






















**TERMS TO BE DEFINED OR IDENTIFIED FOR COMPETENCY 12:**

- Limits
- Straightness
- Flatness
- True position
- Parallelism
- Basic size
- Feature
- Form tolerance
- Positional tolerance
- Circularity
- Datum reference
- Symmetry
- Allowance
- MMC
- LMC
- Clearance
- Tolerance
- Reference dimension
- Orientation tolerance zone
- Profile tolerance control
- Concentricity
- Runout tolerance
- Datum targets
- Decimal metric dimensions
- Actual size

**ITEMS FOR REVIEW for COMPETENCY 12:**

**Standards and Terminology**

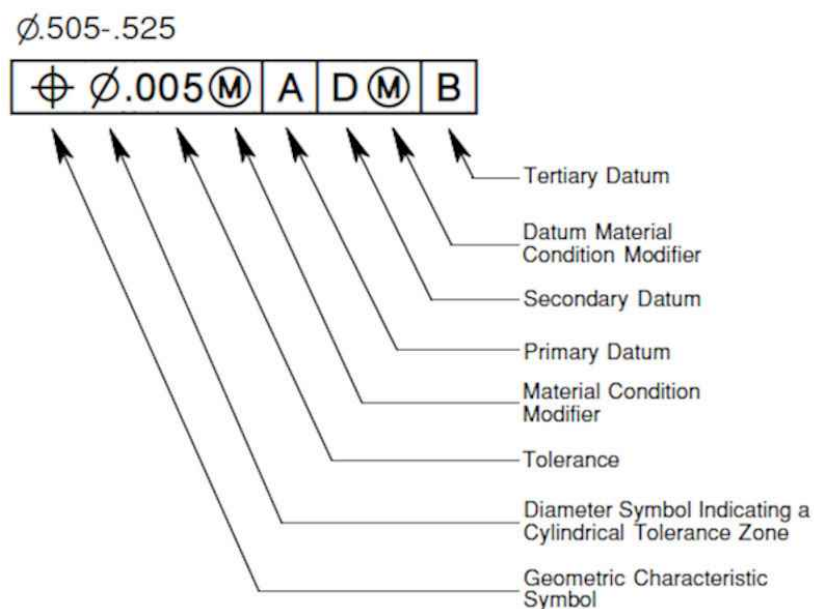
Dimensioning practices and Rules based on ASME Y14.5M 1994. Identify geometric symbols but not Geometric Dimensioning and Tolerancing practices.

	Counterbore		Profile of a Line
	Countersink		Profile of a Surface
	Depth		Angularity
	Diameter		Perpendicularity
	Least Material Condition		Parallelism
	Most Material Condition		Position
	Square		Concentricity
	Straightness		Symmetry
	Flatness		Circular Runout
	Circularity (roundness)		Total Runout
	Cylindricity		

### **Standard Terminology**

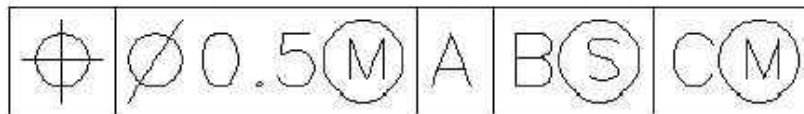
- **Actual Size** – The size of the part as measured.
- **Allowance** – The minimum clearance or maximum interference between two mating parts.
- **Basic Size** - The size from which the limits of size are derived by the application of allowances and tolerance.
- **Clearance** – The space between two mating parts.
- **Feature** - A portion of a part, such as a hole, keyway, or flat surface.
- **Least Material Condition** – The maximum hole diameter or minimum shaft diameter. When a part weighs the least.
- **Limits** - The maximum and minimum allowable sizes of a feature.
- **Maximum Material Condition** – The minimum hole diameter or maximum shaft diameter. When the part weighs the most.
- **Reference Dimension** - A non-toleranced dimension used for information purposes only. It may not govern production or inspection.
- **Tolerance** – The total amount by which the part dimensions are permitted to vary.
- **Unilateral Tolerance** – Variation of size in one direction either positive or negative.
- **Bilateral Tolerance** – Variation of size in both directions positive and negative.
- **Specific Tolerance** – Stated with dimension in field of drawing.
- **General Tolerance** – Stated in title block.

- **Clearance Fit Tolerance** – Internal Dimension maintains a smaller size than external between mating parts. Fit type: RC – Running and Sliding, LC – Locational Clearance
- **Interference Fit Tolerance** – Internal Dimension maintains a larger size than external between mating parts. Fit type: LT – Transition
- **Transition Fit Tolerance** – Condition where a clearance or interference fit may be present between mating parts. Fit type: LN – Locational Interference, FN – Force or Shrink
- **Basic Hole System** – Minimum hole size is used as the base size for fit tolerance calculations.
- **Basic Shaft System** – Maximum shaft size is used as the basic size for fit tolerance calculations.
- Geometry Control Tools are applied to individual part features by placing them in Feature Control Frames. Feature Control Frames are a series of compartments containing symbols and values that describe the geometric tolerance of a feature. The order and purpose of these compartments follow a consistent standard.



- Feature Control Frames can be either “read” or “decoded”.

$\varnothing 12 \pm 0.2$

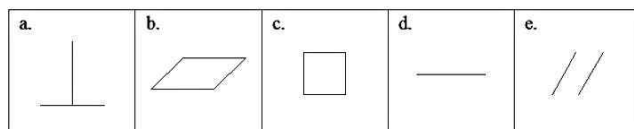


**Reading:** Position within a diameter of 0.5 mm at MMC relative to A,B RFS and C at MMC.

**Decoding:** Position requires the bounded axis of the considered feature to lie within a “cylindrical” tolerance zone of diameter 0.5 mm at MMC, expanding by as much as 0.4 mm as the considered feature departs from MMC toward LMC, oriented and located by Basic dimensions relative to a DRF established using Datum Feature A, Datum Feature B simulated regardless of its size, and Datum Feature C simulated at its Virtual MMC.


### **SAMPLE REVIEW QUESTIONS**

1. When a screw thread is specified as a datum feature, what is used to derive the datum axis unless otherwise specified?  
Pitch diameter  
Major diameter  
Minor diameter  
Screw threads must be used as datum features.
2. Which of the following requires a datum reference?  
Circular runout  
Flatness  
Position  
Profile
3. Identify the FLATNESS symbol. (MCD-CO11-G098)  
A, B, C, D, E



4. Which of the following examples shows the elements of a feature control frame in the correct order?

- A
- B
- C
- D

a. 

b. 

c. 

d. 

5. Which of the following is a two-dimensional geometric control?

- Profile of a line
- Profile of a surface
- Concentricity
- Cylindricity

6. Which of the following is the “between” symbol?

- A
- B
- C
- D

a. 

b. 

c. 

d. 