Introduction to CAD
ITEC 270

Tolerances-2

by

Dr. Ahmed Khattab

Department of Industrial Technology

College of Engineering

University of Louisiana-Lafayette

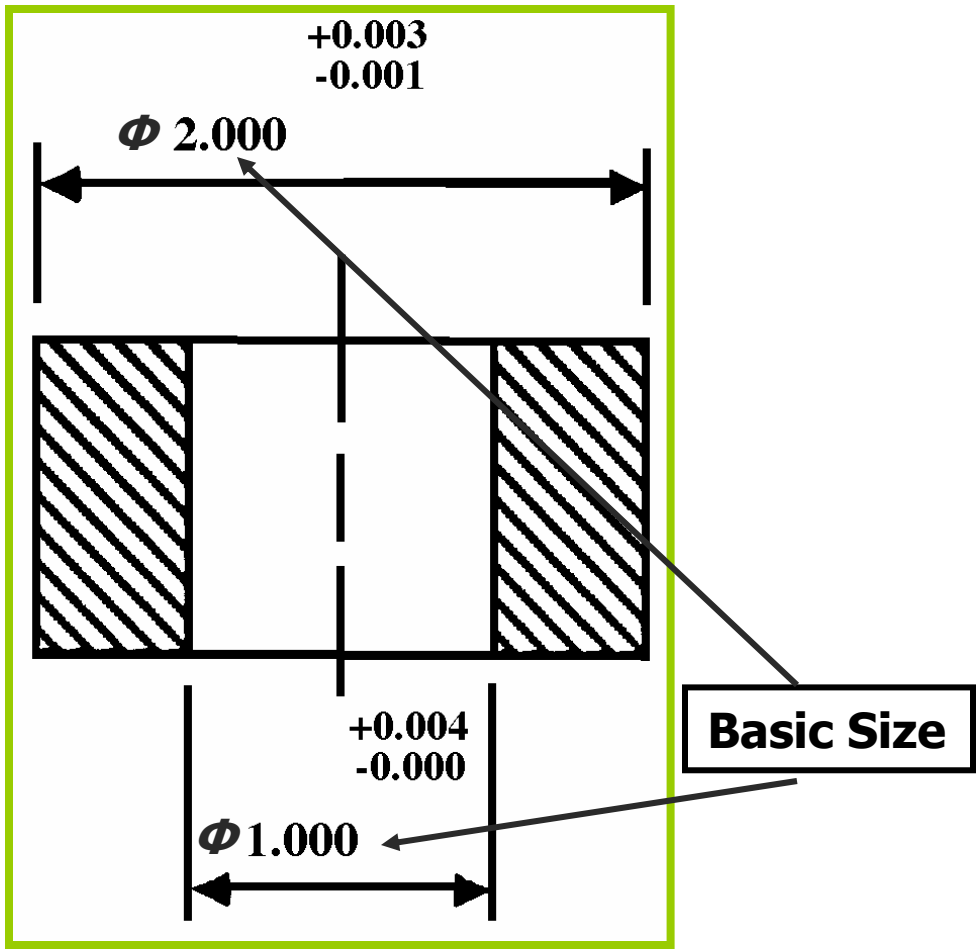
Tolerancing Terms

Nominal or Basic Size

The exact (ideal) size from which the limits are figured

Actual Size

The measured size of the finished parts



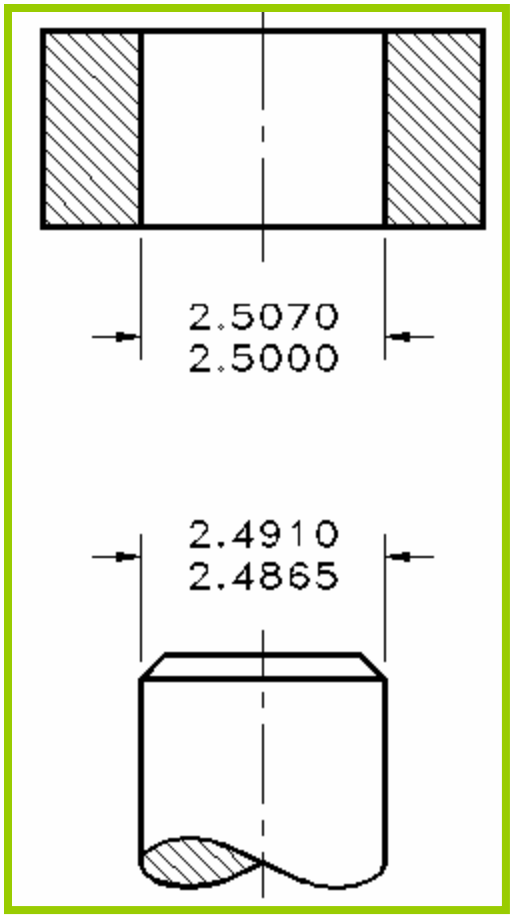
Tolerancing Terms

Limits

The ***Max*** and ***Min*** permissible sizes. 2.5070 & 2.5000 for the hole
2.4865 & 2.4910 for the shaft

Allowances

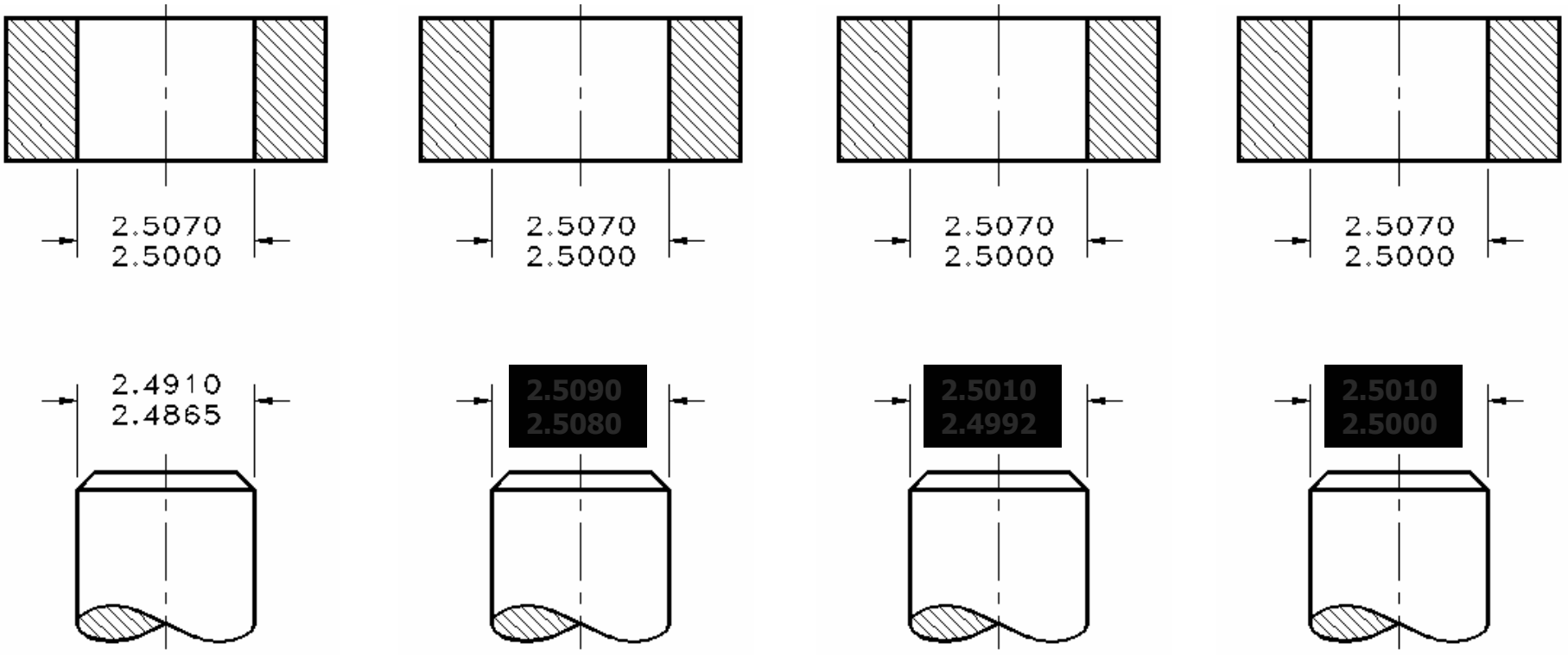
The ***Tightest*** fit between 2 mating parts. Smallest hole – Largest shaft
2.5000 – 2.4910



Tolerancing Terms

Fits

The degree of tightness or looseness between 2 assembled parts.



Clearance fit

Interference fit

Transition fit

Line fit

Basic Hole/Shaft System

System in which tolerances are based on a zero line on the hole or shaft.

- Basic hole system – uses the smallest hole size as the basic diameter.

  Best when machining tools are used.

- Basic shaft system – uses the largest diameter as the basic diameter.

  Best when available standard shafts are used.

Cylindrical Fits

ANSI B4.1, Standard for Cylindrical Fits:

- RC: Running or sliding clearance fit
- L-: Location fits (LC, LT, LN)
- FN: Force and shrink fits

Fit Types

Running and Sliding Fits (RC)

All fits in this class are intended to operate under running performance conditions.

Force and Shrink Fits (FN)

FN1 through FN5 produce a series of force or shrink fits. Used to transmit motion.

Fit Types

Location Fits (L -)
Used for stationary assemblies.

Three types:

- Clearance (LC)
- Transition (LT)
- Interference (LN)

Clearance Fit (LC)

Always results in a ***clearance*** allowance between the two parts.

Used for stationary assemblies which can be freely assembled or disassembled.

Transition Fit (LT)

Produces either light ***clearance*** or light ***interference*** between the two mating parts. Used when greater accuracy in assembly is needed.

Interference Fit (LN)

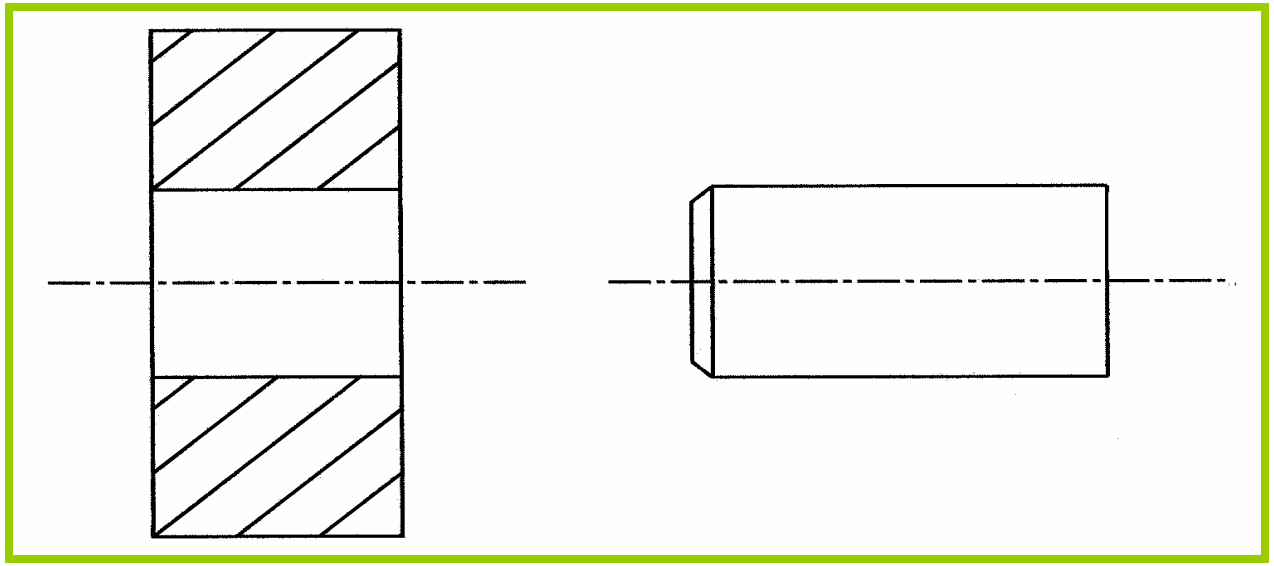
Always produces an ***interference*** between two mating parts. Has a higher accuracy than LT fits.

Example:

Determine the dimensions of both the shaft and the hole.

Select the tolerance and do an allowance analysis.

Assume an RC5 fit for a basic diameter of 2.0000”



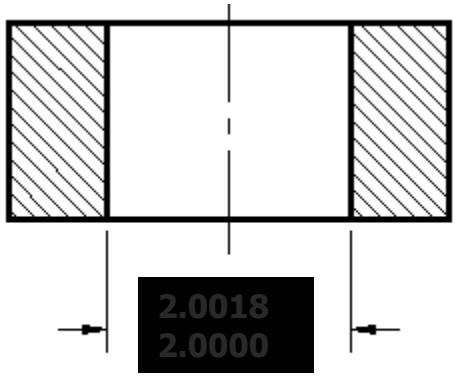
Solution:

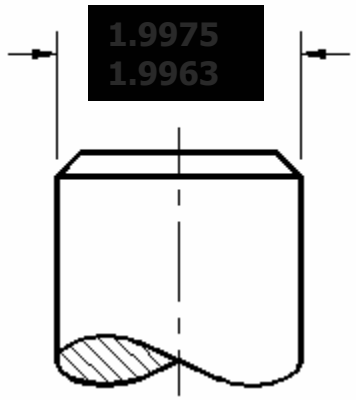
Class RC5: Running & Clearance fit

Basic Diameter = 2.0000

Hole { +1.8 .0018 ; 0 0000 }

Shaft { -2.5 -.0025 ; -3.7 -.0037 }





Max Clear. = .0055

(2.0018 – 1.9963)

Min Clear. = .0025

(2.0000 – 1.9975)

Nominal Size Range, inches Over	To	Class RC 5 Limits of Clearance	Standard Limits Hole H8	Standard Limits Shaft e7
0	0.12	**0.6** **1.6**	**+0.6** **−0**	**−0.6** **−1.0**
0.12	0.24	**0.8** **2.0**	**+0.7** **−0**	**−0.8** **−1.3**
0.24	0.40	**1.0** **2.5**	**+0.9** **−0**	**−1.0** **−1.6**
0.40	0.71	**1.2** **2.9**	**+1.0** **−0**	**−1.2** **−1.9**
0.71	1.19	**1.6** **3.6**	**+1.2** **−0**	**−1.6** **−2.4**
1.19	1.97	**2.0** **4.6**	**+1.6** **−0**	**−2.0** **−3.0**
1.97	3.15	**2.5** **5.5**	**+1.8** **−0**	**−2.5** **−3.7**
3.15	4.73	**3.0** **6.6**	**+2.2** **−0**	**−3.0** **−4.4**

Note:
The given values are in thousandths.

Geometric Tolerance

- Specify tolerance for the geometry of a part separate from its size
- GDT (Geometric Dimensioning and Tolerancing) uses special symbols to control different geometric features of a part

GEOMETRIC TOLERANCING

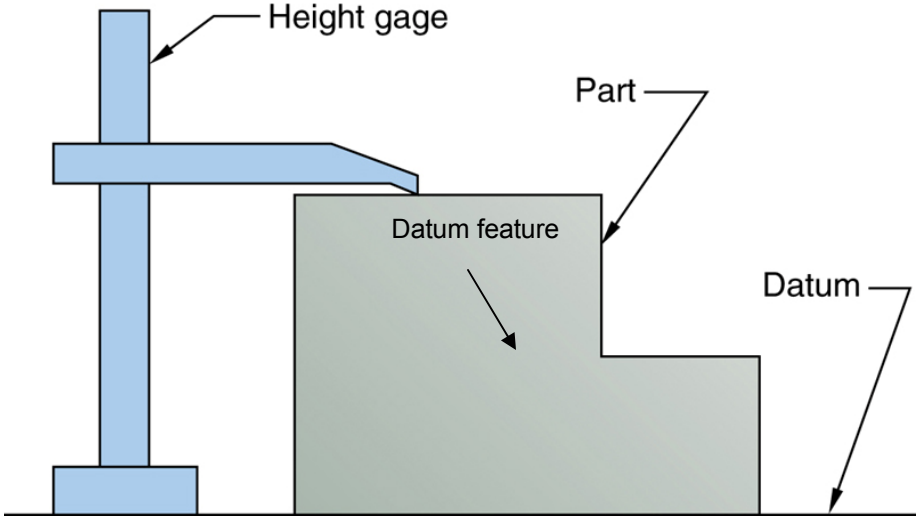
- There are 13 geometric symbols used in the language of GD&T
- There are 5 separate categories:
  - Form
  - Orientation
  - Location
  - Runout
  - Profile

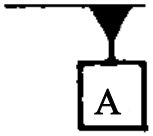
GEOMETRIC TOLERANCING

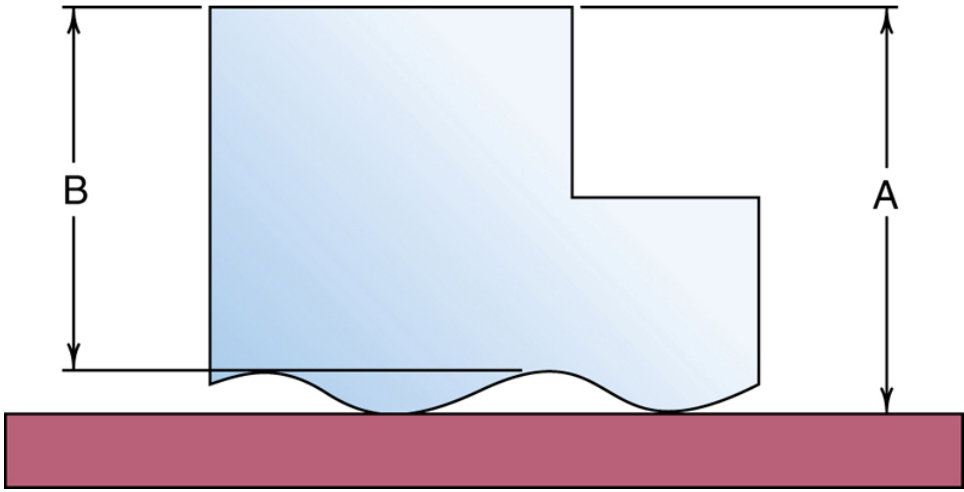
CATEGORY	CHARACTERISTIC	SYMBOL	DATUM * REFERENCS
FORM	FLATNESS		NEVER USES A DATUM REFERENCE
	STRAIGHTNESS		
	CIRCULARITY		
	CYLINDRICITY		
ORIENTATION	PERPENDICULARITY		ALWAYS USES A DATUM REFERENCE
	ANGULARITY		
	PARALLELISM		
LOCATION	POSITION		
	CONCENTRICITY		
RUNOUT	CIRCULAR RUNOUT		
	TOTAL RUNOUT		
PROFILE	PROFILE OF A LINE		MAY USE A DATUM REFERENCE
	PROFILE OF A SURFACE		

Datum

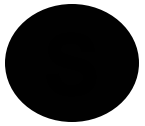
A point, axis (centerline) or surface (plane) used as a starting point for dimensions and tolerances.



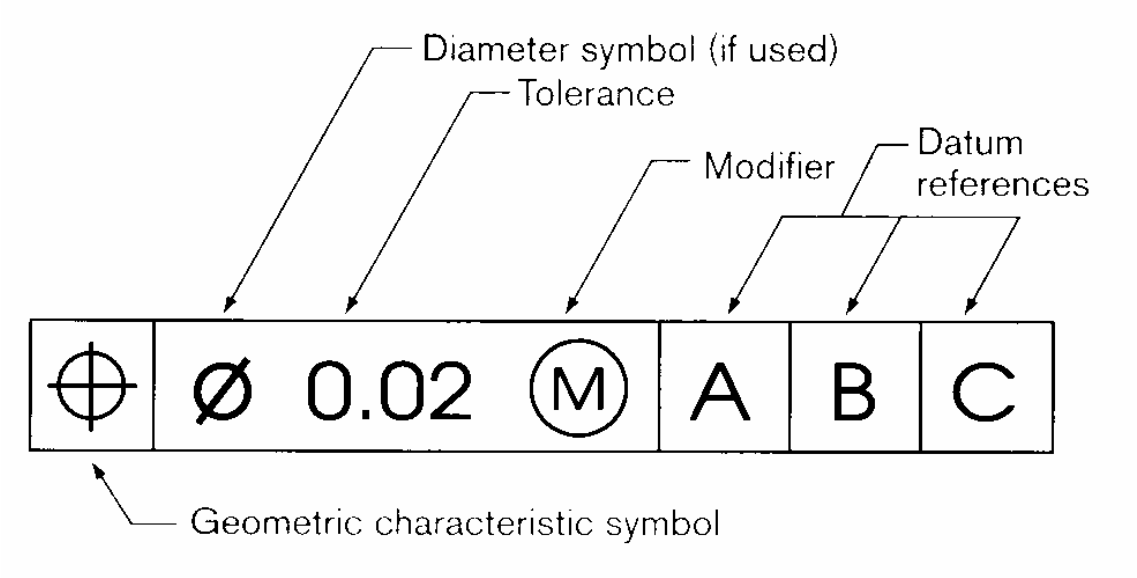




Modifiers

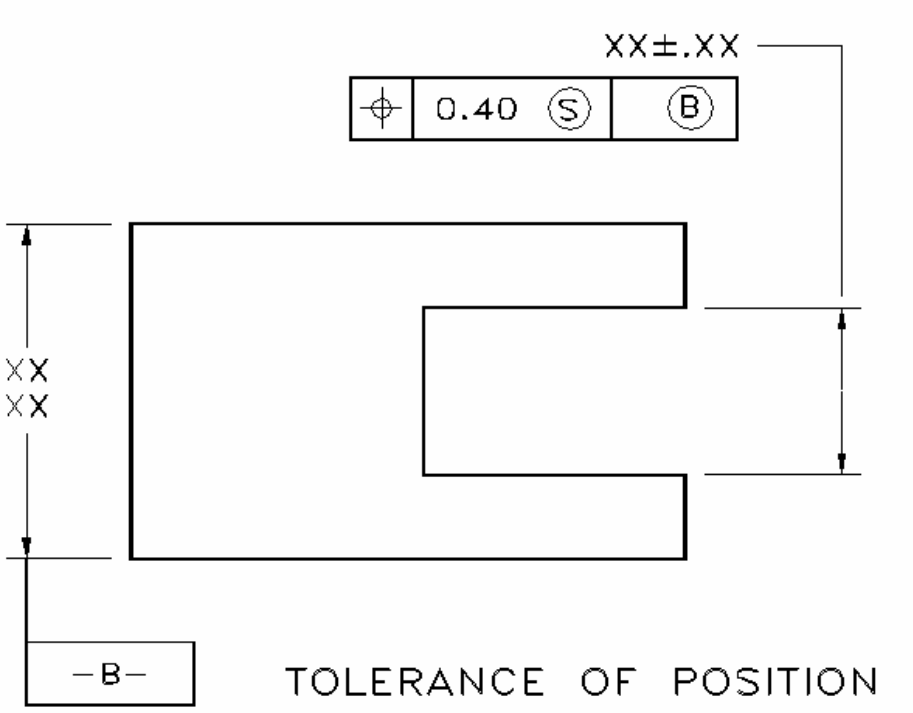
- Maximum Material Condition (MMC) Ⓜ
- Least Material Condition (LMC) Ⓛ
- Regardless of Feature Size (RFS) 

GEOMETRIC TOLERANCING

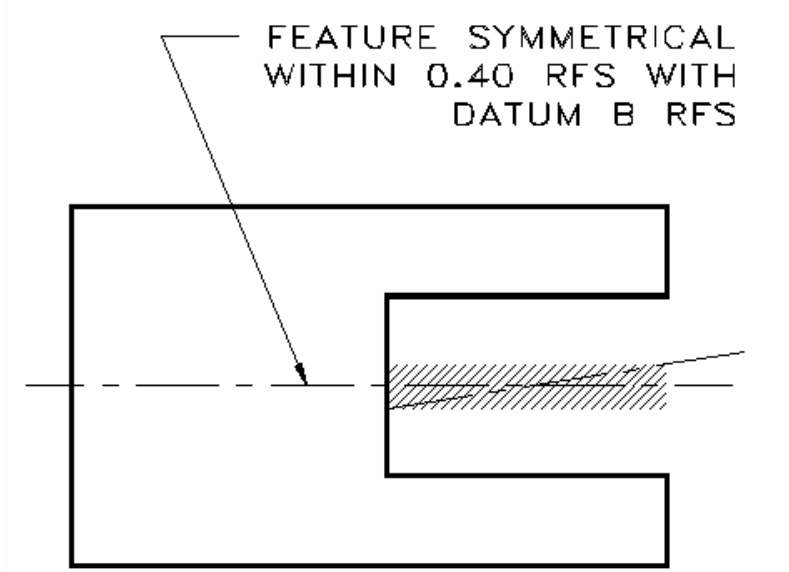


Feature Control Frame

Position

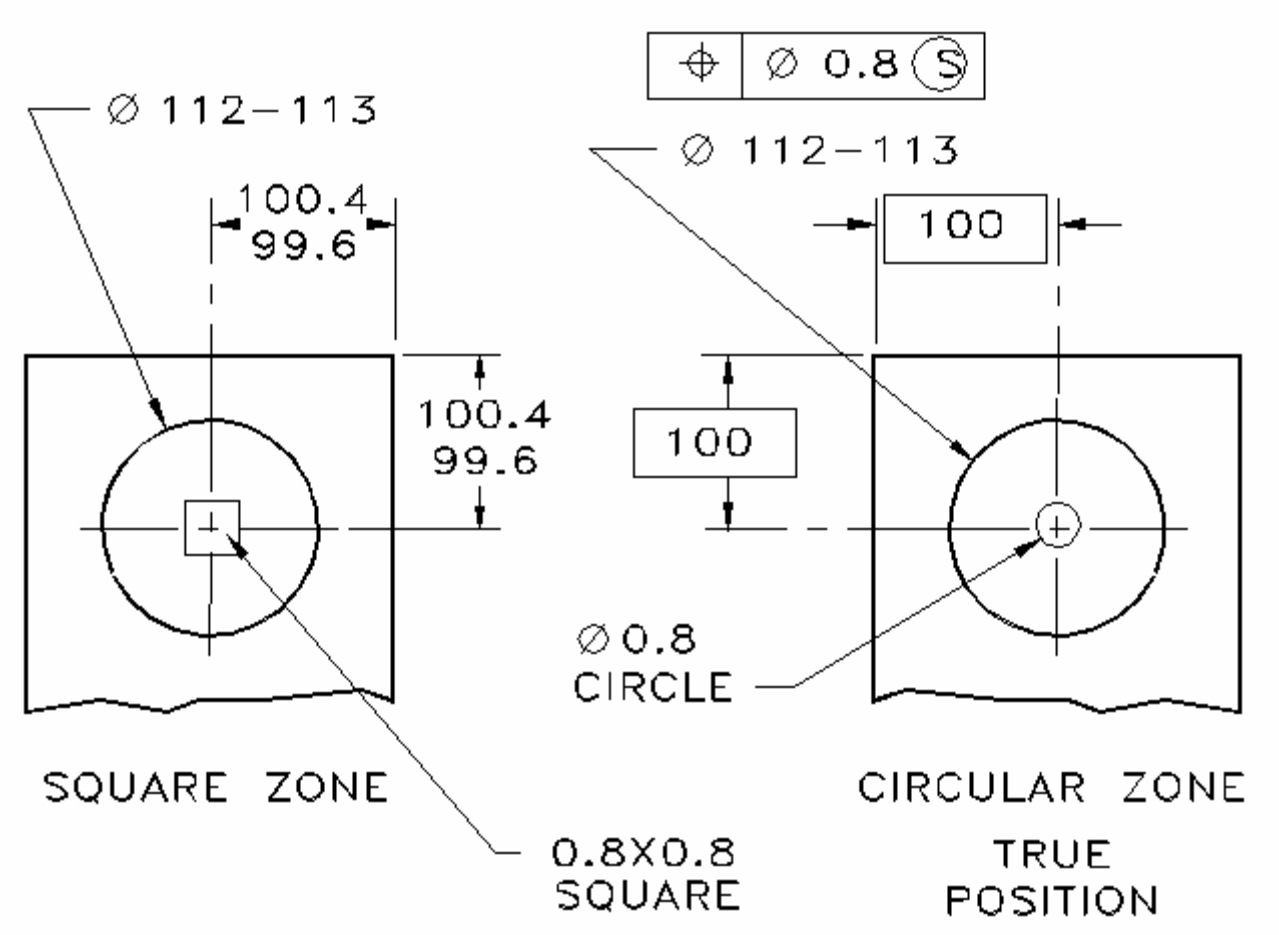


TOLERANCE OF POSITION

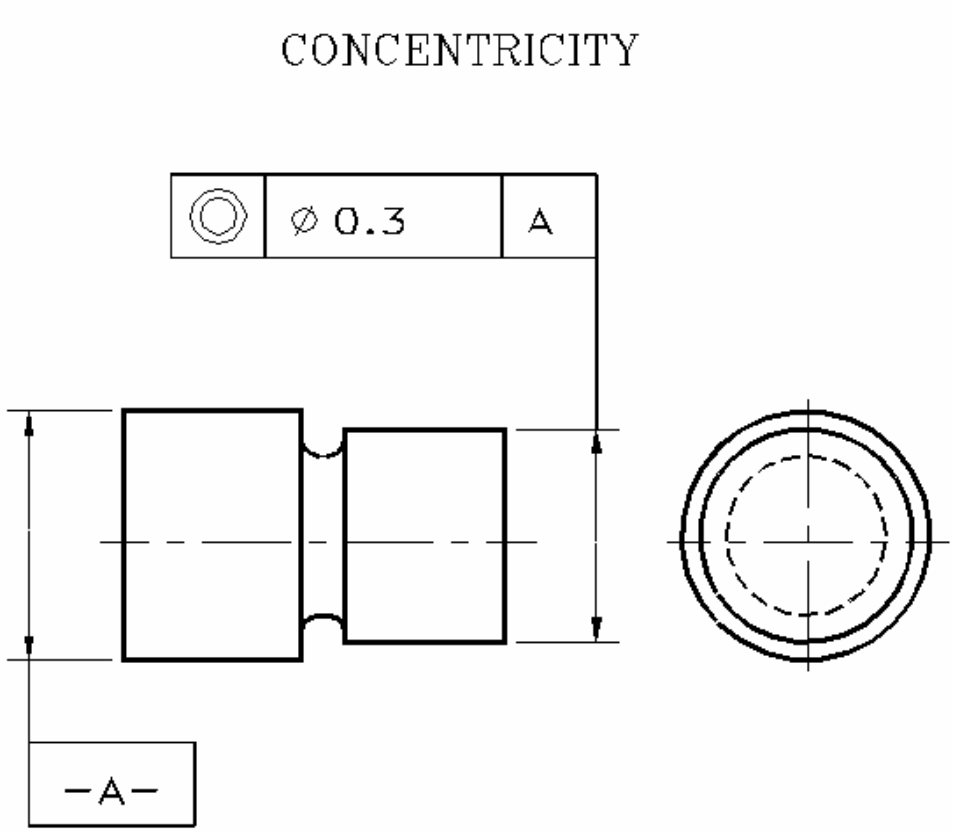


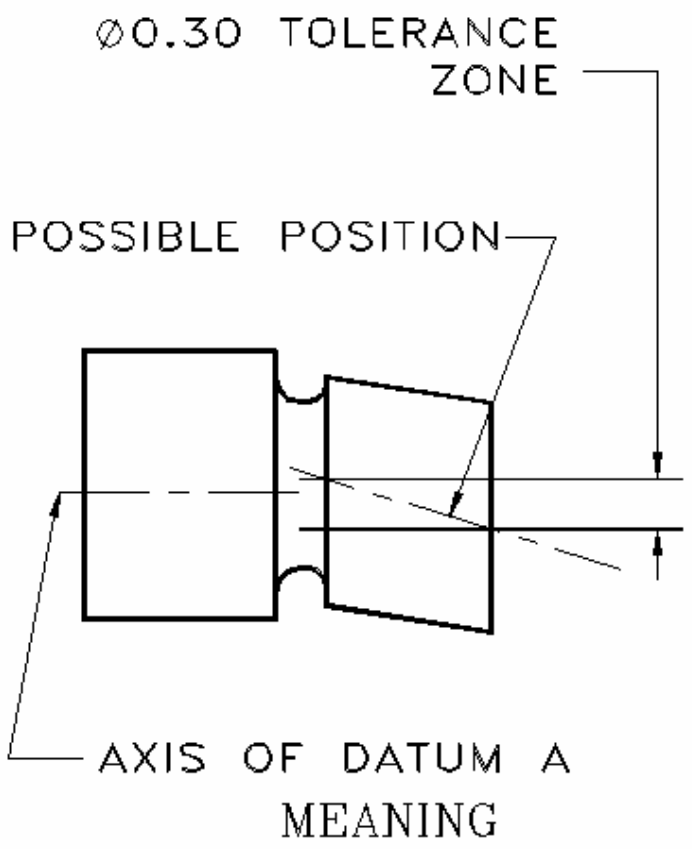
TOLERANCE OF POSITION

Position

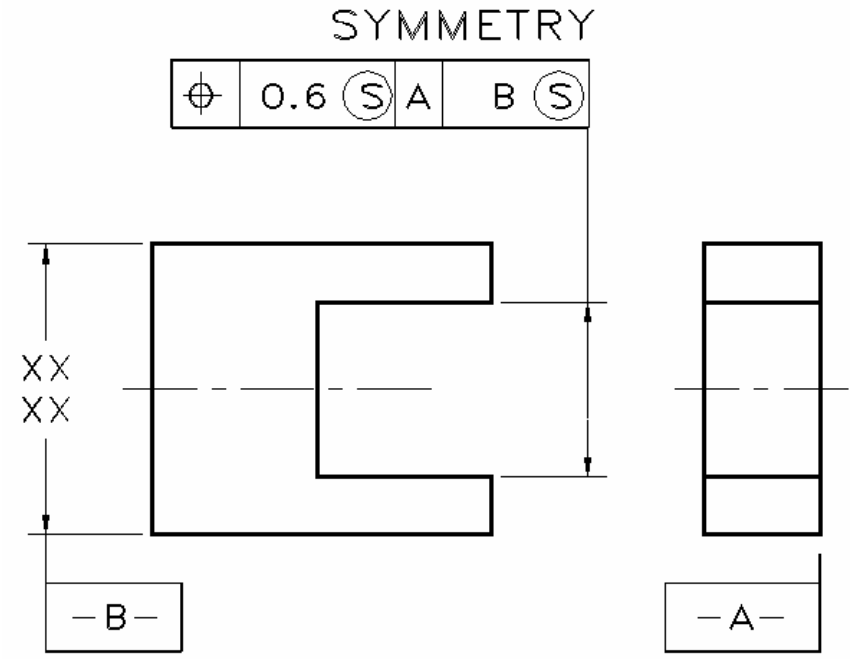

⌖ ∅ 0.8 S
∅ 112–113
100.4
99.6
100.4
99.6
∅ 112–113
100
100
∅0.8
CIRCLE
SQUARE ZONE
0.8X0.8
SQUARE
CIRCULAR ZONE
TRUE
POSITION

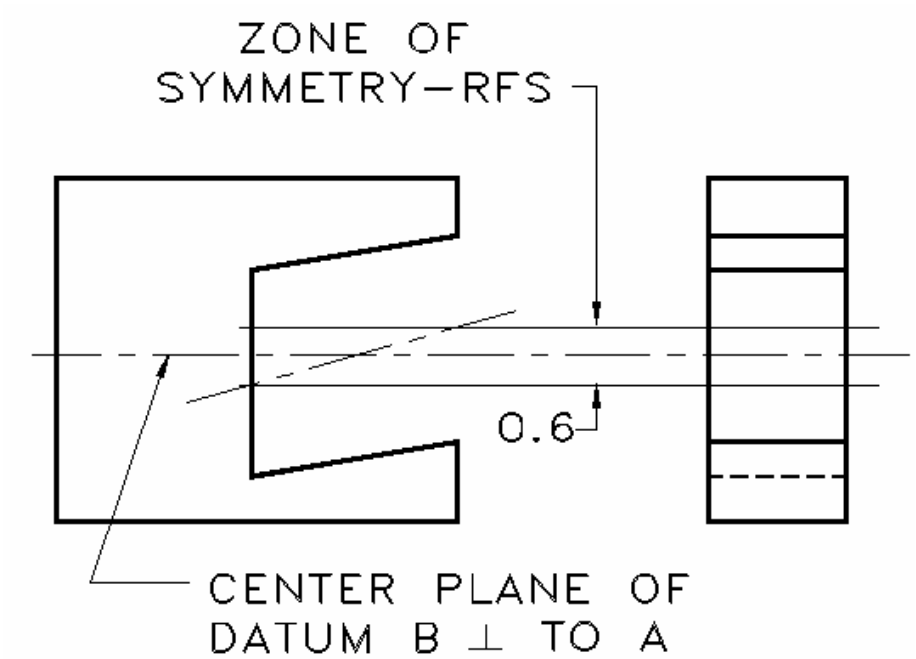
Concentricity



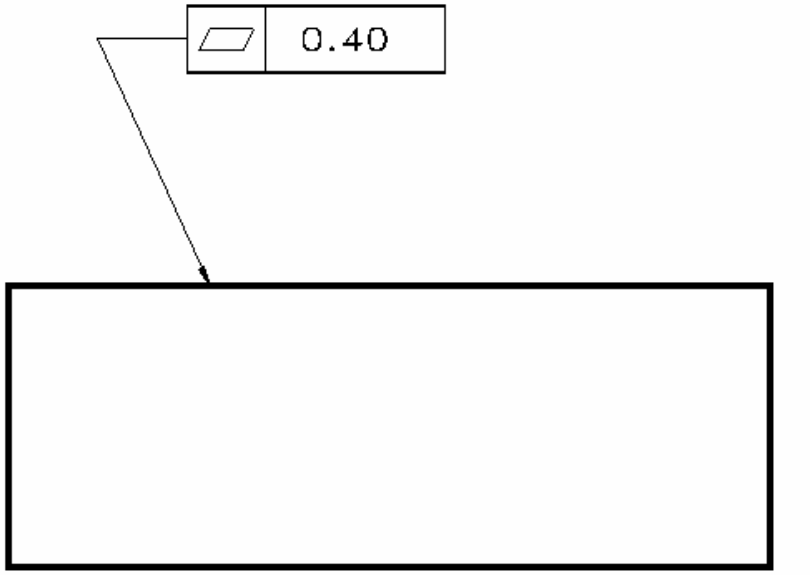


Symmetry

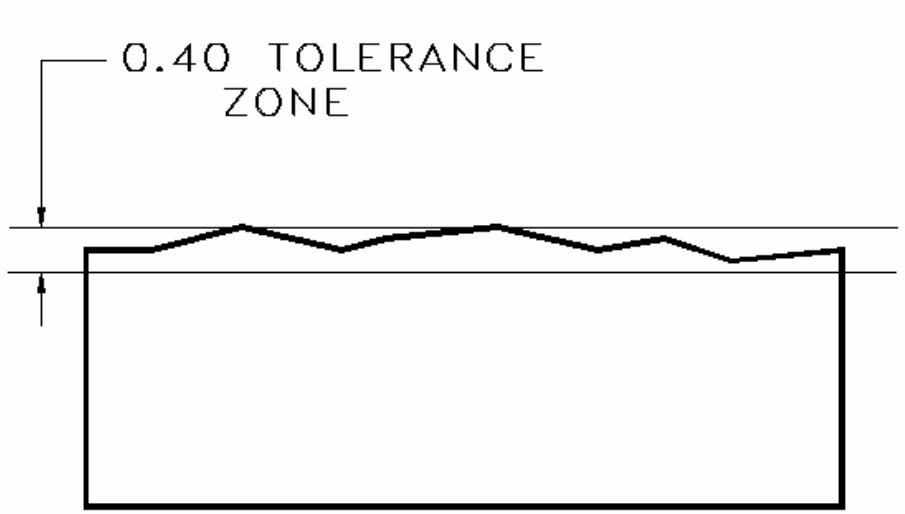

SYMMETRY
0.6 S A B S
XX
XX
–B–
–A–


ZONE OF
SYMMETRY–RFS
0.6
CENTER PLANE OF
DATUM B ⊥ TO A

Flatness

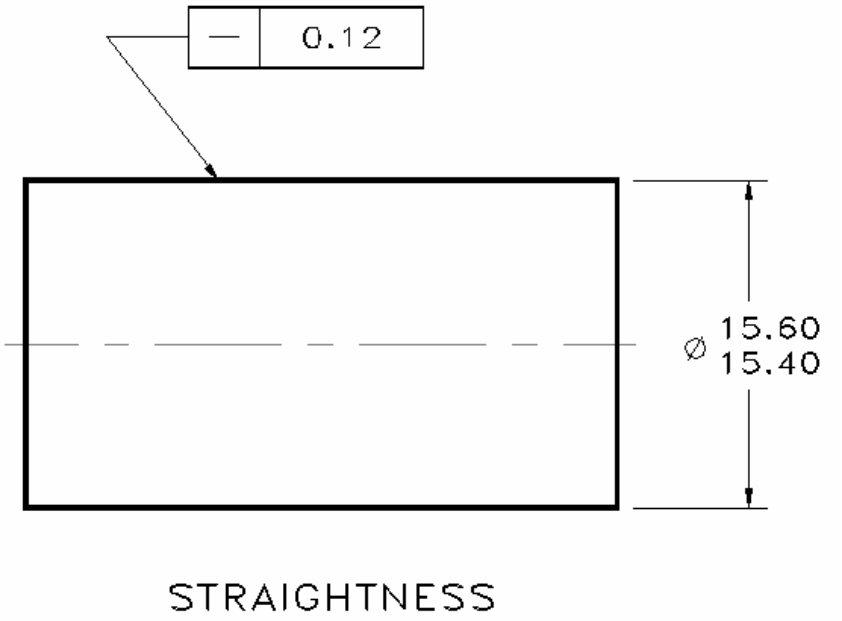


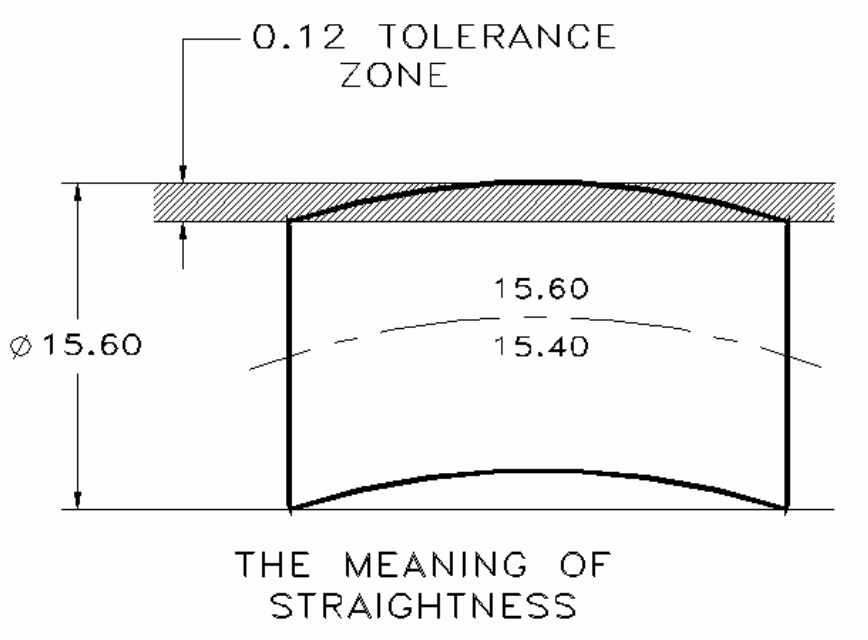
FLATNESS



THE MEANING OF
FLATNESS

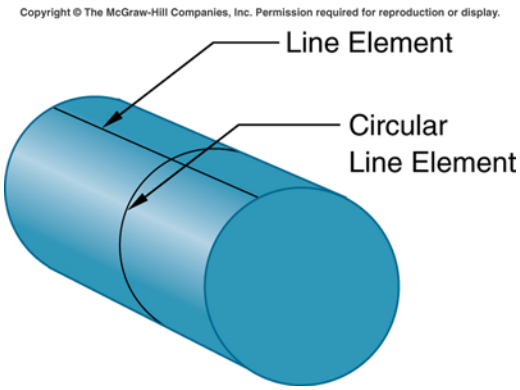
Straightness

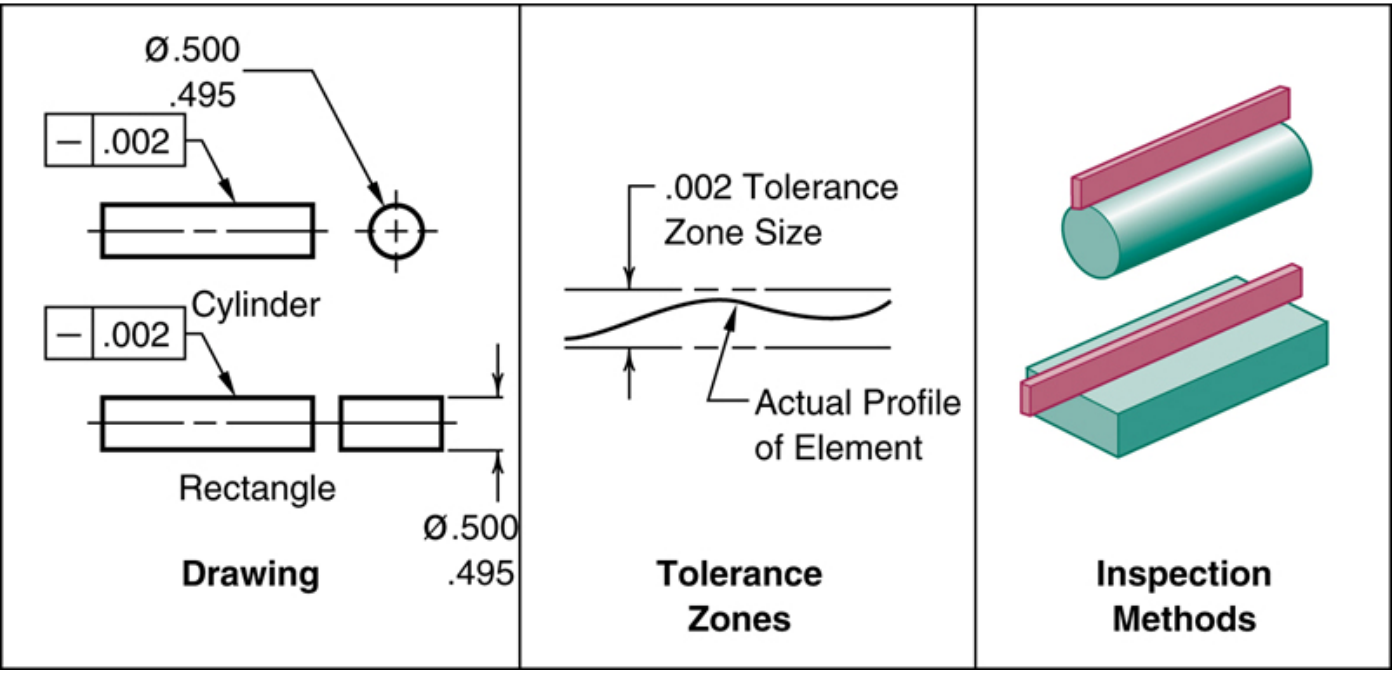




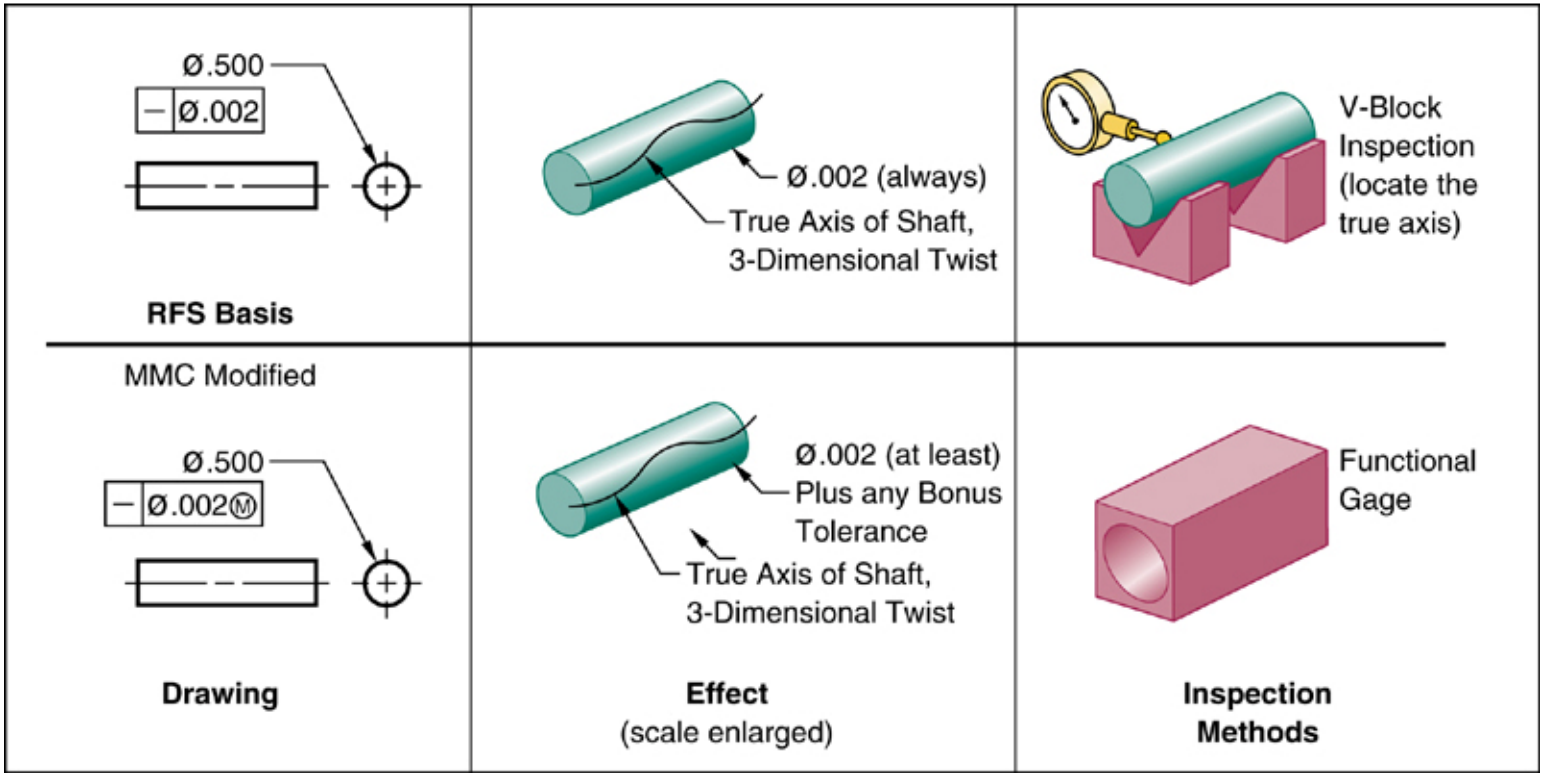
Straightness

Measures the variation of an individual feature along a straight line in a specified direction



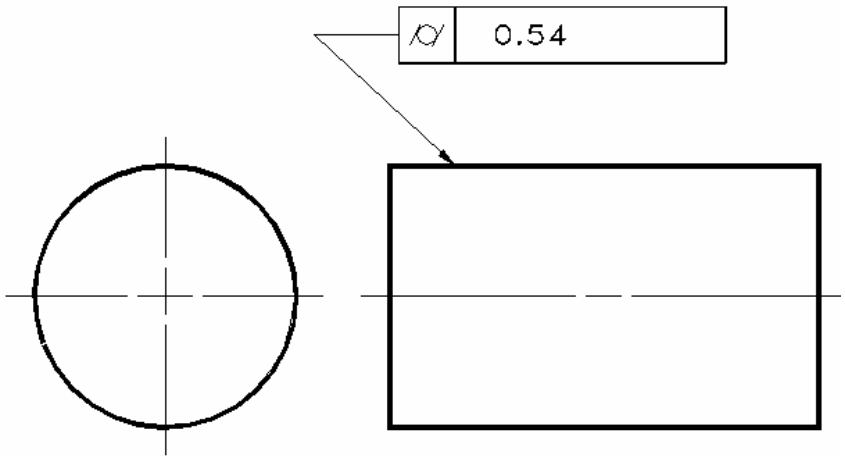


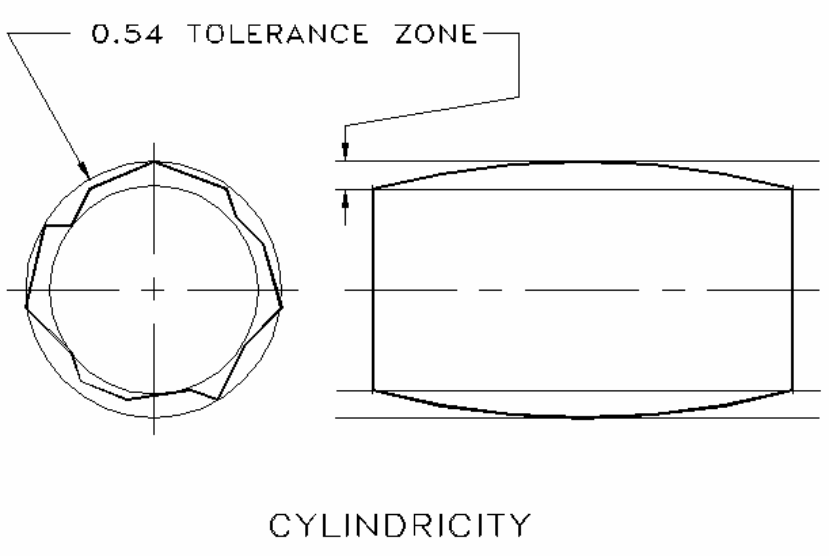
Straightness



Straightness of axis (center plane)

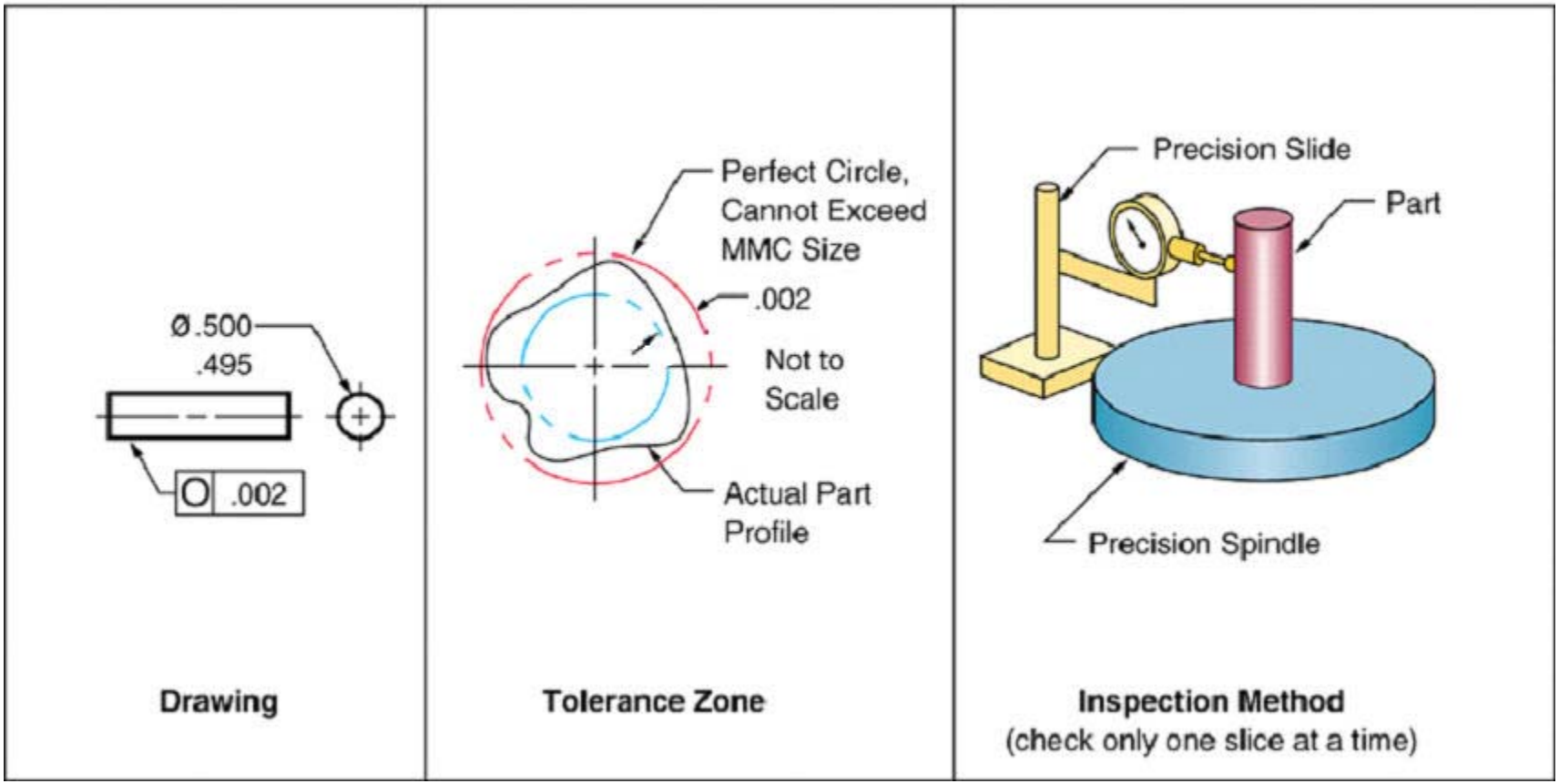
Cylindricity



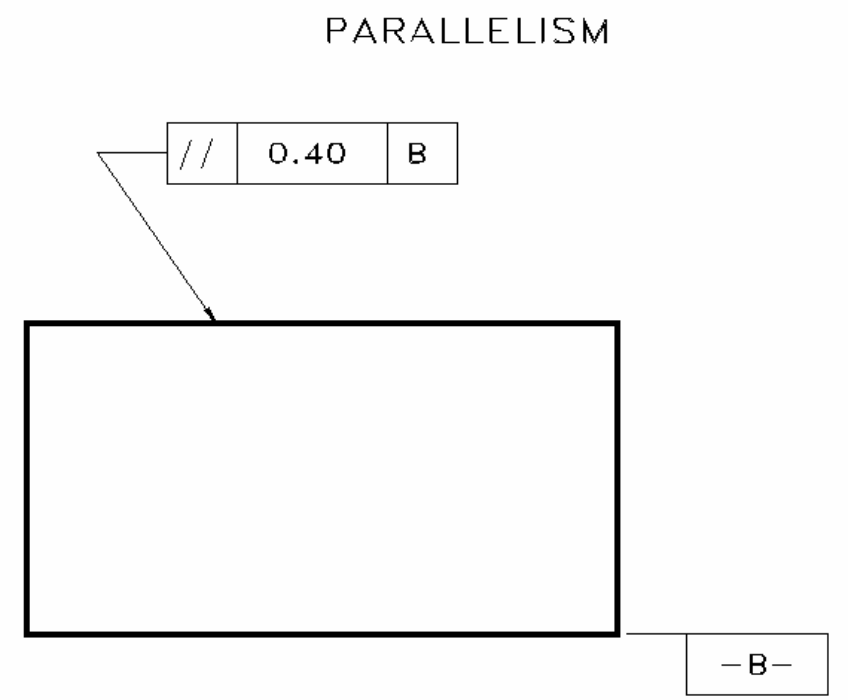


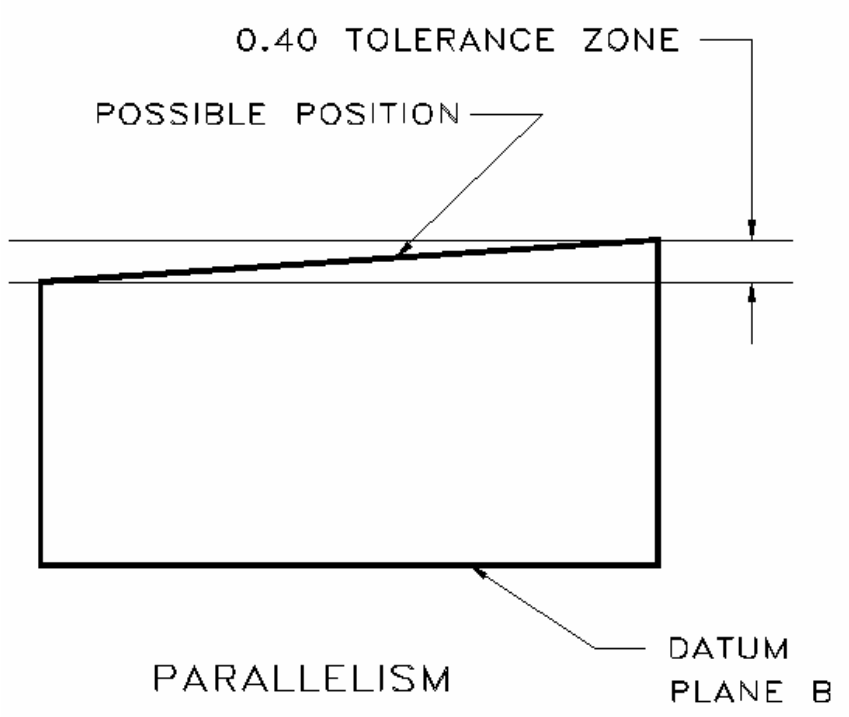
Circularity

Circularity tolerance zone is two concentric circles spaced apart a radial distance equal to the circularity tolerance value

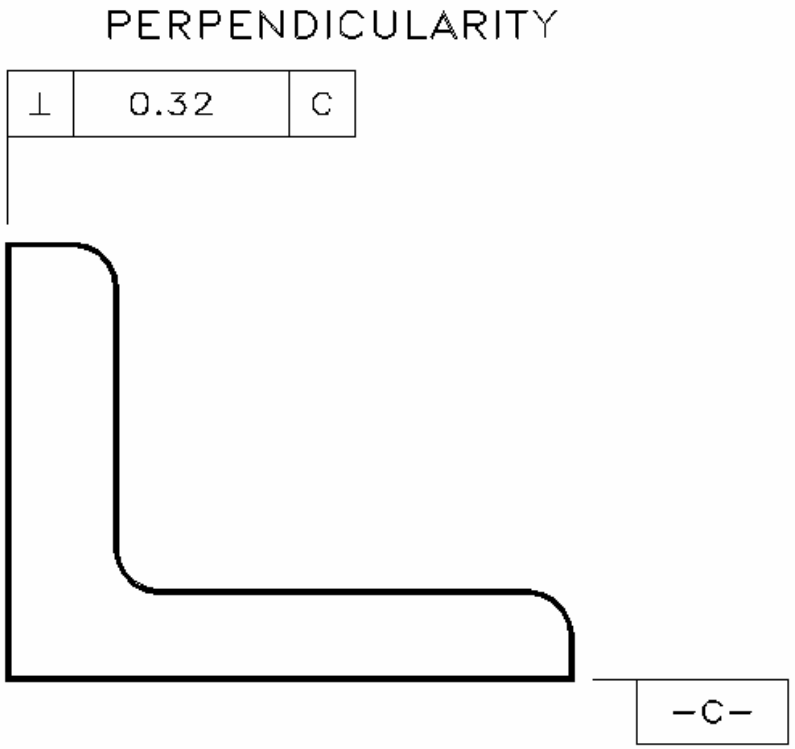


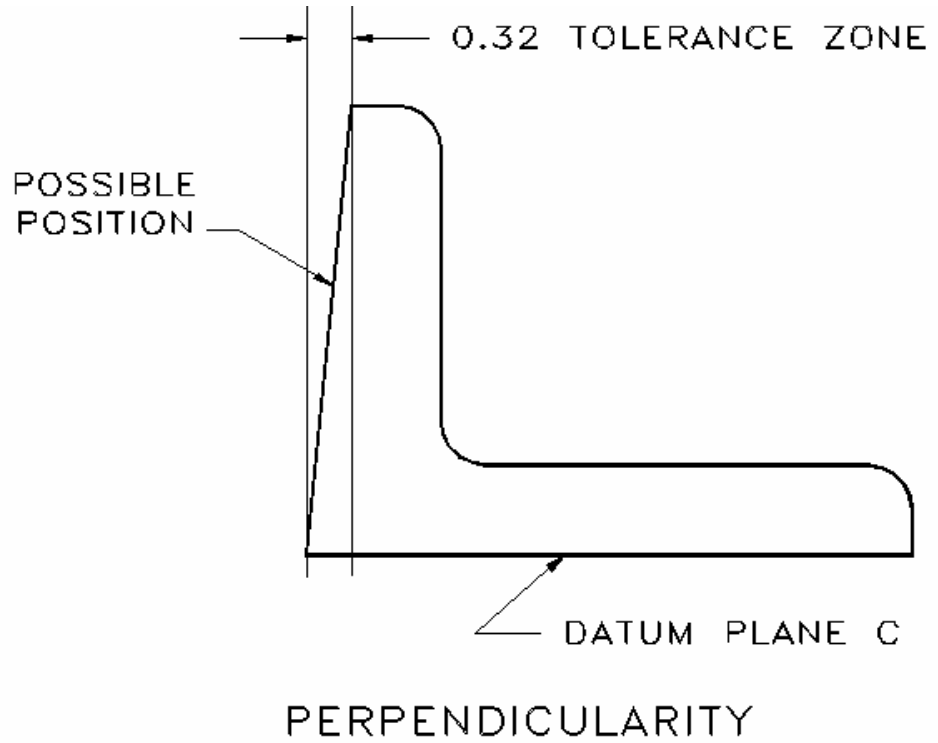
Parallelism



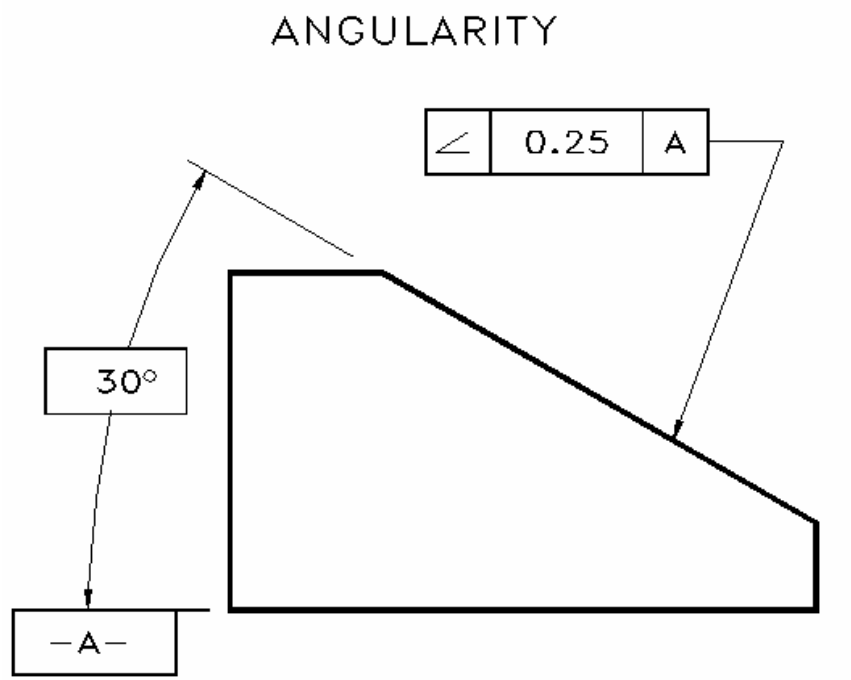


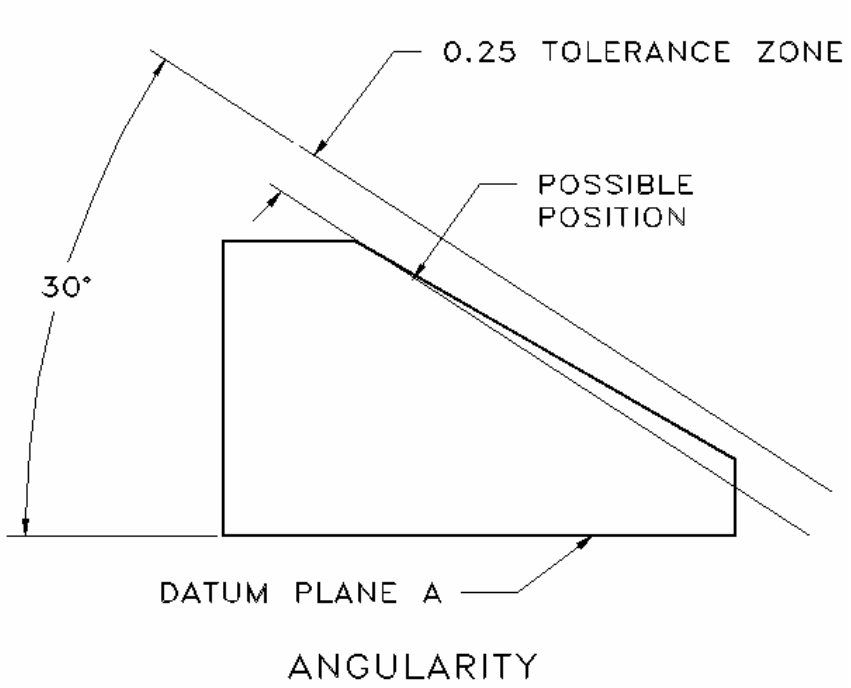
Perpendicularity





Angularity


ANGULARITY
∠ 0.25 A
30°
–A–


0.25 TOLERANCE ZONE
POSSIBLE POSITION
30°
DATUM PLANE A
ANGULARITY

Surface Roughness