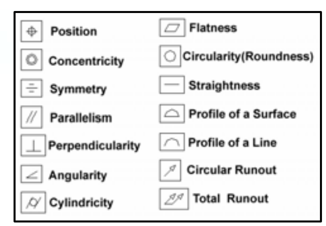
MEM 201

Fundamentals of Computer Aided Design

Geometrical Dimensioning & Tolerancing (GD&T)



Department of Mechanical Engineering and Mechanics

Today's Objectives…..

- Tolerances and why do we need them.
- Different types of tolerances.
- To learn how to effectively tolerance parts in engineering drawings.
- Allowance/Clearance
- Expressing tolerances in AutoCAD.

Tolerancing

- Definition: "Allowance for a specific variation in the size and geometry of part."

- Why is it needed: No one or thing is perfect !

- Hence, engineers have come up with a way to make things close to perfect by specifying Tolerances !
  - Since variation from the drawing is inevitable the acceptable degree of variation must be specified.
  - Large variation may affect the functionality of the part
  - Small variation will effect the cost of the part
    - requires precise manufacturing.
    - requires inspection and the rejection of parts.

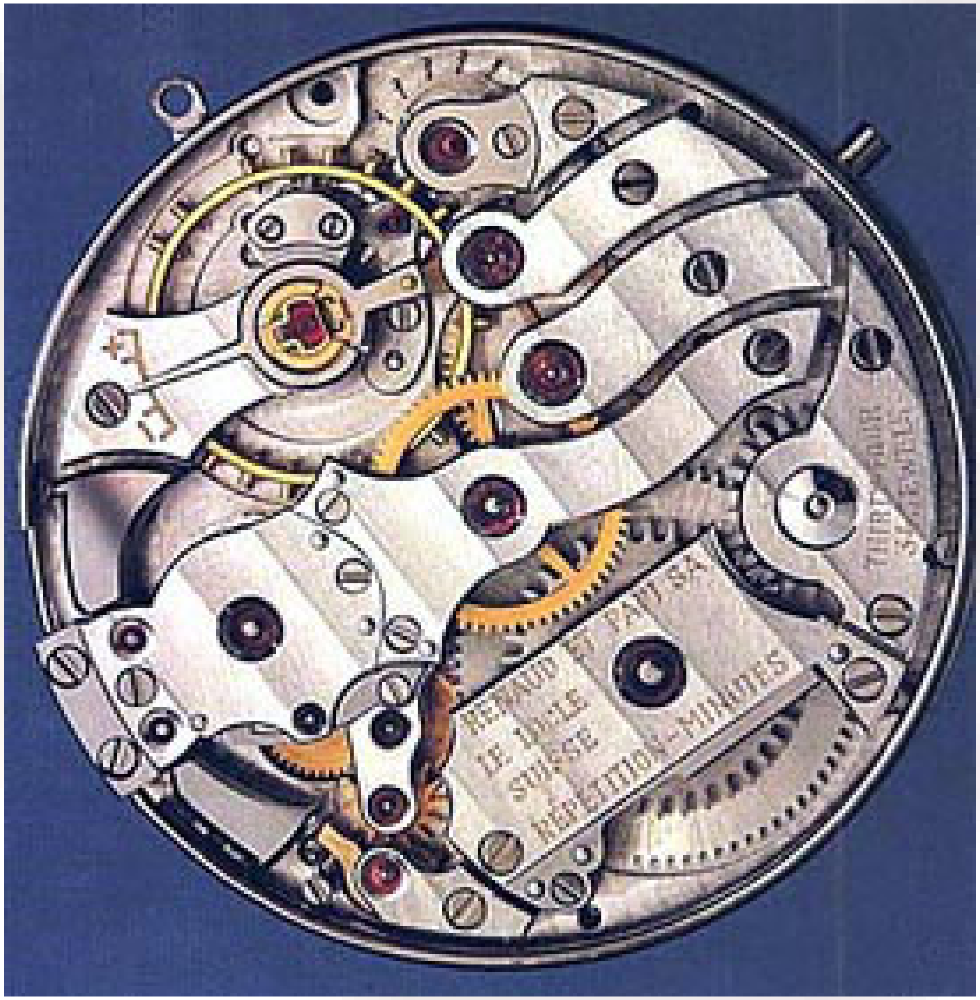
When does Tolerances become important

- Assemblies: Parts will often not fit together if their dimensions do not fall with in a certain range of values.
- Interchangeability: If a replacement part is used it must be a duplicate of the original part within certain limits of deviation.
- The relationship between functionality and size or shape of an object varies from part to part.

Tolerances do not affect its function

Tolerances are important here !

Food for thought: Tolerance levels in this mechanism?



Tolerance in relation to $$$$

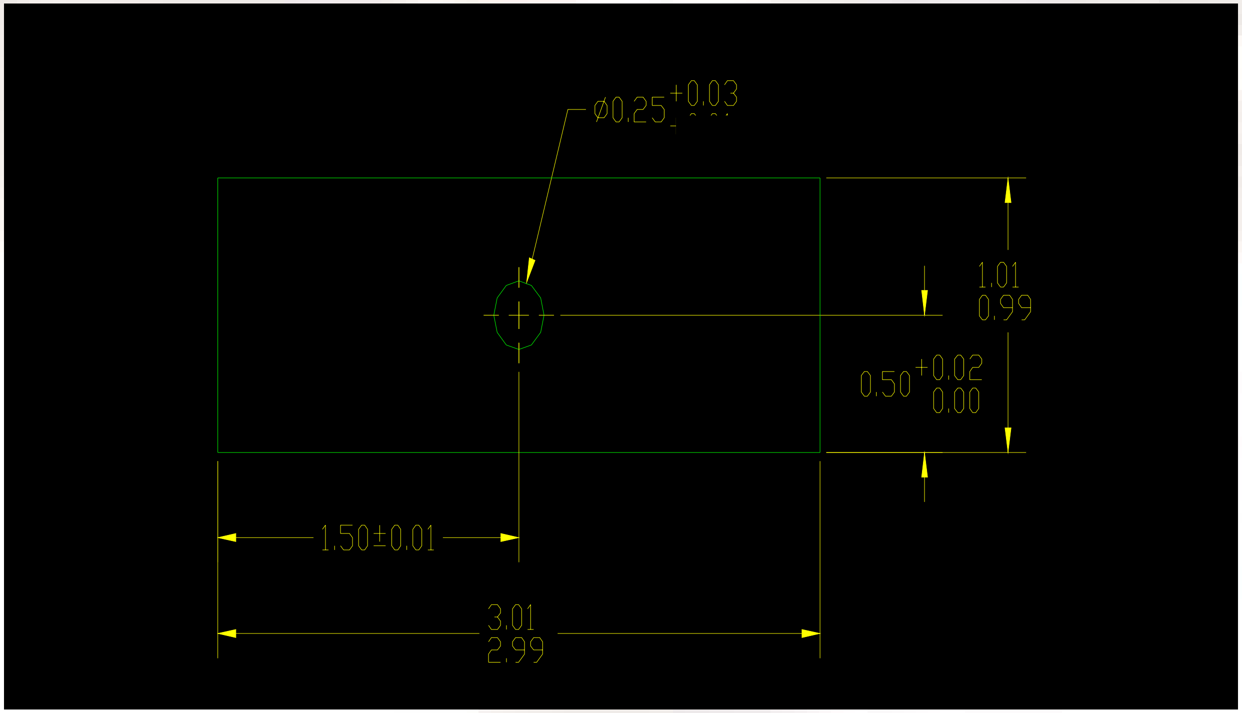
- Cost generally increases with smaller tolerance
  - Small tolerances cause an exponential increase in cost
  - Therefore your duty as an engineer have to consider : Do you need Φ1.0001in or is 1.01in good enough?
- Parts with small tolerances often require special methods of manufacturing.
- Parts with small tolerances often require greater inspection and call for the rejection of parts → Greater Quality Inspection → Greater cost.
- **Do not specify a smaller tolerance than is necessary!**

How are Tolerances Specified

- Size
  - Limits specifying the allowed variation in each dimension (length, width, height, diameter, etc.) are given on the drawing
- Geometry
  - Geometric Tolerancing
    - Allows for specification of 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

Value of Tolerance

- The tolerance for a single dimension may be specified with the dimension and then the tolerance.
  - The tolerance is total variation between the upper and lower limits.



General Tolerances

- These are specified when all dimension in the drawings have the same tolerance.
- These notes are used to reduce the number of dimensions required on a drawing and to promote drawing clarity.

1 EXCEPT WHERE STATED OTHERWISE
TOLERANCES ON DIMENSIONS ±.010

2 UNLESS OTHERWISE SPECIFIED
±.007 TOLERANCE ON MACHINED DIMENSIONS
±.10 TOLERANCE ON CAST DIMENSIONS
ANGULAR TOLERANCE ±.1°

Tolerances specified for size

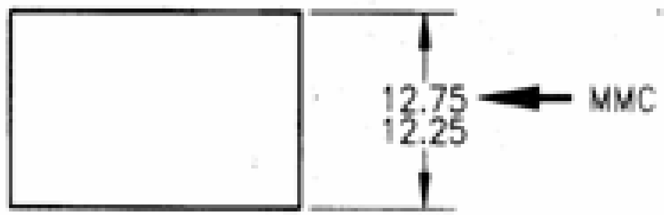
- Limit Tolerances – (12.75/12.25)
- Plus/Minus Tolerances
  - Unilateral Tolerances - (12.00 + or - xxx)
  - Bilateral Tolerances - (12.00 +xxx/- xxx)

These tolerance values indicate the:

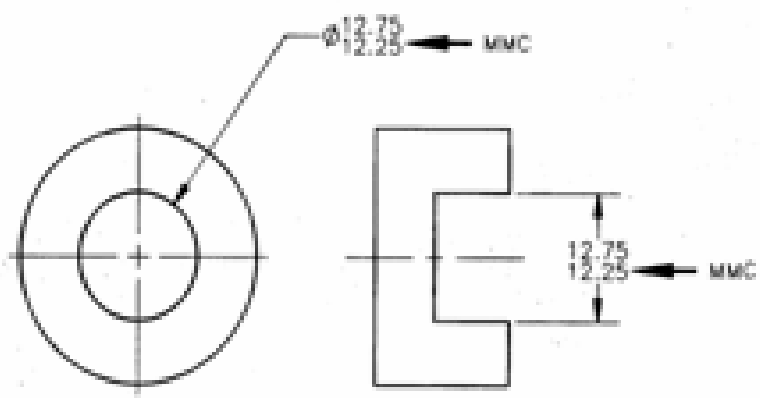
MMC: Maximum Material Condition

LMC: Least Material Condition

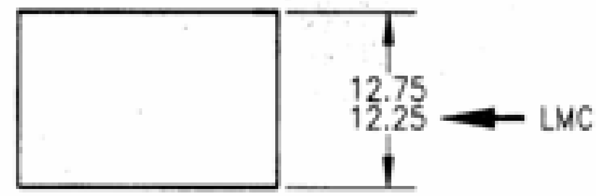
Limit Tolerances



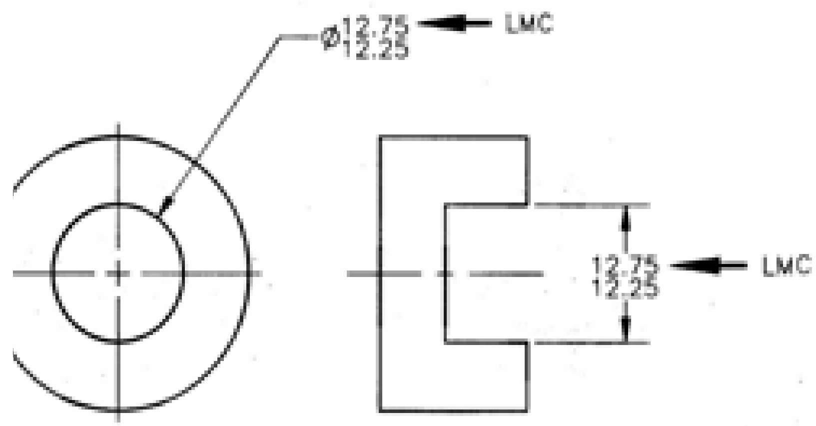
Example 1-8. Maximum Material Condition (MMC) for external features.



Example 1-9. Maximum Material Condition (MMC) for internal features.



Example 1-10. Least Material Condition (LMC) for external features.

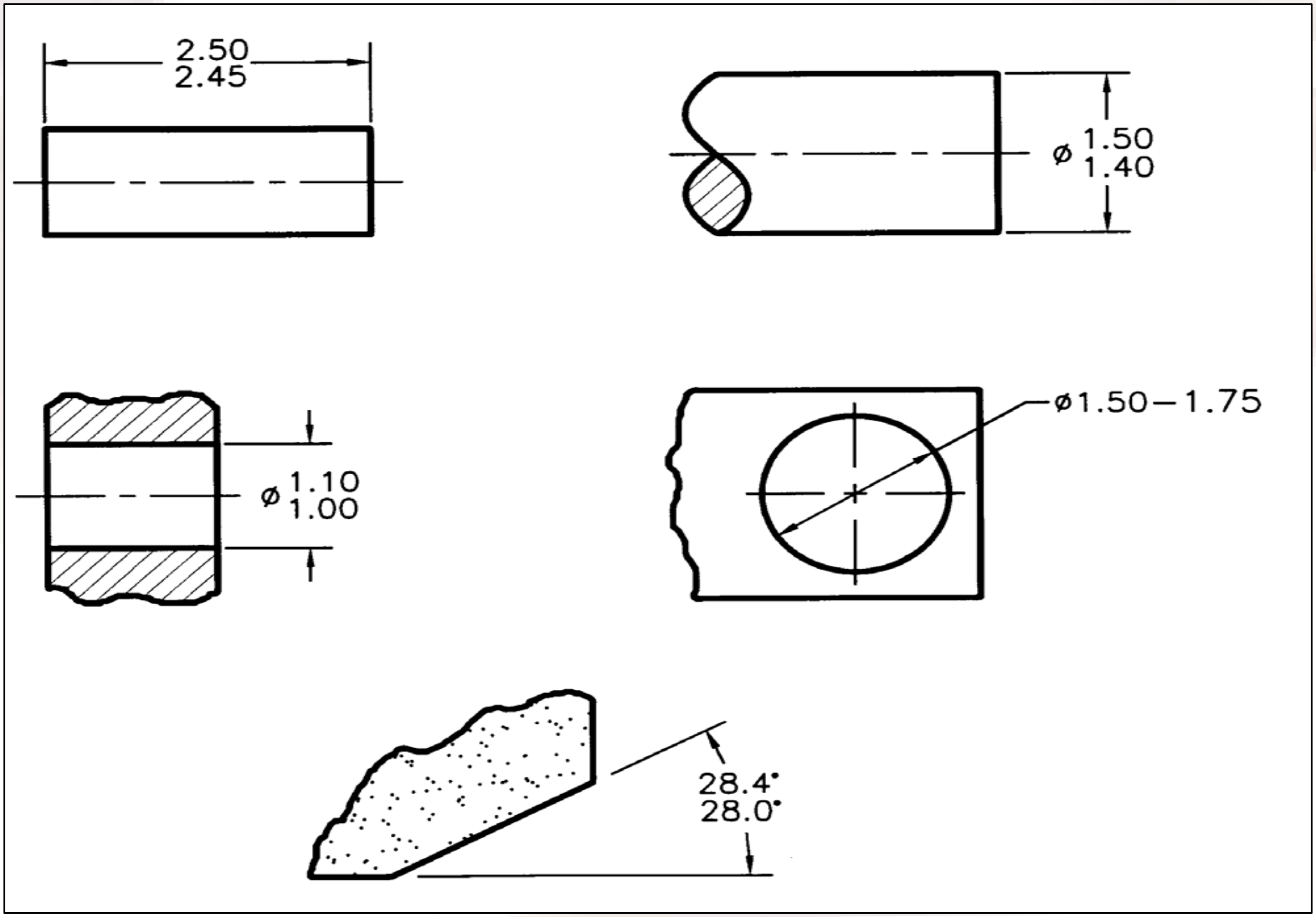


nple 1-11. Least Material Condition (LMC) for internal features.

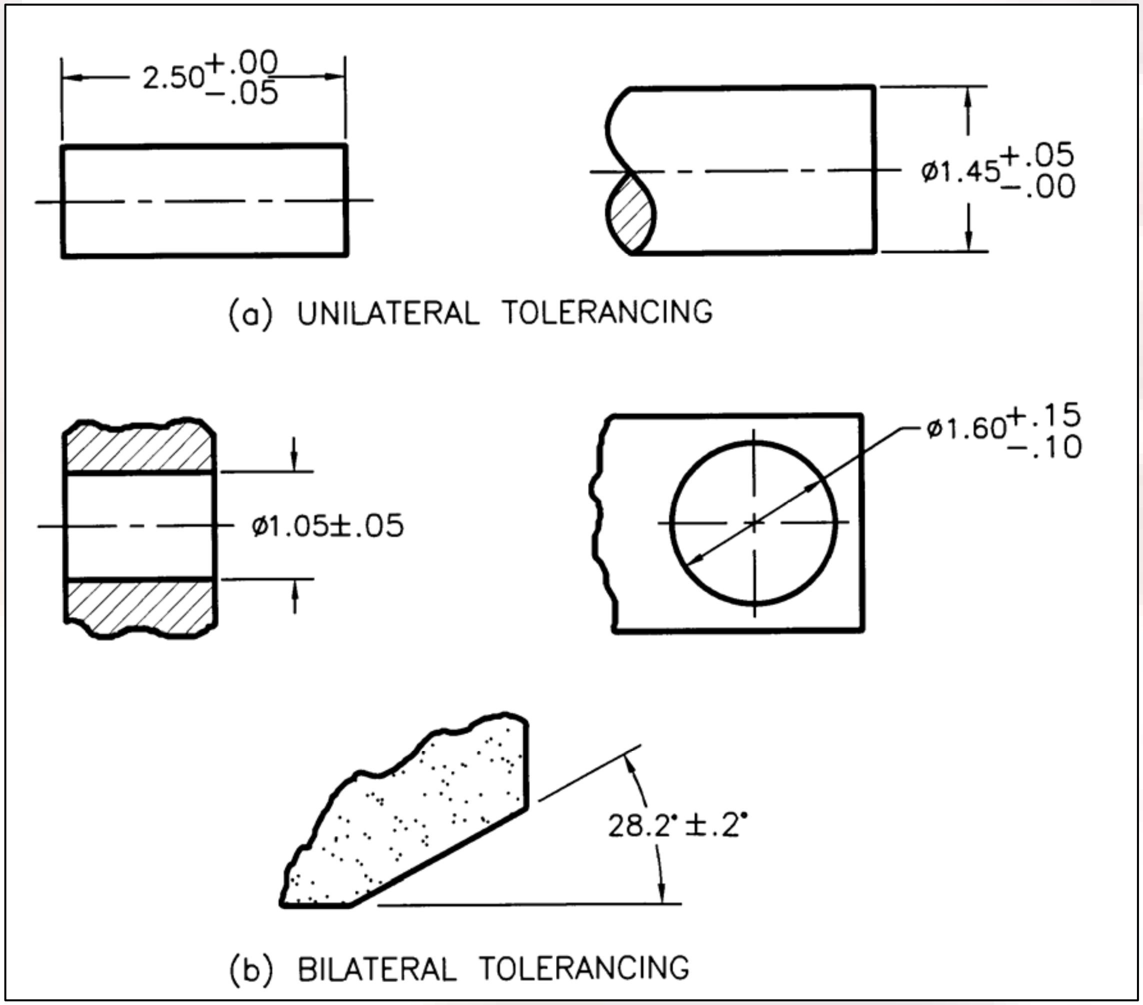
MMC: Maximum Material Condition

LMC: Least Material Condition

Limit Tolerances



Plus/Minus Tolerances



Department of Mechanical Engineering and Mechanics

Allowance and Clearance

- <u>**ALLOWANCE**</u>
- ***Allowance*** is defined as an intentional difference between the maximum material limits of mating parts. Allowance is the minimum clearance (positive allowance), or maximum interference (negative allowance) between mating parts. The calculation formula for allowance is:

  ALLOWANCE = MMC HOLE – MMC SHAFT

- <u>**CLEARANCE**</u>
- ***Clearance*** is defined as the loosest fit or maximum intended difference between mating parts.
- The calculation formula for clearance is:

  CLEARANCE = LMC HOLE – LMC SHAFT

Types of Fit

- Types of Fit
  - Clearance fit
    - The parts are toleranced such that the largest shaft is smaller than the smallest hole
    - The allowance is positive and greater than zero
  - Interference fit
    - The max. clearance is always negative
    - The parts must always be forced together
  - Transition fit
    - The parts are toleranced such that the allowance is negative and the max. clearance is positive
    - The parts may be loose or forced together

BASIC FITS OF MATING PARTS

Standard ANSI Fits:

Running and Sliding fits (RC) are intended to provide a running performance with suitable lubrication allowance. The range is from RC1 to RC9.

Force fits (FN) or Shrink fits constitute a special type of interference fit characterized by maintenance of constant pressure. The range is from FN1 to FN5.

A ***force fit*** is referred to as interference fit or a shrink fit. The smallest amount of interference is:

MIN INTERFERENCE = LMC SHAFT - LMC HOLE

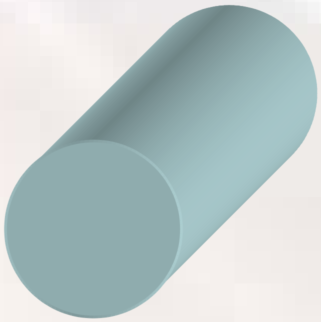
The greatest amount of interference is:

MAX INTERFERENCE = MMC SHAFT - MMC HOLE

Locational fits are intended to determine only the location of the mating parts.

Sample Calculation

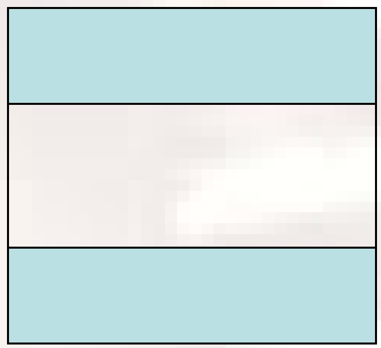
Given: Diameter of shaft: 1.5mm



Upper Limit Tolerance: 0.03mm

Lower Limit Tolerance : 0.04mm

Given: Diameter of Hole: 1.48mm



Upper Limit Tolerance : 0.03mm

Lower Limit Tolerance : 0.05mm

Allowance: MMC-Hole - MMC-Shaft

= 1.43 – 1.53 = - 0.1mm

Clearance: LMC-Hole – LMC-Shaft

= 1.51 – 1.46 = 0.05mm

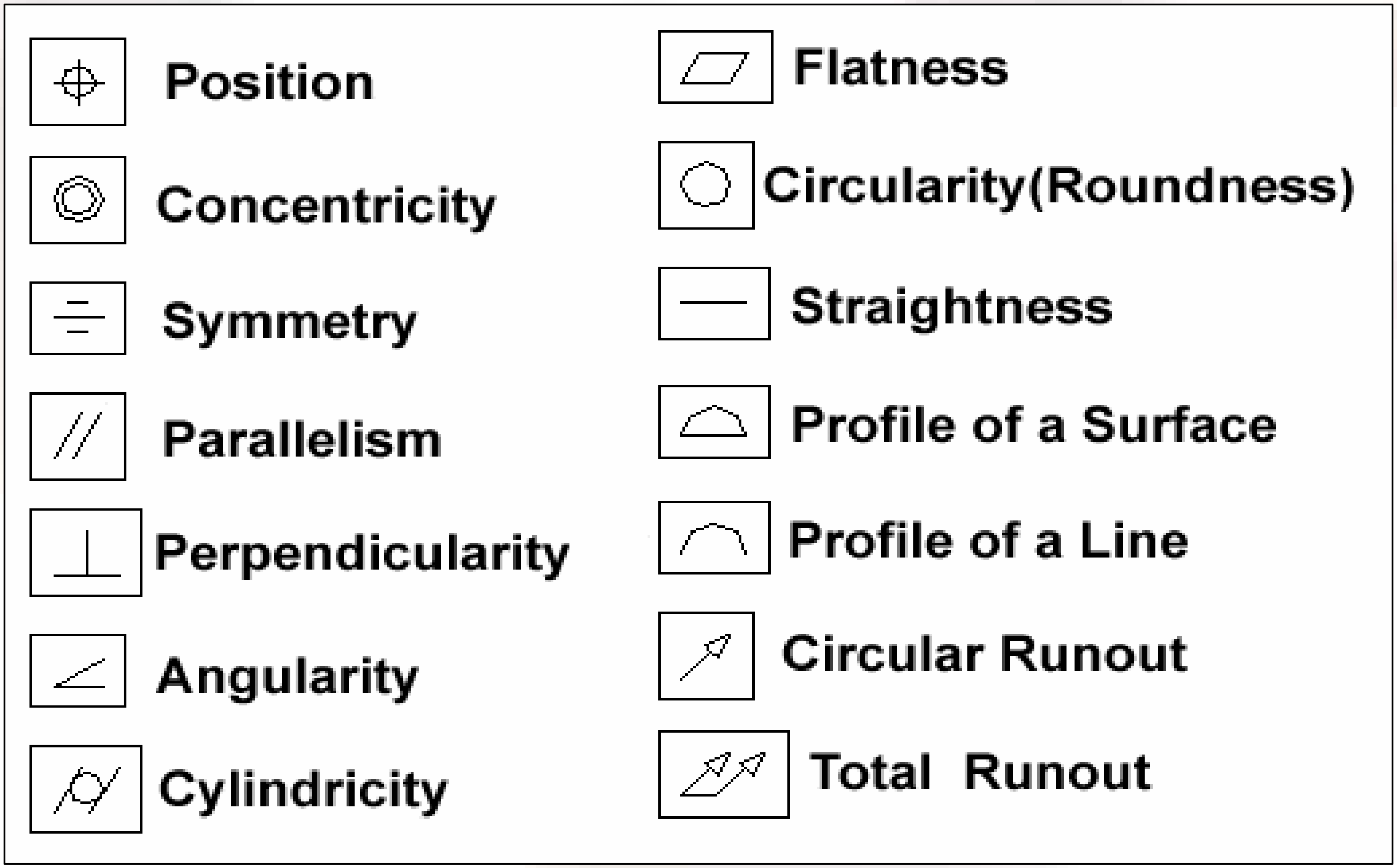
Answer

Allowance: -0.1mm

Clearance: 0.05mm

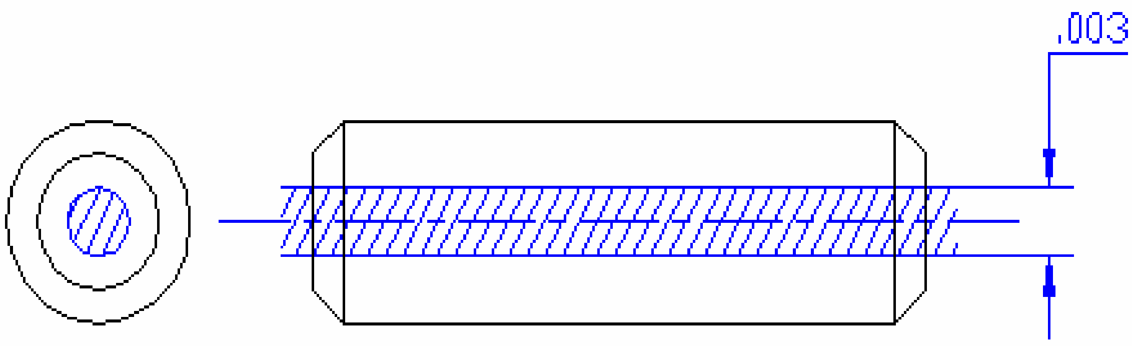
Type of Fit: Transition Fit

Geometric Dimensioning & Tolerancing (GD&T)

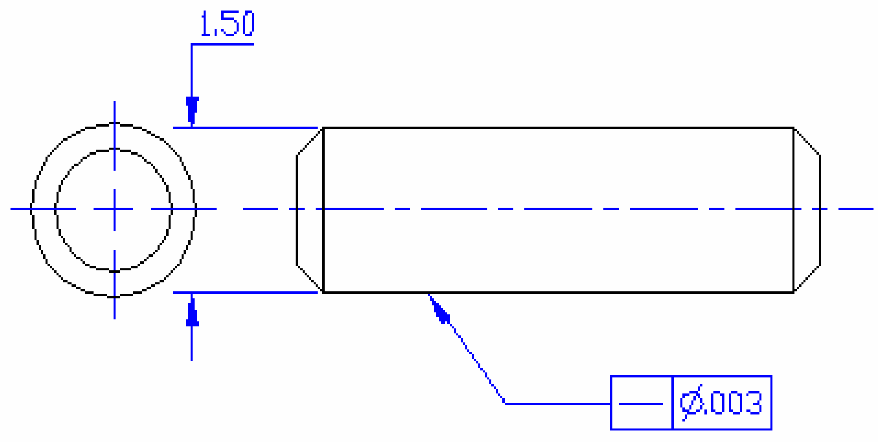


Tolerance of Form

Straightness

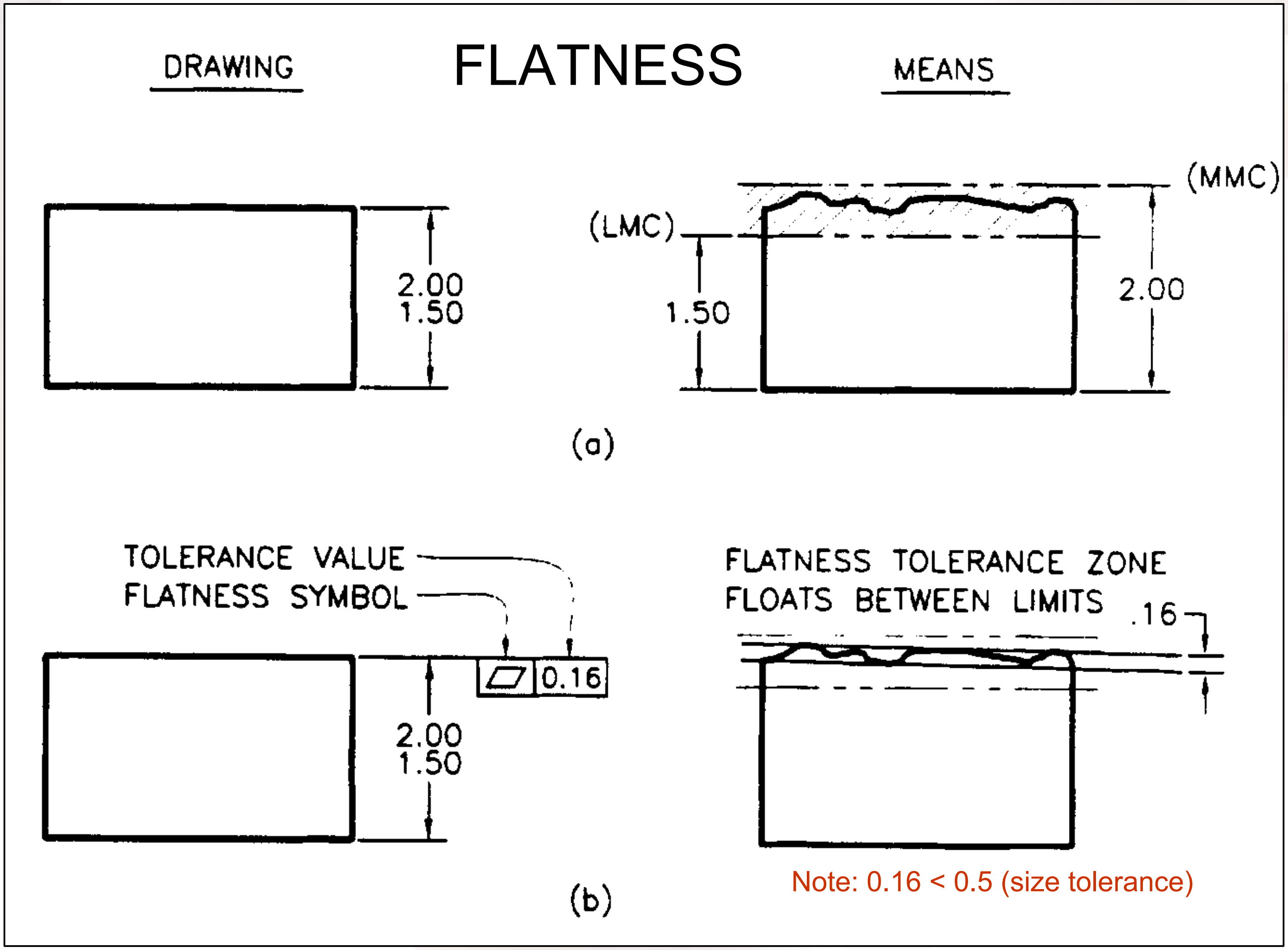


Straightness Tolerance Zone



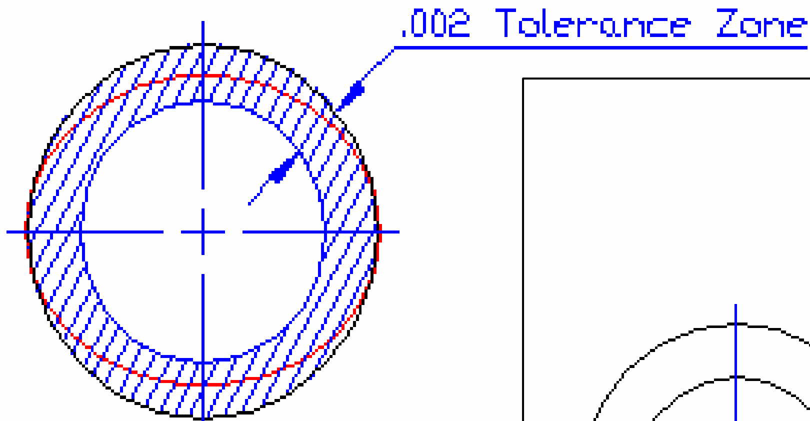
Straightness Tolerance

Tolerance of Form

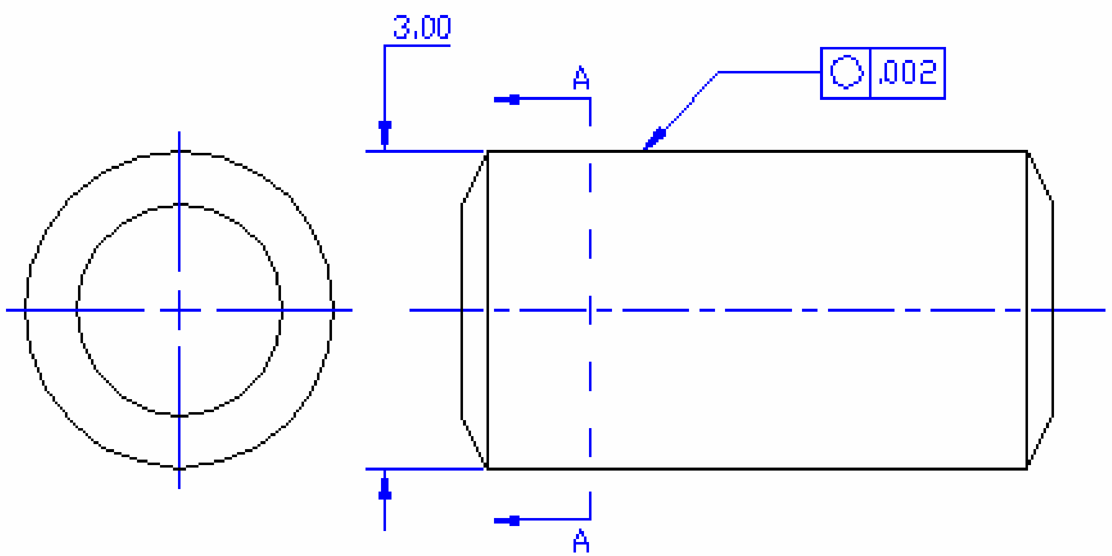


Tolerance of Form

Circularity ◯



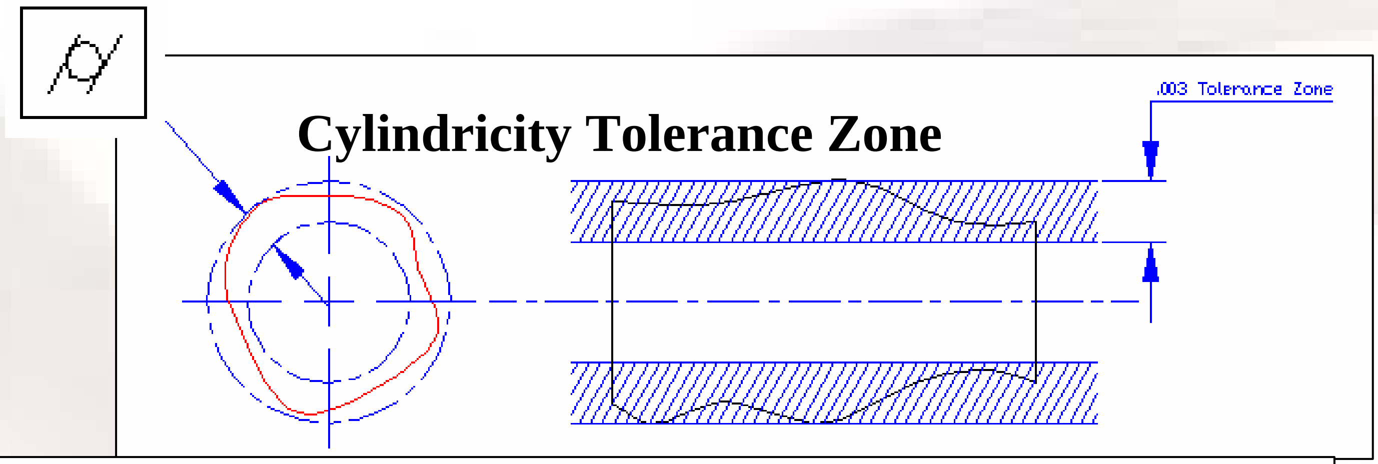
Circularity Tolerance Zone

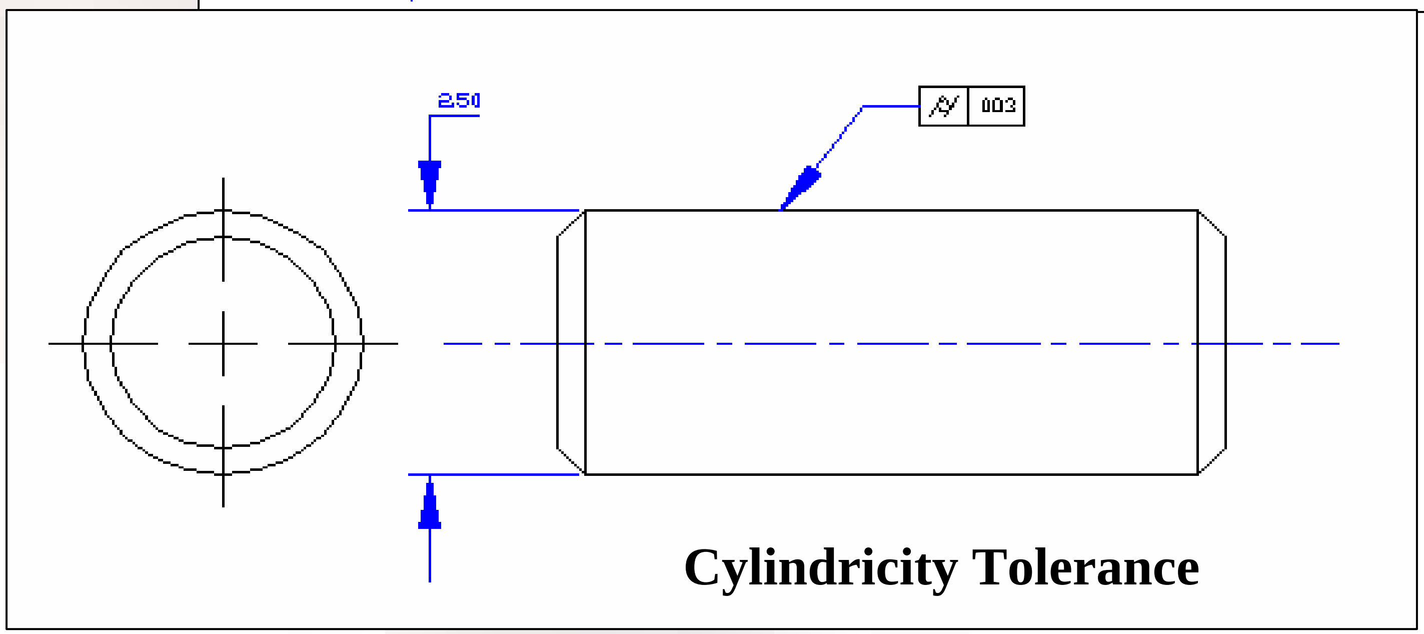


Circularity Tolerance

Tolerance of Form

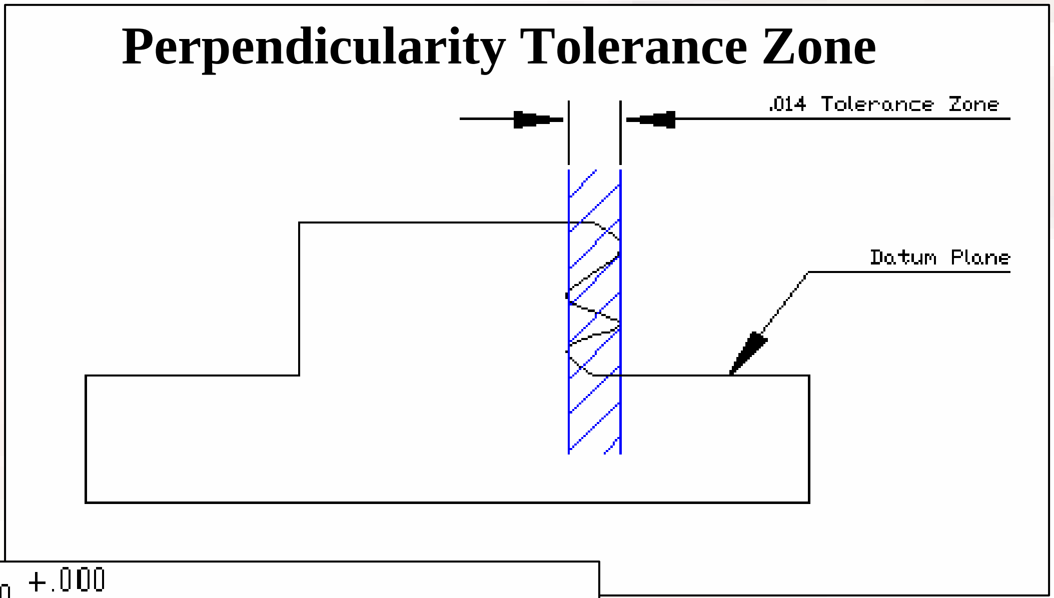
Cylindricity

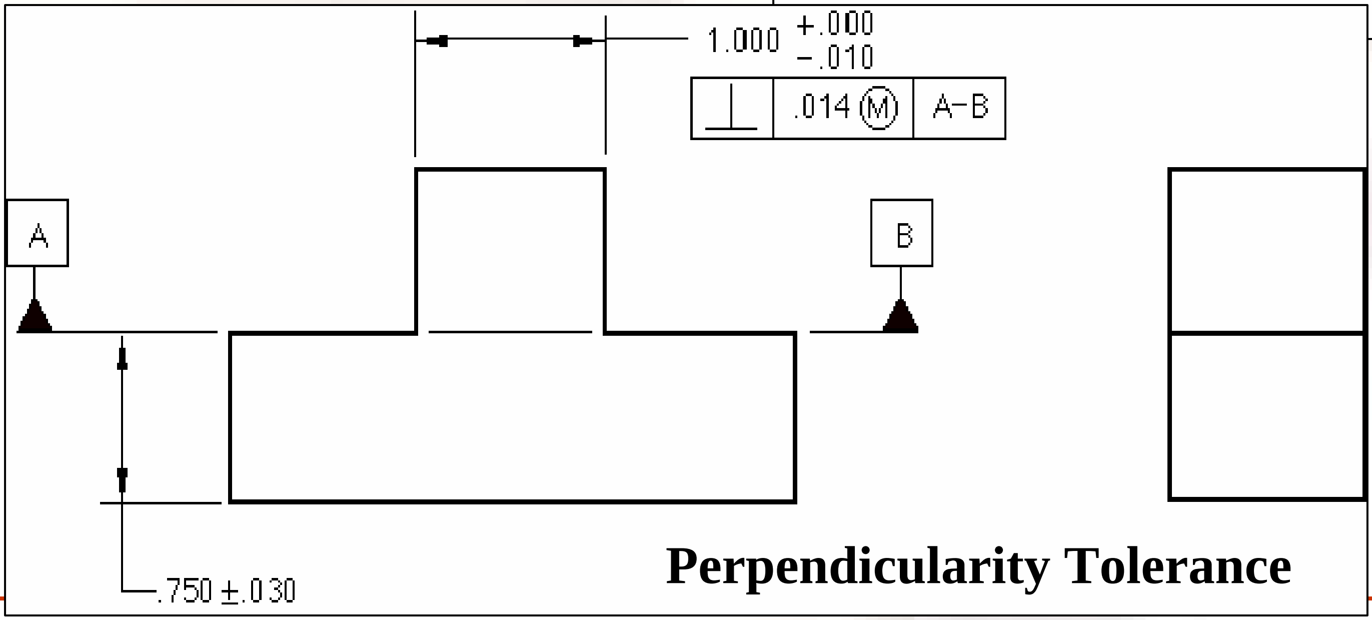




Tolerance of Orientation

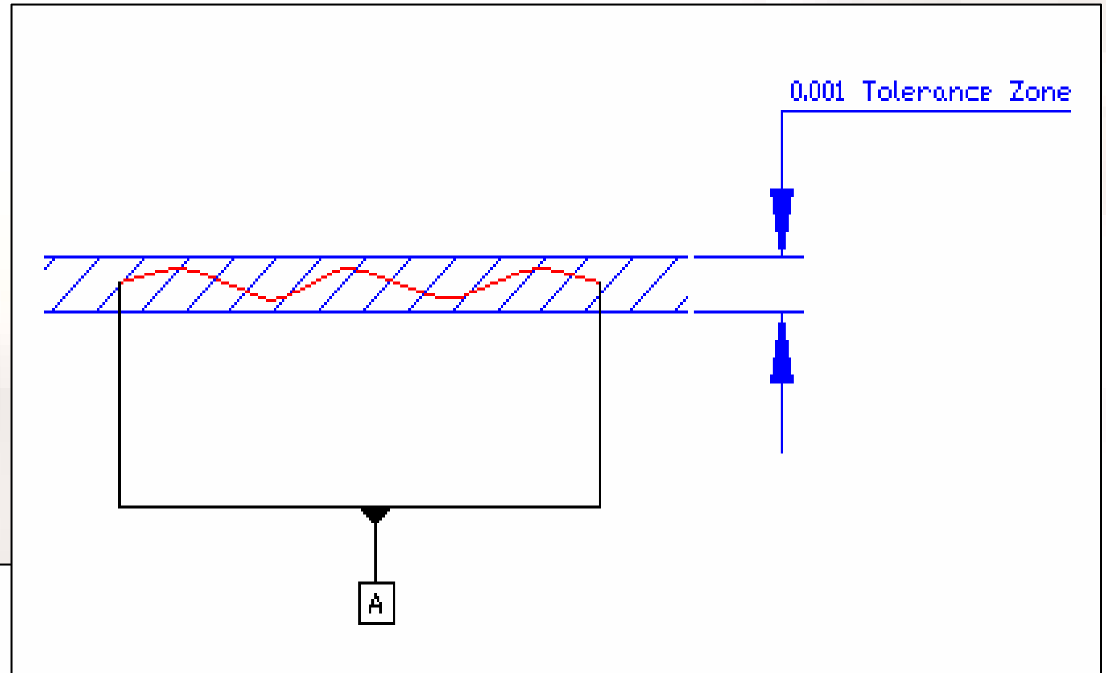
Perpendicularity ⊥

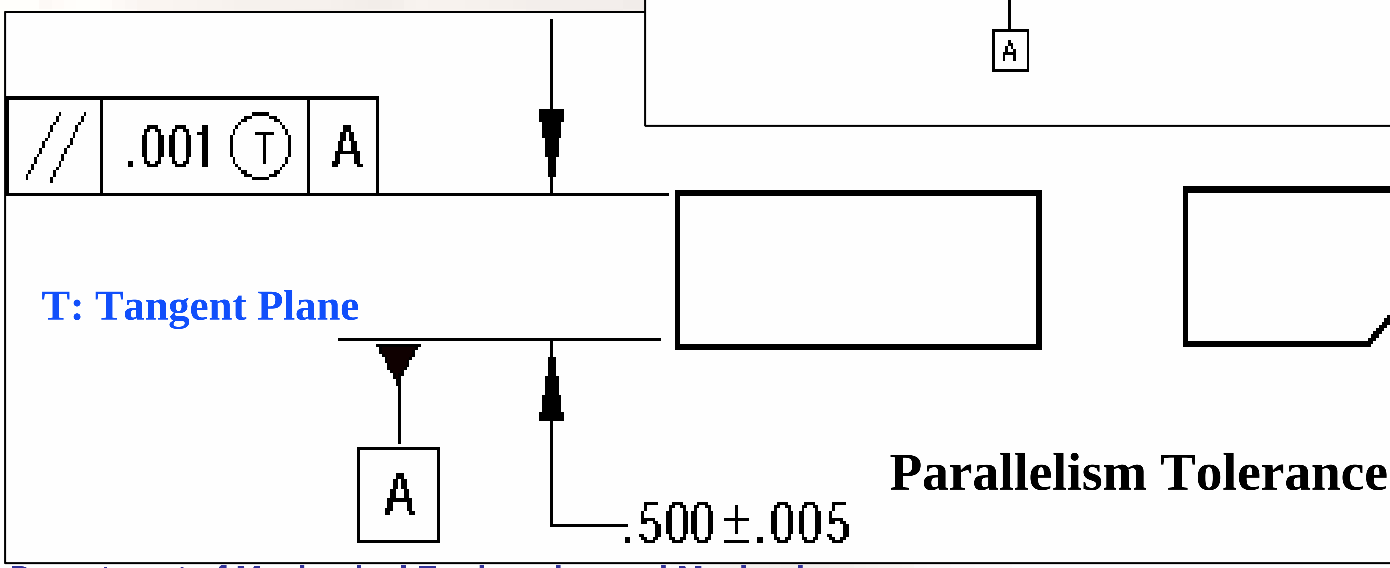




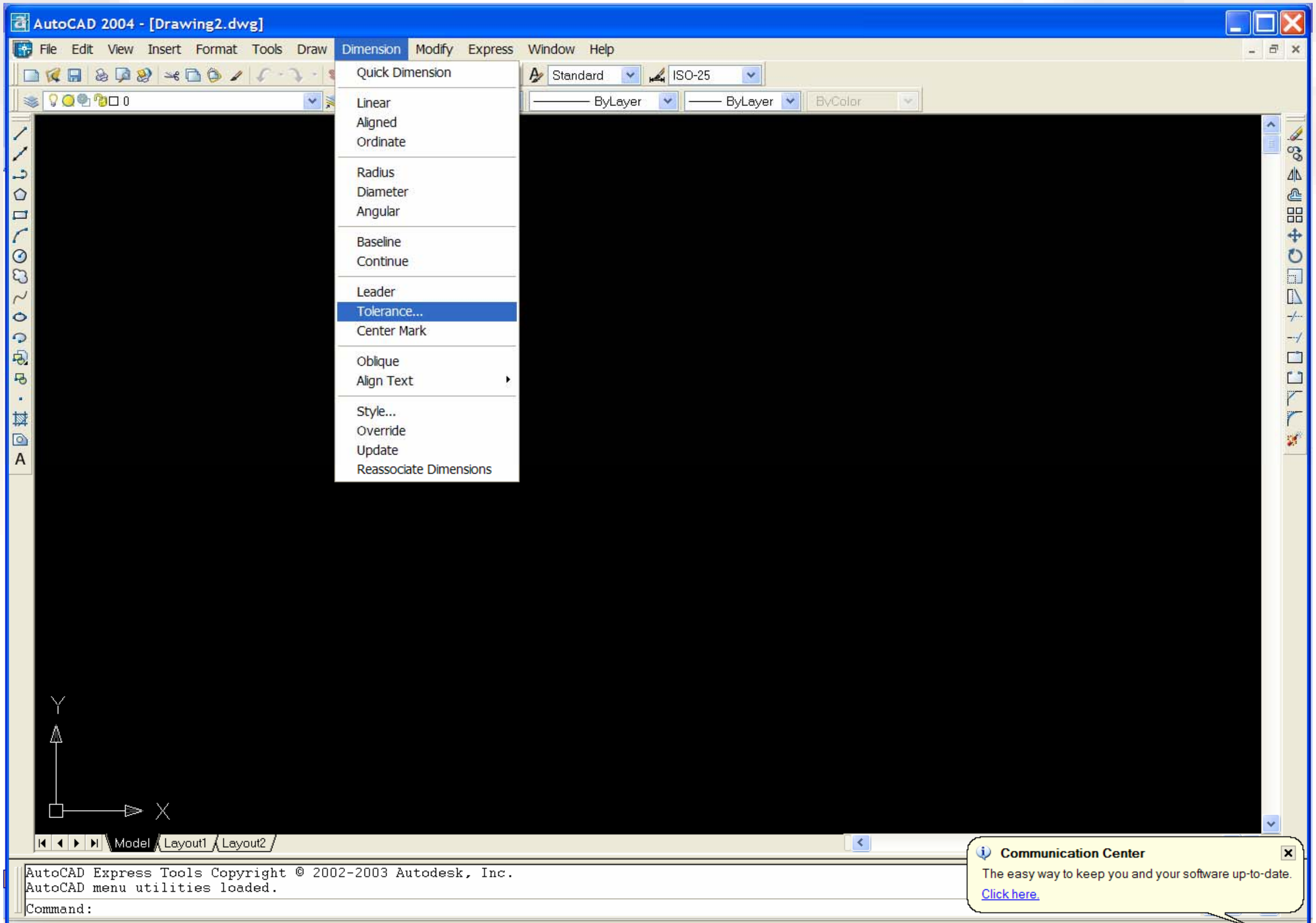
Tolerance of Orientation

Parallelism //

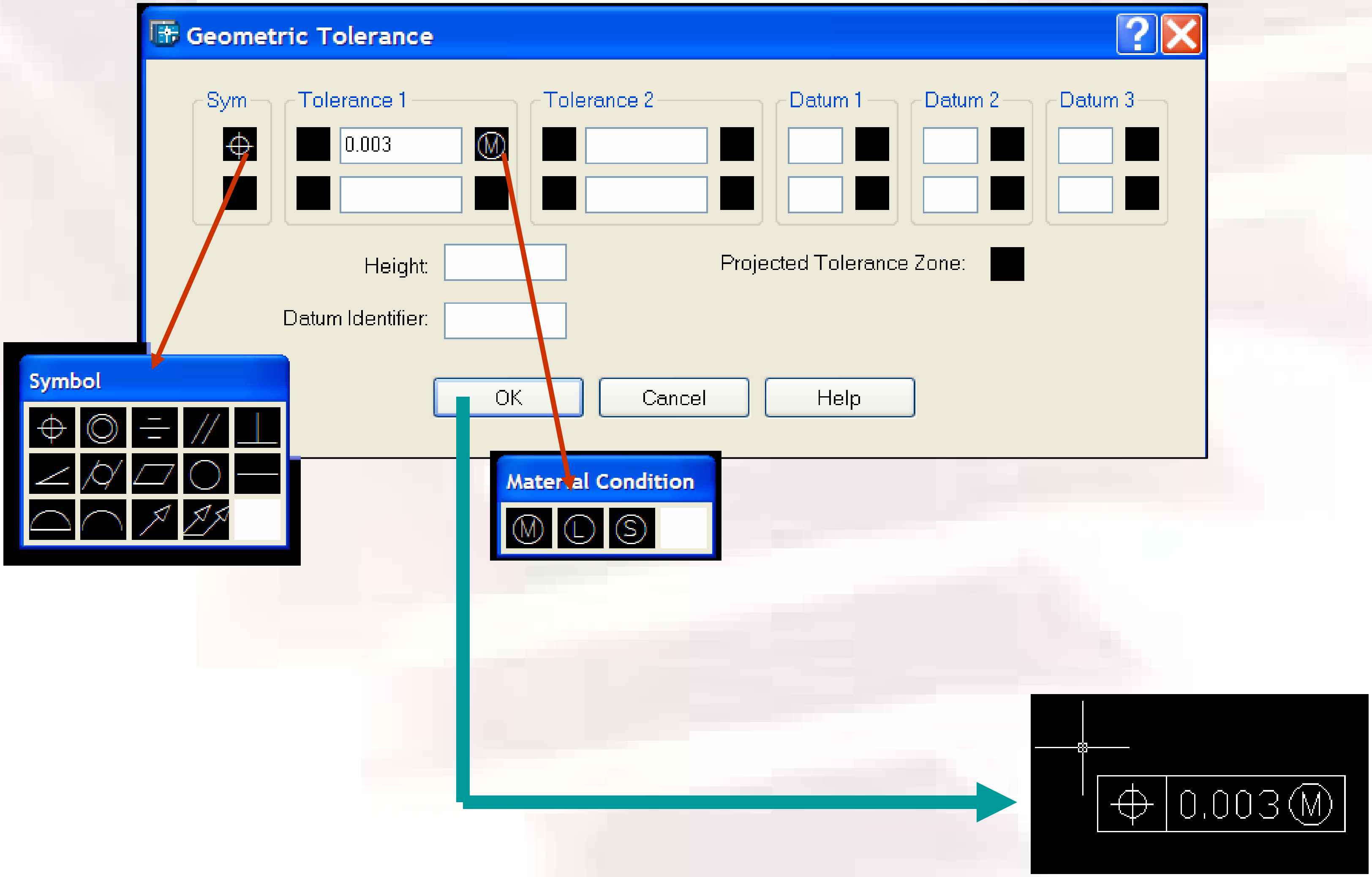




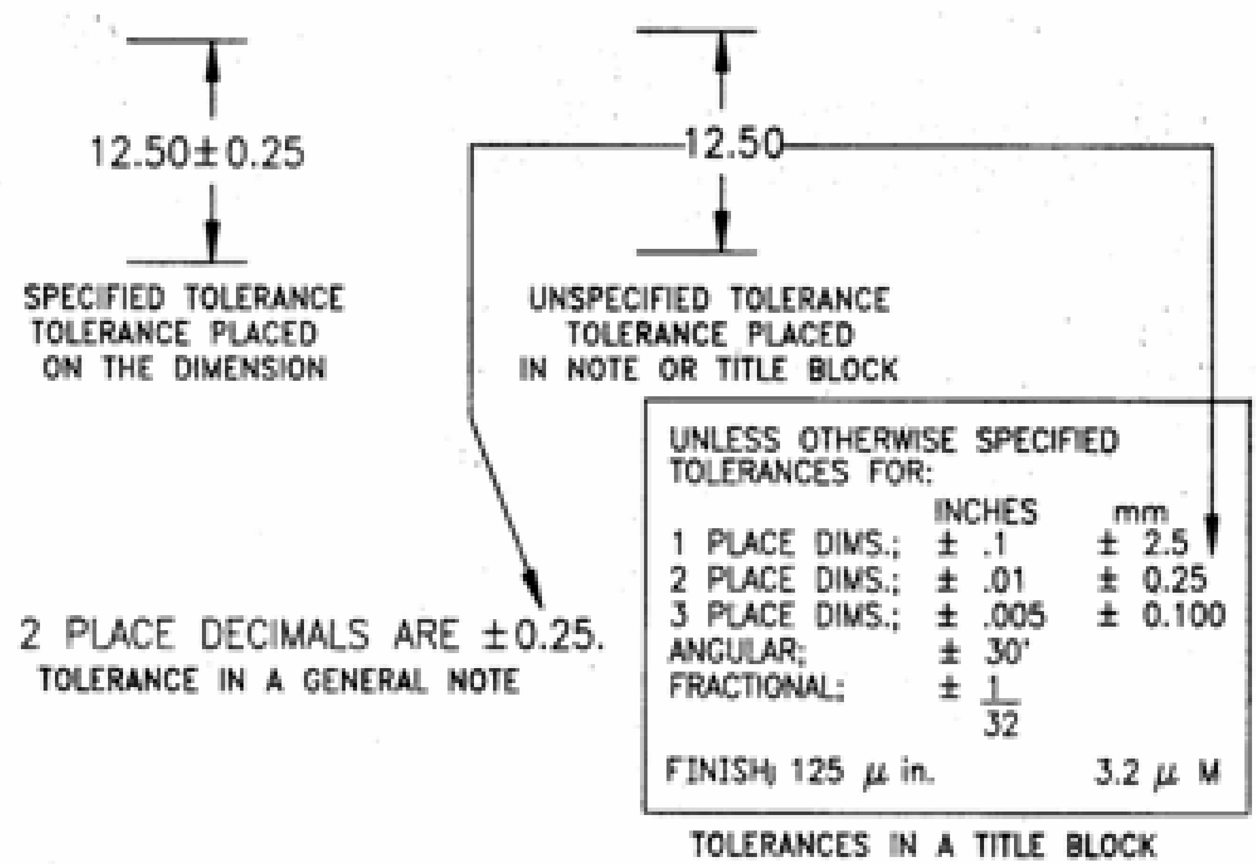
Tolerances in AutoCAD



Tolerances in AutoCAD



Tolerances in AutoCAD



Example 1-4. Specifying the tolerance on the dimension, in a general note, or in the drawing title block.

GEOMETRY DIMESIONING AND TOLERANCE FOR CADD/CAM

Some dimensioning and tolerance guidelines for use in conjunction with CADD/CAM:

- Geometry tolerancing is necessary to control specific geometric form and location.
- Major features of the part should be used to establish the basic coordinate system, but are not necessary defined as datum.
- Subcoordinated systems that are related to the major coordinates are used to locate and orient features on a part.
- Define part features in relation to three mutually perpendicular reference plans, and along features that are parallel to the motion of CAM equipment.
- Establish datum related to the function of the part, and relate datum features in order of precedence as a basis for CAM usage.
- Completely and accurately dimension geometric shapes. Regular geometric shapes may be defined by mathematical formulas. A profile feature that is defined with mathematical formulas should not have coordinate dimensions unless required for inspection or reference.
- Coordinate or tabular dimensions should be used to identify approximate dimensions on an arbitrary profile.
- Use the same type of coordinate dimensioning system on the entire drawing.
- Continuity of profile is necessary for CADD. Clearly define contour changes at the change or point of tangency. Define at least four points along an irregular profile.
- Circular hole patterns may be defined with polar coordinate dimensioning.
- When possible, dimension angles in degrees and decimal parts of degrees.
- Base dimensions at the mean of a tolerance because the computer numerical control (CNC) programmer normally splits a tolerance and works to the mean. While this is theoretically desirable, one can not predict where the part will be made. Dimensions should always be based on design requirements. If it is known that a part will be produced always by CNC methods, then establish dimensions without limits that conform to CNC machine capabilities. Bilateral profile tolerances are also recommended for the same reason.

Further Reading……

- Interpretation of Geometric Dimensioning & Tolerancing by Daniel E. Puncochar
- Geometric Dimensioning and Tolerancing by Alex Krulikowski
- Geo-Metrics III : The Application of Geometric Dimensioning and Tolerancing Techniques (Using the Customary Inch Systems) by Lowell W. Foster
- Tolerance design : a handbook for developing optimal specifications by C.M. Creveling.
- Dimensioning and Tolerancing Handbook by Paul J. Drake
- Inspection and Gaging by Clifford W. Kennedy
- Geometric Dimensioning and Tolerancing by Cecil H. Jensen
- Tolerance Stack-Up Analysis by James D. Meadows

Home Work #2

1. Find T_H, T_s, Allowance, C_{max}, C_{min}, and what kind of fit it is ?
   Hole F 66 upper deviation +0.051, lower deviation 0.0
   Shaft F 66 upper deviation -0.024, lower deviation -0.050
2. Find T_H, T_s, Allowance , C_{max}, I_{max}, and what kind of fit it is ?
   Hole F 32 upper deviation +0.021, lower deviation 0.0
   Shaft F 32 upper deviation +0.029, lower deviation +0.016
3. If a shaft is 10±0.05 inch what is its maximum and least material conditions.
4. Please draw circularity and perpendicularity symbol blocks with geometric tolerance of 0.005 for each, and sketch their tolerance zones for a cylinder and a upside down T shape block respectively.

Home Work #2 contd..

T_h= tolerance of hole
T_s= Tolerance of shaft
C_{max}= Maximum clearance
C_{min}= Minimum clearance
I_{max}= Maxiumum interference

F66 and F32 indicates the nominal dimensions of the hole or shaft

- **Refer Notes and AutoCAD text book for help in solving problems.**
- **Home works should include your names and the section you belong to.**
- **Will be due during the next Lecture Class.**