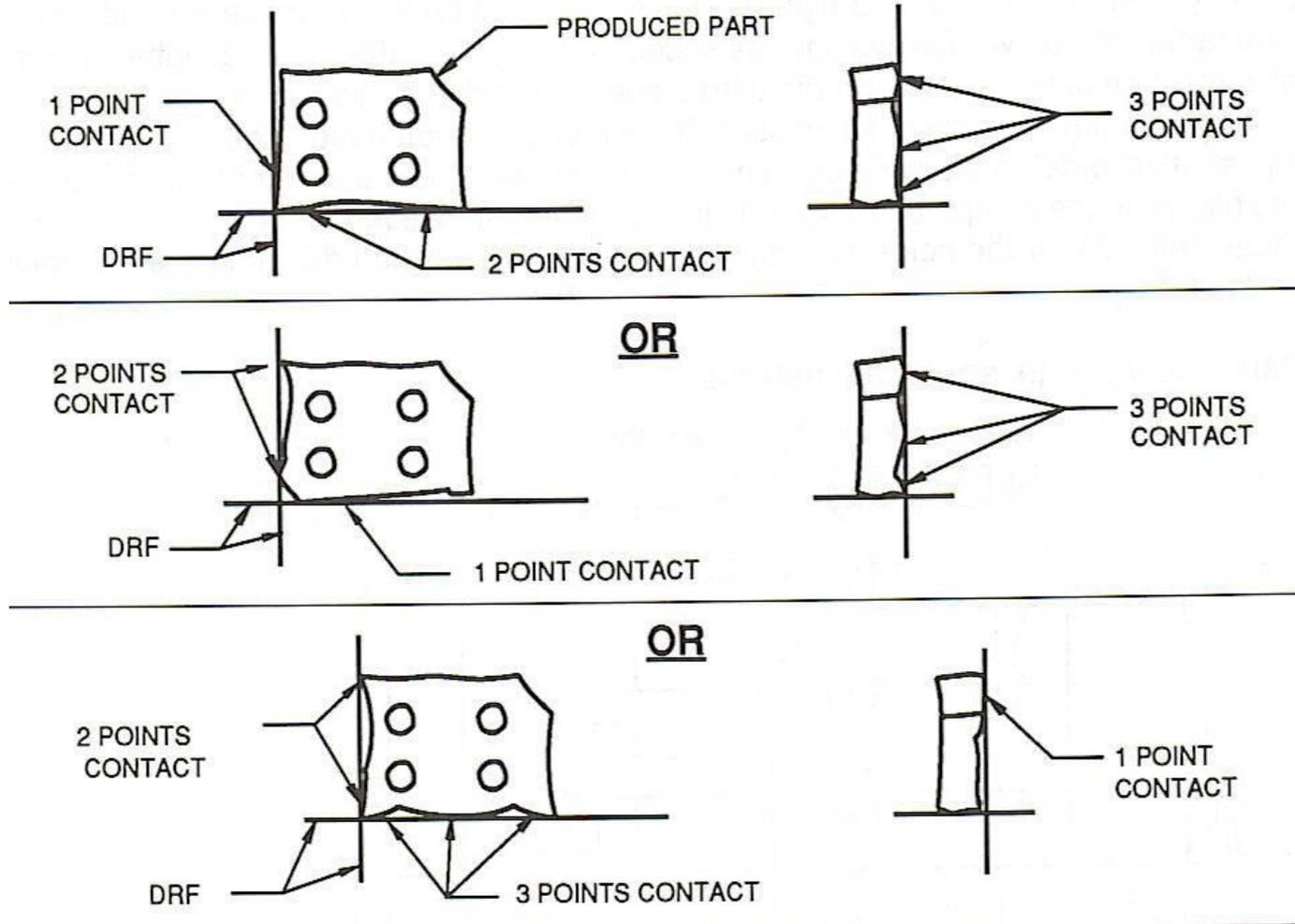
# Datum Planes



# Datum Planes and Axis



# 3 - 2 - 1 Part Datum Location



# Form Tolerances

- Control the form with reference to *Ideal* shape
- Applies to *Line, Arc, Circle, Profile, Surface*
- Tolerance is a *Peak-to – Valley* measure

# Types of Form Tolerances

Straightness



Flatness



Roundness



Cylindricity



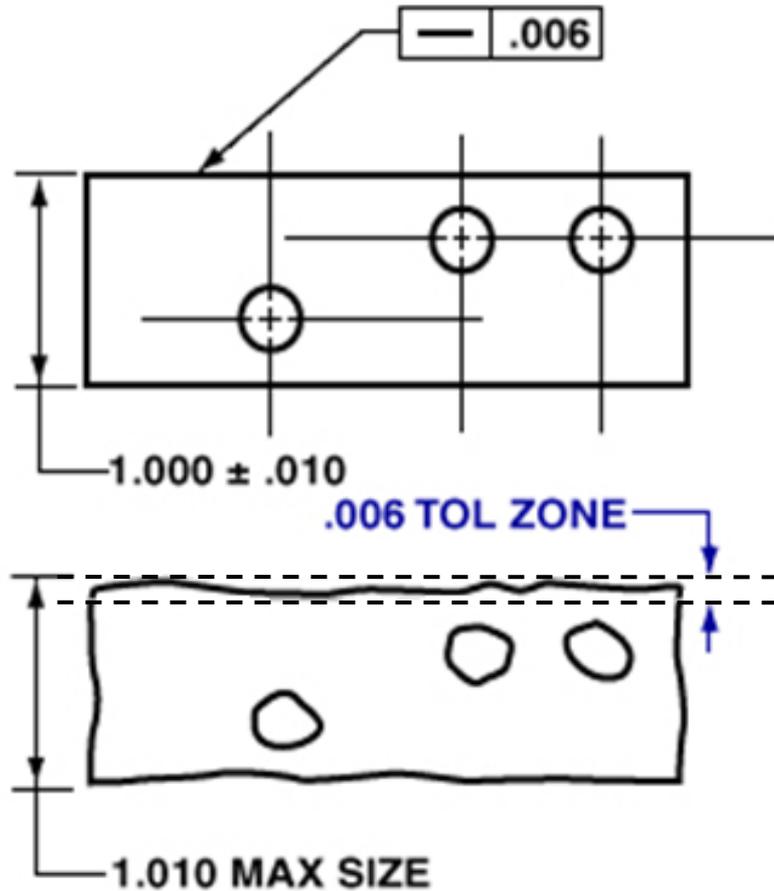
Profile of line



Profile of Surface

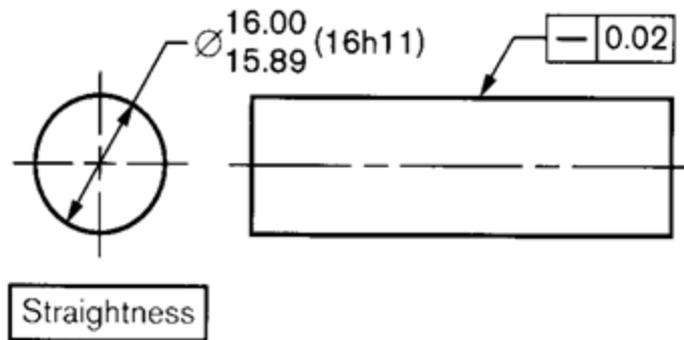


# Straightness Tolerance

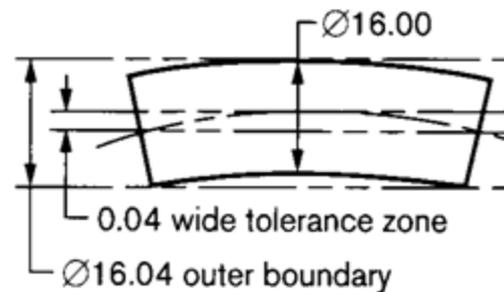
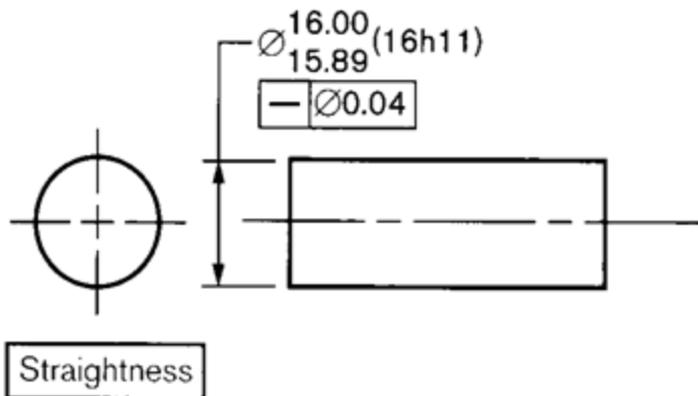
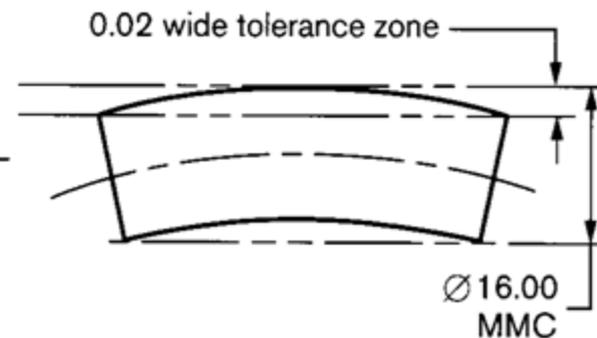


# Straightness Tolerance – Line / Axis

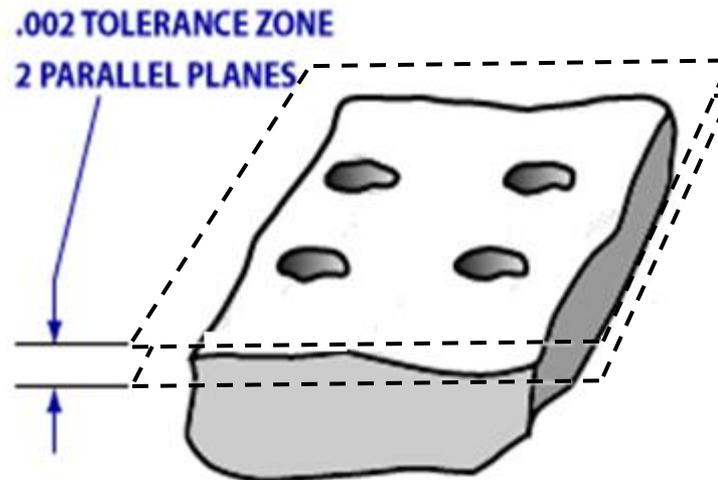
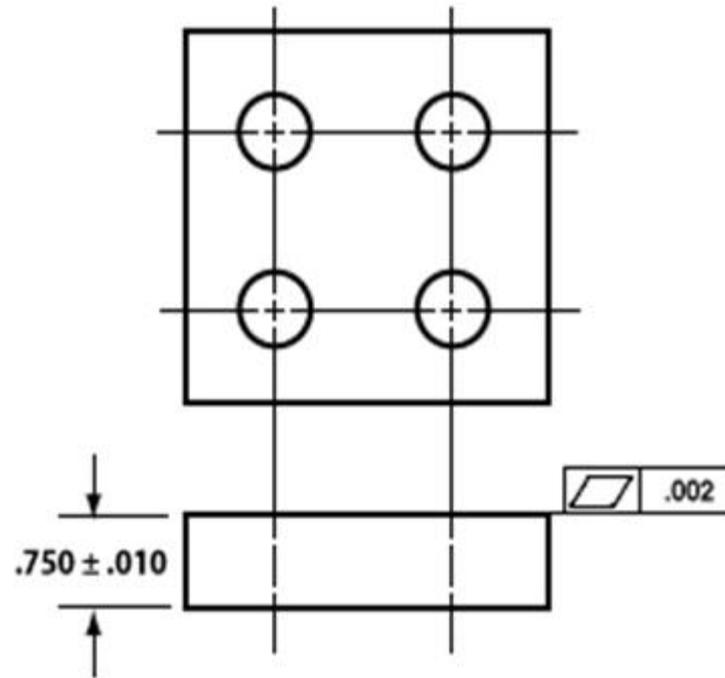
## On the Drawing



## Interpretation

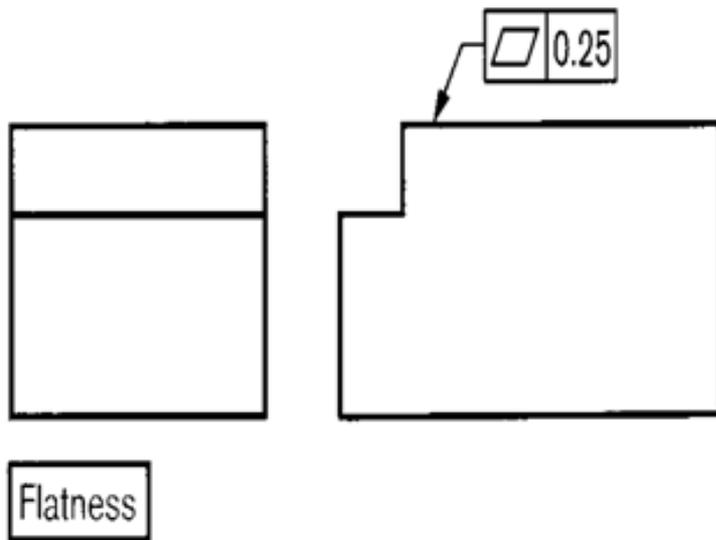


# Flatness Tolerance

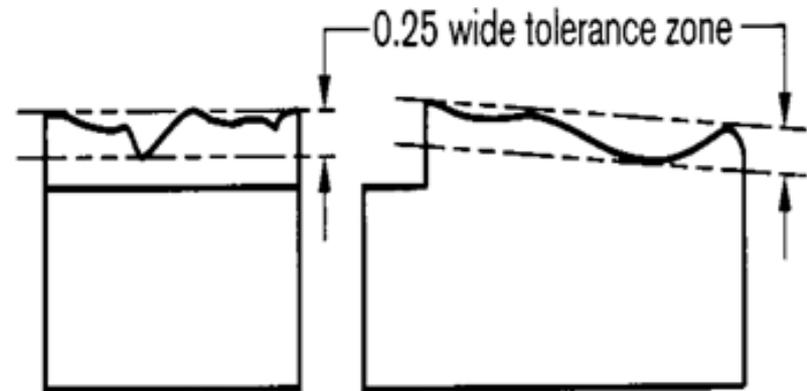


# Flatness Tolerance – Plane

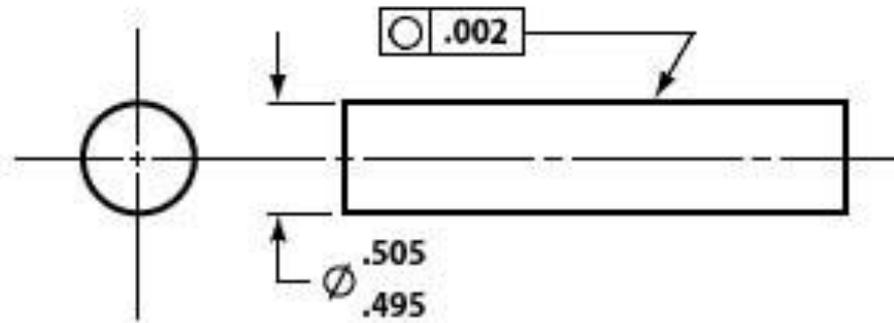
## On the Drawing



## Interpretation

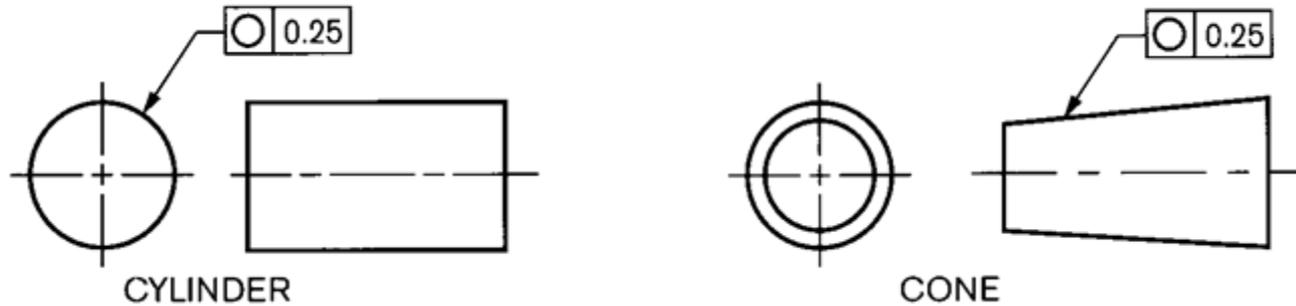


# Circularity (Roundness) Tolerance

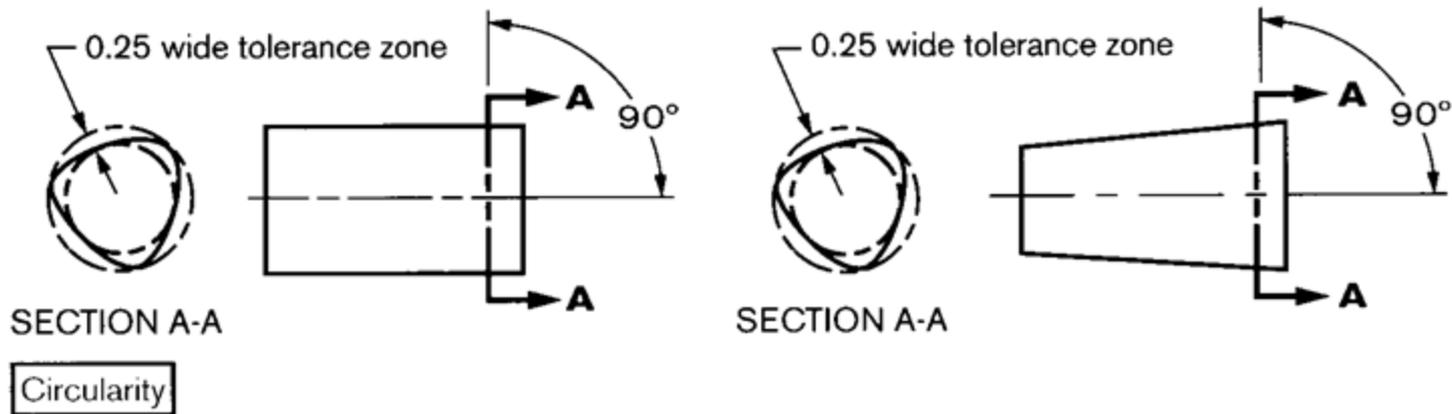


# Circularity (Roundness) Tolerance

## On the Drawing

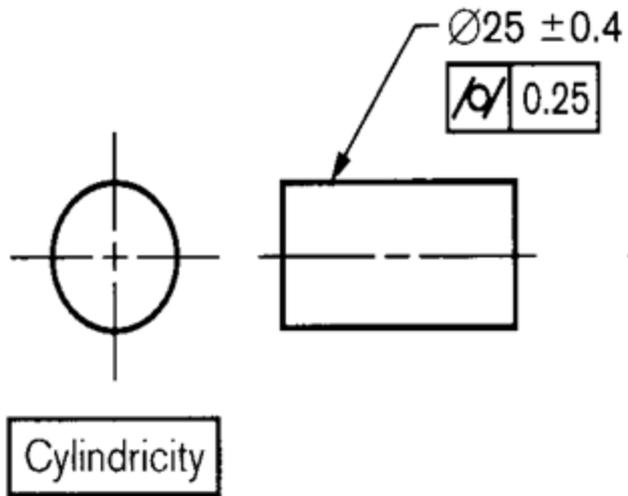


## Interpretation

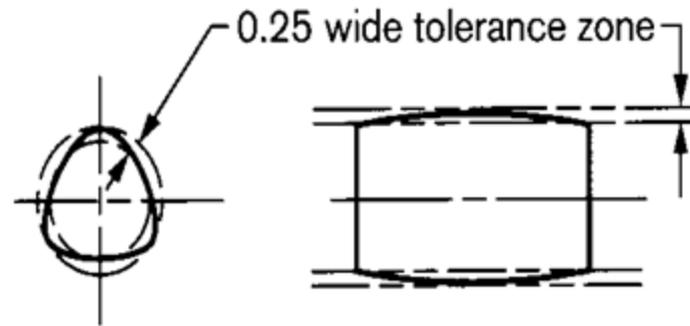


# Cylindricity Tolerance

## On the Drawing



## Interpretation



# Orientation Tolerance

- Specifies relative orientation (Attitude) between entities
  - *Edges, Axes, Surfaces, Features*
- Always need a Datum (Reference) Plane / Feature.

# Types of Orientation Tolerances

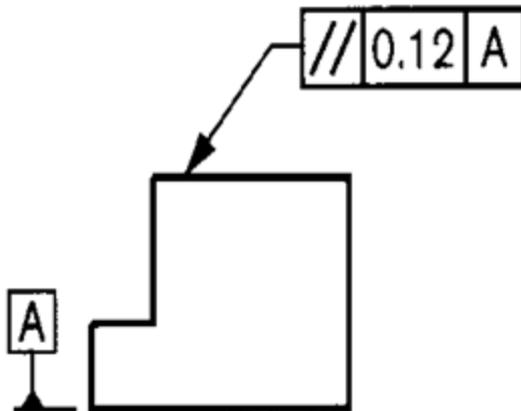
Parallelism //

Perpendicularity  $\perp$

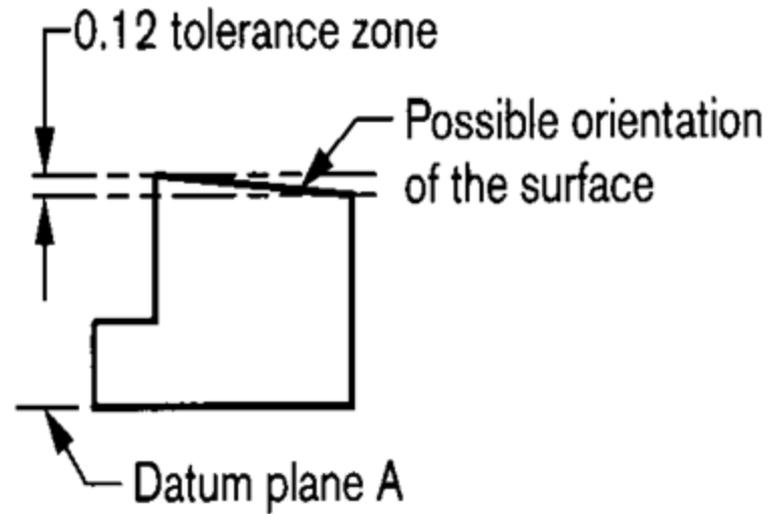
Angularity  $\sphericalangle$

# Parallelism Tolerance

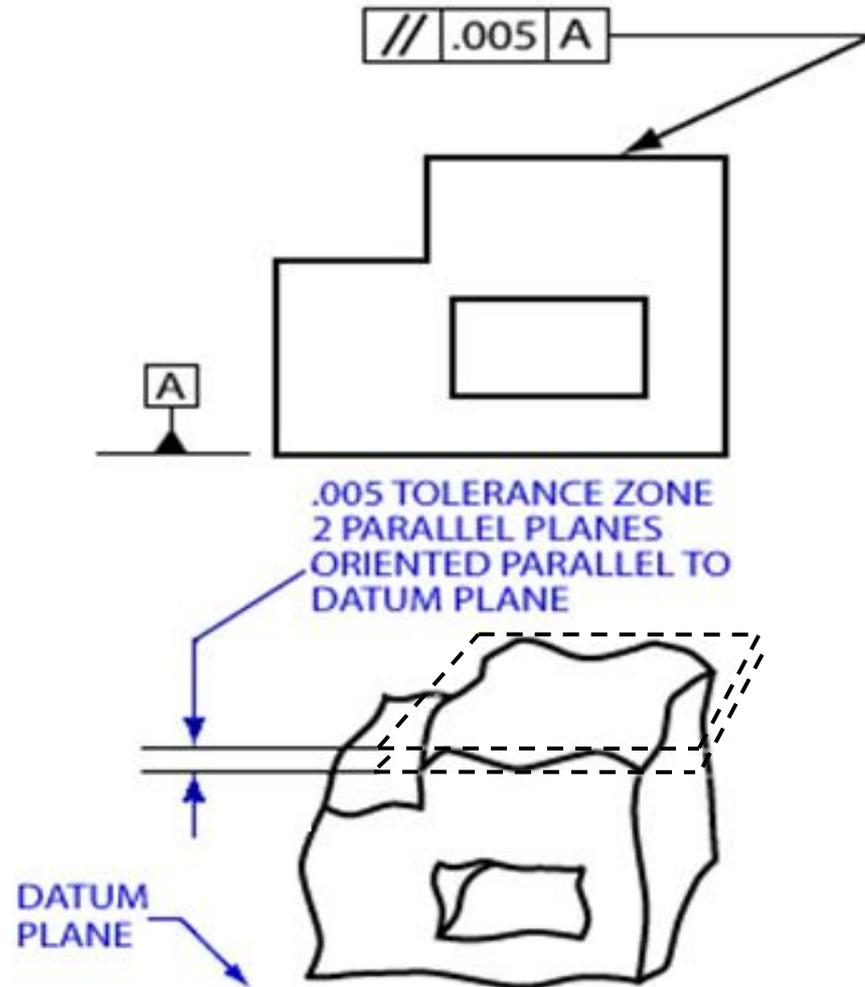
## On the Drawing



## Interpretation

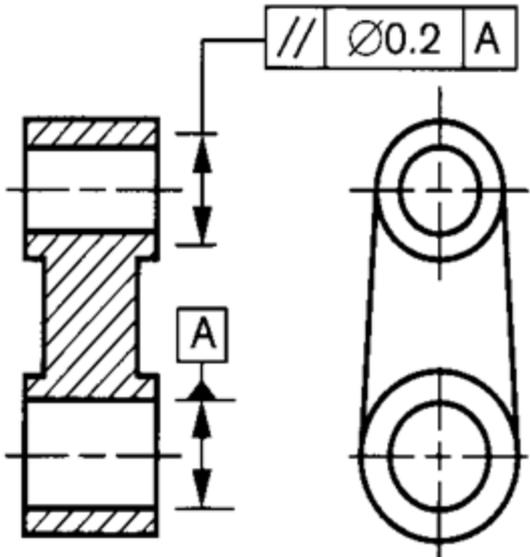


# Parallelism Tolerance - Planes

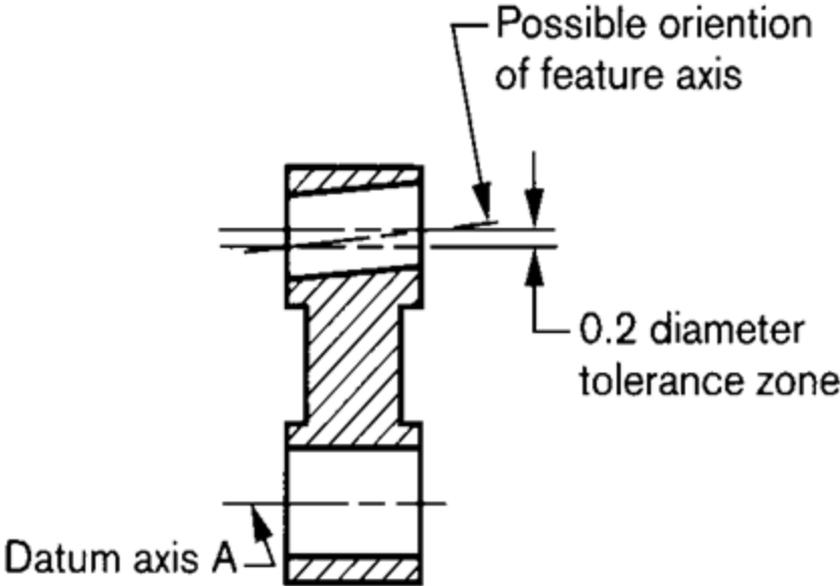


# Parallelism Tolerance - Features

## On the Drawing

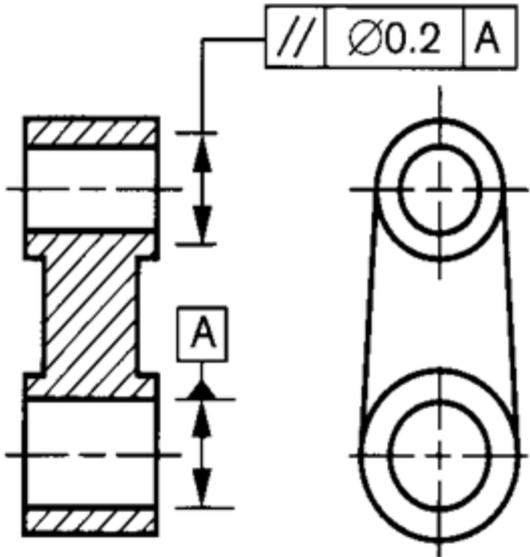


## Interpretation

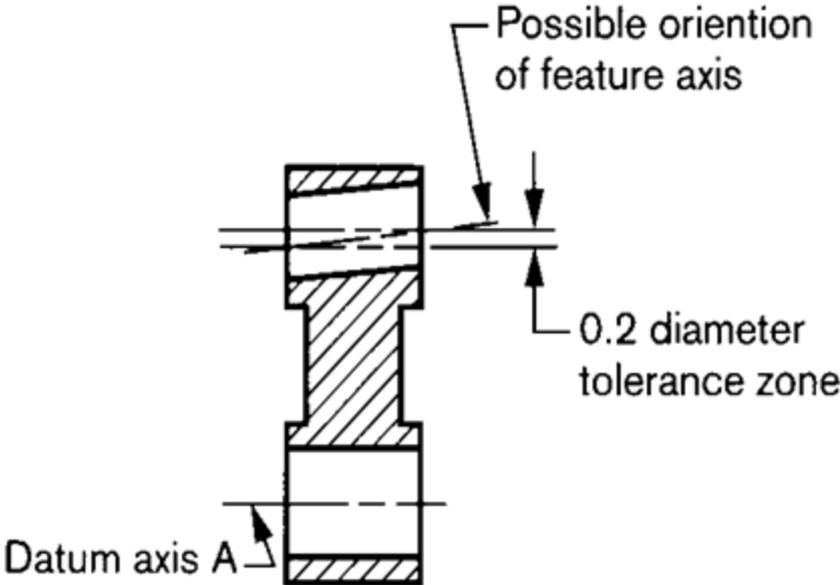


# Parallelism Tolerance - Features

## On the Drawing

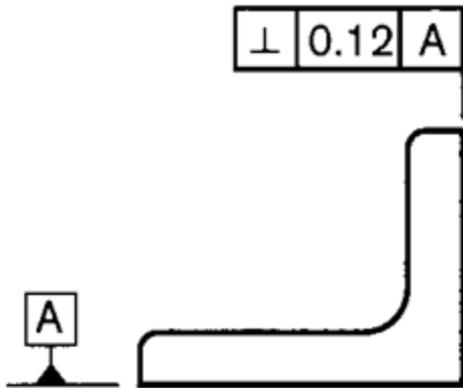


## Interpretation

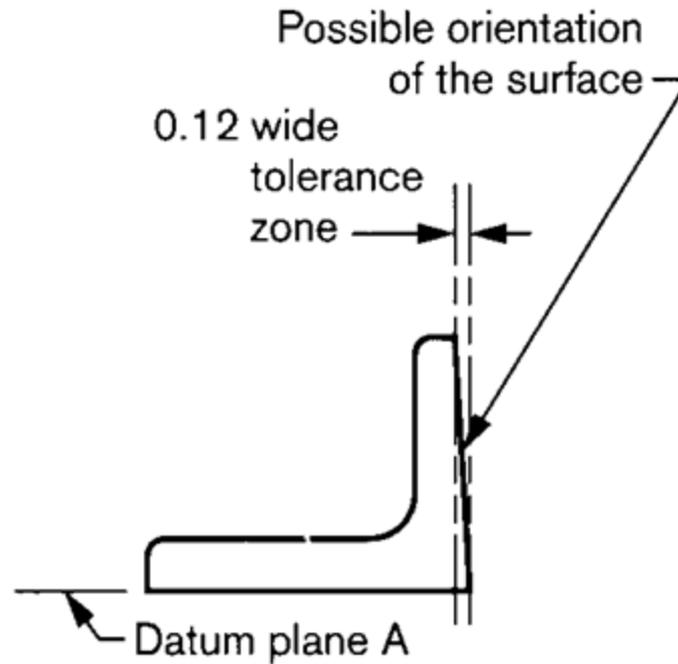


# Perpendicularity Tolerance

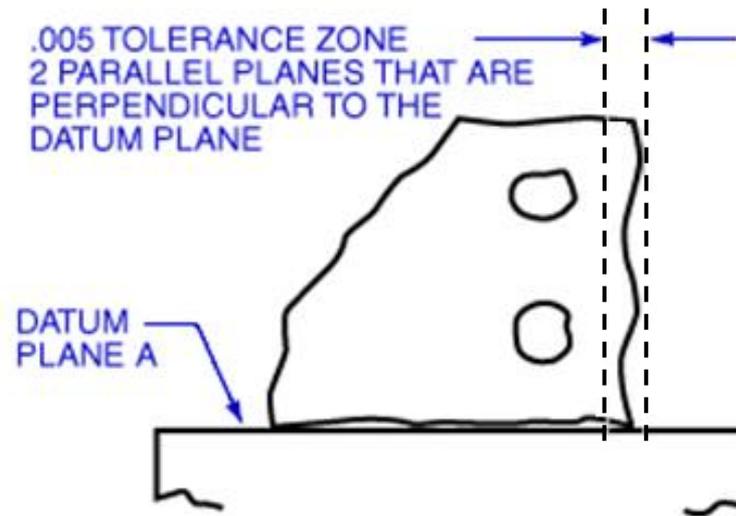
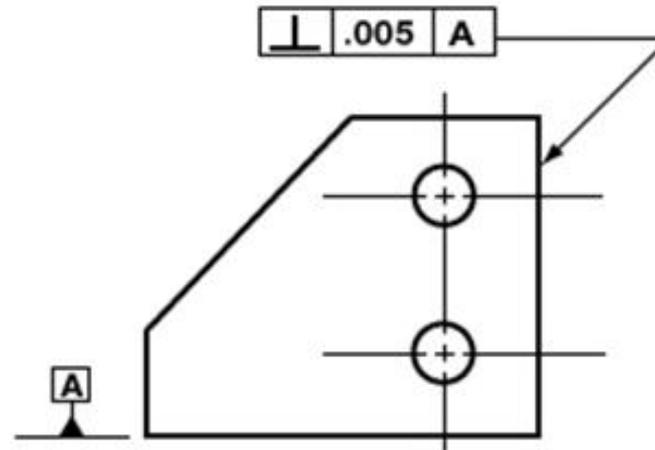
## On the Drawing



## Interpretation



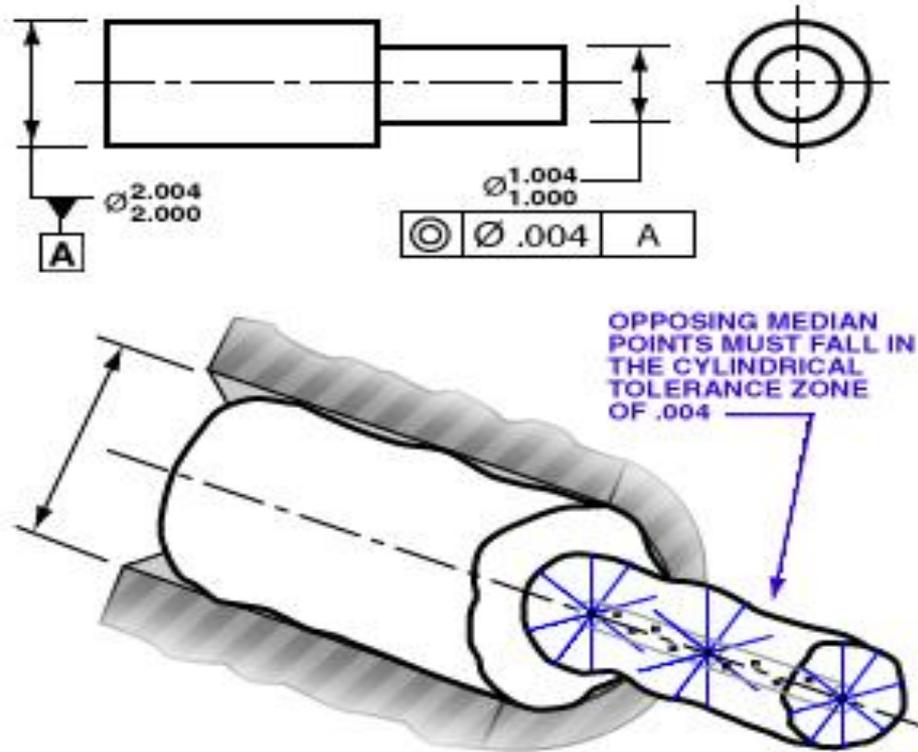
# Perpendicularity Tolerance



# Location Tolerance

- Specifies relative Location (Attitude) between entities
  - *Edges, Axes, Surfaces, Features*
- Always need a Datum (Reference) Plane / Feature.

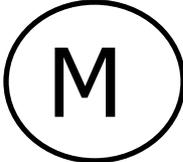
# Concentricity Tolerance



# Material Condition Modifiers

## Maximum Material Condition (MMC)

- Largest (size) shaft
- Smallest (size) Hole

Symbol : 

## Regardless of Feature size (RFS)

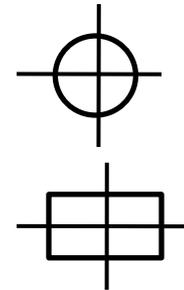
# MMC – RFS Condition

ANSI/ ISO recommends

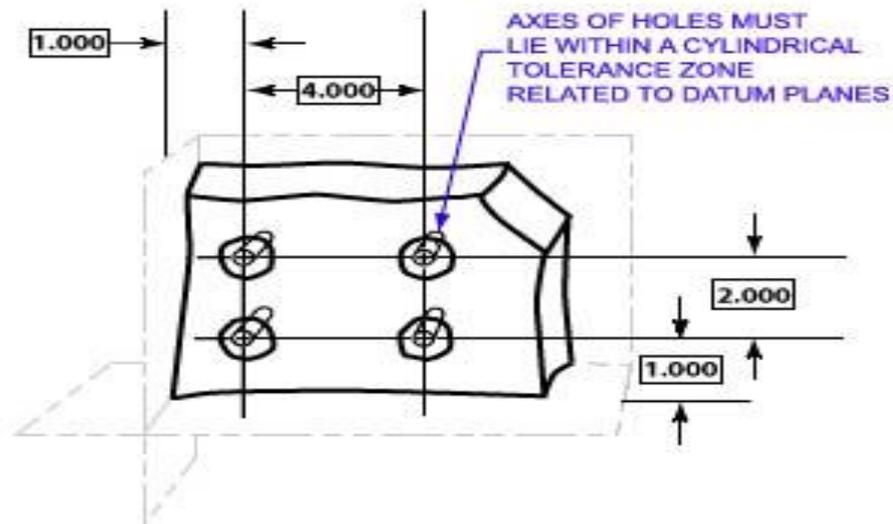
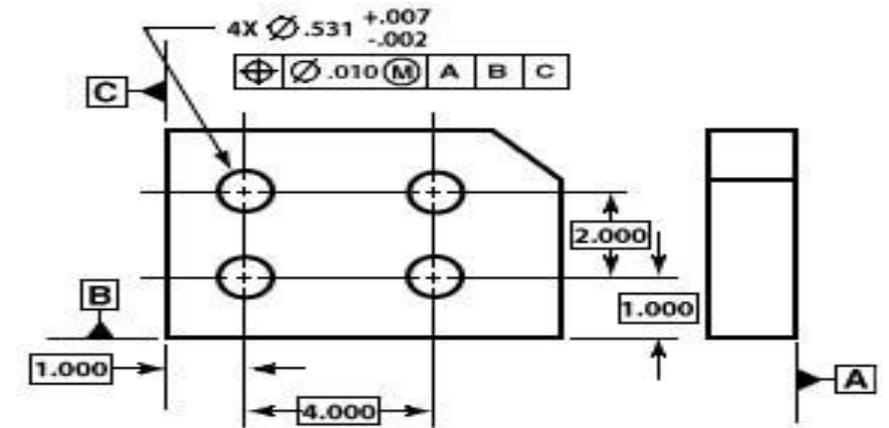
- Form of a feature is Perfect (Ideal) at MMC
- If not specified, treat Feature Tolerance as RFS.

# Position Tolerance

- Controls Position (Centre or Axis) of Holes
- Needs Datums
  - Primary/ Secondary/ Tertiary
- Tolerance Zone
  - Circular
  - Rectangular



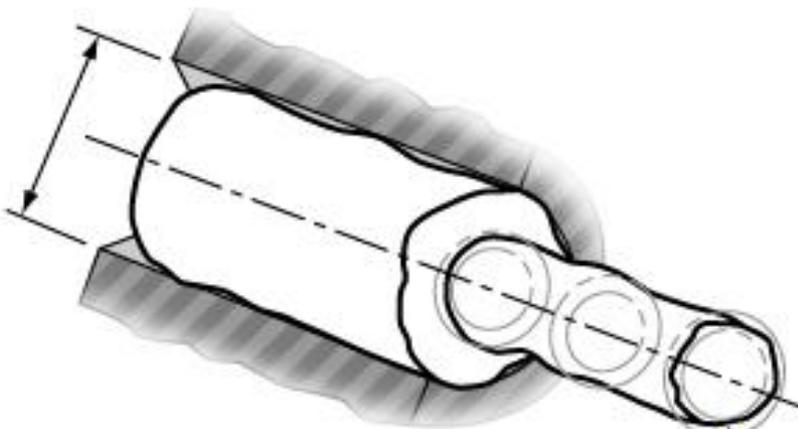
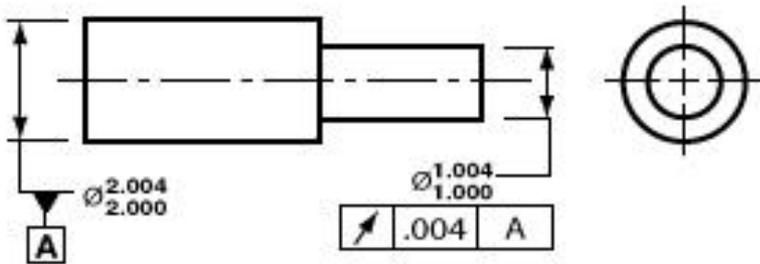
# Position Tolerance



# Run Out Tolerance

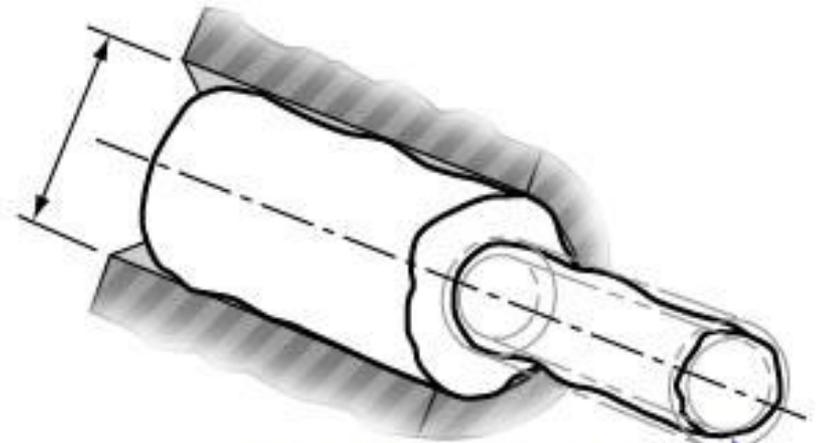
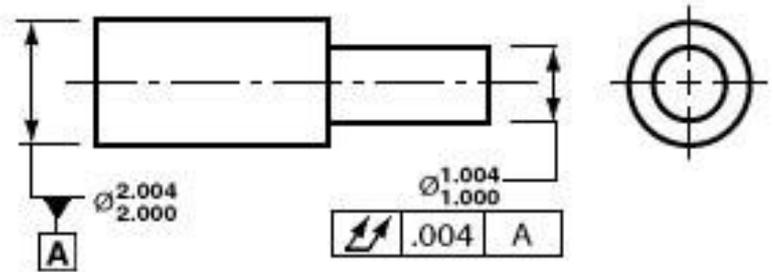
- Primarily for cylindrical rotating parts
- Types of Run Out
  - Circular Run Out
  - Total Run Out

# Run Out Tolerance



EACH CIRCULAR ELEMENT OF THE SURFACE MUST LIE BETWEEN TWO CONCENTRIC CIRCLES, ONE HAVING A RADIUS OF  $.004$  LARGER THAN THE OTHER

Circular



EACH ELEMENT OF THE SURFACE MUST LIE BETWEEN TWO CONCENTRIC CYLINDERS, ONE HAVING A RADIUS OF  $.004$  LARGER THAN THE OTHER

Total

# Functional Gaging

- To test the combined effect of *All* tolerances
  - Dimensional Tolerance
  - Geometric Tolerance
- Gage - *Extreme Virtual Size*

# Gage - Extreme Virtual Size

