Geometric Dimensioning and Tolerancing

Engineering III

GD&T

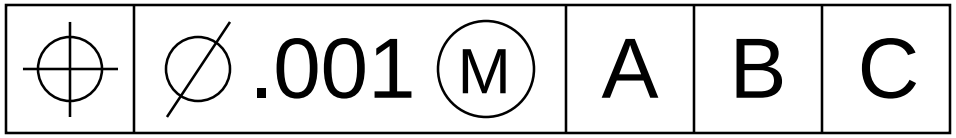
- Geometric Dimensioning & Tolerancing is an international engineering language that is used on engineering drawings to describe products in three dimensions.
- GD&T is a precise mathematical language that describes the ***form***, ***orientation*** and ***location*** of part features in zones of tolerance.

Geometric Dimensioning

- Geometric dimensioning and tolerancing is a three-dimensional, mathematically-based system.
- Within this system, features on an object are oriented or located relative to a Cartesian Coordinate System or ***Datum Reference Frame***.

Geometric Dimensioning

- ***Feature Control Frames*** are used to specify acceptable tolerance zones for the features relative to the Datum Reference Frame.



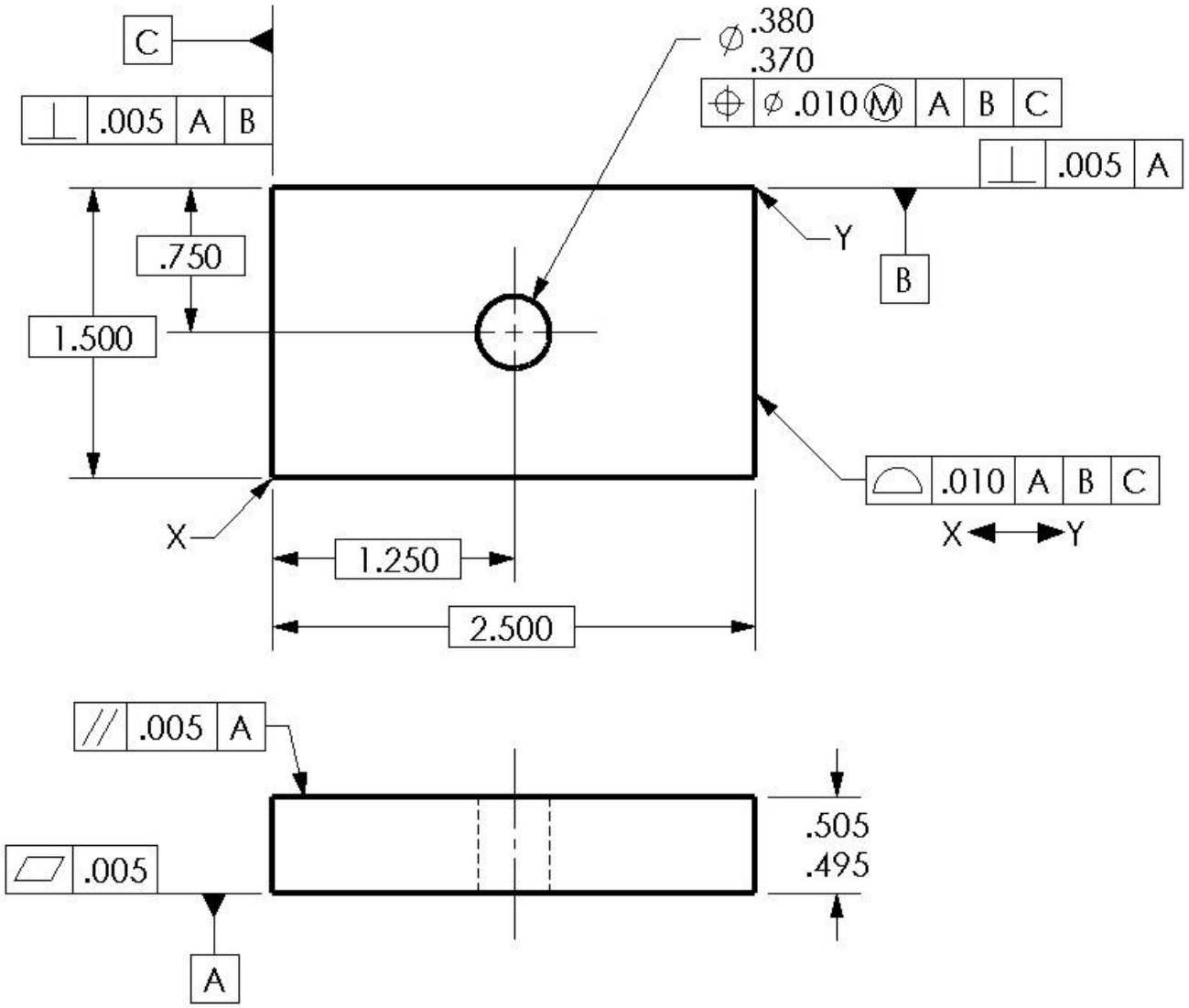
GD&T

- Like any language, it takes some time to learn it well.
- Some personnel might learn it in a conversational way.
- Others can read it but not write it.
- Others might be experts.

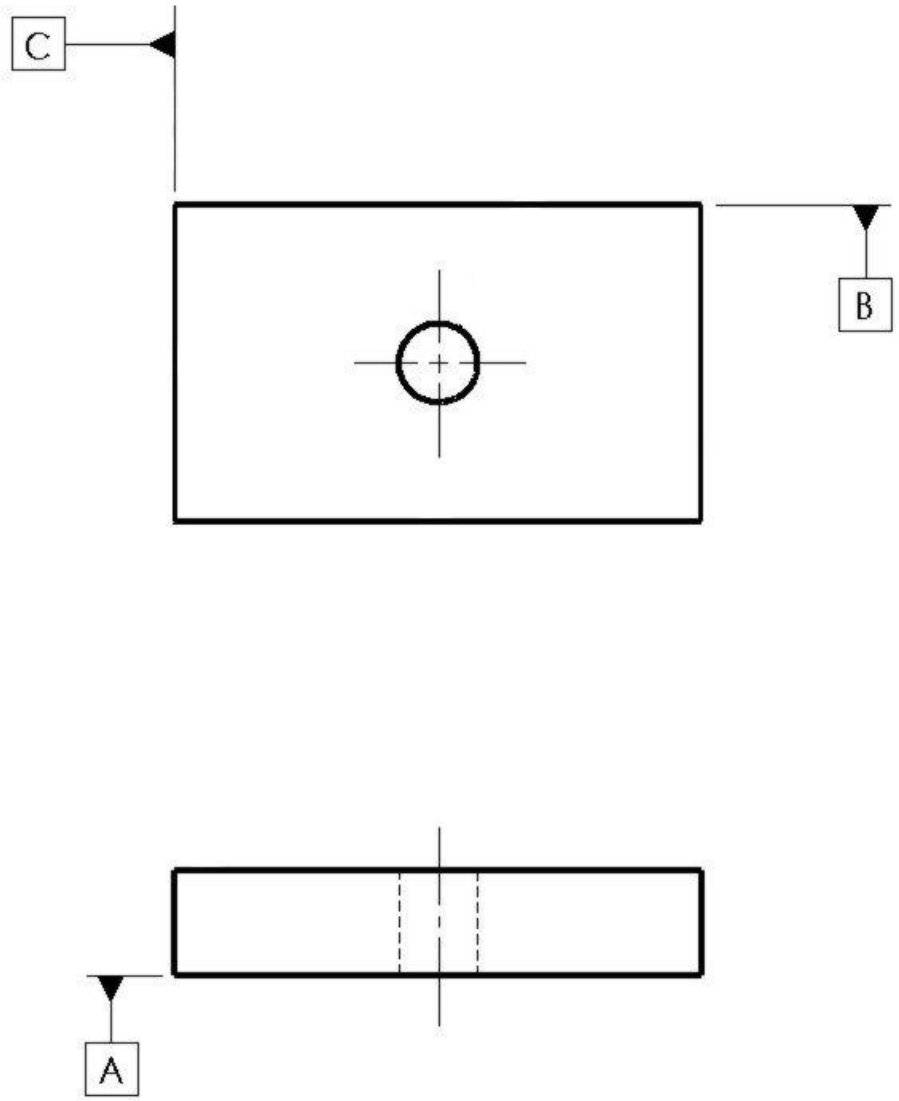
Advantages of GD&T

- Clearer intent of the designer.
- Better communication throughout the design process.
  - Better choices for manufacturing/machining.
  - Better/more accurate choices for inspection.
- Leaves almost nothing that can be interpreted more than one way.

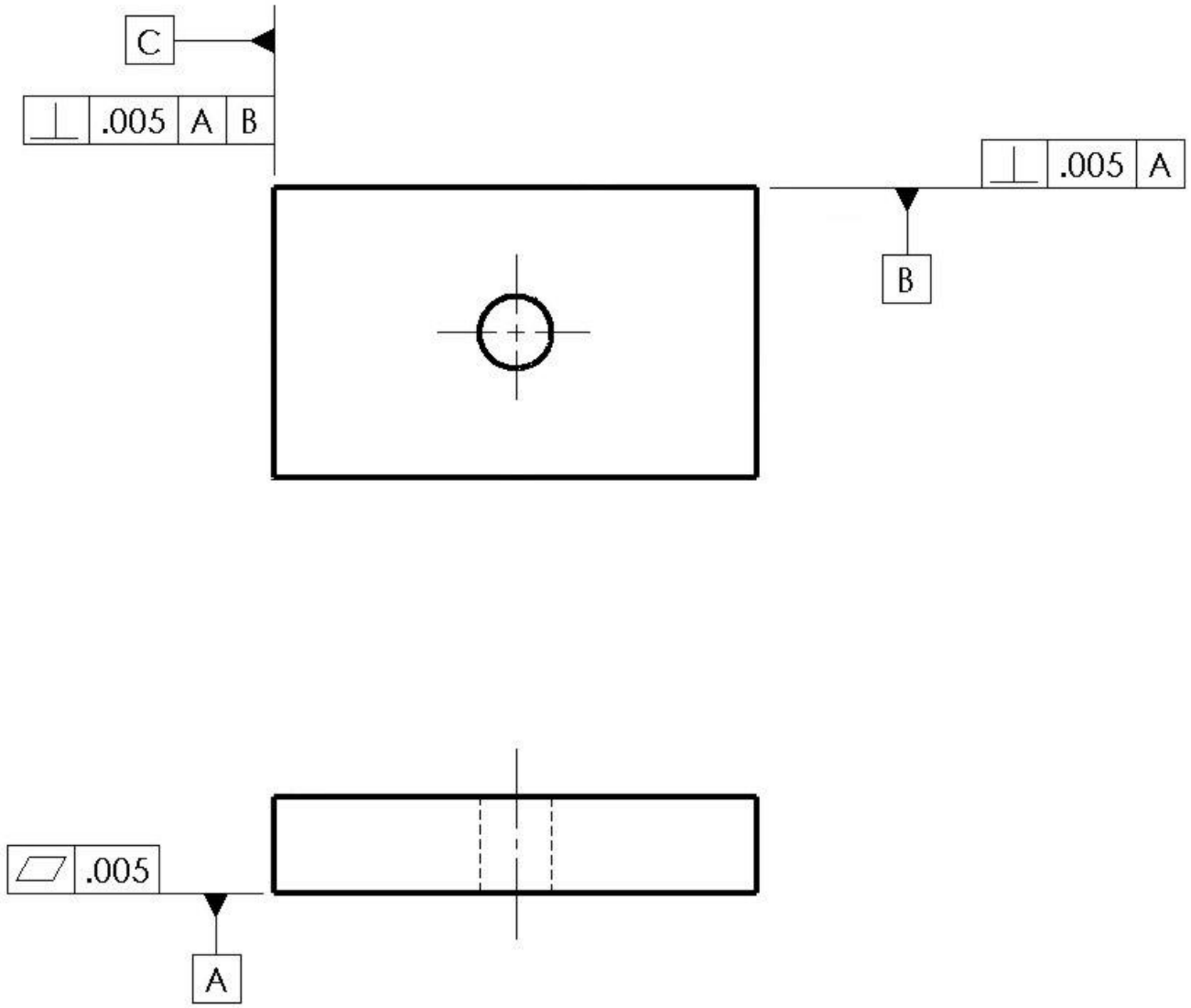
Example Drawing



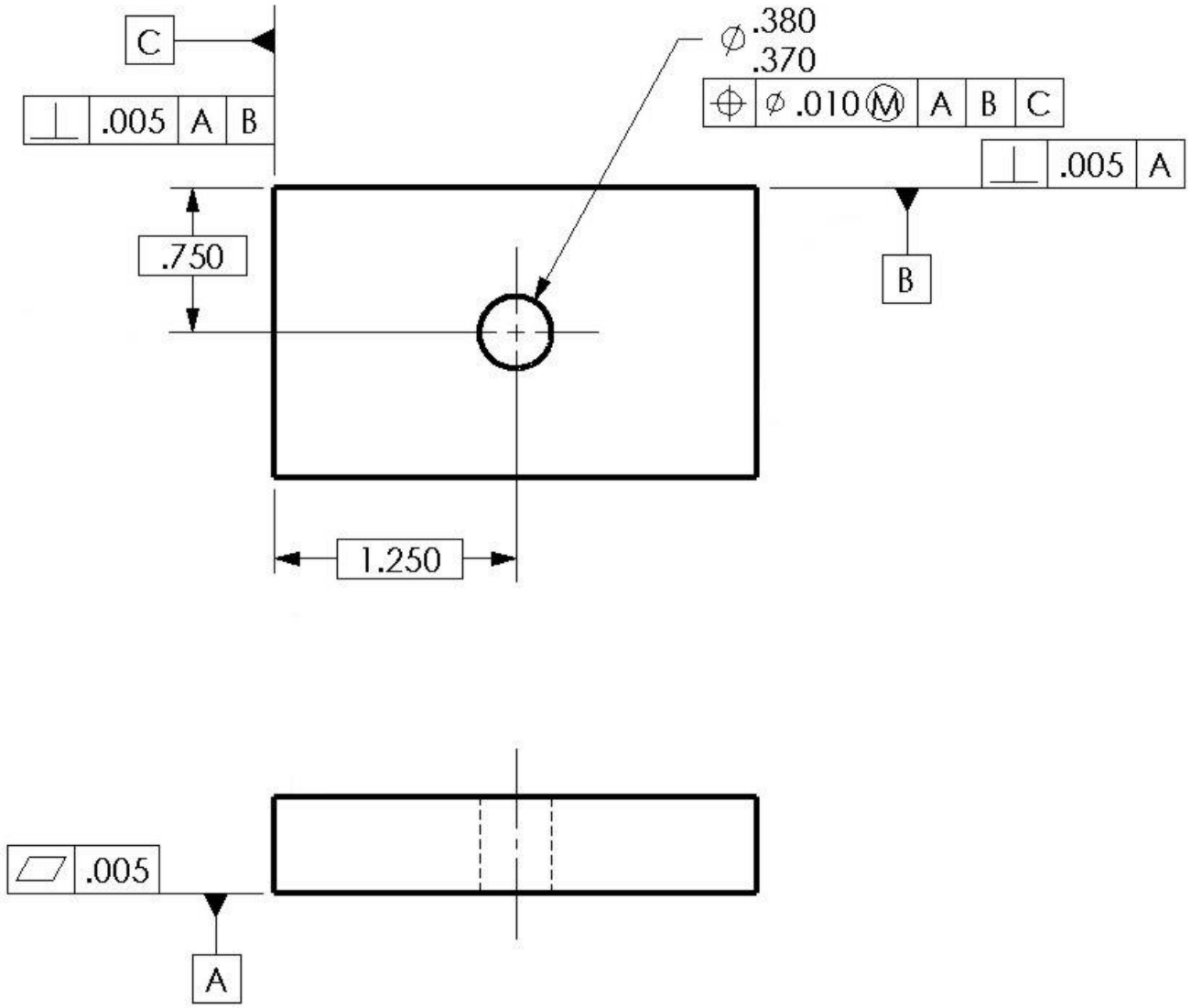
Establish Datums



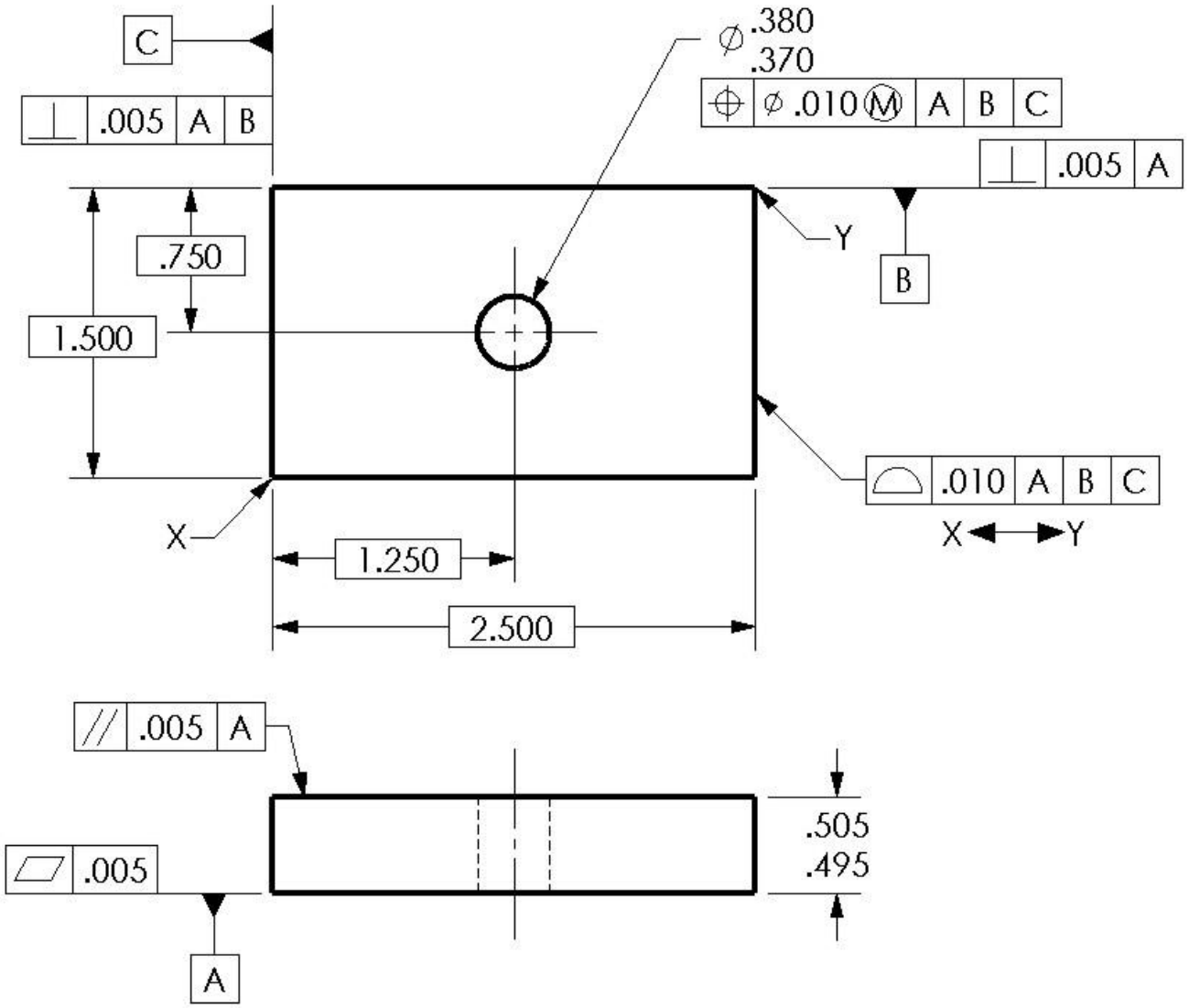
Control Datums



Position the Machined Hole

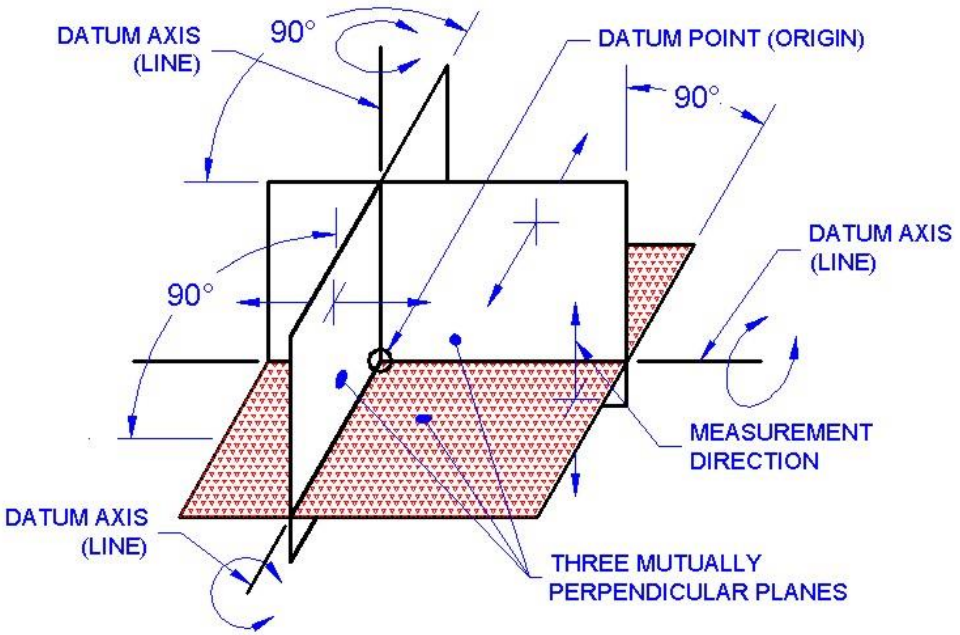


Control the Other Surfaces



Datum Reference Frame

- The DRF is made up of three mutually perpendicular planes (similar to a Cartesian Coordinate System).
- These planes exist in theory only.
- Planes are established relative to features on the actual object.



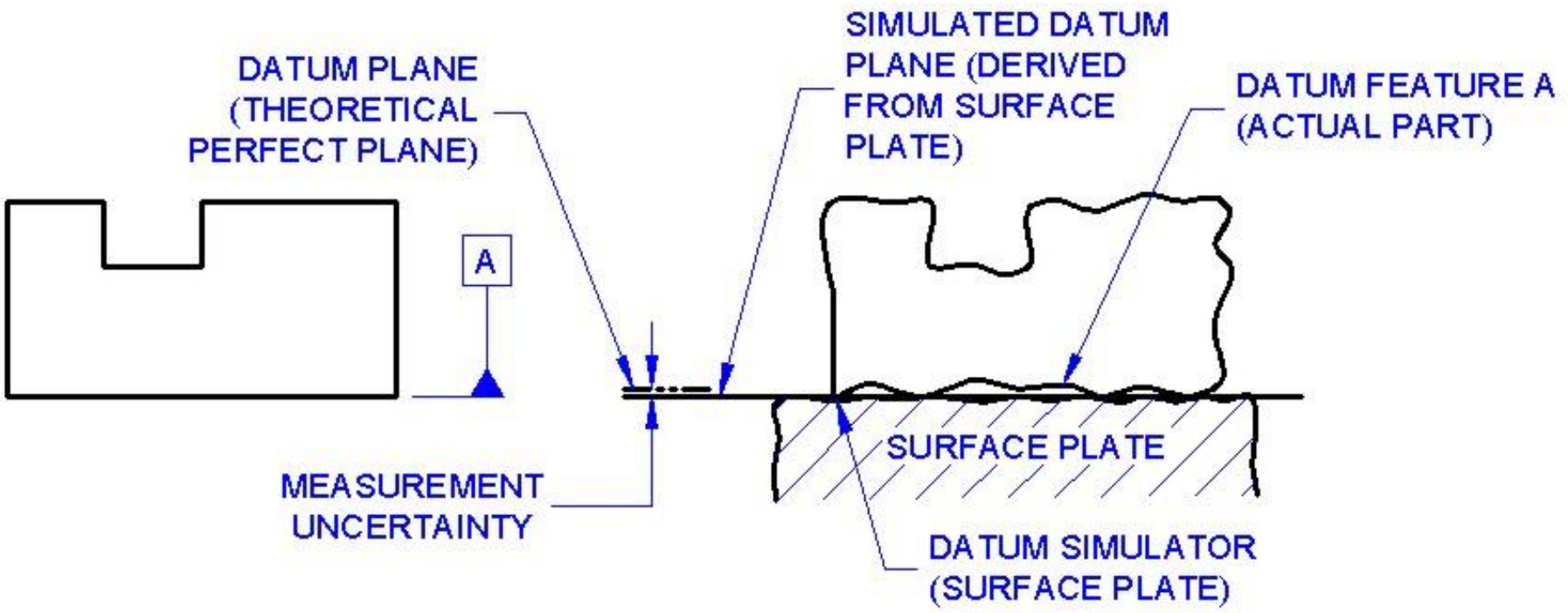
Datum Terminology

- ***Datum*** - A theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a part are established.
- ***Datum Feature*** - An actual feature of a part that is used to establish a datum.

Datum Terminology

- ***Datum Feature Symbol*** - The symbolic means of indicating a datum feature. It consists of a capital letter enclosed in a square frame and a leader line extending from the frame to the concerned feature, terminating with a triangle.
- ***Datum Feature Simulator*** - A surface of adequately precise form contacting the datum feature(s) and used to establish the simulated datum(s). Typically this surface must be at least 10 times better in quality (flatness) than the tolerances specified on the drawing

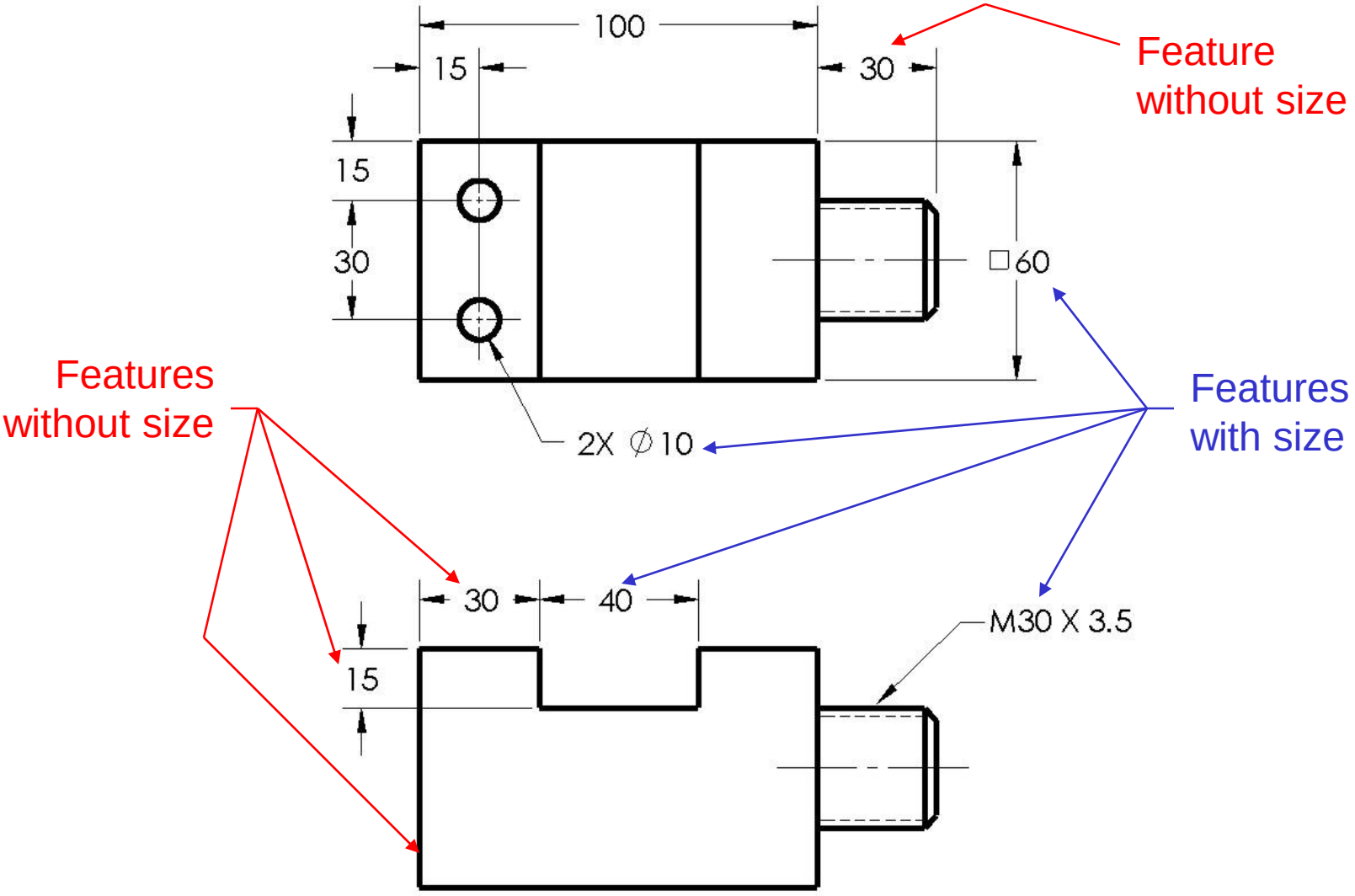
Datum Terminology



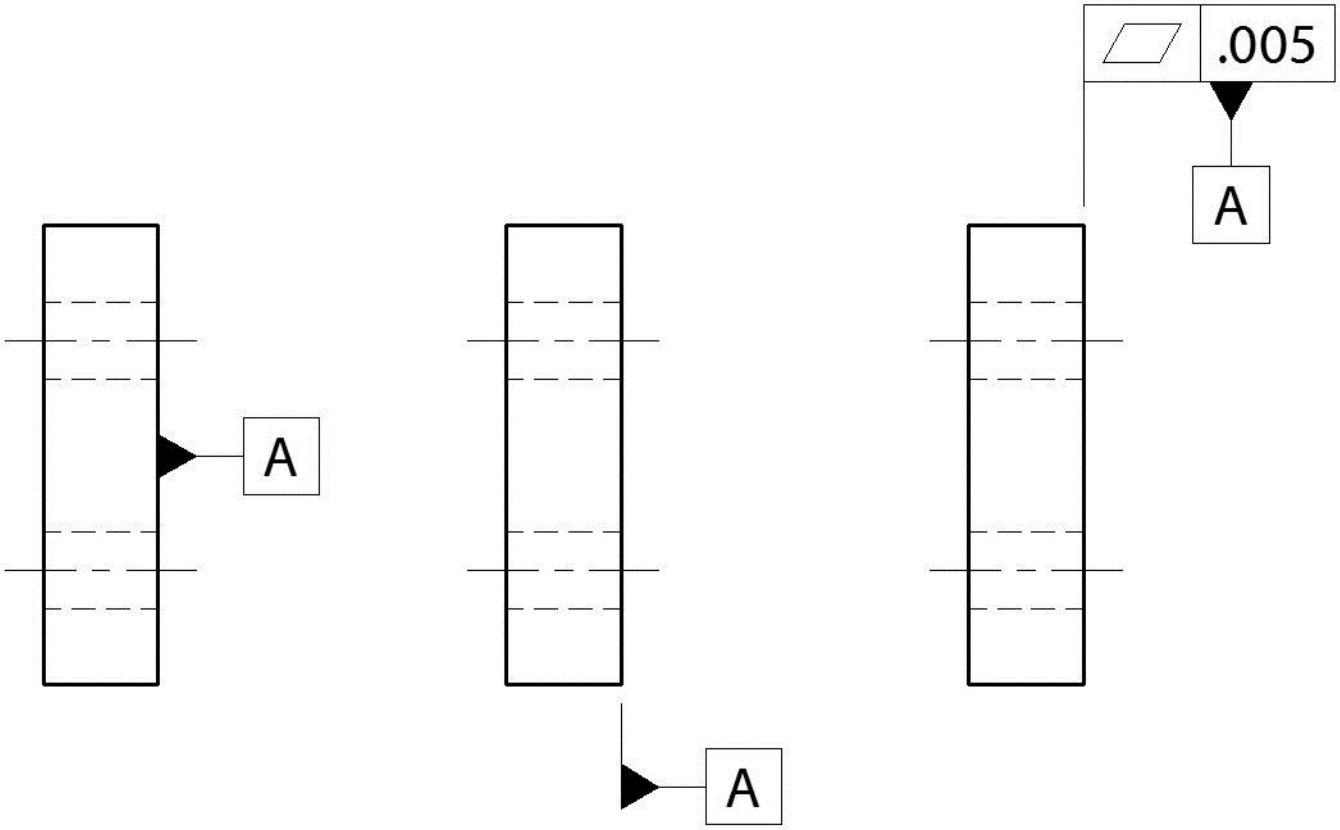
Features

- ***Feature*** - The general term applied to a physical portion of a part, such as a surface, pin, tab, hole, or slot.
- ***Feature of size (feature with size)*** - A cylindrical or spherical surface, or a set of two opposed elements or opposed parallel surfaces, associated with a size dimension.
- ***Feature without size*** - Typically this is a planar surface.

Features With and Without Size



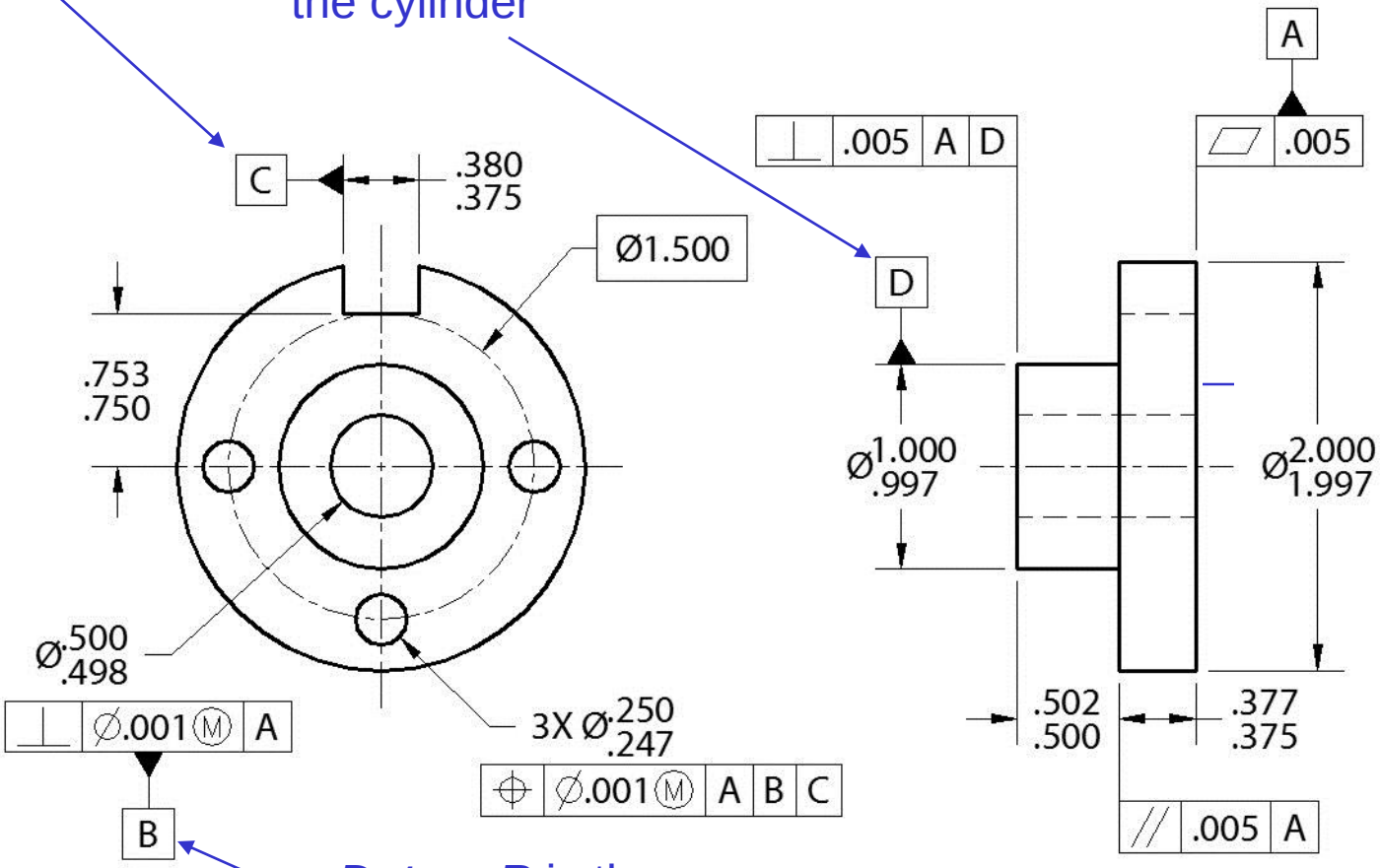
Applying Datums to a Feature without Size



Applying Datums to a Feature with Size

Datum C is the median plane of the slot

Datum D is the axis of the cylinder



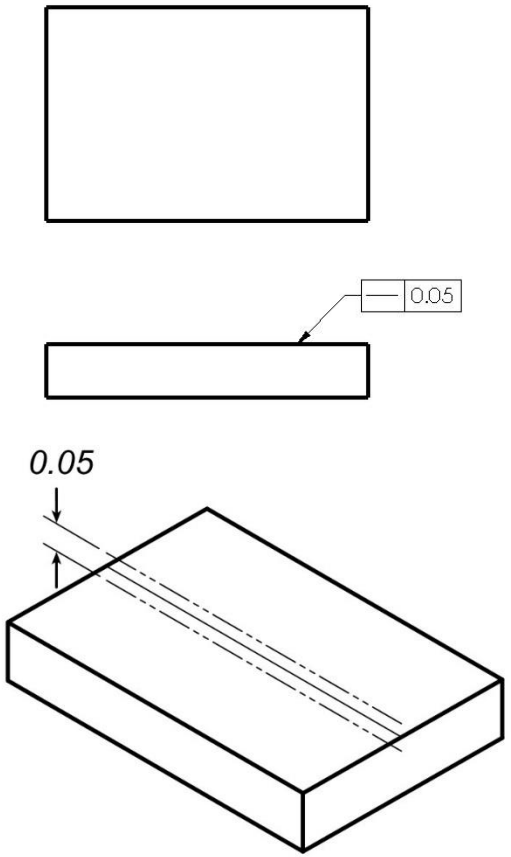
Datum B is the axis of the hole

Geometric Characteristic Symbols

	TYPE OF TOLERANCE	CHARACTERISTIC	SYM
FOR INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	
		FLATNESS	
		CIRCULARITY	
		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	
		SYMMETRY	
	RUNOUT	CIRCULAR RUNOUT	
		TOTAL RUNOUT	

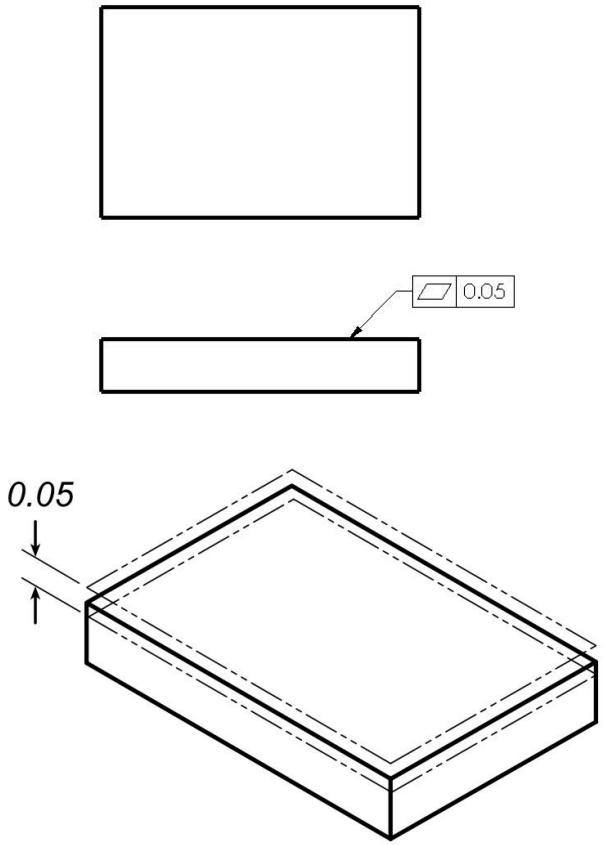
Form Tolerances - Straightness

- Each longitudinal element on the surface must lie between two parallel lines 0.05 apart.
- The shape of the tolerance zone is a *2D area* between two parallel lines.



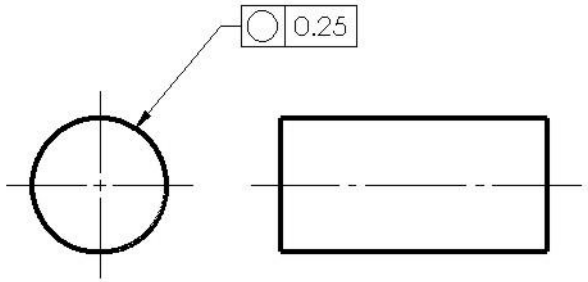
Form Tolerances - Flatness

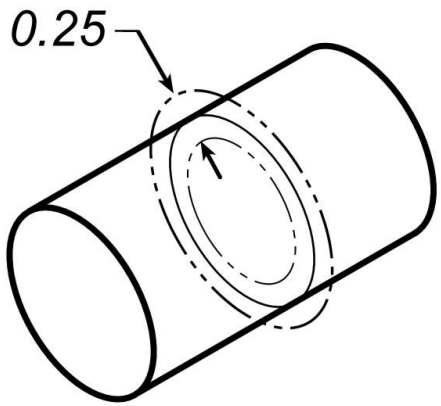
- All points on the surface must lie between two parallel planes 0.05 apart.
- The shape of the tolerance zone is a *3D area* between two parallel planes.



Form Tolerances - Circularity

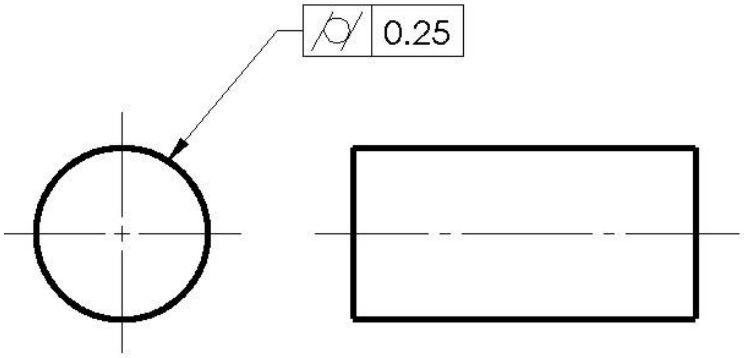
- All points on the surface must lie between two concentric circles 0.05 apart.
- The shape of the tolerance zone is a *2D area* between two concentric circles.

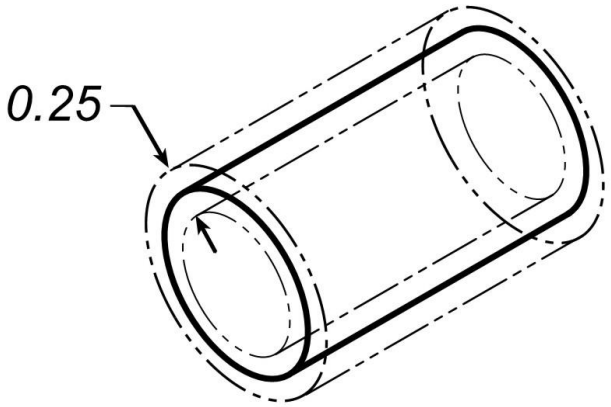




Form Tolerances - Cylindricity

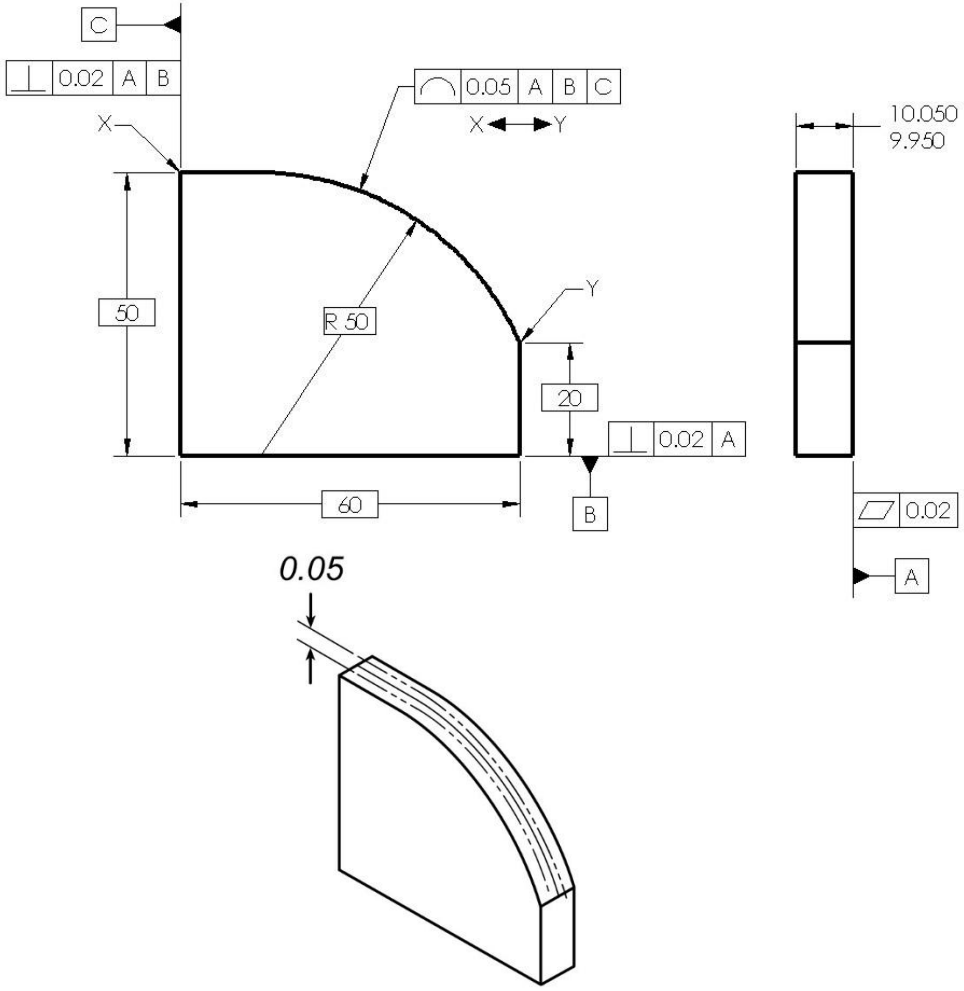
- All points on the surface must lie between two concentric cylinders 0.05 apart.
- The shape of the tolerance zone is a *3D area* between two concentric cylinders.





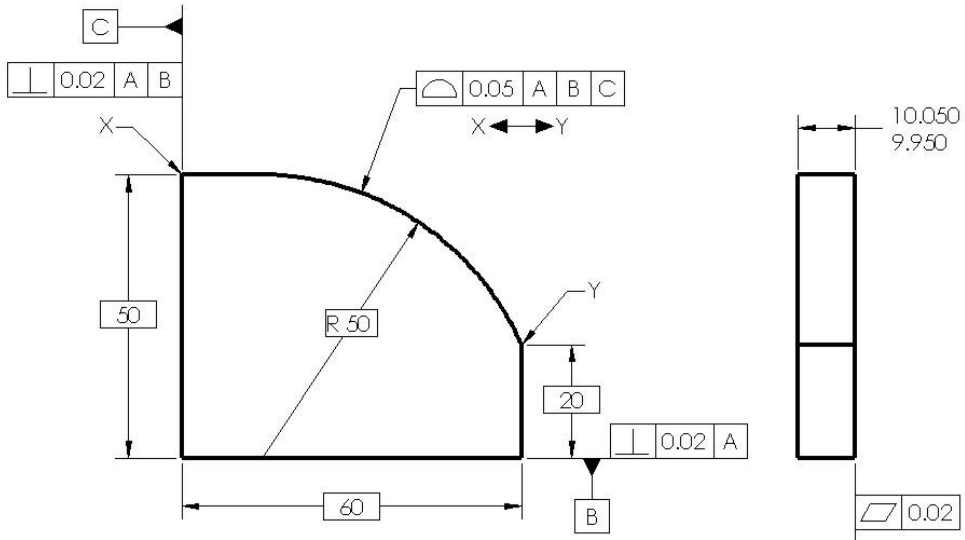
Profile of a Line Tolerance

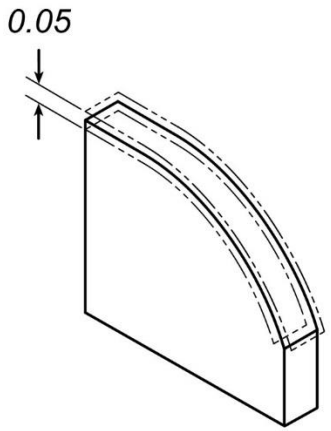
- Each point on the specified path must lie between two parallel contours 0.05 apart (0.025 on each side of path).
- The shape of the tolerance zone is a *2D area* between the two contours.
- Perfect geometry is located with basic dimensions.



Profile of a Surface Tolerance

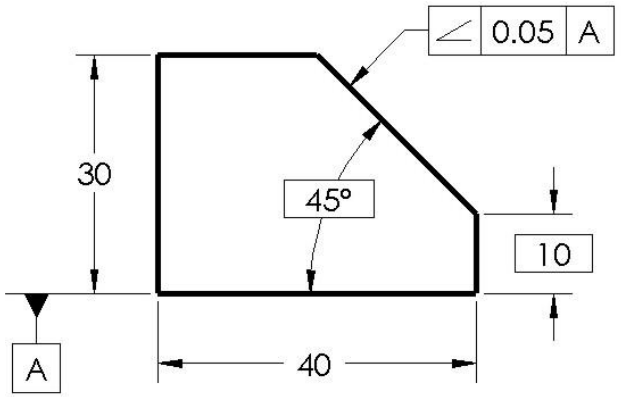
- Each point on the surface must lie between two parallel/ concentric contours 0.05 apart (0.025 on each side of surface).
- The shape of the tolerance zone is a *3D area* between the two contours.
- Perfect geometry is located with basic dimensions.

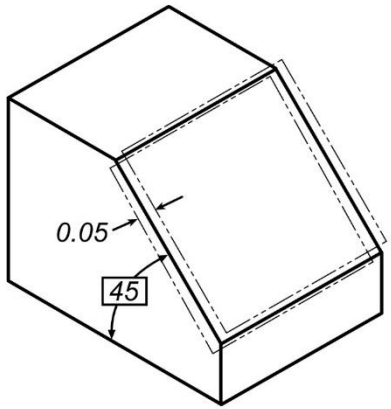




Orientation Tolerances - Angularity

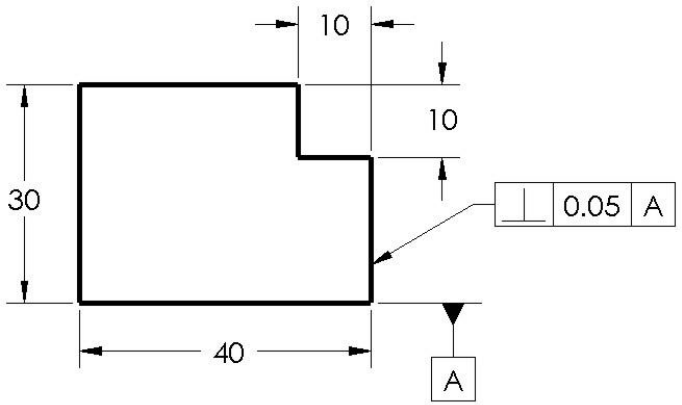
- All points on the surface must lie between two parallel planes 0.05 apart.
- Perfect geometry is located using basic dimensions.
- The shape of the tolerance zone is a *3D area* between two parallel planes.

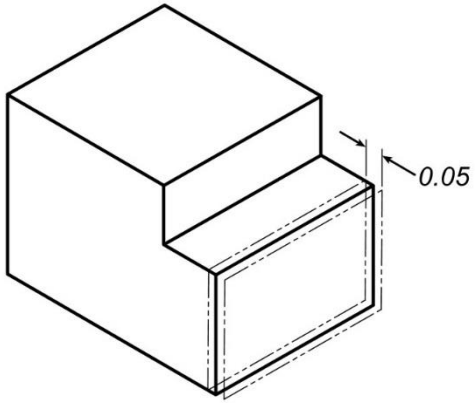




Orientation Tolerances - Perpendicularity

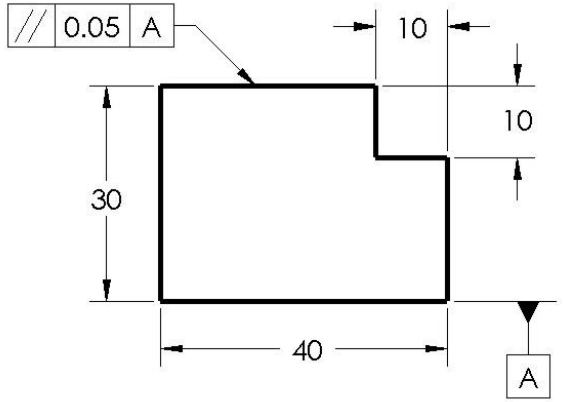
- All points on the surface must lie between two parallel planes 0.05 apart.
- The shape of the tolerance zone is a *3D area* between two parallel planes.

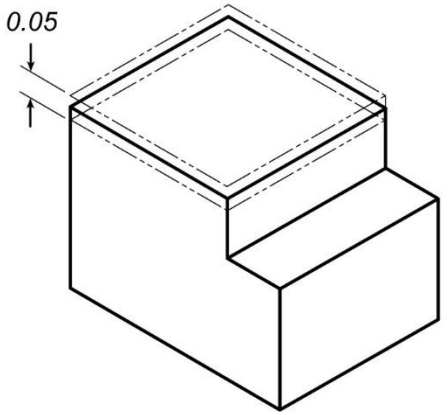




Orientation Tolerances - Parallelism

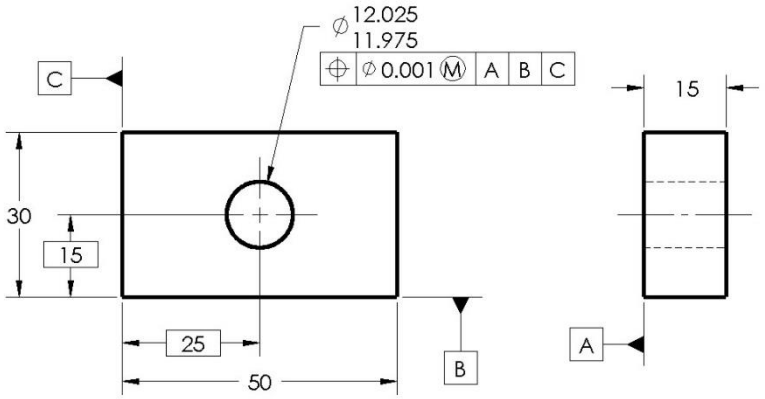
- All points on the surface must lie between two parallel planes 0.05 apart.
- The shape of the tolerance zone is a *3D area* between two parallel planes.

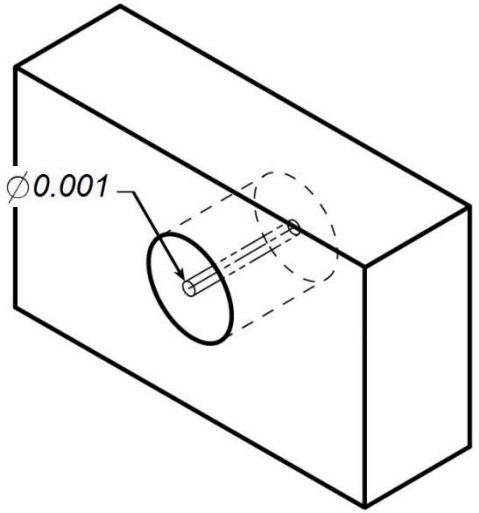




Location Tolerances - Position

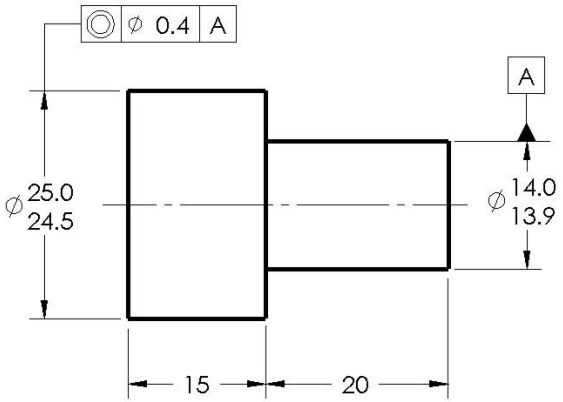
- All points on the axis must lie within a cylinder with a diameter of 0.001 at maximum material condition.
- The cylinder is located with basic dimensions from the datums.
- The shape of the tolerance zone is a *3D area* within the cylinder.

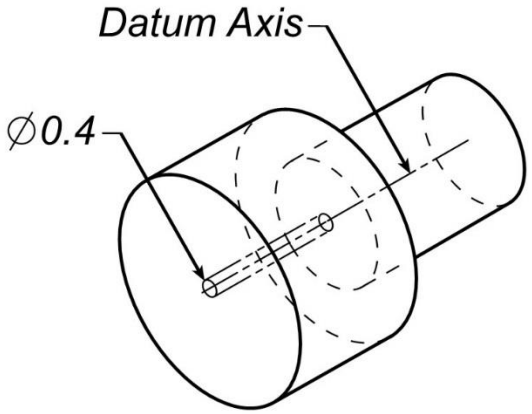




Location Tolerances - Concentricity

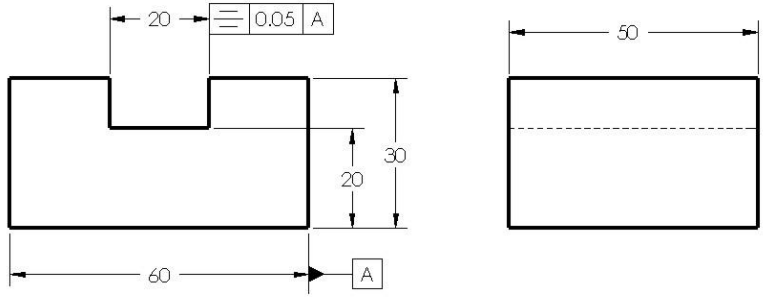
- All points on the axis must lie within a cylinder with a diameter of 0.4 relative to the datum axis.
- The shape of the tolerance zone is a *3D area* within the cylinder.

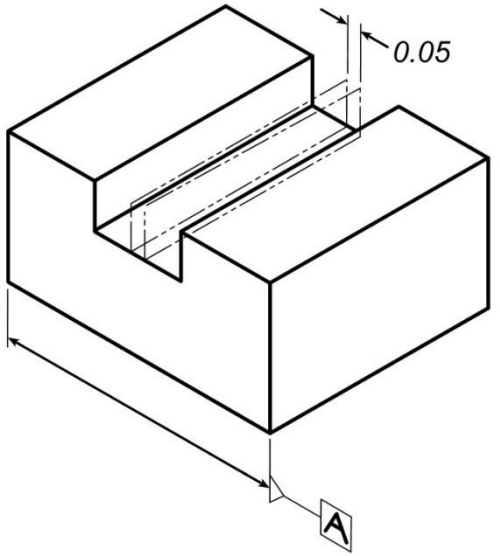




Location Tolerances - Symmetry

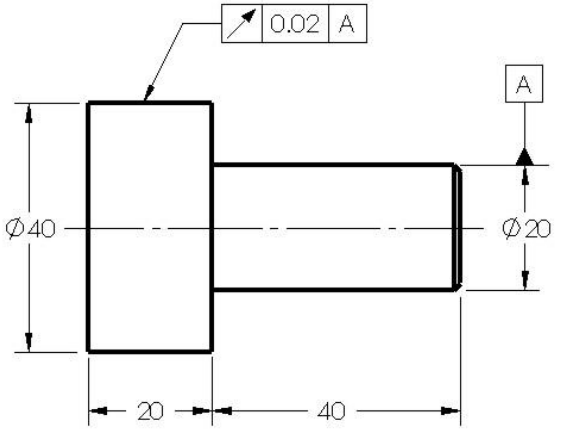
- All points on the feature's median plane must lie between two parallel planes defined by the datum's median plane.
- The shape of the tolerance zone is a *3D area* between the two planes.

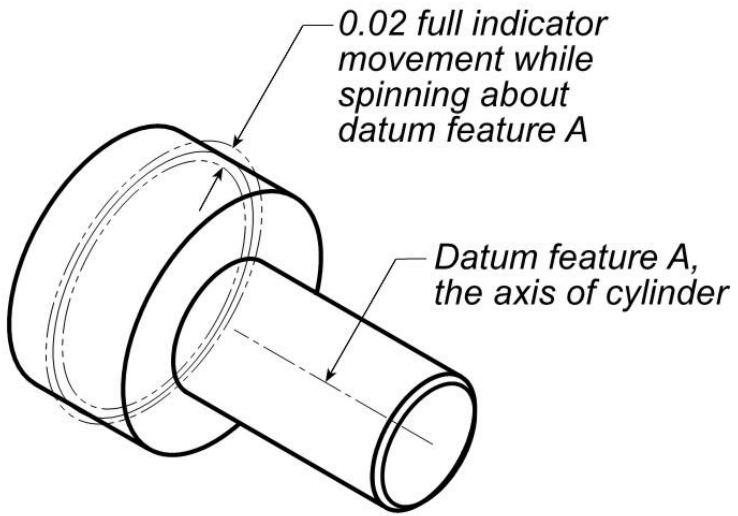




Runout Tolerances - Circular

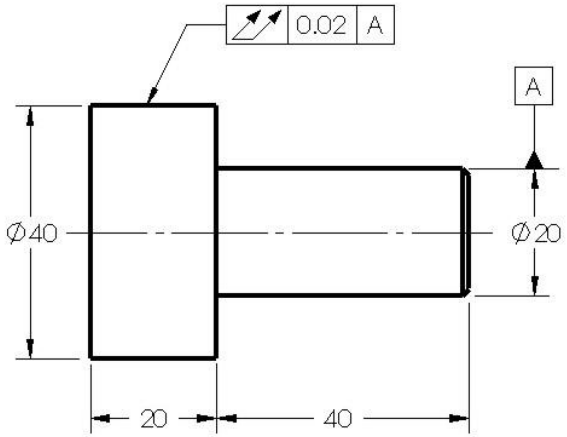
- All points on the surface must lie between two concentric circles 0.02 apart relative to the datum feature.
- The shape of the tolerance zone is the *2D area* between the two concentric circles.

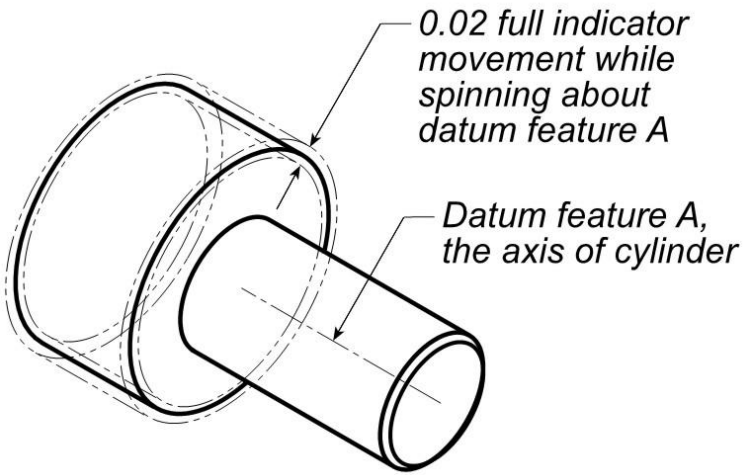




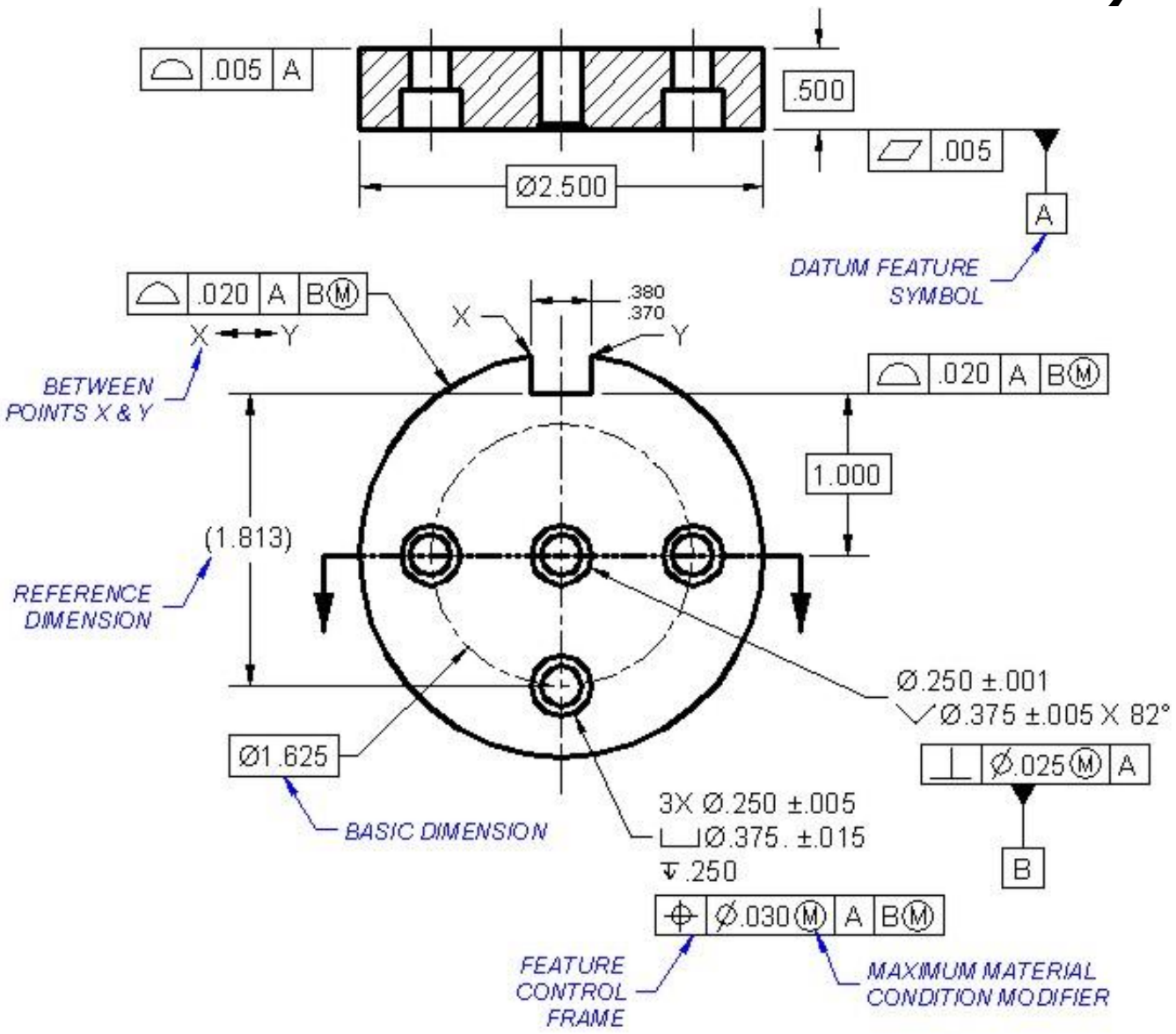
Runout Tolerances - Total

- All points on the surface must lie between two concentric cylinders 0.02 apart relative to the datum feature.
- The shape of the tolerance zone is the *3D area* between the two concentric cylinders.





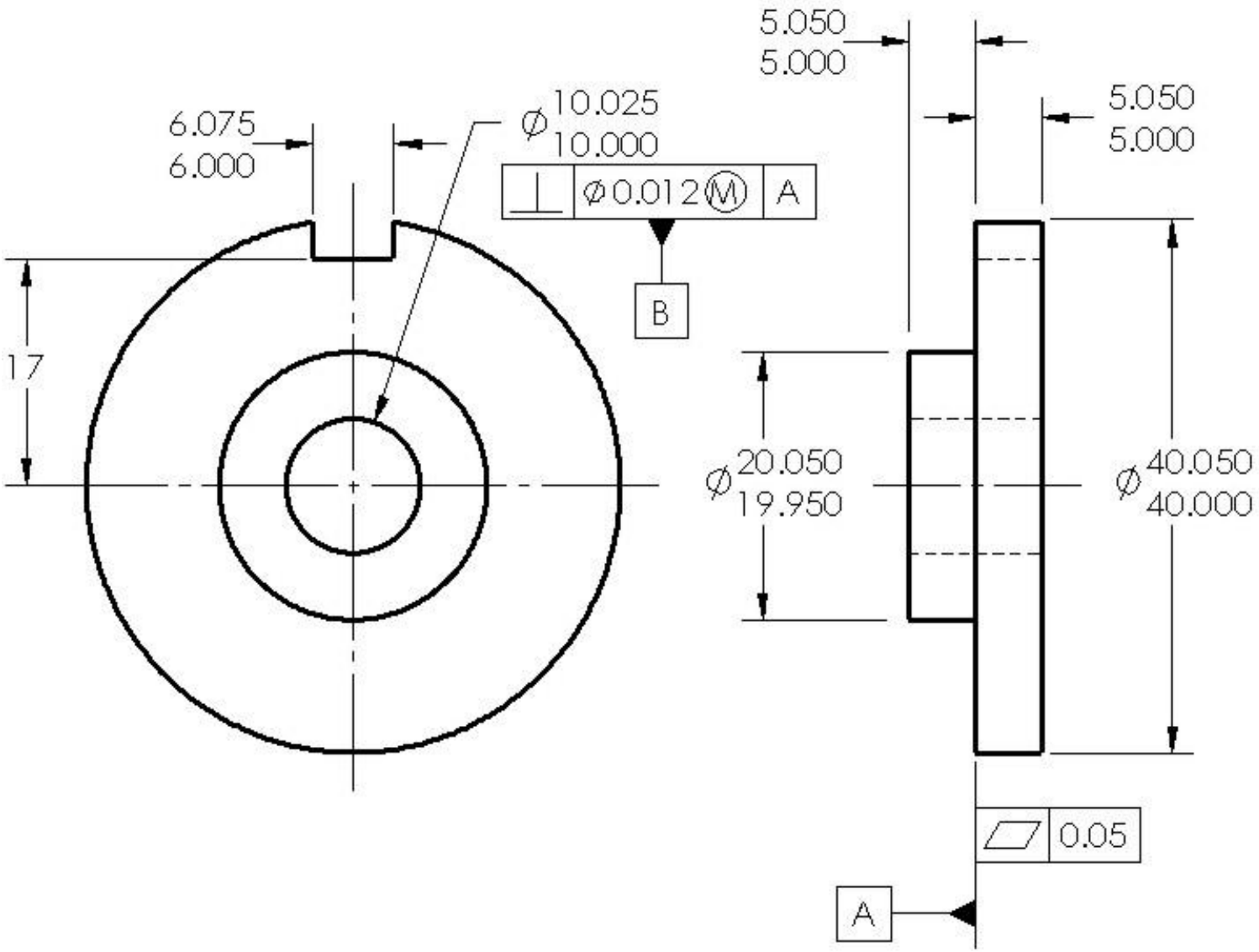
Symbols


.005 A
.500
Ø2.500
.005
A
DATUM FEATURE SYMBOL
.020 A BⓂ
X Y
BETWEEN POINTS X & Y
.380
.370
X
Y
.020 A BⓂ
1.000
(1.813)
REFERENCE DIMENSION
Ø.250 ±.001
Ø.375 ±.005 X 82°
Ø.025Ⓜ A
B
Ø1.625
BASIC DIMENSION
3X Ø.250 ±.005
Ø.375 . ±.015
↧ .250
Ø.030Ⓜ A BⓂ
FEATURE CONTROL FRAME
MAXIMUM MATERIAL CONDITION MODIFIER

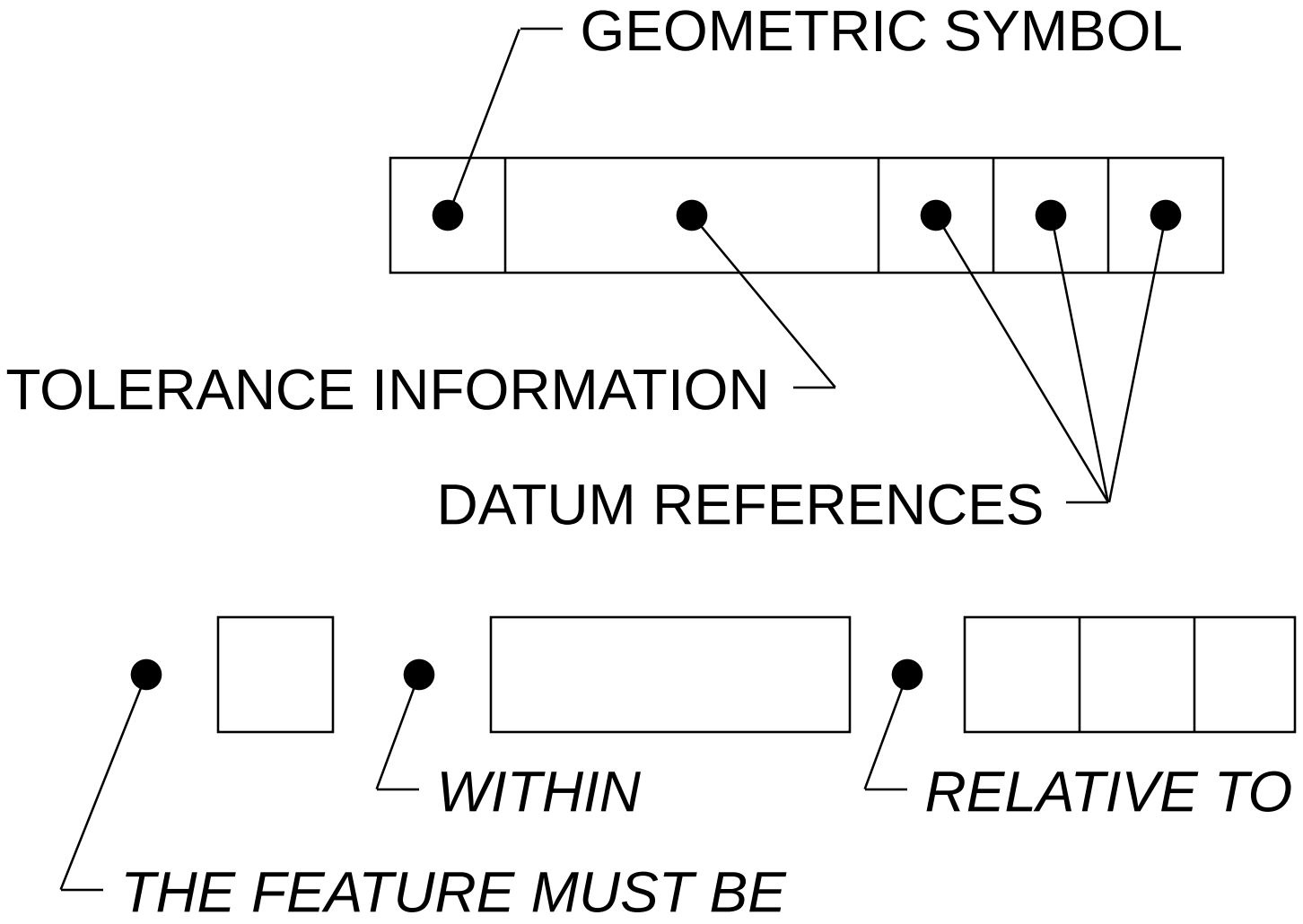
Material Condition Modifiers

- ***Maximum Material Condition*** – The geometric tolerance applies only at the feature's maximum material within the stated limits of size.
- ***Least Material Condition*** – The geometric tolerance applies only at the feature's least material within the stated limits of size.
- ***Regardless of Feature Size*** – The geometric tolerance applies at any increment of size of the feature within its size tolerance.

MMC & LMC


6.075
6.000
Ø10.025
10.000
⊥ Ø0.012 Ⓜ A
B
17
5.050
5.000
5.050
5.000
Ø20.050
19.950
Ø40.050
40.000
0.05
A

Feature Control Frame

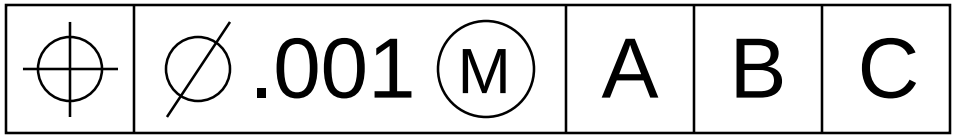


Feature Control Frame

//	0.05	A

- The feature must be ***parallel***
- within a ***five-hundredths of a millimeter*** tolerance zone
- relative to datum feature ***A***

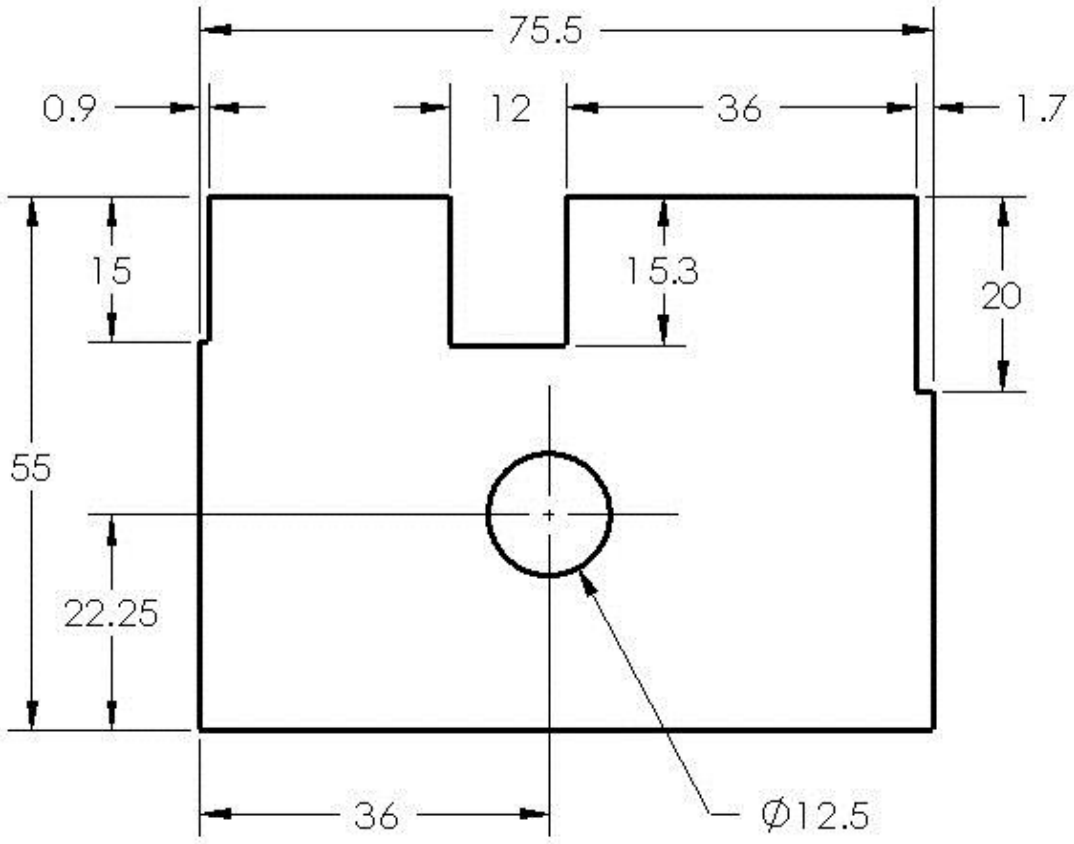
Feature Control Frame



- The feature must be ***positioned***
- within a ***one-thousandth of an inch, cylindrical*** tolerance zone ***at maximum material condition***
- relative to ***primary datum feature A*****, *secondary datum feature B*****, and *tertiary datum feature C***.

Standards for Millimeters

- Show leading zeros for values less than 1 unit
- Do not show trailing zeros



Standards for Inches

- Do not show leading zeros for values less than 1 unit
- Show trailing zeros equal to the precision of the drawing