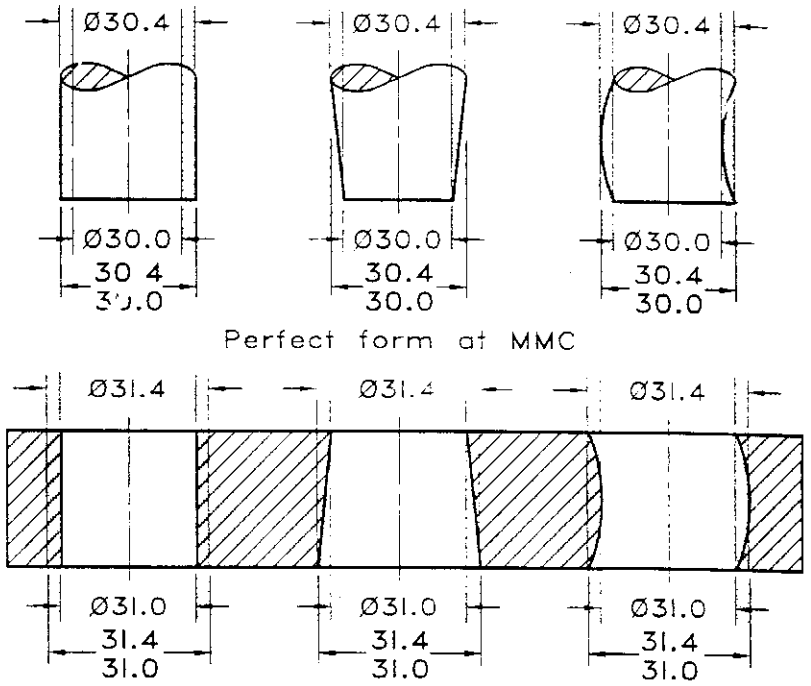
Geometric Dimensioning and Tolerancing - GD&T

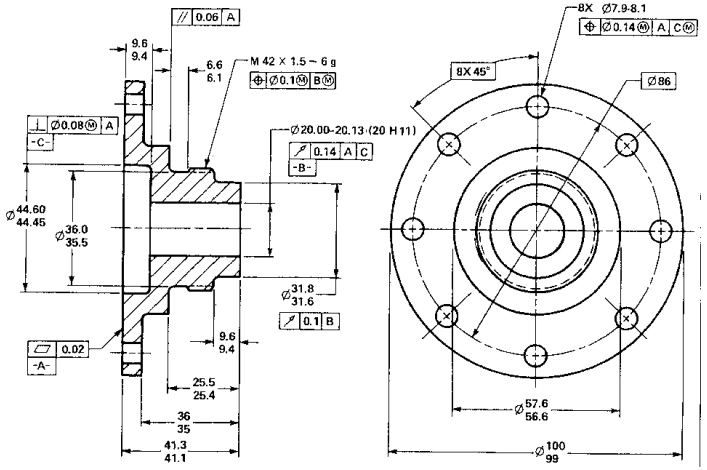
GD&T

- The need to tolerance more then size led to a more precise system



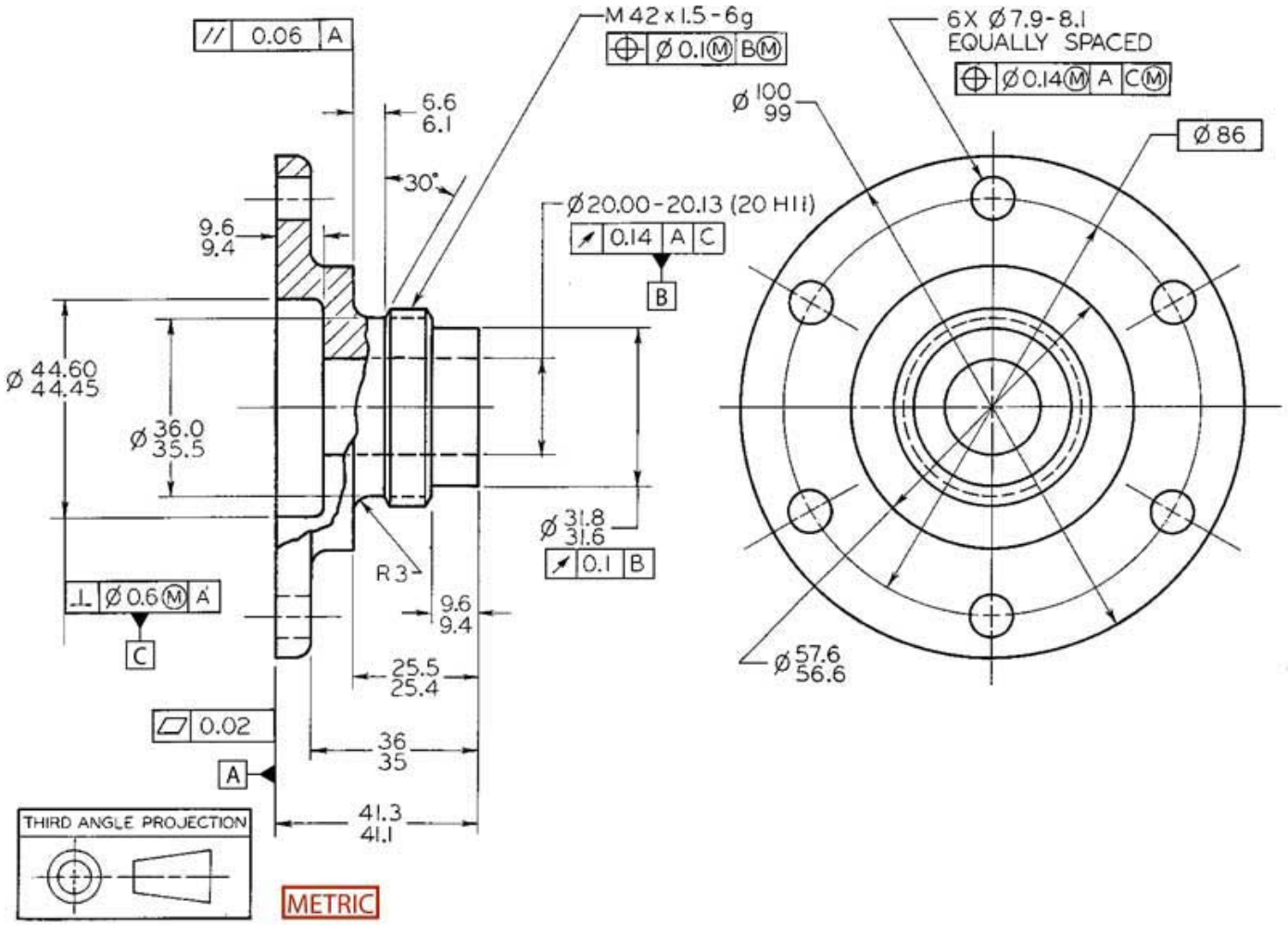
GD&T

- Allows for setting tolerance limits not just for size but for shape features as well.
- Dependant on function of part and relationship to mating parts.
- ASME Y14.5



GD&T - Example

Technical Drawing 13th Edition
Giesecke, Mitchell, Spencer, Hill Dygdon, Novak, Lockhart

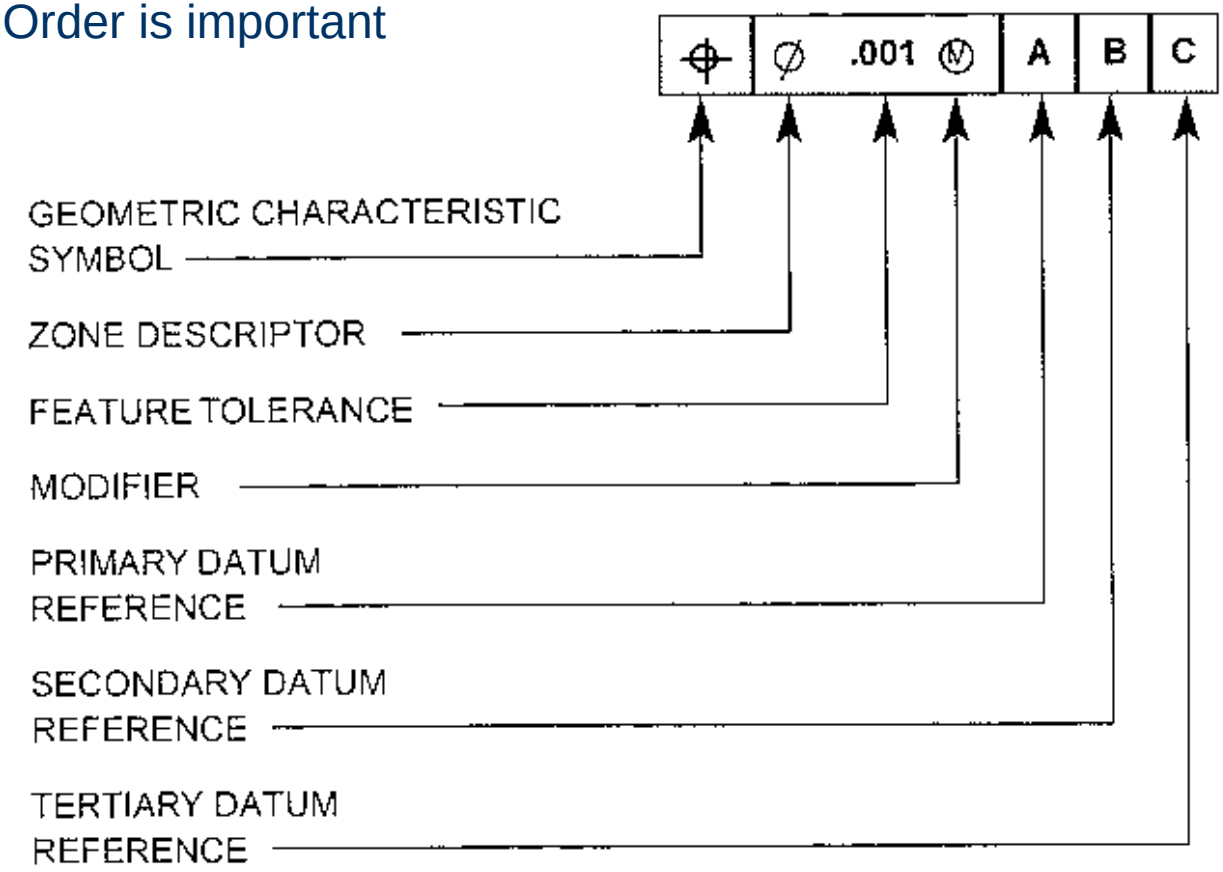


GD&T

- GD&T described using feature control frame
- Need to understand:
  - Geometric Characteristics
  - Modifiers
  - Datums

Feature Control Frame

Note: Order is important



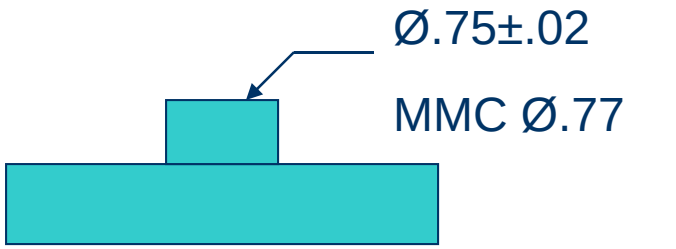
Geometric Characteristics

	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL
FOR INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	
		FLATNESS	
		CIRCULARITY (ROUNDNESS)	
		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	

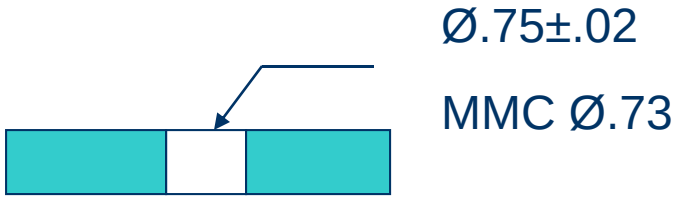
Modifiers

- Symbols that help to define characteristics
  - Example: MMC – Maximum Material Condition- condition when most material exists

External Feature



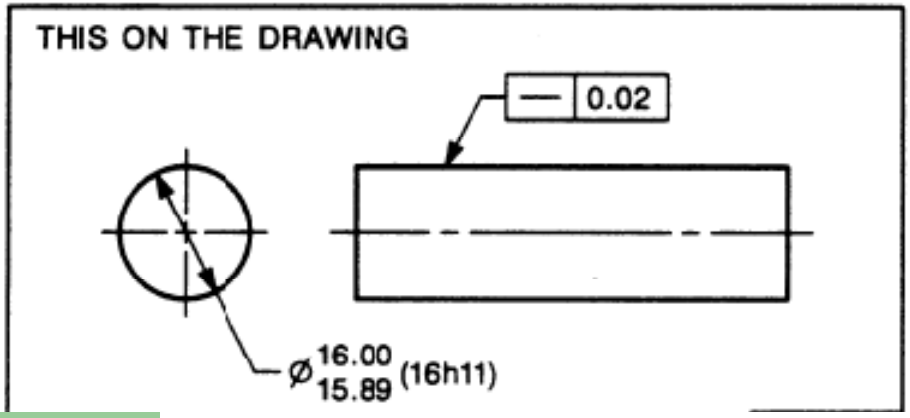
Internal Feature



Compare MMC to RFS (Regardless of Feature Size)

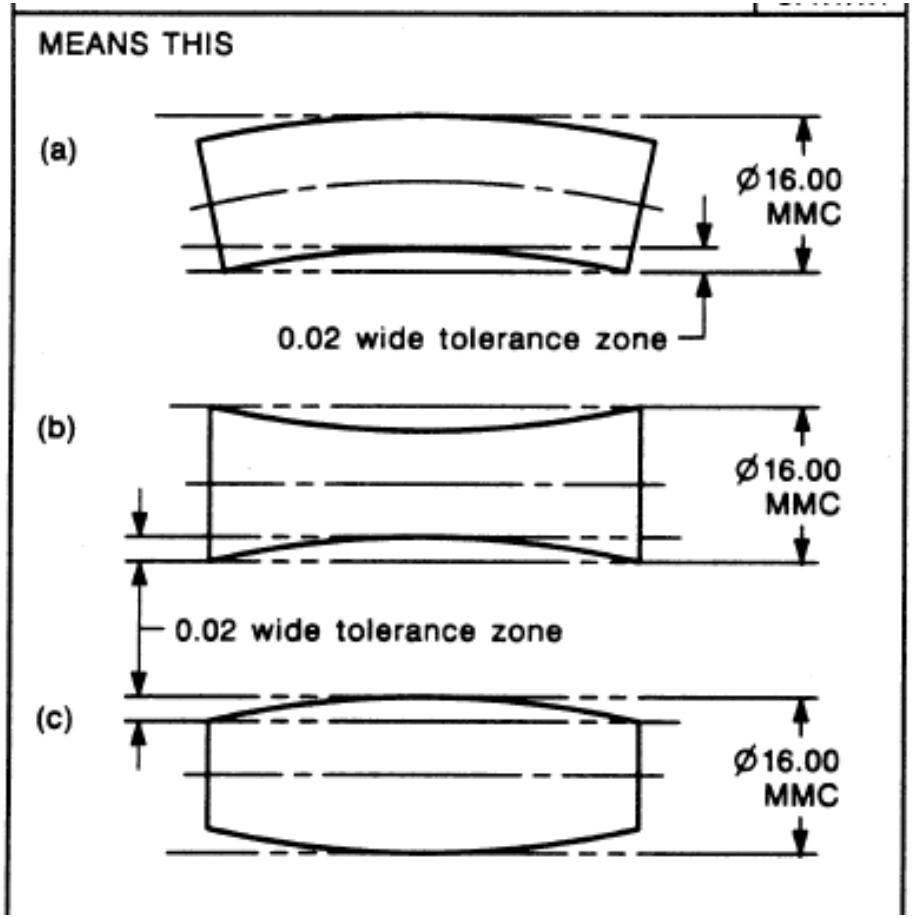
Produced size	Tolerance RFS	Tolerance MMC
1.002	.002	.002
1.001	.002	.003
1.000	.002	.004
.999	.002	.005
.998	.002	.006

Straightness Example

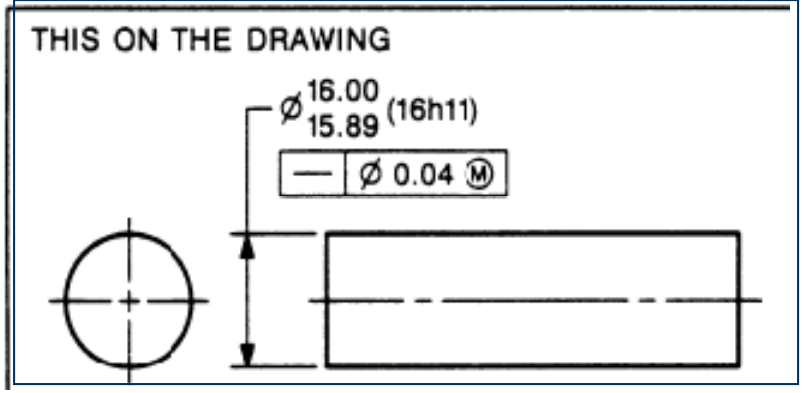


Each longitudinal element of the surface must lie between two parallel lines (0.02 apart) where the two lines and the nominal axis of the part share a common plane. The feature must be within the specified limits of size and the boundary of perfect form at MMC (16.00).

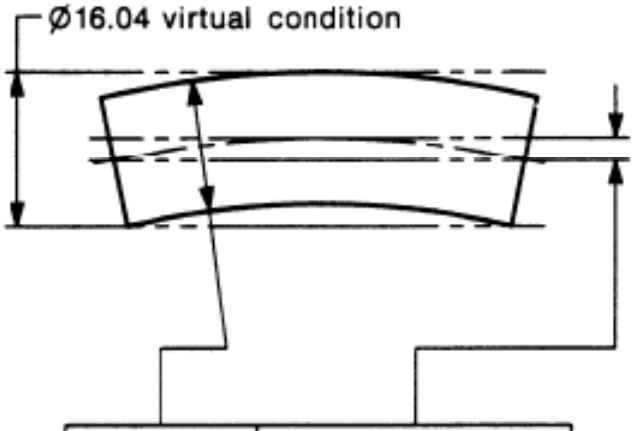
Note: Waisting (b) or barreling (c) of the surface, though within the straightness tolerance, must not exceed the limits of size of the feature.



Straightness at MMC

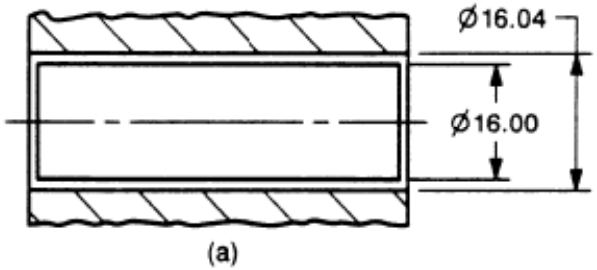


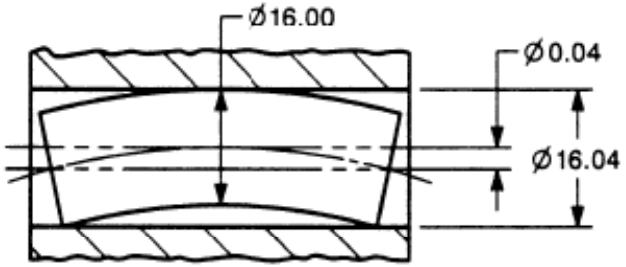
MEANS THIS

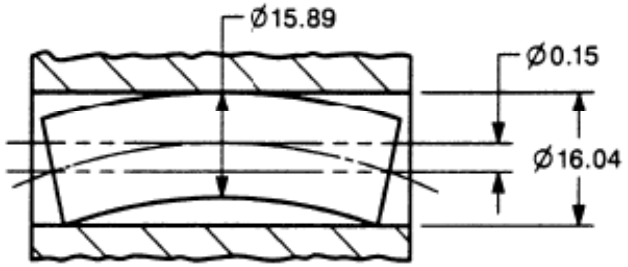


Feature size	Diameter tolerance zone allowed
16.00	0.04
15.99	0.05
15.98	0.06
↓	↓
15.90	0.14
15.89	0.15

ACCEPTANCE BOUNDARY

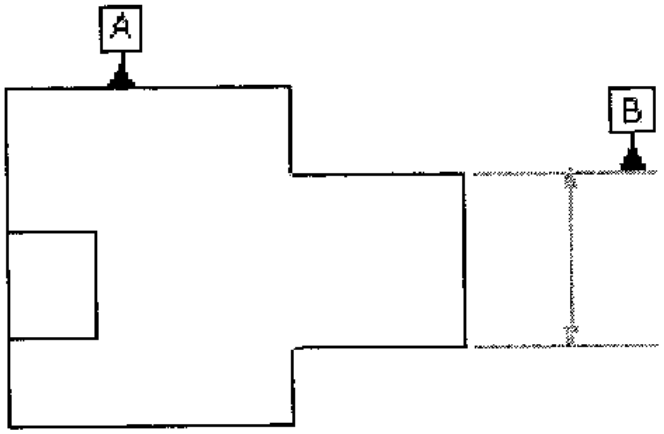


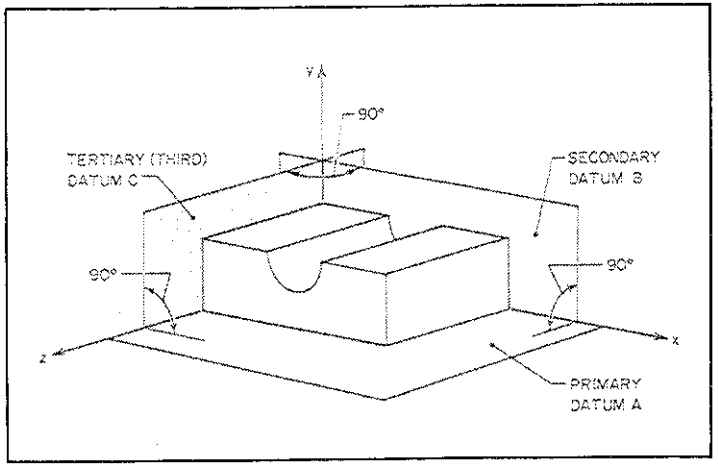




Datums

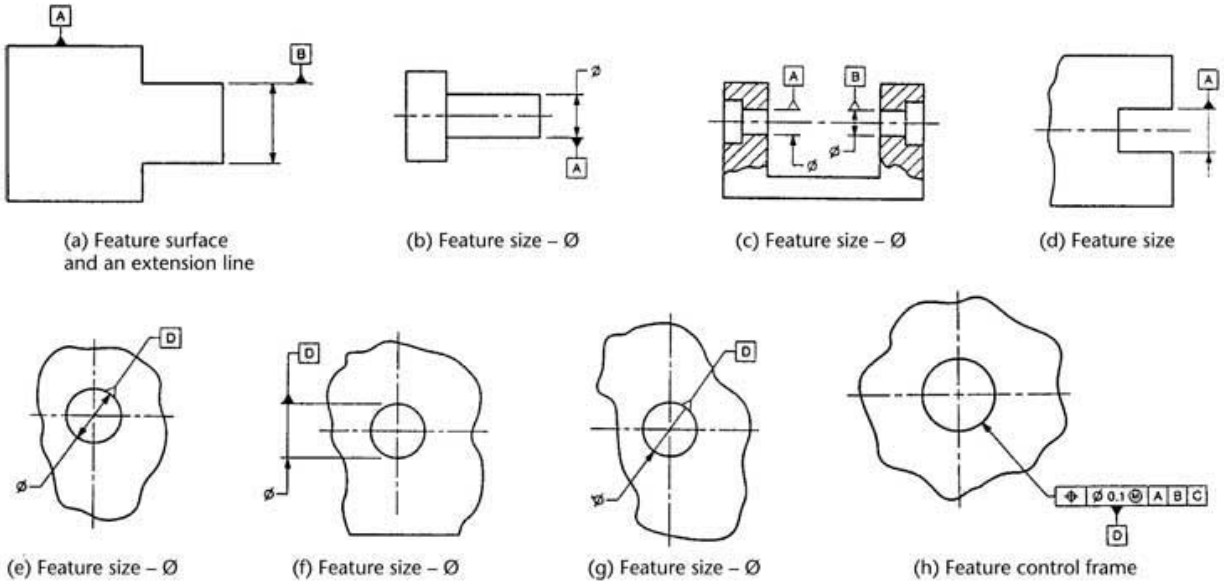
- Theoretically perfect points, lines and planes used for reference.
- Again function of part very important
- Classified as primary, secondary or tertiary.



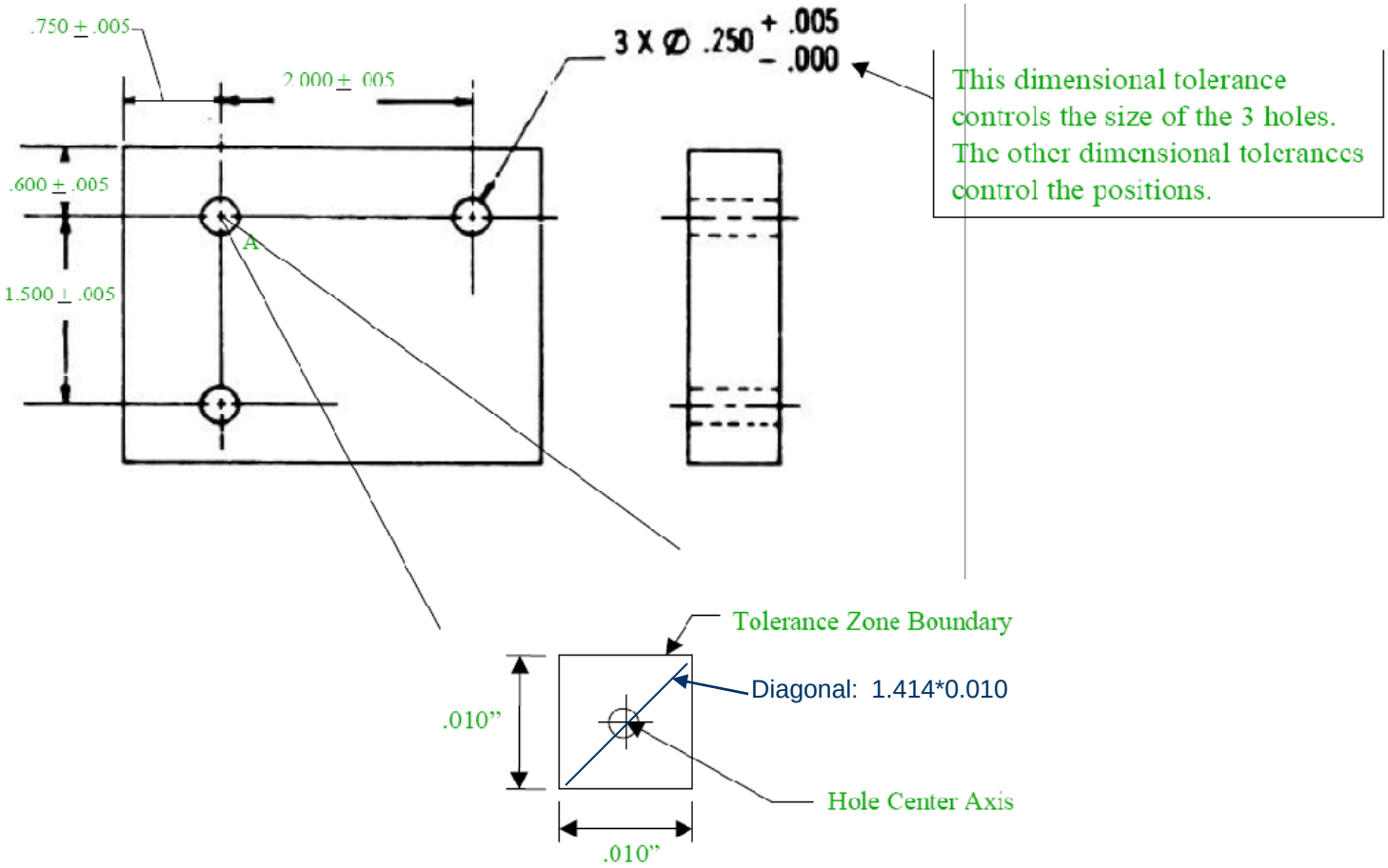


Datum Surfaces and Features

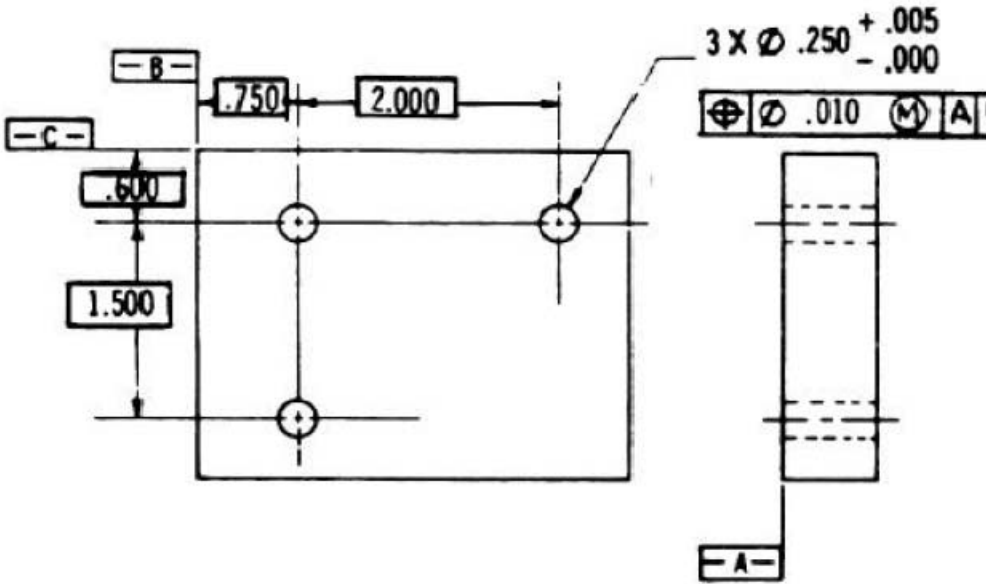
- Datum surfaces and features are used as references to control other features



Coordinate Tolerancing



Position Tolerancing

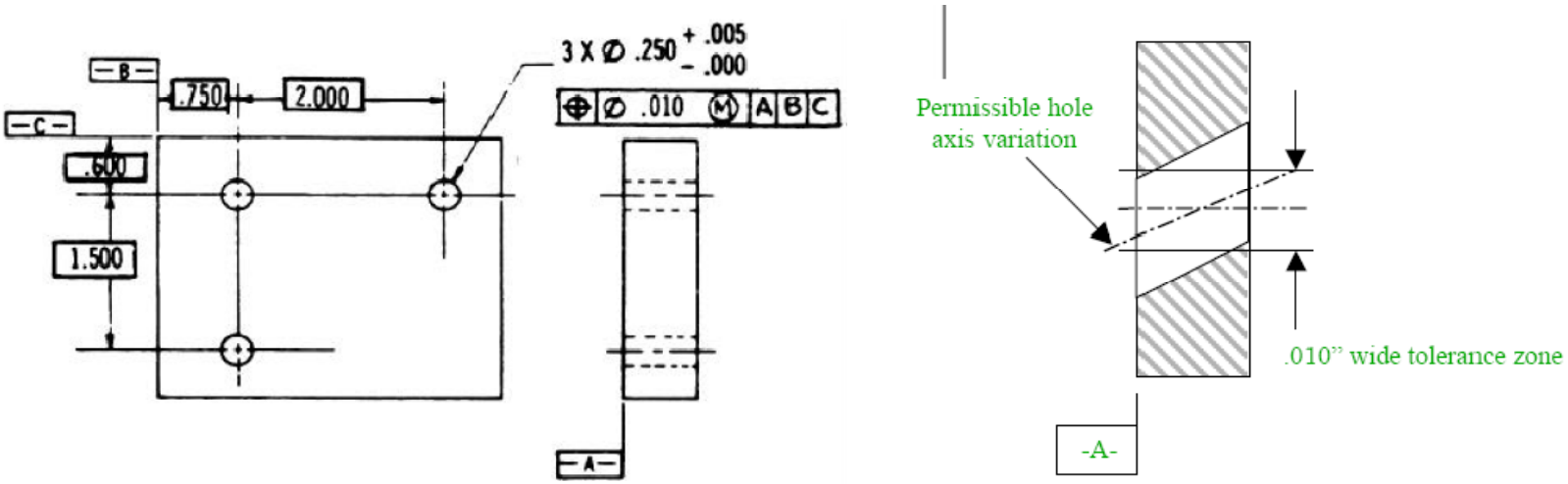


MMC of hole = .250

LMC of hole = .255

Hole diameter	Tolerance Zone
.250 (MMC)	.010
.251	.011
.252	.012
.253	.013
.254	.014
.255 (LMC)	.015

True Position and Perpendicularity



Referencing datum A means that the center axis of the hole must be perpendicular to datum plane A. The axis must intersect datum plane A inside the tolerance zone

Using Geometric Dimensioning and Tolerancing

- Applying GDT principles requires:
  - Defining the part's functions
  - Listing the functions by priority
  - Defining the datum reference frame
  - Control selection
  - Calculating tolerances