Engineering Communication

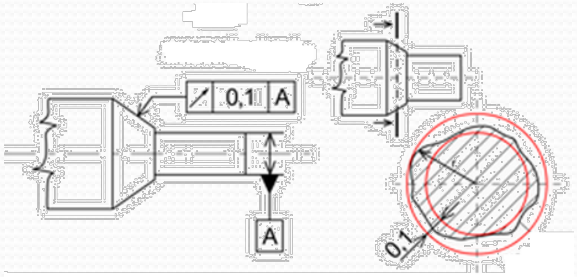
2012.03.13.

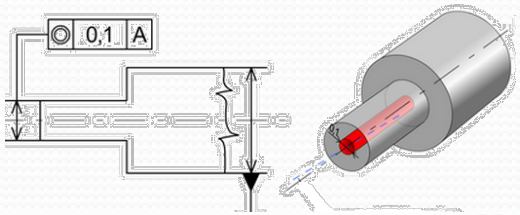
Geometric Dimensioning and Tolerancing - Part II

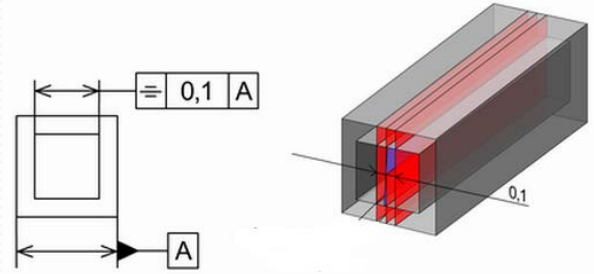
Categories of Dimensioning

Features requiring datum reference

- Orientation
  - Perpendicularity
  - Angularity
  - Parallelism
- Runout
  - Circular Runout
  - Total Runout
- Location
  - Position
  - Concentricity
  - Symmetry



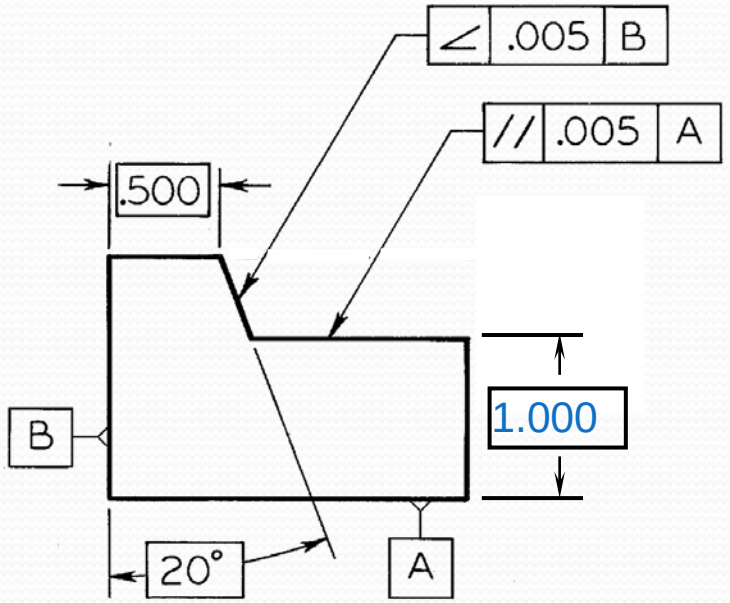




Geometric Dimensioning & Tolerancing (GD&T)

Datums

- Datums are features (points, axis, and planes) on the object that are used as reference surfaces from which other measurements are made.
- They are used in *designing, tooling, manufacturing, inspecting*, and *assembling* components and sub-assemblies.



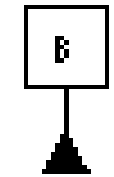
Geometric Dimensioning & Tolerancing (GD&T)

Datums

- Features are identified with respect to a datum.
- Always start with the letter **A**
- Do not use letters I, O, or Q
- May use double letters AA, BB, etc.
- This information is located in the feature control frame.



- Datums on a drawing of a part are represented using the symbol seen below.



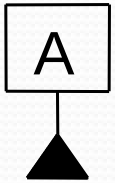
Geometric Dimensioning & Tolerancing (GD&T)

Datum reference symbols

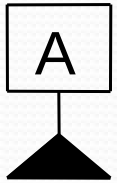
- The datum feature symbol identifies a surface or feature of size as a datum.



ANSI
1982



ASME
1994

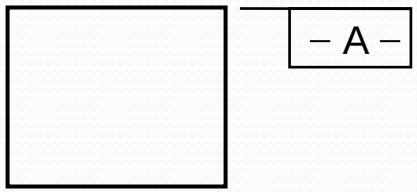


ISO

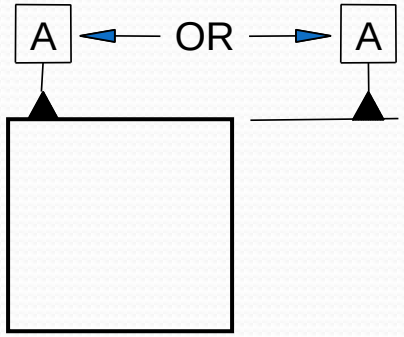
Geometric Dimensioning & Tolerancing (GD&T)

Placement of datums

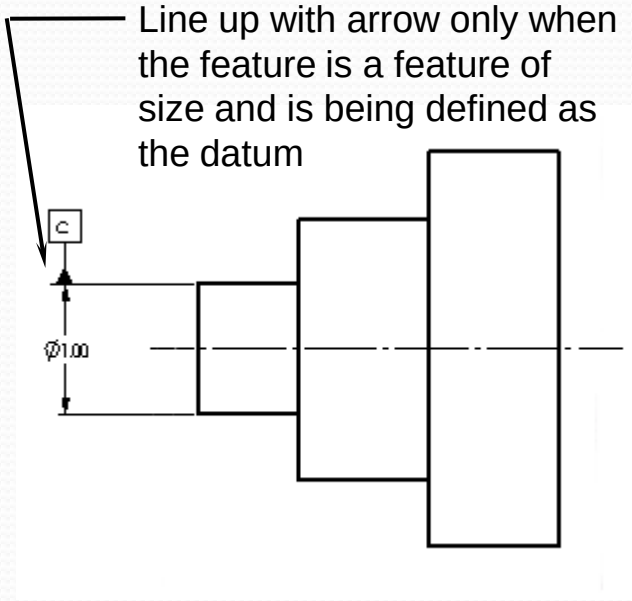
- Datums are generally placed on a feature, a centerline, or a plane depending on how dimensions need to be referenced.



ANSI 1982



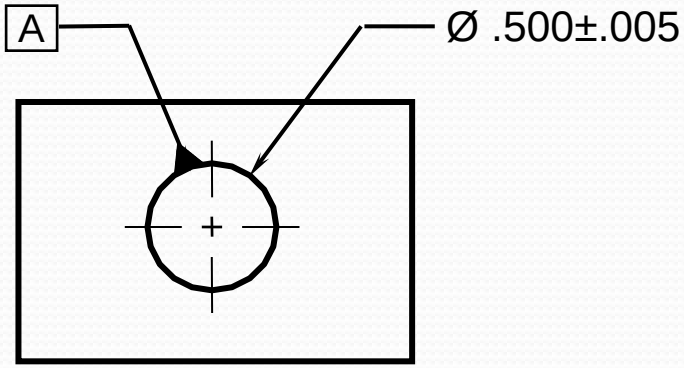
ASME 1994



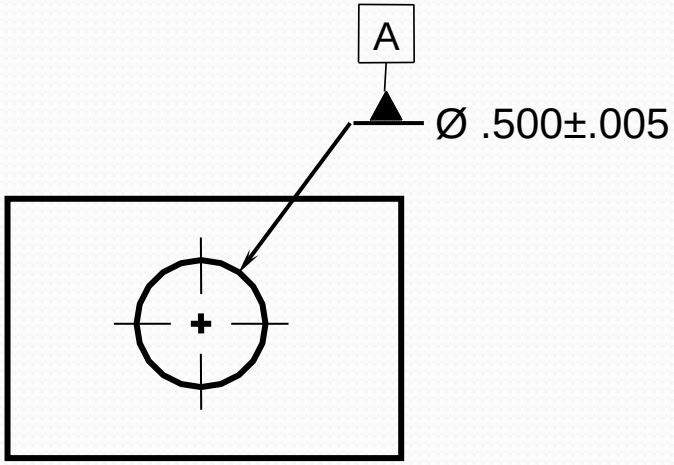
Geometric Dimensioning & Tolerancing (GD&T)

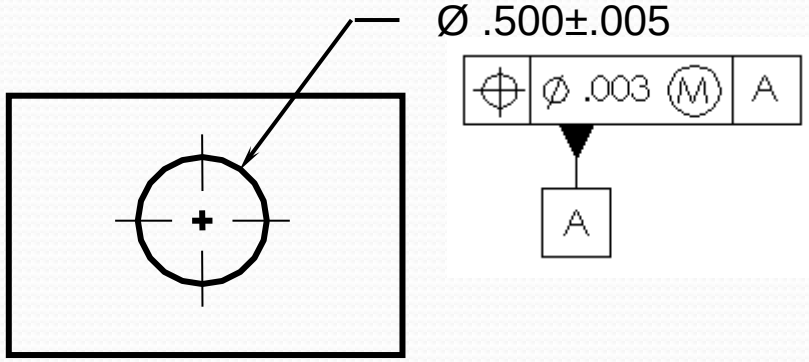
Placement of datums

- Feature sizes, such as holes



- Sometimes a feature has a GD&T and is also a datum



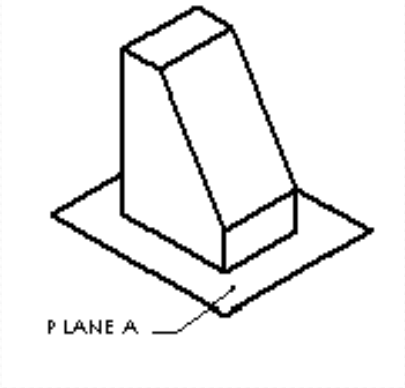


Geometric Dimensioning & Tolerancing (GD&T)

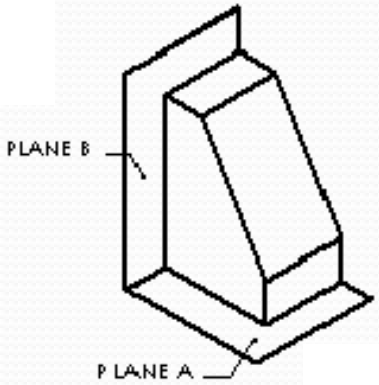
Example Datums

- Datums must be perpendicular to each other

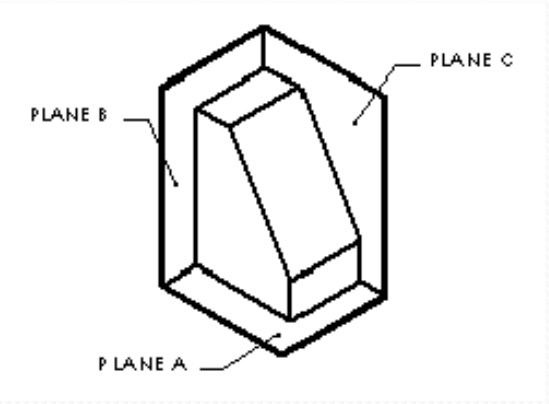
  - Primary



  - Secondary



  - Tertiary



Geometric Dimensioning & Tolerancing (GD&T)

Primary datum

- A primary datum is selected to provide functional relationships, accessibility, and repeatability.
  - Functional Relationships
    - A standardization of size is desired in the manuf. of a part
    - Consideration of how parts are orientated to each other is very important
    - For example, legos are made in a standard size in order to lock into place. A primary datum is chosen to reference the location of the mating features.
  - Accessibility
    - Does anything, such as, shafts, get in the way?

Geometric Dimensioning & Tolerancing (GD&T)

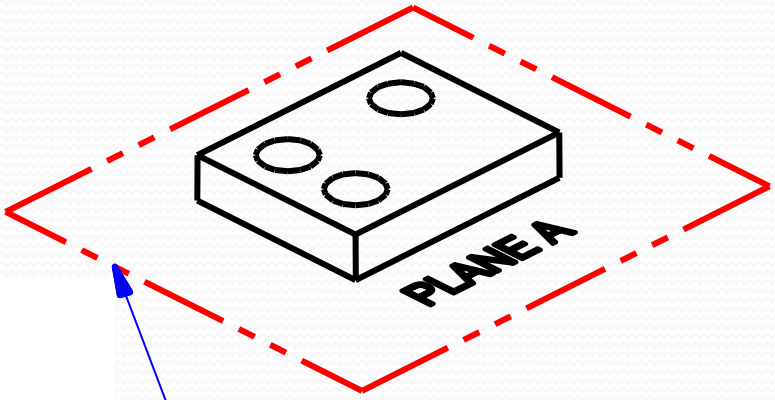
Primary datum

- A primary datum is selected to provide functional relationships, accessibility, and repeatability.
  - Repeatability
    - The primary datum chosen must insure precise measurements. The surface established must produce consistent
    - Measurements when producing many identical parts to meet requirements

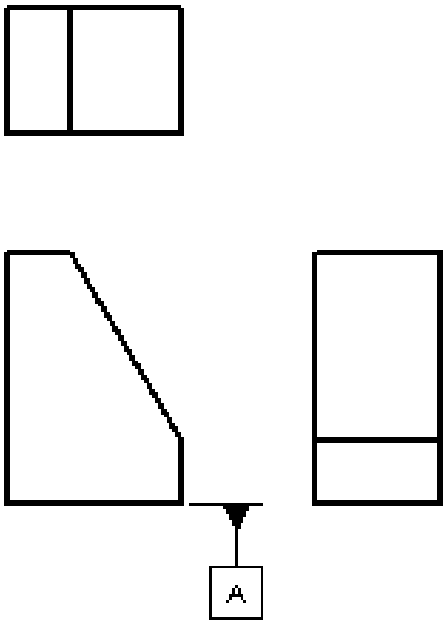
Geometric Dimensioning & Tolerancing (GD&T)

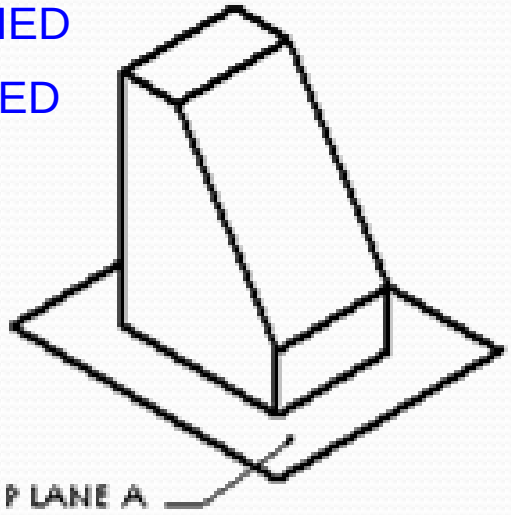
Primary datum

- Restricts 6 degrees of freedom



FIRST DATUM ESTABLISHED BY THREE POINTS (MIN) CONTACT WITH SIMULATED DATUM A

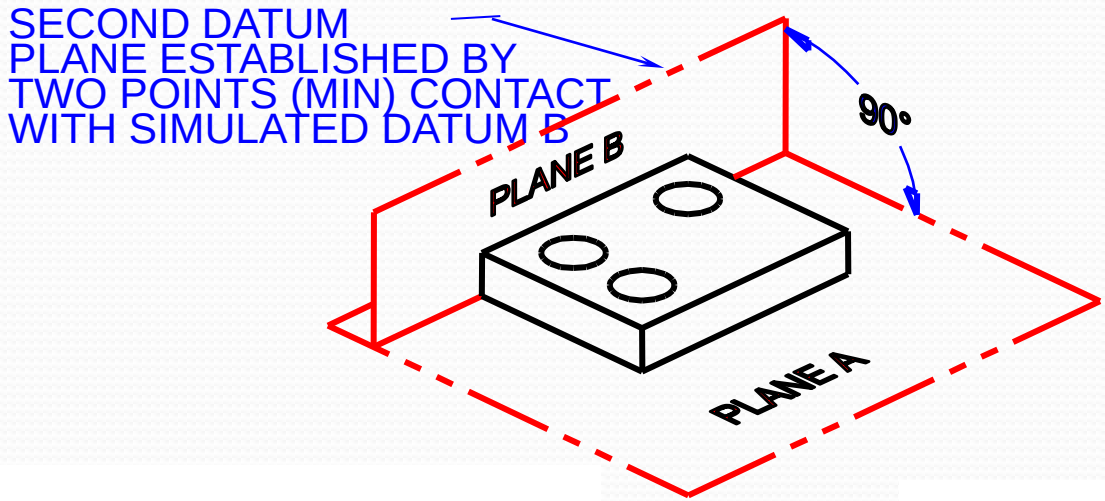


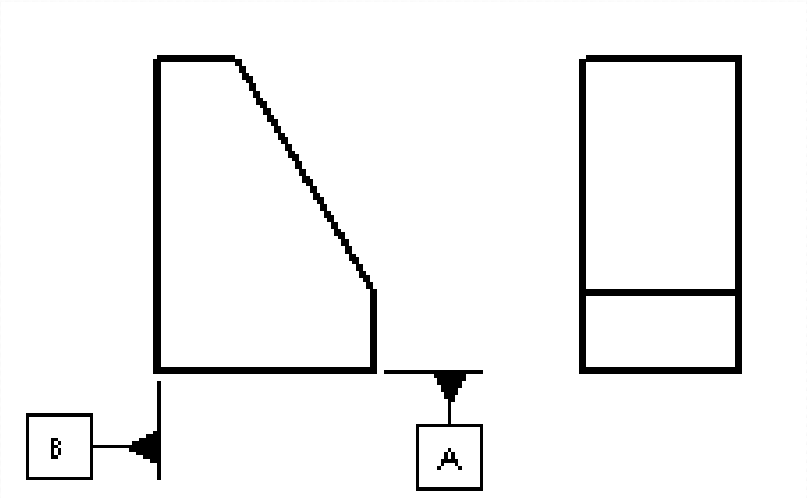


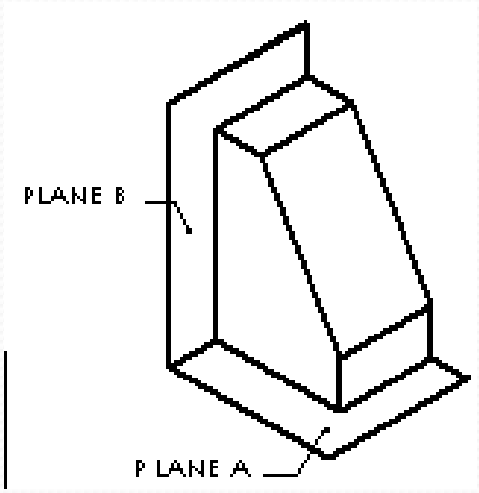
Geometric Dimensioning & Tolerancing (GD&T)

Secondary datum

- Restricts 10 degrees of freedom



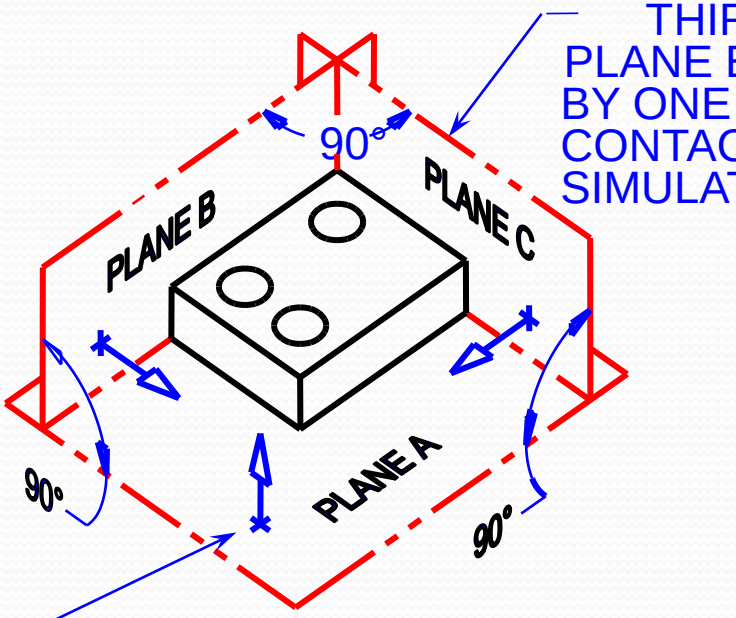


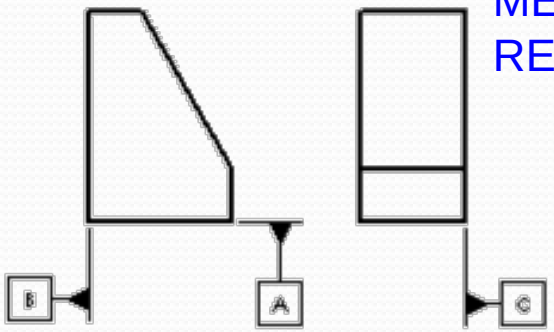


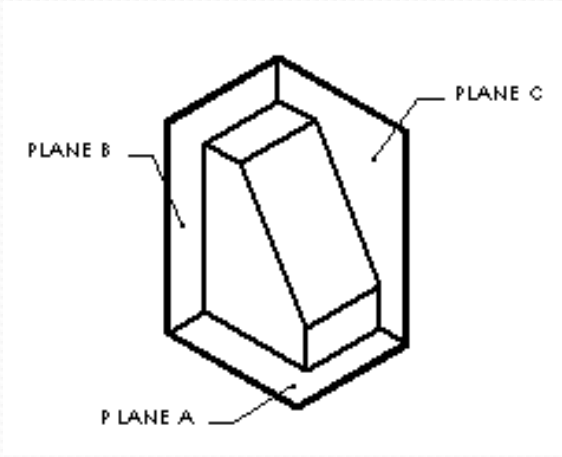
Geometric Dimensioning & Tolerancing (GD&T)

Tertiary datum

- Restricts 12 degrees of freedom

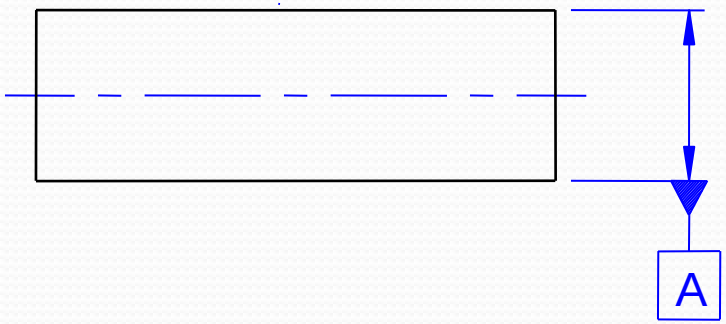




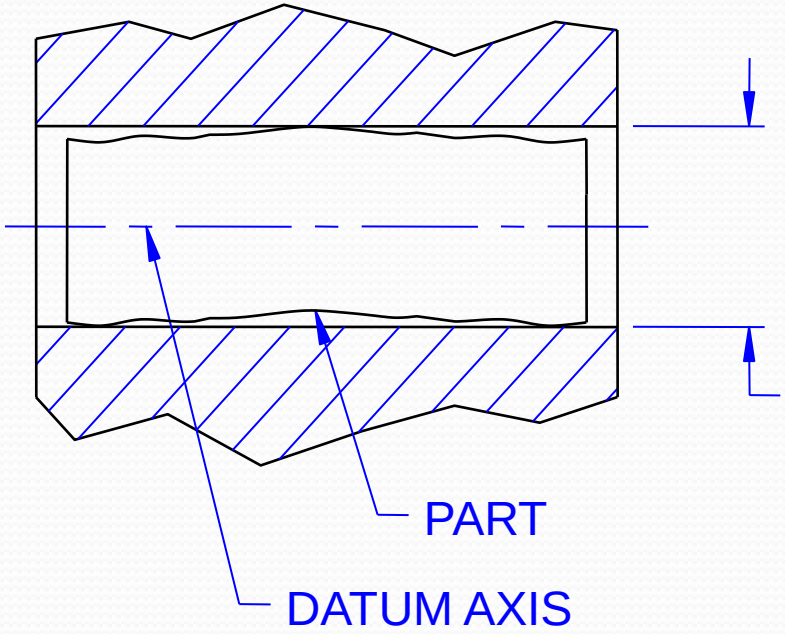


Geometric Dimensioning & Tolerancing (GD&T)

Size datum (circular)



THIS ON
THE DRAWING

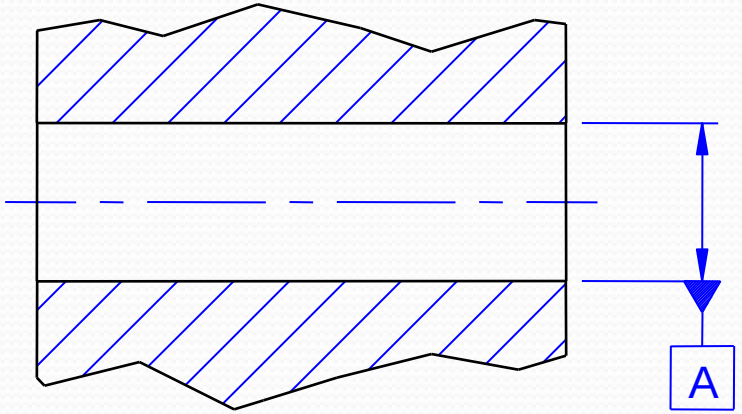


MEANS THIS

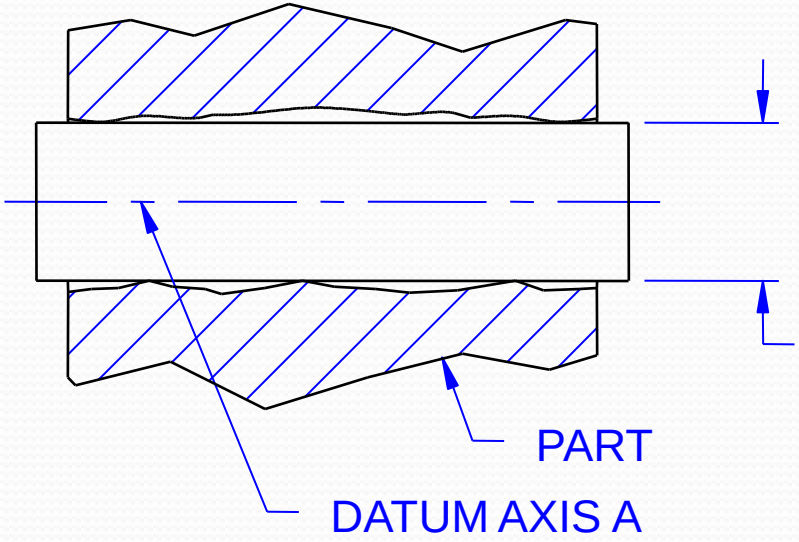
SIMULATED DATUM-
SMALLEST
CIRCUMSCRIBED
CYLINDER

Geometric Dimensioning & Tolerancing (GD&T)

Size datum (circular)



THIS ON
THE DRAWING



MEANS THIS

SIMULATED DATUM-
LARGEST
INSCRIBED
CYLINDER

Geometric Dimensioning & Tolerancing (GD&T)

Orientation tolerances

Perpendicularity 

Angularity 

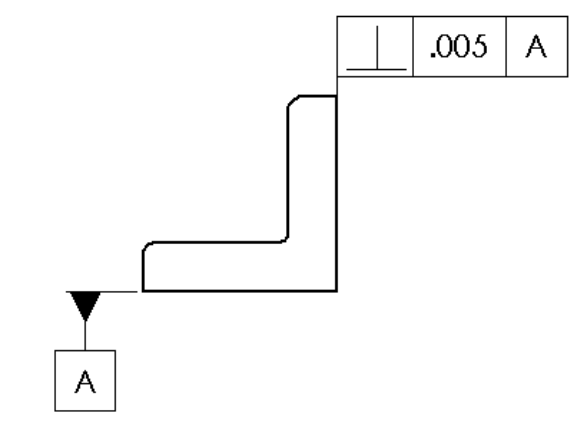
Parallelism 

- Controls the orientation of individual features
- Datums are required
- Shape of tolerance zone: 2 parallel lines, 2 parallel planes, and cylindrical

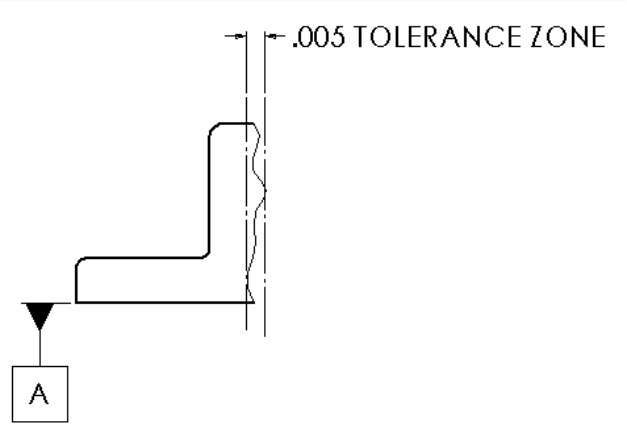
Geometric Dimensioning & Tolerancing (GD&T)

Perpendicularity

- is the condition of a surface, center plane, or axis at a right angle (90°) to a datum plane or axis.



The perpendicularity of this surface must be within a .005 tolerance zone relative to datum A.

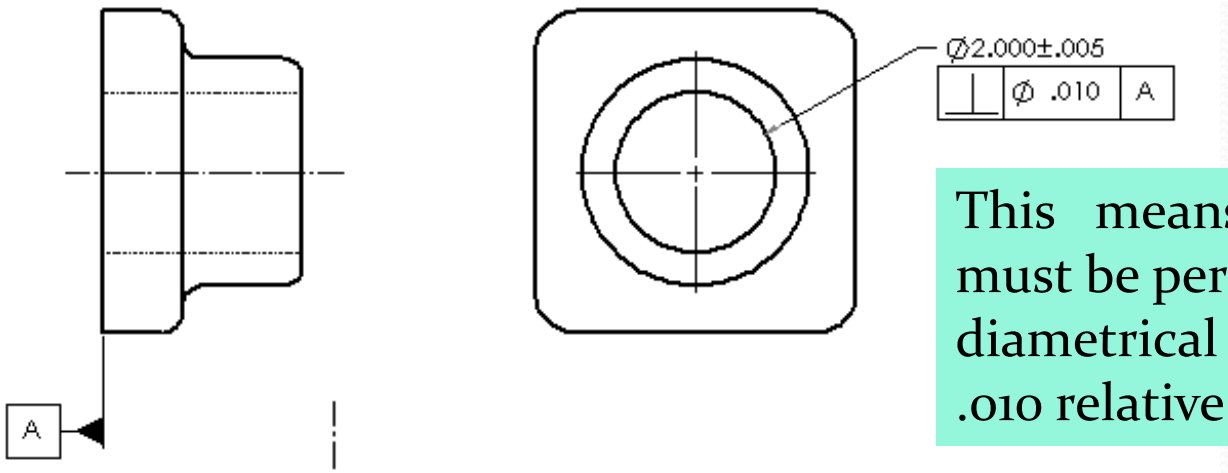


The tolerance zone is the space between the 2 parallel lines. They are perpendicular to the datum plane and spaced .005 apart.

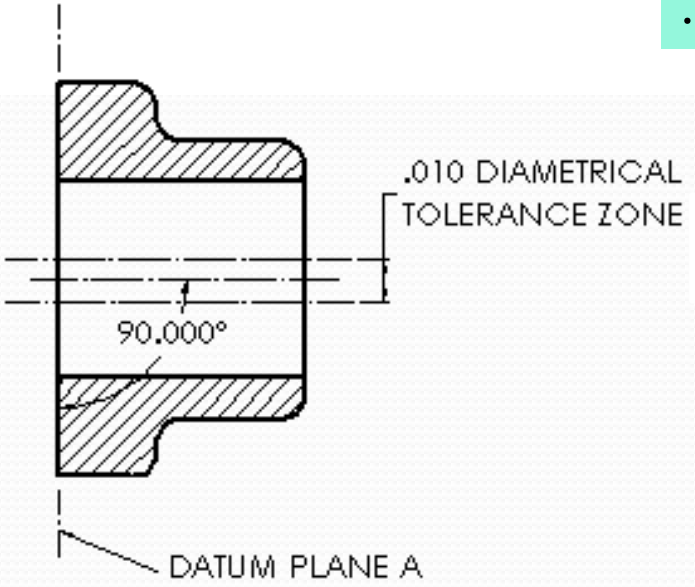
Geometric Dimensioning & Tolerancing (GD&T)

Perpendicularity

- Location of hole (axis).



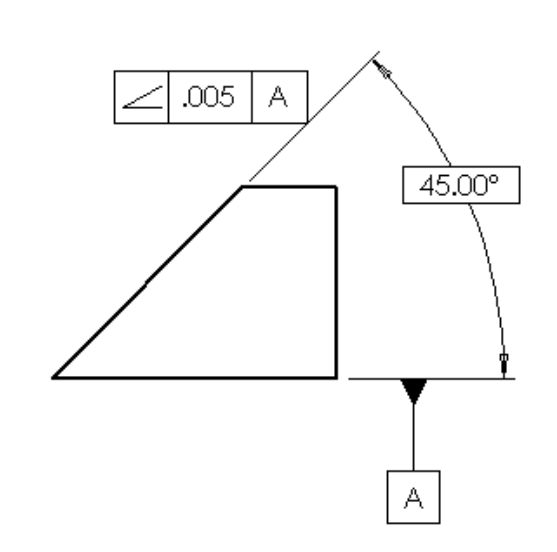
This means 'the hole (axis) must be perpendicular within a diametrical tolerance zone of .010 relative to datum A'



Geometric Dimensioning & Tolerancing (GD&T)

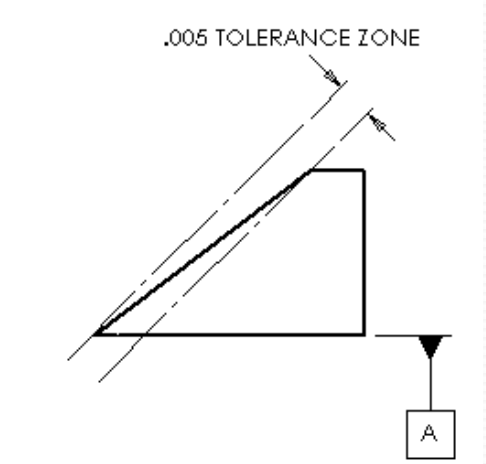
Angularity

- is the condition of a surface, axis, or median plane which is at a specific angle (other than 90°) from a datum plane or axis.



The surface is at a 45º angle with a .005 tolerance zone relative to datum A.

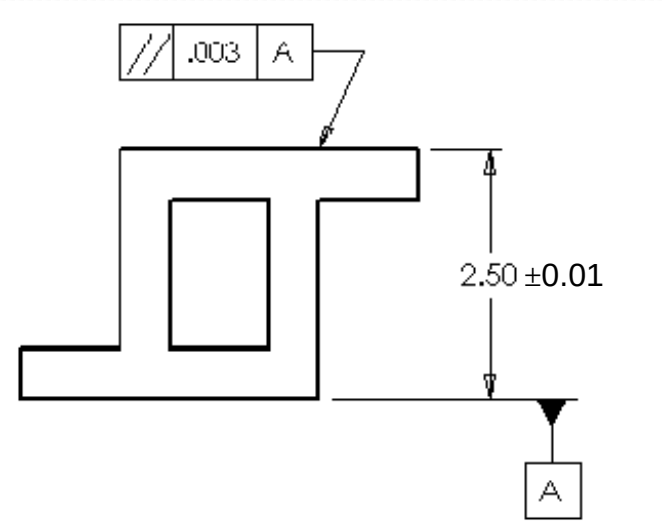
- Can be applied to an axis at MMC
- Typically must have a **basic** dimension

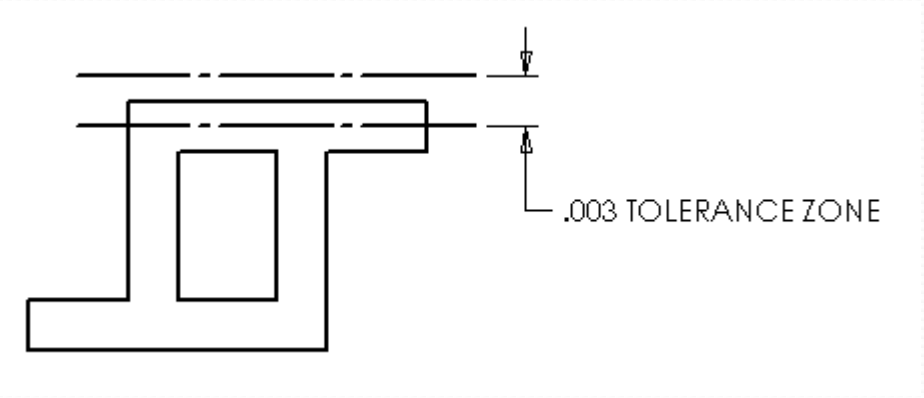


Geometric Dimensioning & Tolerancing (GD&T)

Paralellism

- The condition of a surface or center plane equidistant at all points from a datum plane, or an axis.
- The distance between the parallel lines, or surfaces, is specified by the geometric tolerance.





Geometric Dimensioning & Tolerancing (GD&T)

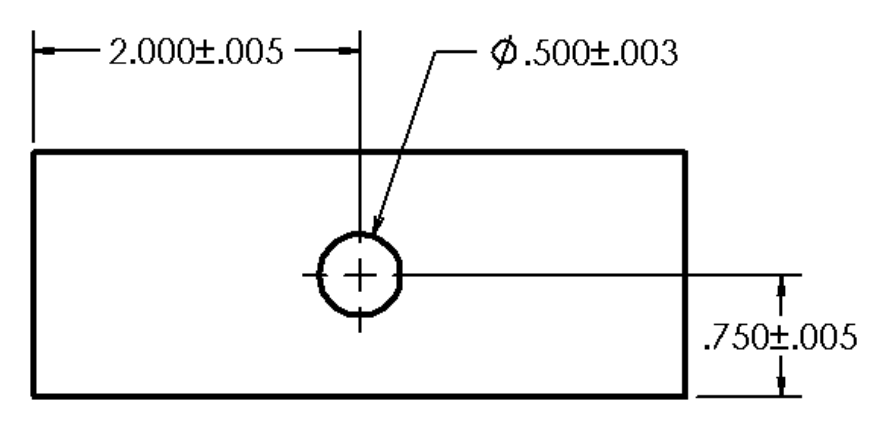
Position tolerances

- A position tolerance is the total permissible variation in the location of a feature about its exact true position.
- For cylindrical features, the position tolerance zone is typically a cylinder within which the axis of the feature must lie.
- For other features, the center plane of the feature must fit in the space between two parallel planes.
- The exact position of the feature is located with basic dimensions.
- The position tolerance is typically associated with the size tolerance of the feature.
- Datums are required.

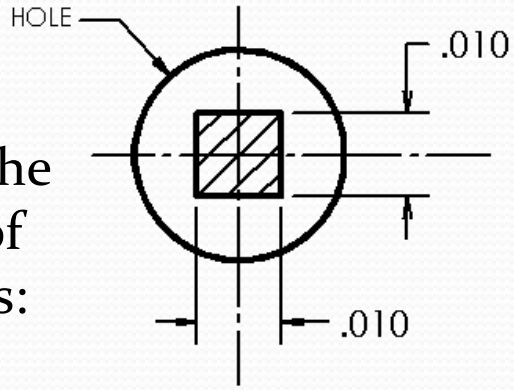
Geometric Dimensioning & Tolerancing (GD&T)

Coordinate system position

- Consider the following hole dimensioned with coordinate dimensions:



The tolerance zone for the location of the hole is:

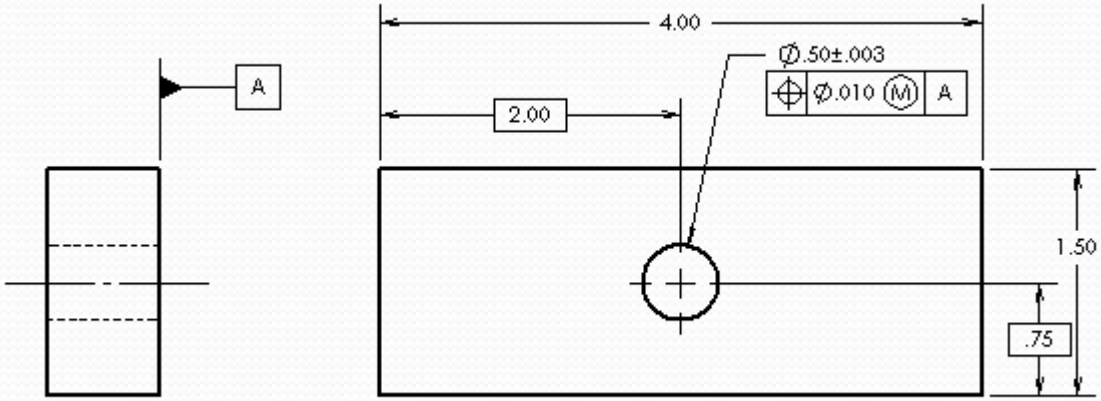


- Problems:
  - Two points, equidistant from true position may not be accepted.
  - Total tolerance diagonally is .014, which may be more than was intended.

Geometric Dimensioning & Tolerancing (GD&T)

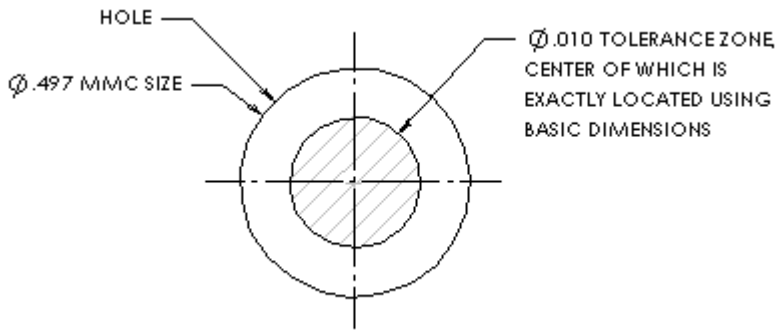
Position tolerancing

- Consider the same hole, but add GD&T:



- Now, overall tolerance zone is:

MMC = .500 - .003 = .497

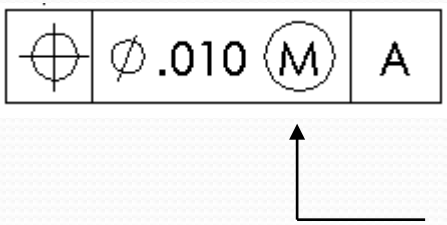


- The actual center of the hole (axis) must lie in the round tolerance zone. The same tolerance is applied, regardless of the direction

Geometric Dimensioning & Tolerancing (GD&T)

Bonus tolerance

- The specified tolerance was:

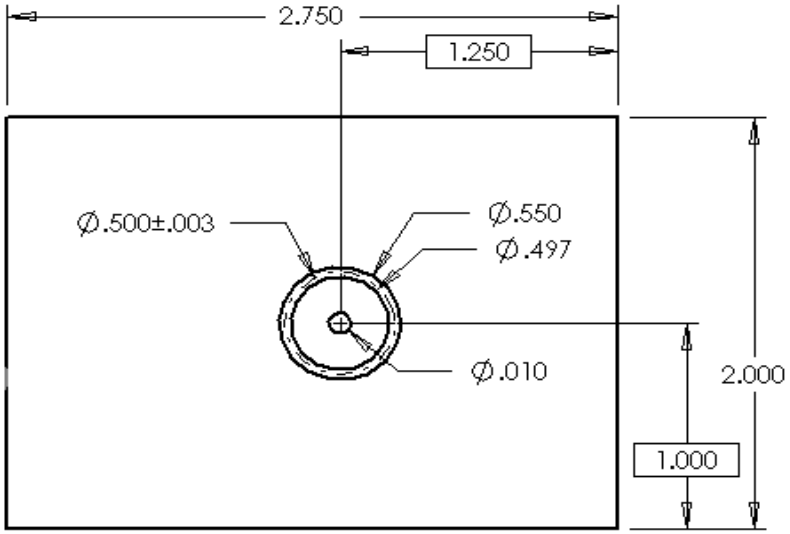


This means that the tolerance is .010 <u>if</u> the hole size is the MMC size, that is: 0.497. If the hole is bigger, we get a <u>bonus</u> tolerance equal to the difference between the MMC size and the actual size.

Geometric Dimensioning & Tolerancing (GD&T)

Bonus tolerance example

- The specified tolerance was:



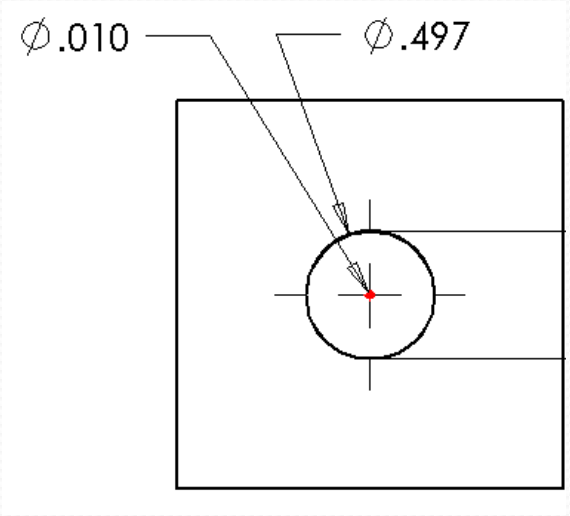
Actual Hole Size	Bonus Tol.	Φ of Tol. Zone
Ø .497 (MMC)	0	.010
Ø .499 (.499 - .497 = .002)	.002 (.010 + .002 = .012)	.012
Ø .500 (.500 - .497 = .003)	.003 (.010 + .003 = .013)	.013
Ø .502	.005	.015
Ø .503 (LMC)	.006	.016
Ø .504	?	?

This means that the tolerance is .010 if the hole size is the MMC size, or .497. If the hole is bigger, we get a bonus tolerance equal to the difference between the MMC size and the actual size.

This system makes sense… the larger the hole is, the more it can deviate from true position and still fit in the mating condition!

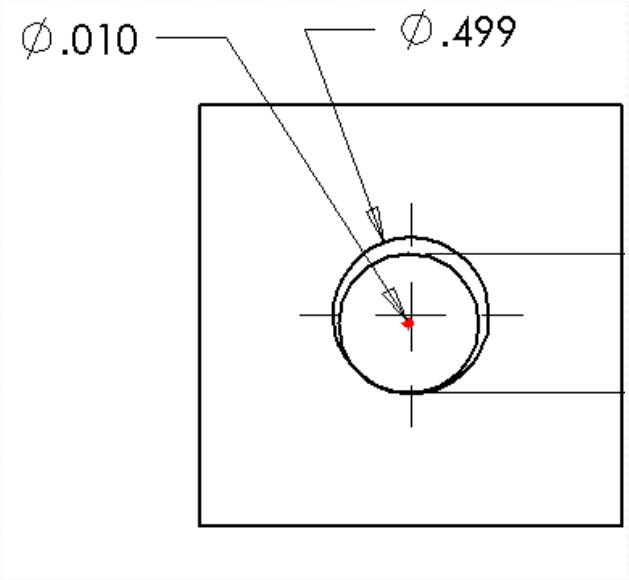
Geometric Dimensioning & Tolerancing (GD&T)

Bonus tolerance example



.497 = BONUS 0

TOL ZONE .010

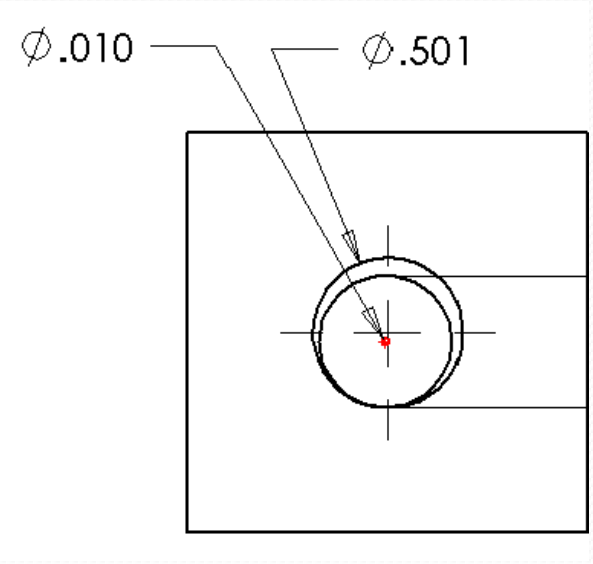


.499 - .497 = BONUS .002

BONUS + TOL. ZONE = .012

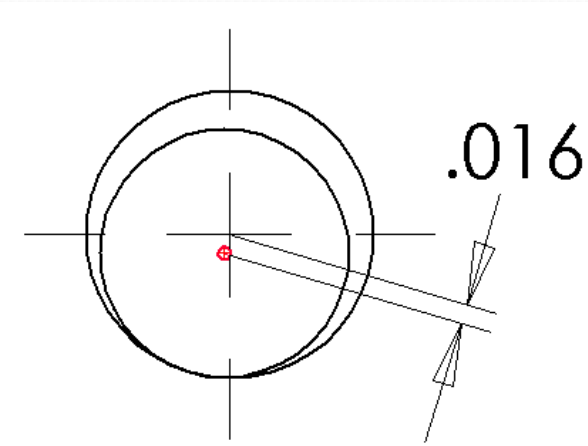
Geometric Dimensioning & Tolerancing (GD&T)

Bonus tolerance example



.501 - .497 = BONUS .004

BONUS + TOL. ZONE = .014

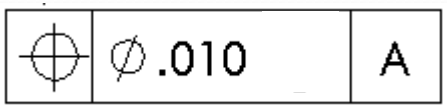


.503 - .497 = BONUS .006

BONUS + TOL. ZONE = .016

Geometric Dimensioning & Tolerancing (GD&T)

What if the tolerance had been specified as:

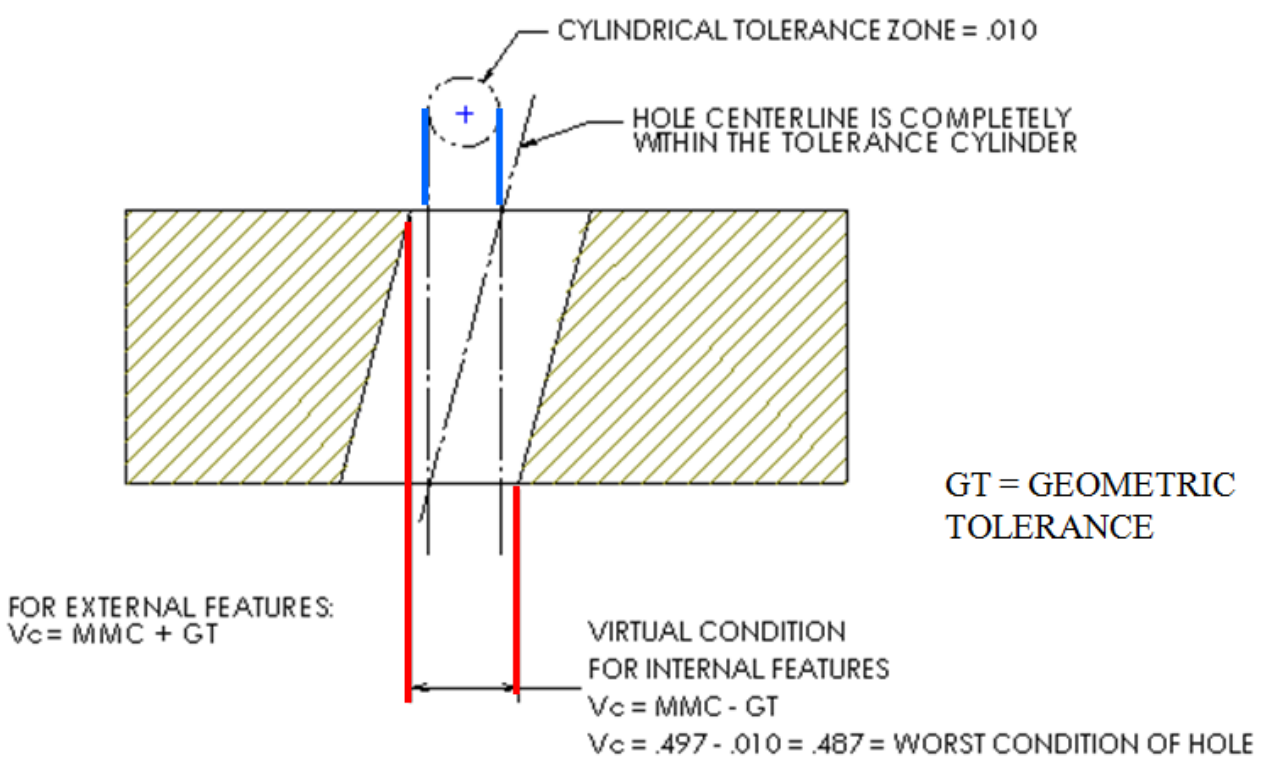


Since there is NO material modifier, the tolerance is RFS, which stands for regardless of feature size. This means that the position tolerance is .010 at all times. There is no bonus tolerance associated with this specification.

Geometric Dimensioning & Tolerancing (GD&T)

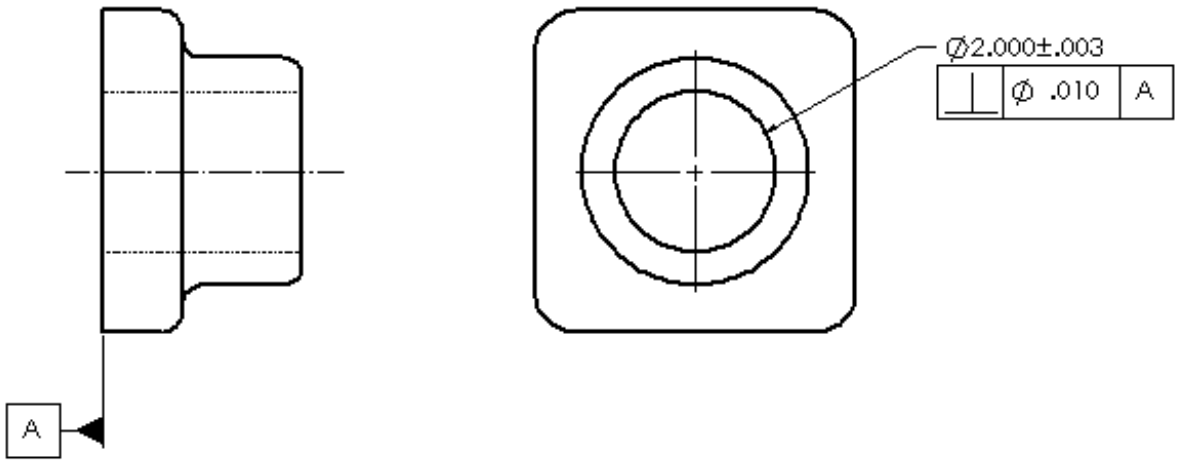
Virtual condition

The **worst** case boundary generated by the collective effects of a size feature's specified MMC or LMC material condition and the specified geometric tolerance

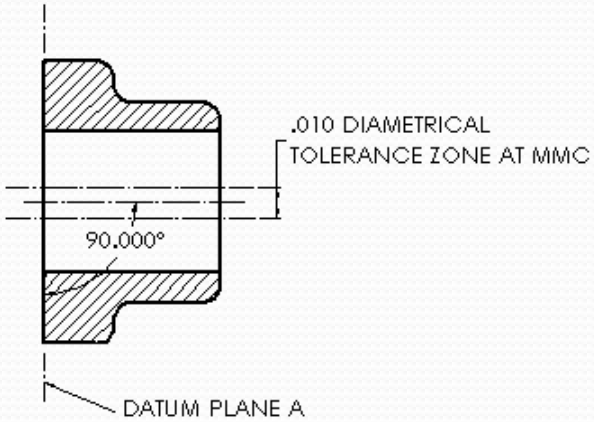


Geometric Dimensioning & Tolerancing (GD&T)

Homework



Means "the hole (AXIS) must be perpendicular within a diametrical tolerance zone of .010 at MMC relative to datum A."



Vc =

Actual Hole Size	Bonus Tol.	Ø of Tol. Zone
1.997 (MMC)		
1.998		
1.999		
2.000		
2.001		
2.002		
2.003		