DASSAULT SYSTEMES

GD&T ENHANCEMENTS:

Navigating the Latest Position Tolerancing Updates in ASME Y14.5

3DEXPERIENCE

3DEXPERIENCE®
WORLD

Dr. Thomas Allsup
Assistant Professor of Instruction
University of Texas at Arlington
thomas.allsup@uta.edu

ORIGINAL TITLE:

THREE RULES OF GD&T ARE LOCATION, LOCATION, LOCATION

Stolen from Dr. Raul Fernandez, UTA MAE

PREVIOUSLY ON SW/3DEXPERIENCE WORLDS:

- 2009 How to Spell GD&T (geometric controls)
- 2010 The Revenge of the Circled Letters (modifiers)
- 2013 Hitchhiker's Guide to GD&T (tolerance zones)
- 2014 The Big GD&T Theory (datums)
- 2015 I Know Everything About GD&T Except…
- 2016 How to Measure GD&T (inspection)

AGENDA

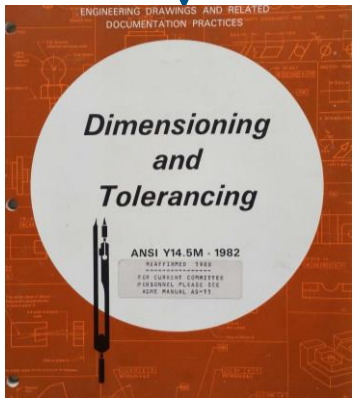
- Standard History
- Positional Tolerance
- Datum Translation
- FRTZF & PLTZF
- Projected Tolerance Zone
- Zero Position Tolerance
- Discussion

WHY IS POSITIONAL TOLERANCE IMPORTANT?

- Stanley Parker, in the first book on Geometric Dimensioning and Tolerances in1956, ***Drawings and Dimensions*** described the issues with creating inspection gauges based on traditional plus or minus tolerances while assembling ordinance in World War II.
- He needed a better circular tolerance zone based on the “true position” of the feature which formed the basis of positional geometric control.
- Although Stanley Parker did not propose any symbols, I find it humorous that the symbol chosen for the most important tolerance looks like an ordinance bullseye.

THE ASME GD&T STANDARD

A Brief History

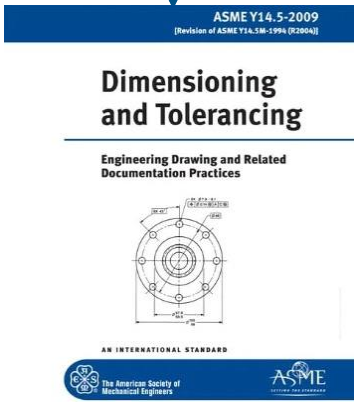


ANSI Y14.5M-1982

- Added “M” for Metric system

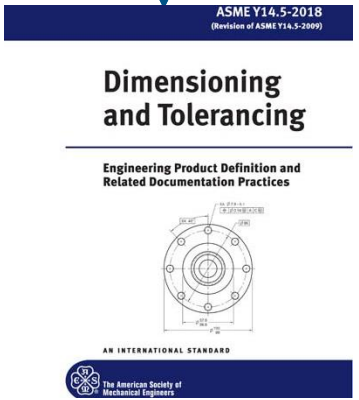
ASME Y14.5M-1994

- Change from ANSI to ASME



ASME Y14.5-2009

- Major changes to standard’s organization



ASME Y14.5-2018

- Removal of symmetry and concentricity
- Reaffirmed in 2024

DASSAULT SYSTEMES

“INCORPORATED BY REFERENCE” LEGAL CHALLENGES

- ASME is a Standards Development Organization (SDO)
- SDO’s develop “voluntary consensus standards”
- SDOs have substantial costs for developing and maintaining standards
- Revenues from the sales and licensing of copyrighted standards help fund SDOs (along with training and certification)
- The IBR (Incorporated by Reference) – Code of Federal Regulations (CFR) lawsuits are in flux
- This impacted not only ASME, but also IEEE, ANSI, AWS, and many more.

CURRENT STANDARD?

- *When you start using the new standard, your drawing formats should be revised to have words like:*
  - “Interpret this drawing using ASME Y14.5-2018"
- *Don't throw your old standards away:*
  - You might need it to interpret the drawings you created or receive from others that were created using an old standard.
  - Don’t use the drawing date to determine what standard to use, look for the note on the drawing.

MODEL BASED DEFINITION

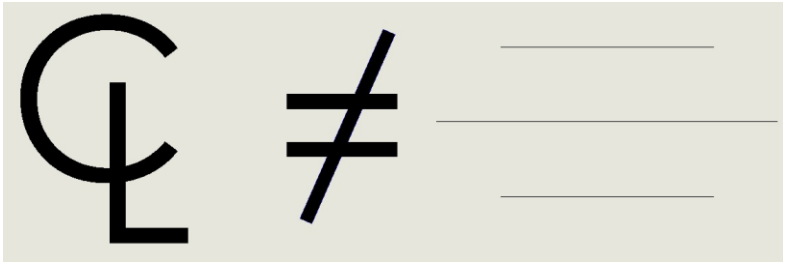
- The concepts of dimensions and tolerances do not depend upon there being a 2D drawing or not.
- Start getting in the habit of modeling your parts with the linear tolerances and GD&T in 3D.
- You can decide later if you want to
  - “Show” dimensions in your 2D drawings and reduce your drafting work
  - “Publish” your 3D model with tolerances and eliminate your drafting work

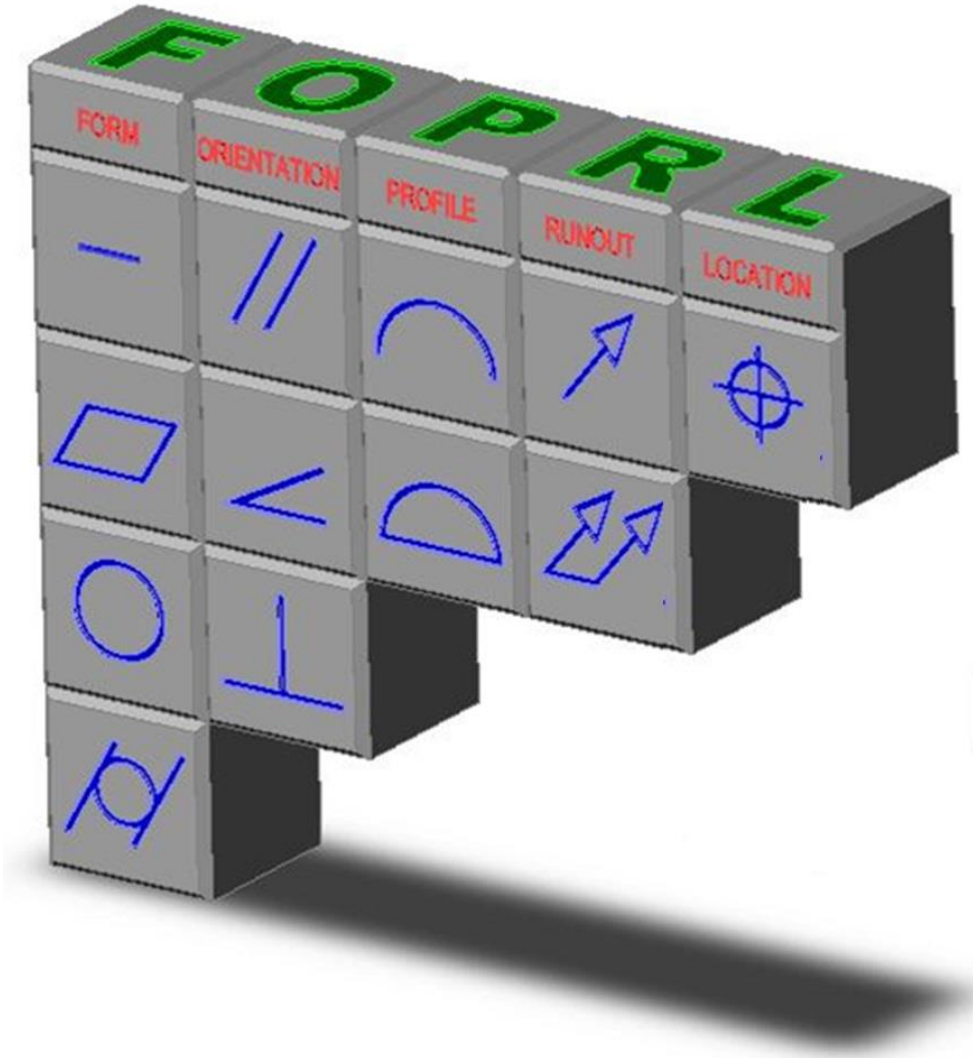
BIGGEST CHANGE

- The two most misapplied, misunderstood, and hard to measure symbols have been removed from the standard.
- No more Symmetry
  - You probably should have used position anyway.
- No more Concentricity
  - You probably should have used runout anyway.

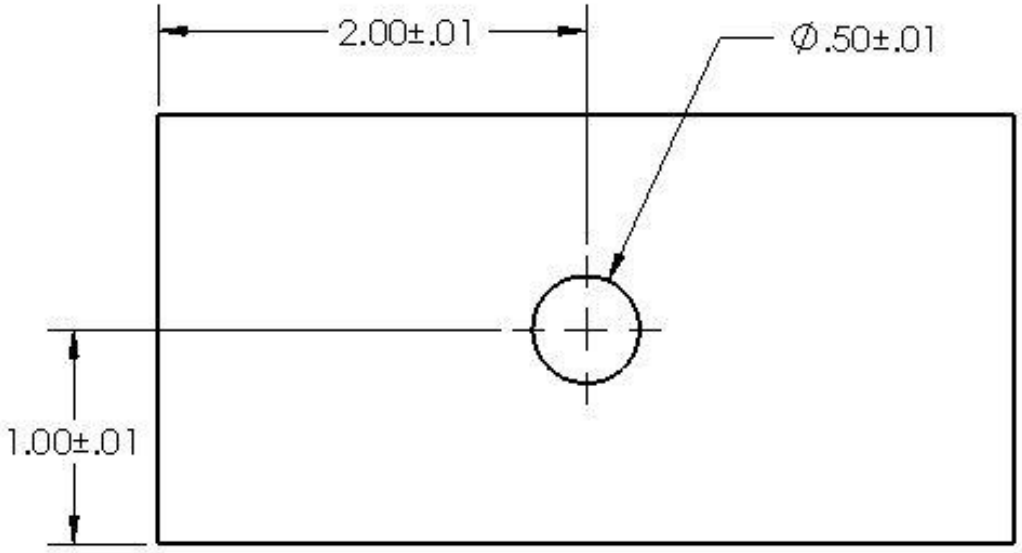
SYMMETRICAL OUTLINES

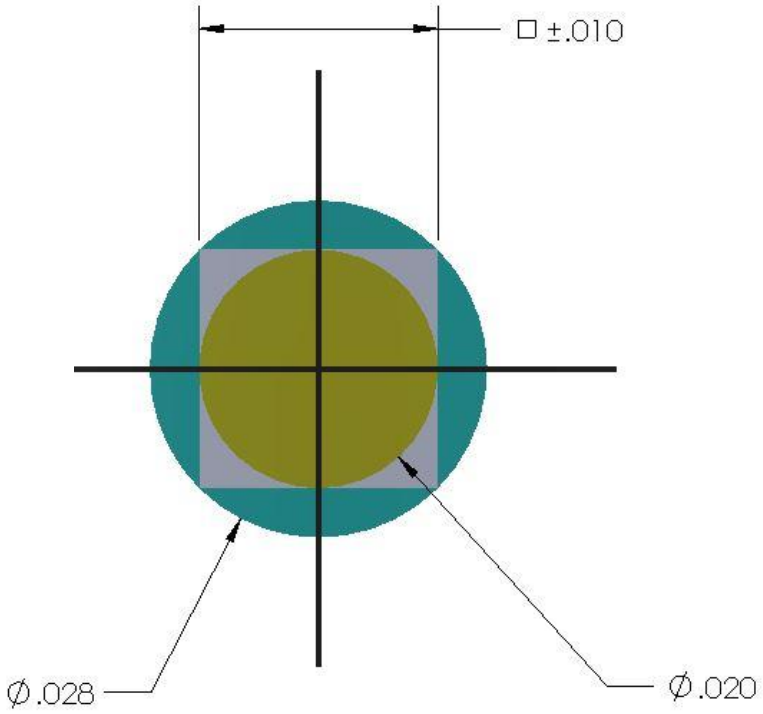
- Centerline not the same as GD&T control symmetry
- Incidentally, the symbol CL has been replaced with a “equal sign” on the center line somewhere along the way in the ISO1101.
  - Imagine me saying in an old man voice “I don’t like that change”…



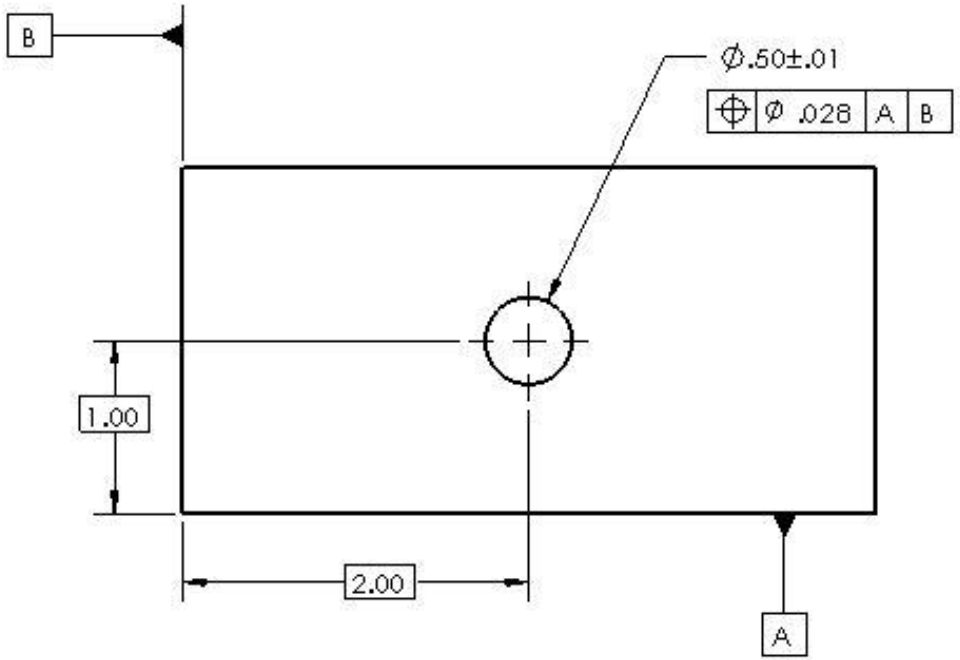


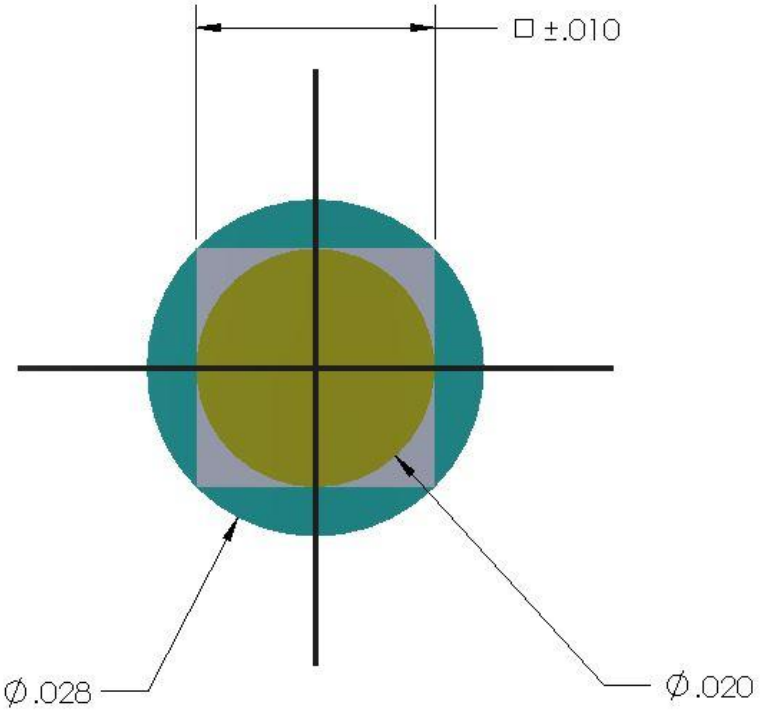
WHAT DO WE MEAN BY POSITIONAL TOLERANCE?





CONVERTING TO POSITIONAL TOLERANCE



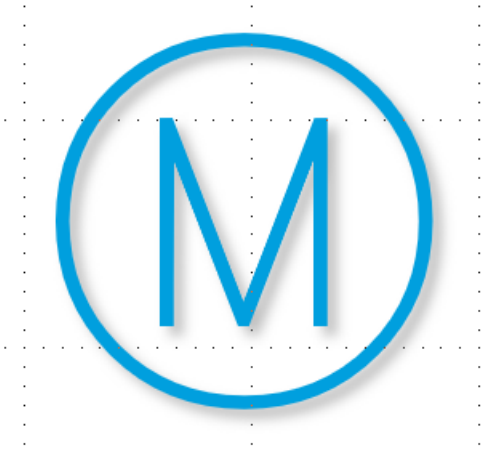


FIRST BONUS TOLERANCE

- Switching from traditional cartesian tolerances to GD&T polar tolerances either:
  - Tightens a feature by taking the inscribed circle within the square tolerance.
  - Loosens a feature by taking the circle with the diagonal of the square tolerance.
- This tolerance refinement is the reason for the polar to rectangular conversion charts.

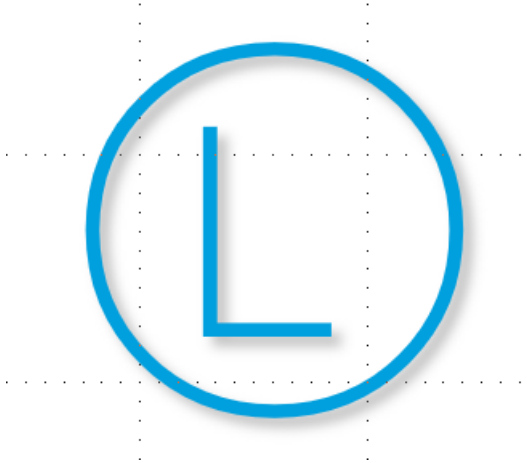
MAXIMUM MATERIAL CONDITION

- The stated tolerance applies when the most material is there.
  - The tolerance zone increases when there is less material – you get ***bonus tolerance*** if a hole is large.
- Examples
  - Thickest plate
  - Smallest hole

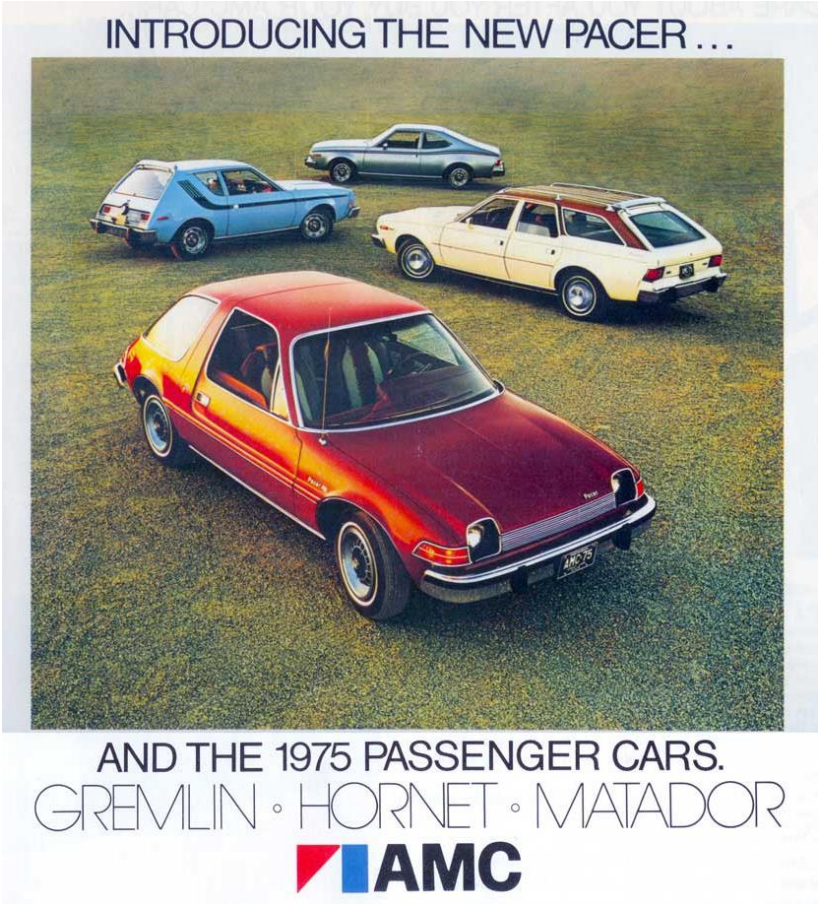


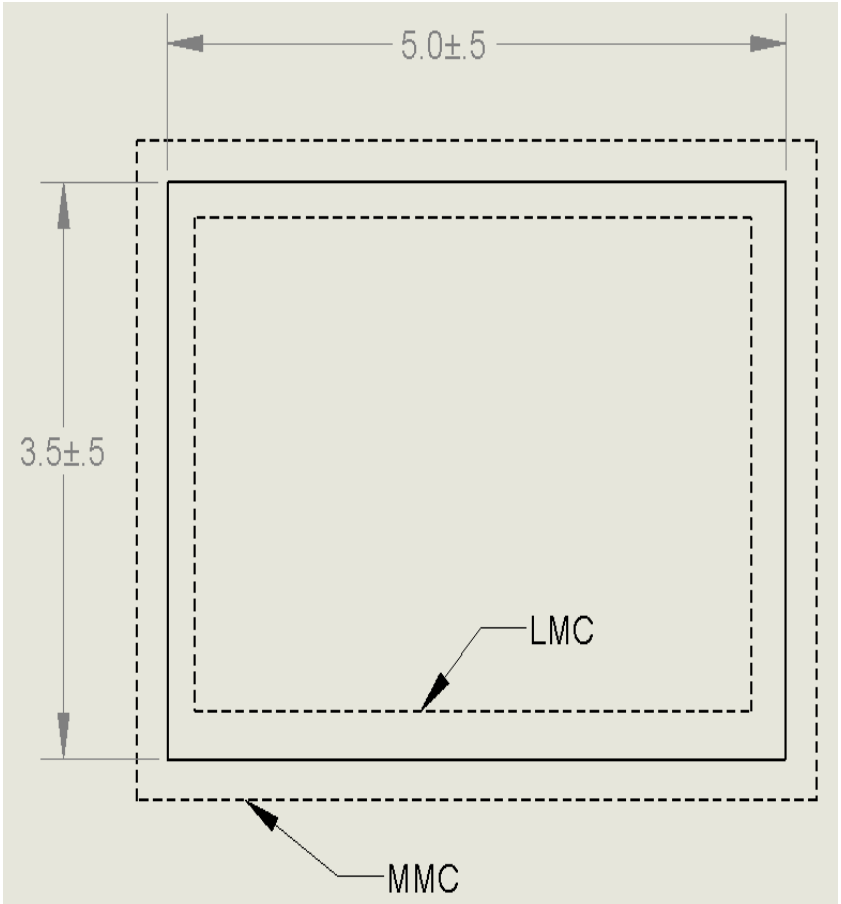
LEAST MATERIAL CONDITION

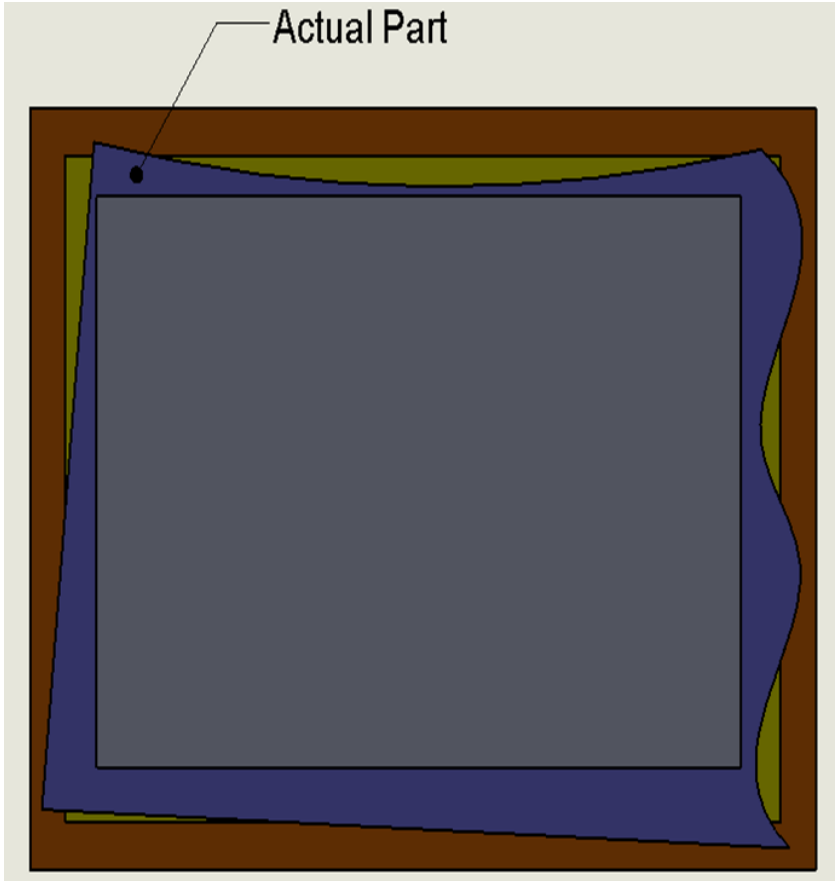
- The stated tolerance applies when the least material is there.
  - This is a rarely used modifier.
- Examples:
  - Thinnest plate
  - Largest hole



AMC didn’t do LMC





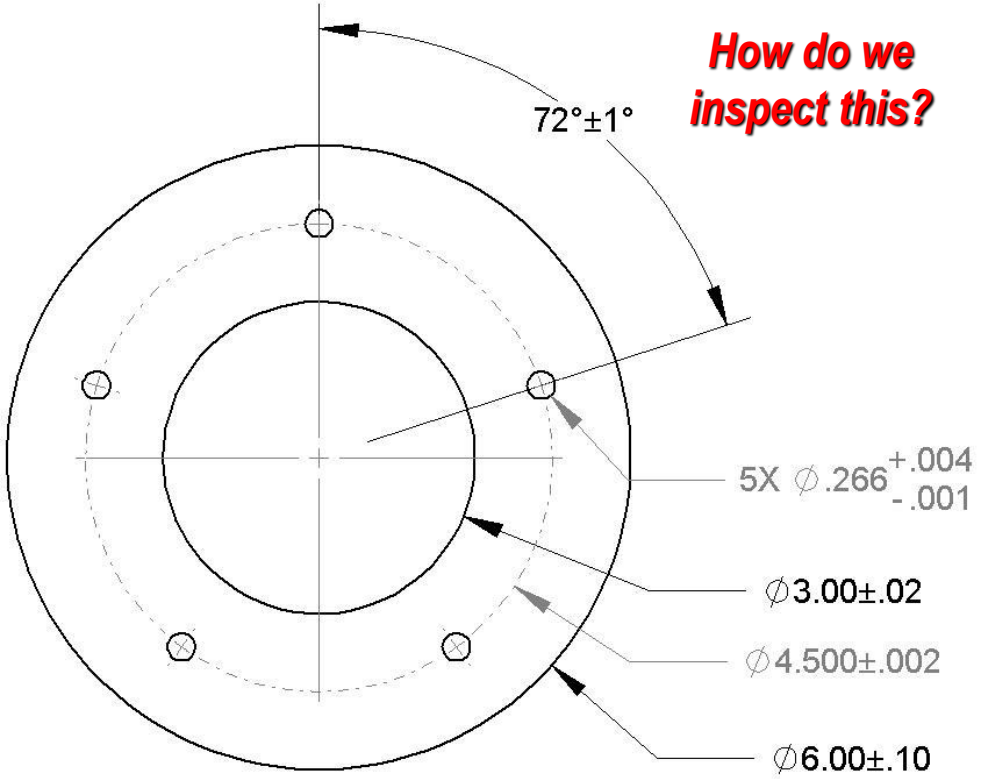


POSITION MEASUREMENTS

- DIRECT MEASUREMENT
  - Works with tolerances at RFS, MMC, or LMC
  - Measure the actual feature size and location and use math (trigonometry) to determine if feature is acceptable or rejected.
- GAGING
  - Works at tolerances modified at MMC and LMC
  - Uses a **GO** / **No GO** gage to determine if feature is acceptable or rejected.

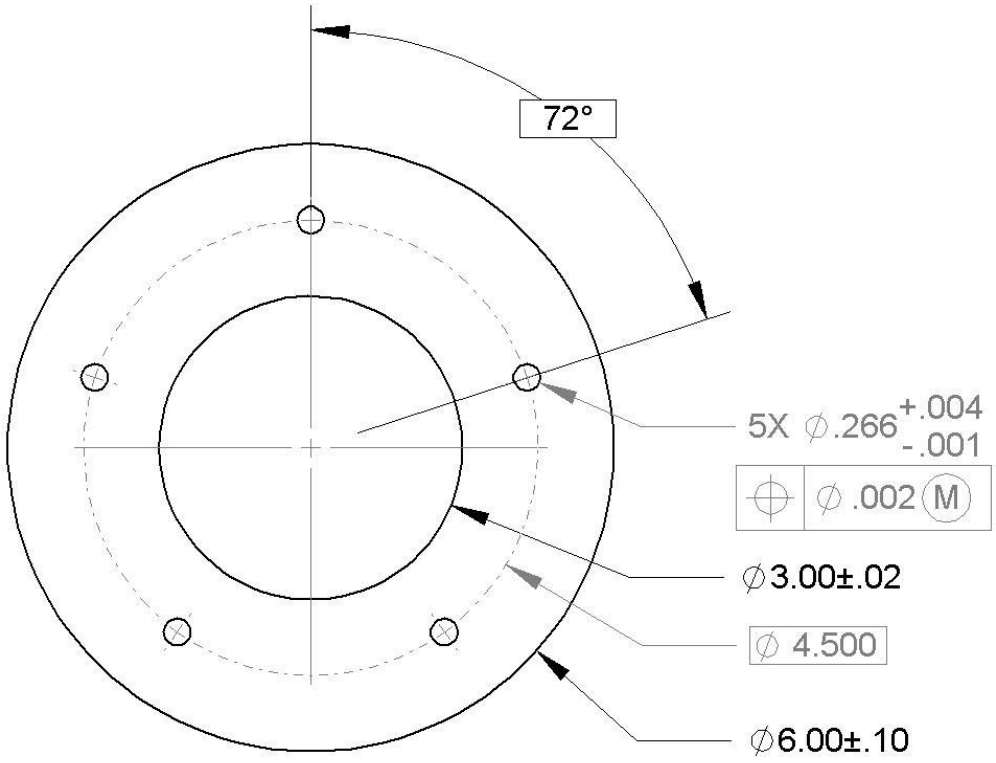
AN INSPECTION NIGHTMARE

- Here's what we want.
- A plate with a bolt pattern.

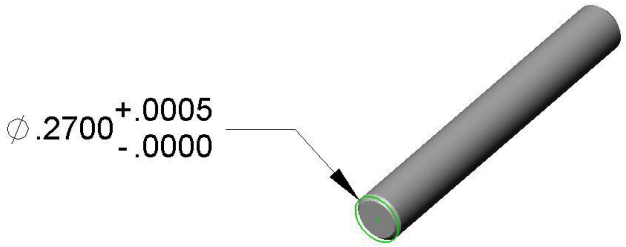


CHANGE TO POSITIONAL TOLERANCE

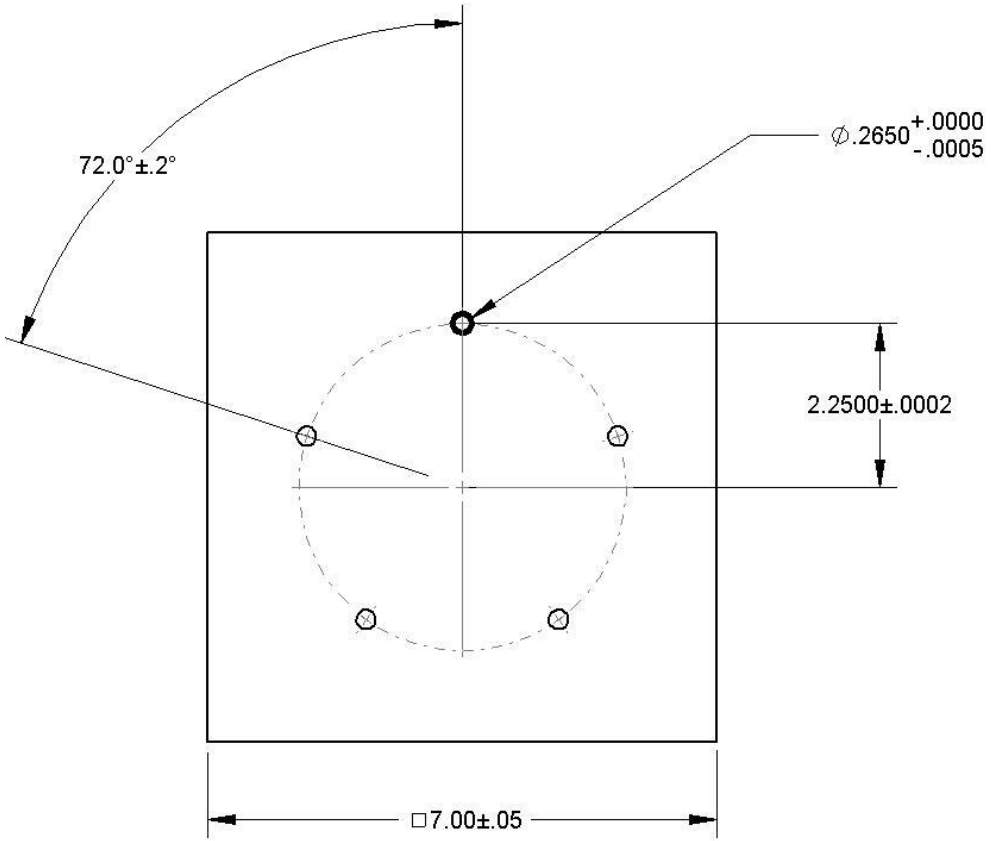
Change the drawing to ***MMC*** so we can use gaging.



GAGES



- No Go Plug
- Go Fixture



VIRTUAL GAGING

- If you use a vision or contact coordinate machine to perform metrology then you don't actually build physical gauges.
- The gauges you use are only mathematical or CAD constructs that exist in a computer.
- Virtual gauges are perfect for LMC fixtures which can not be built in the "real" world.

REMEMBER MMC - LMC?

- MMC is Maximum Material Condition.
- MMC is the condition where the part weighs the most.
  - The thickest plate
  - The smallest hole
- LMC is Least Material Condition.
- LMC is the condition where the part weighs the least.
  - The thinnest plate
  - The largest hole

BOUNDARY

- Datums don’t have physical size or “condition” so MMC, LMC, and regardless of feature size were deemed inappropriate in 2009.
- Datums now have:
  - Maximum Material Boundary (MMB)
  - Least Material Boundary (LMB)
  - Regardless of Material Boundary (RMB)

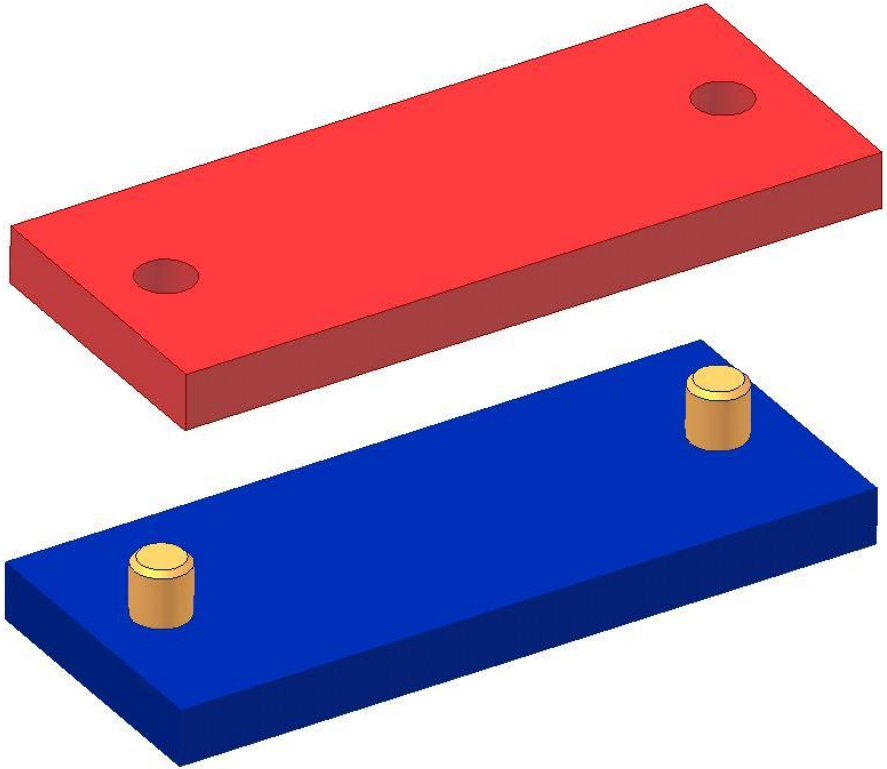
EUROPEAN ODDITY: RECIPROCITY

- Y14.5M-2018 (Reaffirmed 2024) does not have a circled R symbol for Reciprocity or circled E for Envelope positioning method as ISO 1101 now has in it (they mean the same).
- Normally when specifying a position, we say the size of a hole or pin is bounded by the size tolerance and that position tolerance can be variable depending upon size.
- When you invoke reciprocity, you allow the actual size to violate the size tolerance depending on position.

Don't use this on a drawing yet!

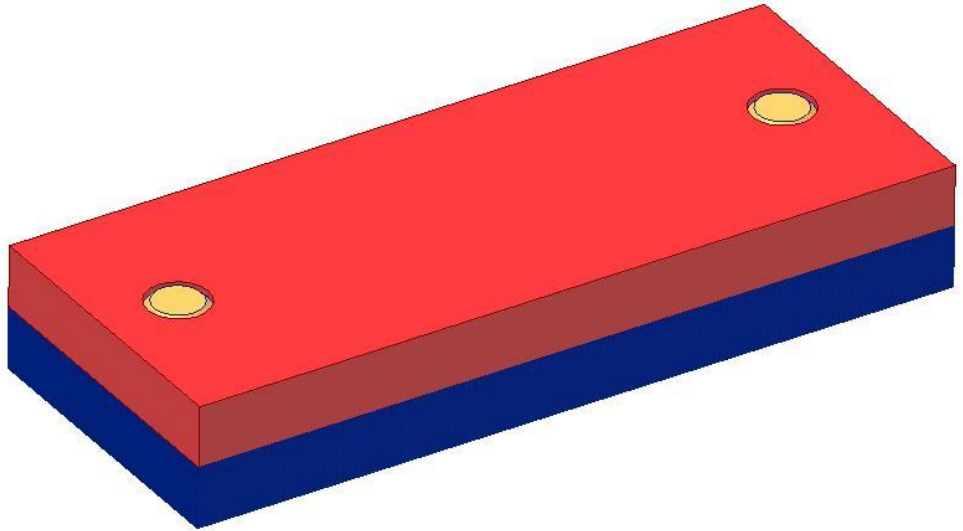
RED PLATE DOF EXAMPLE

- The blue piece with pins is fixed in space.
  - Zero degrees of freedom.
- The red piece with holes is floating free in space.
  - Six degrees of freedom.
  - You can spin it around and move it anywhere.



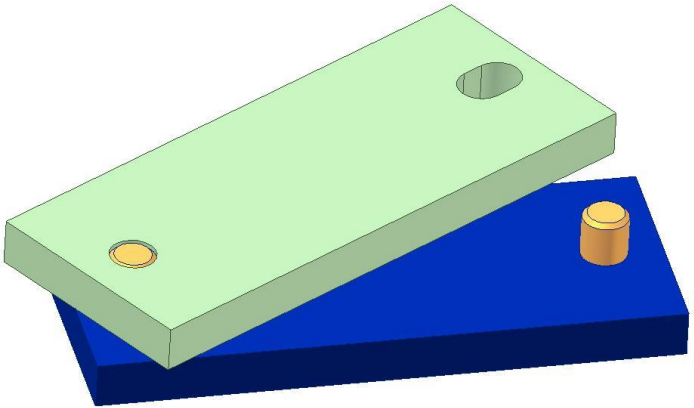
RED PLATE DOF EXAMPLE

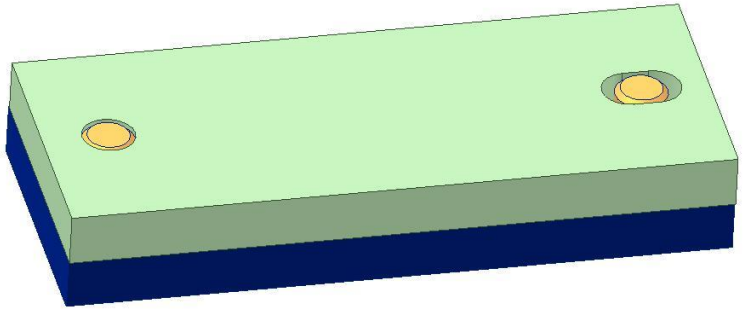
- Once the red piece with holes slides onto the pins of the blue piece, it can no longer rotate about any axis and translation is limited to the direction of the pins.
  - One degree of freedom.
- If you pull up on the red piece it will move in only one direction.



GREEN PLATE DOF EXAMPLE

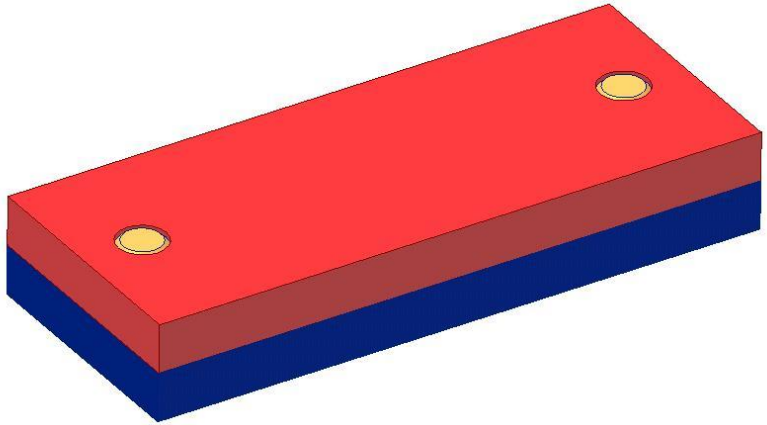
- The second pin on the blue plate stops the green plate from spinning.
- The second pin is said to remove one degree of freedom from the green plate.
  - Eliminates rotation about the hole on the pin.

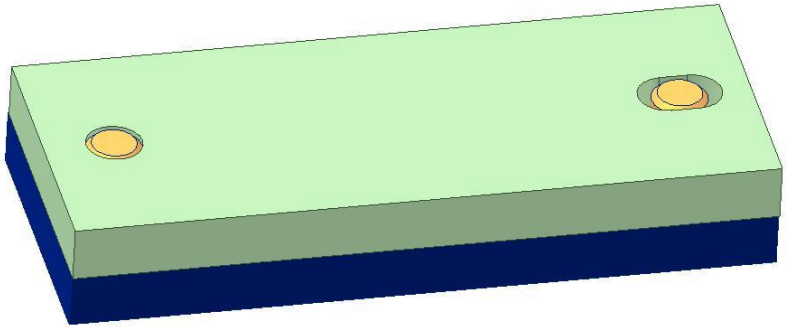




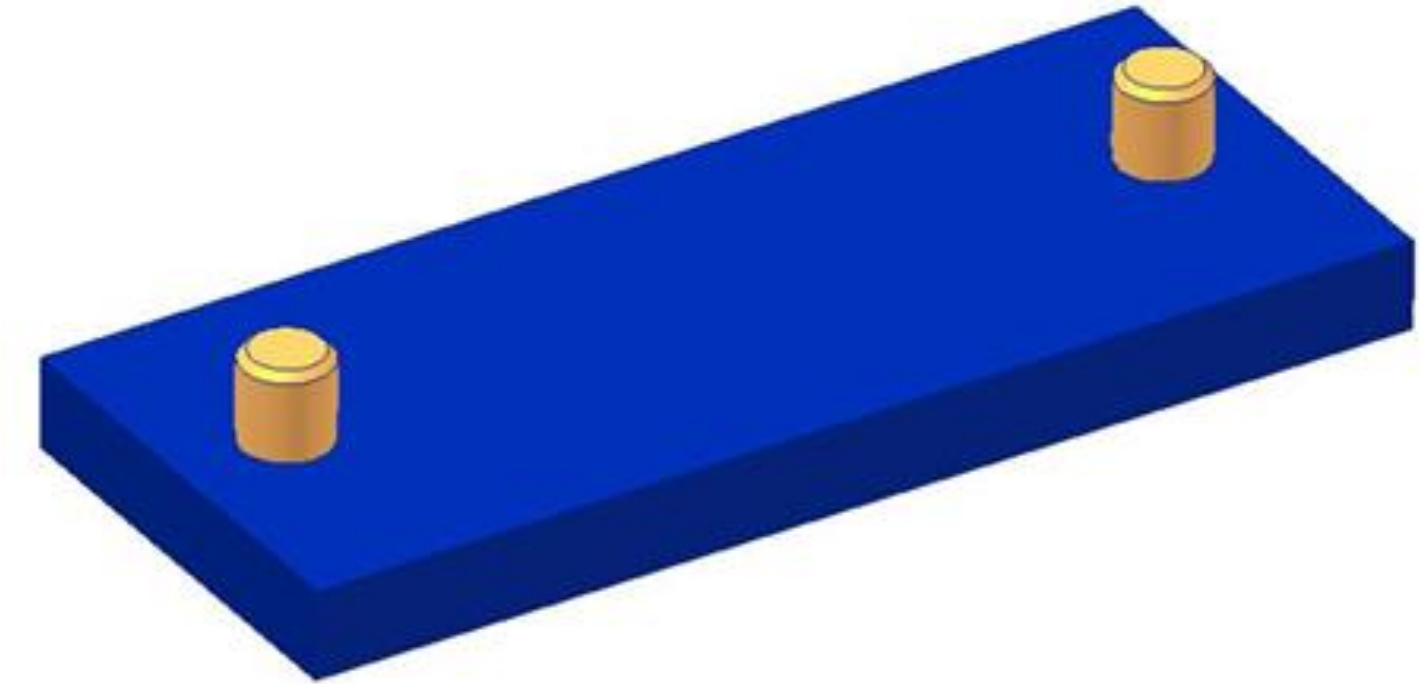
COMPARING DESIGNS

- Both of red plate and the green plate can only move up and down on the blue plate pins
  - Both only have one degree of freedom.
- Is there a disadvantage or advantage to either the red or green plates?



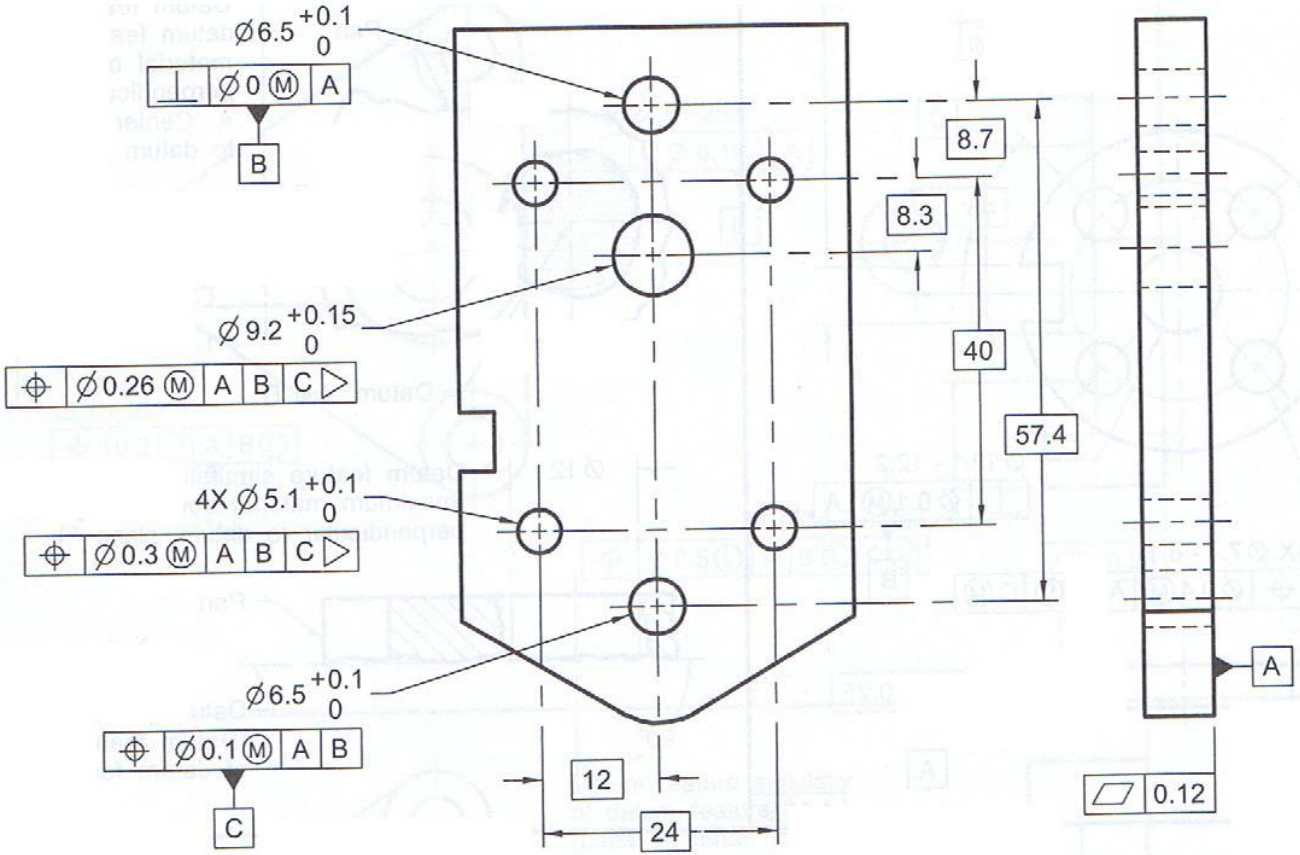


WHAT HAPPENS WHEN YOU NEED TO DESIGN THE BLUE PLATE?

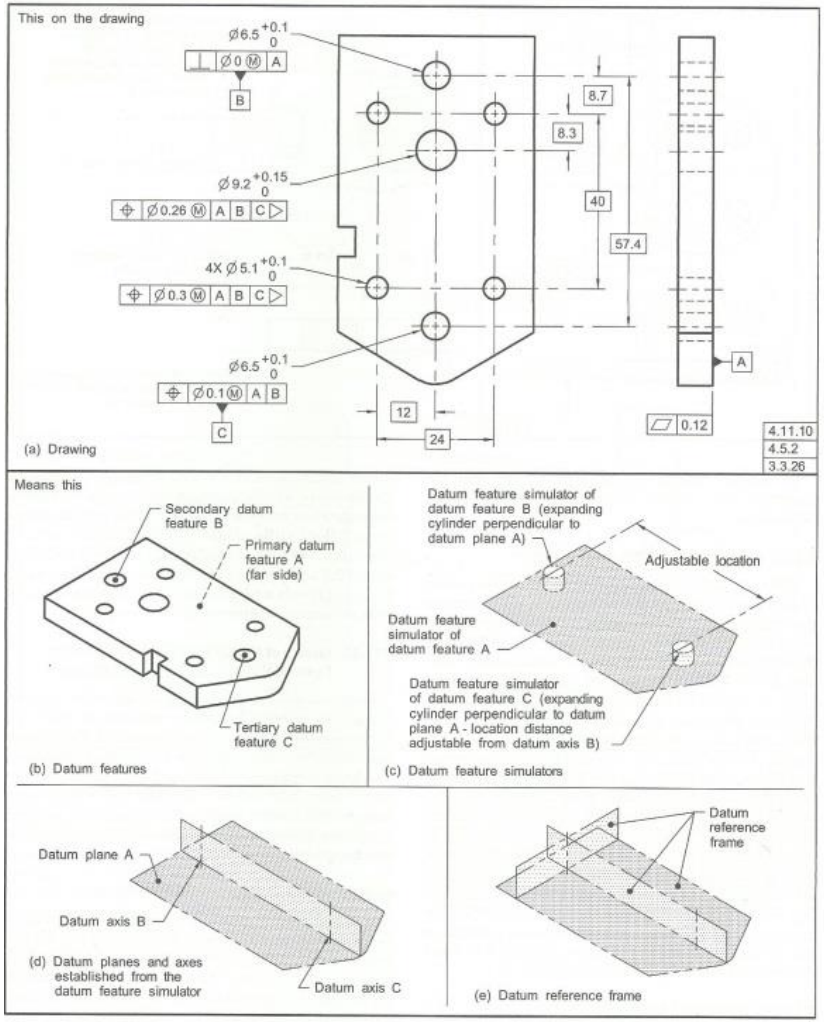


DATUM TRANSLATION SYMBOL

- New Datum Translation Symbol is a triangle on its side like a pointer.
- This overrides the basic dimension for locating a position of a tolerance zone.
- This only makes sense if you have a couple of geometric tolerances on a single feature and you want one of the datum callouts to move with the limits if the tolerance and one of the datum callouts need to be absolute in space.



DATUM TRANSLATION EXAMPLE



SIMULTAENOUS REQUIREMENTS (SIM REQT)

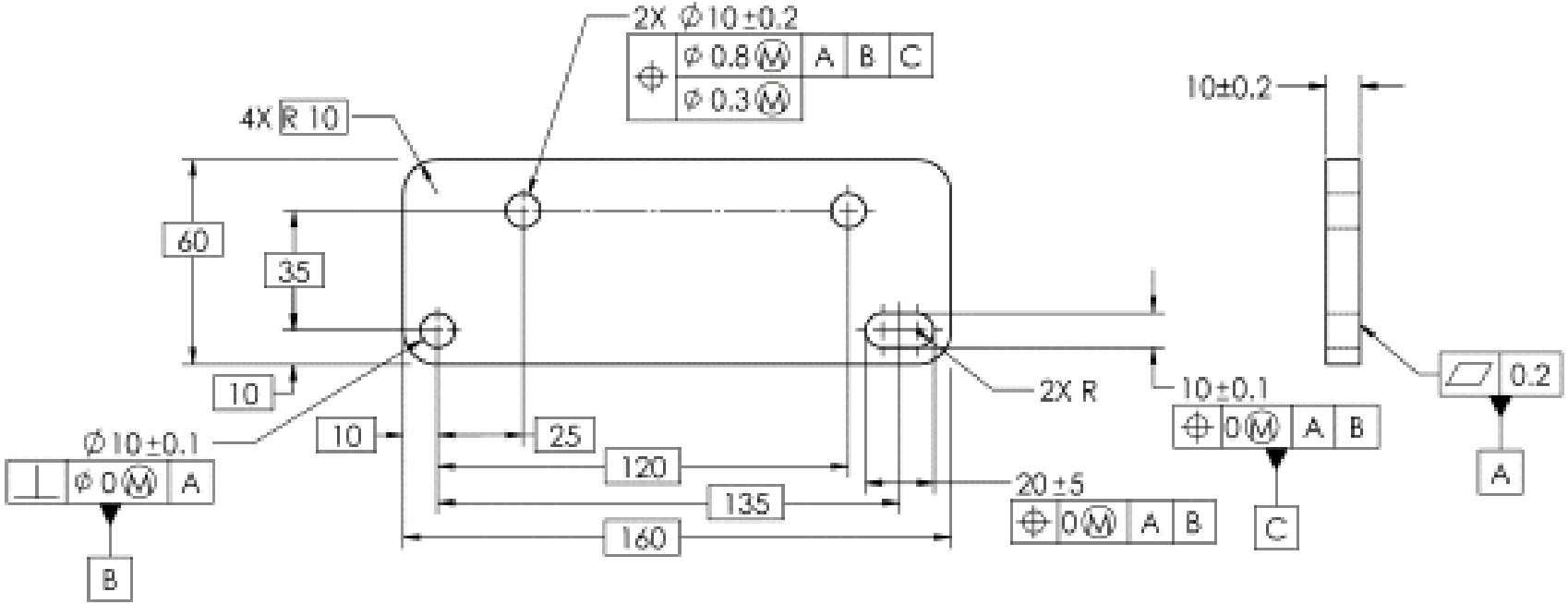
- By default, all feature control frames with the same datums, in the same order, with the same modifiers are all inspected simultaneously unless you state they are separate requirements (SEP REQT).
- Imagine a bowl with features on both inside and outside with the same datum callouts which may be nearly impossible to measure at the same time.

  - Some of the Feature Control Frames need to be separated.

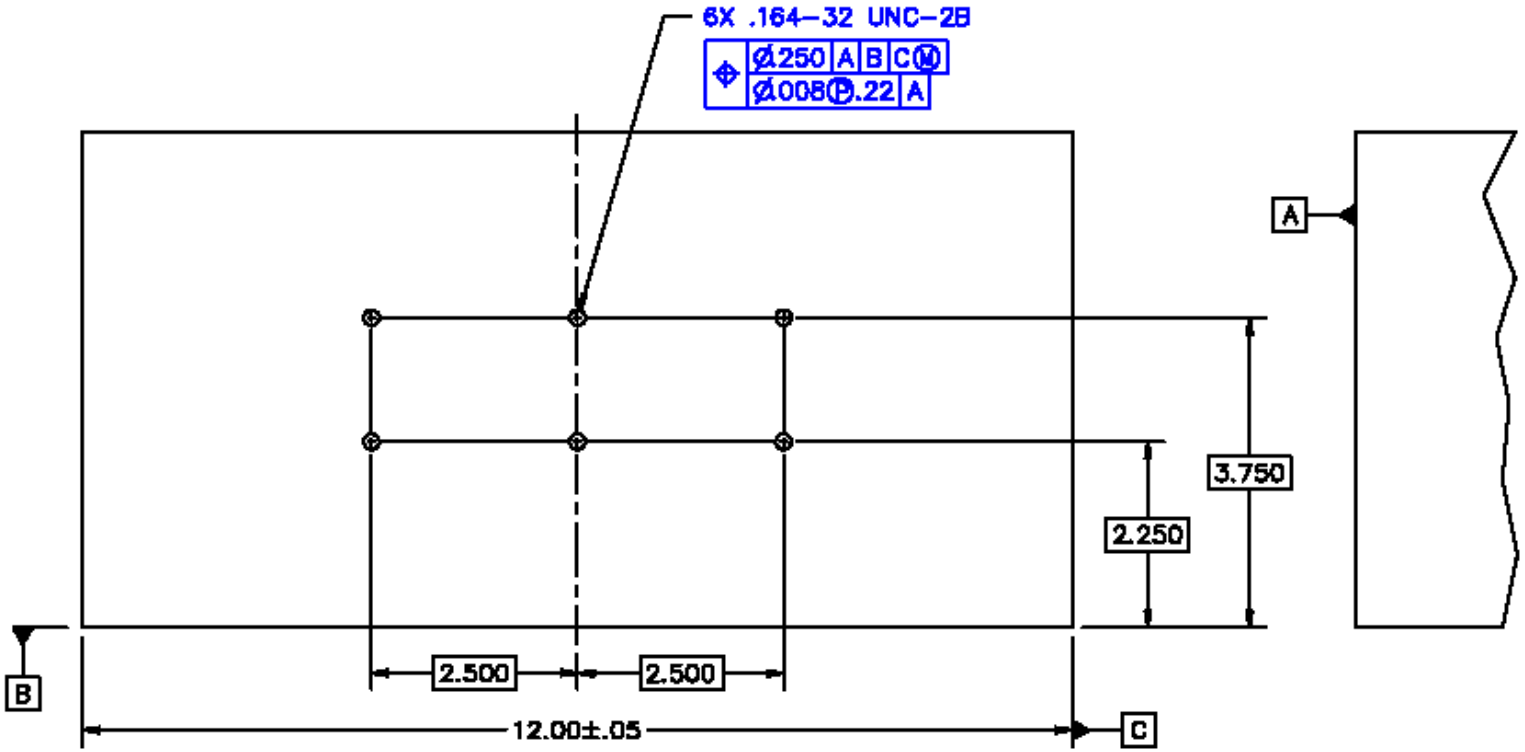
FRITZ AND PLAHTZ?

- Composite positional tolerancing indicates where a pattern is and then the interrelation of the pattern.
- Uses only one Position geometric control symbol in a double tall box of a feature control frame that contains two lines.
  - PLTZF - Pattern-Locating Tolerance Zone Framework (Top line)
  - FRTZF - Feature-Relating Tolerance Zone Framework (Bottom line)
- Imagine using a paper template to mount something where you tape the template to the wall and then drill the fiducials.
  - You don’t care too much about where the mount goes but the hole patten is important!

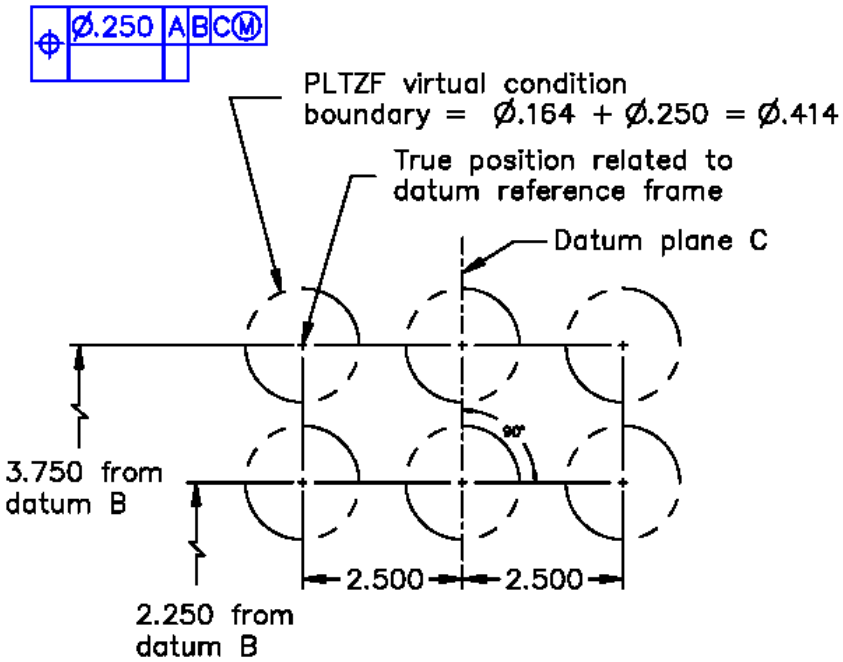
FRTZF / PLTZF EXAMPLE #1



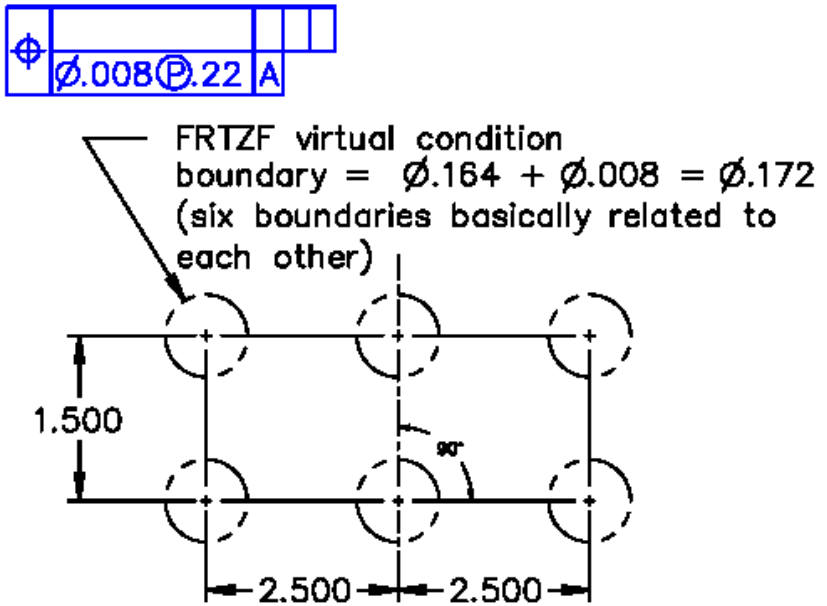
FRTZF / PLTZF EXAMPLE #2



FRTZF / PLTZF EXPLAINED



Note: All boundaries are oriented perpendicular to primary datum A.



Note: All boundaries are oriented perpendicular to primary datum A.

FASTENER POSITIONAL TOLERANCE METHODS

Positional tolerances are calculated differently depending upon how parts are aligned.

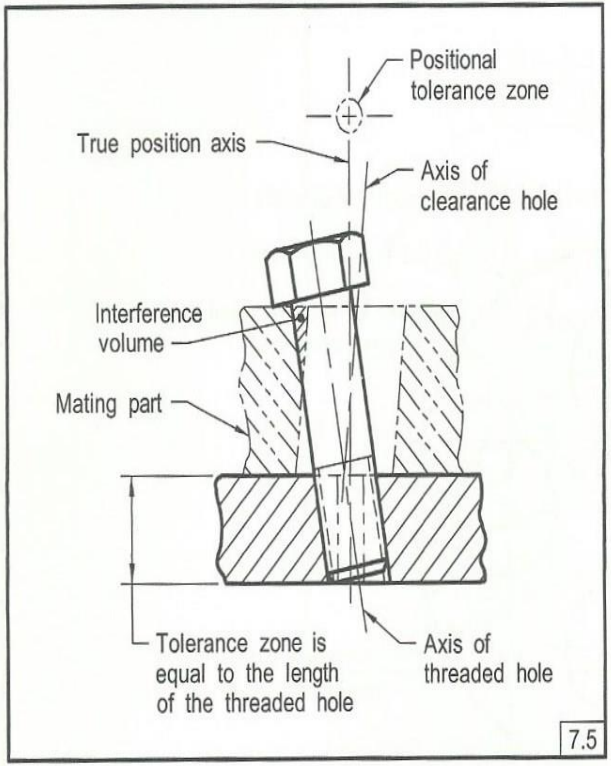
- Floating fastener method: Used when two or more parts with holes must align to accept a separate pin, bolt, or shaft.
  - Positional Tolerance = Hole MMC - Fastener MMC
- Fixed fastener method: Used when two parts must align so that a protrusion on one part must align to a recess on the other.
  - Positional Tolerance = (Hole MMC - Fastener MMC)/2
- Computed tolerance is for ***BOTH*** parts.

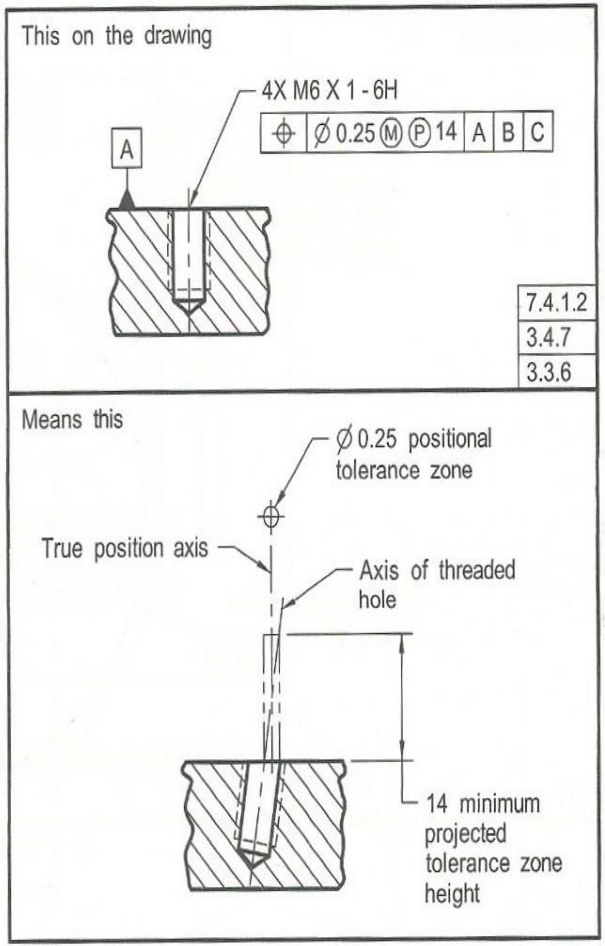
PROJECTED TOLERANCE ZONE

You only measure the surface or edge of the part you wish to inspect except

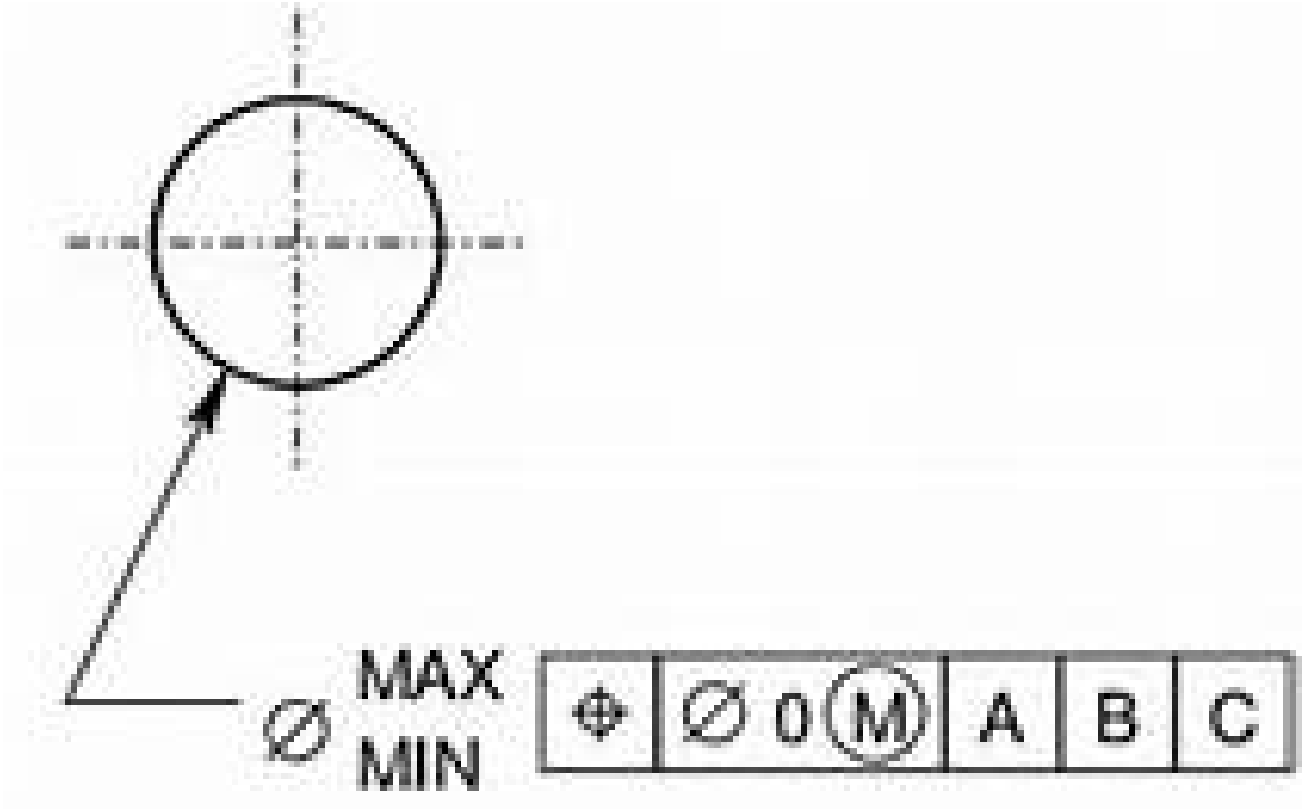
- 7.4.1 Projected Tolerance Zone
  - Moves the tolerance inside the part to someplace off the part out in space - hence the name Projected Tolerance Zone.
  - Projected Tolerance Zone only works on the three orientation controls applied to a feature of size and position.

PROJECTED TOLERANCE EXAMPLE

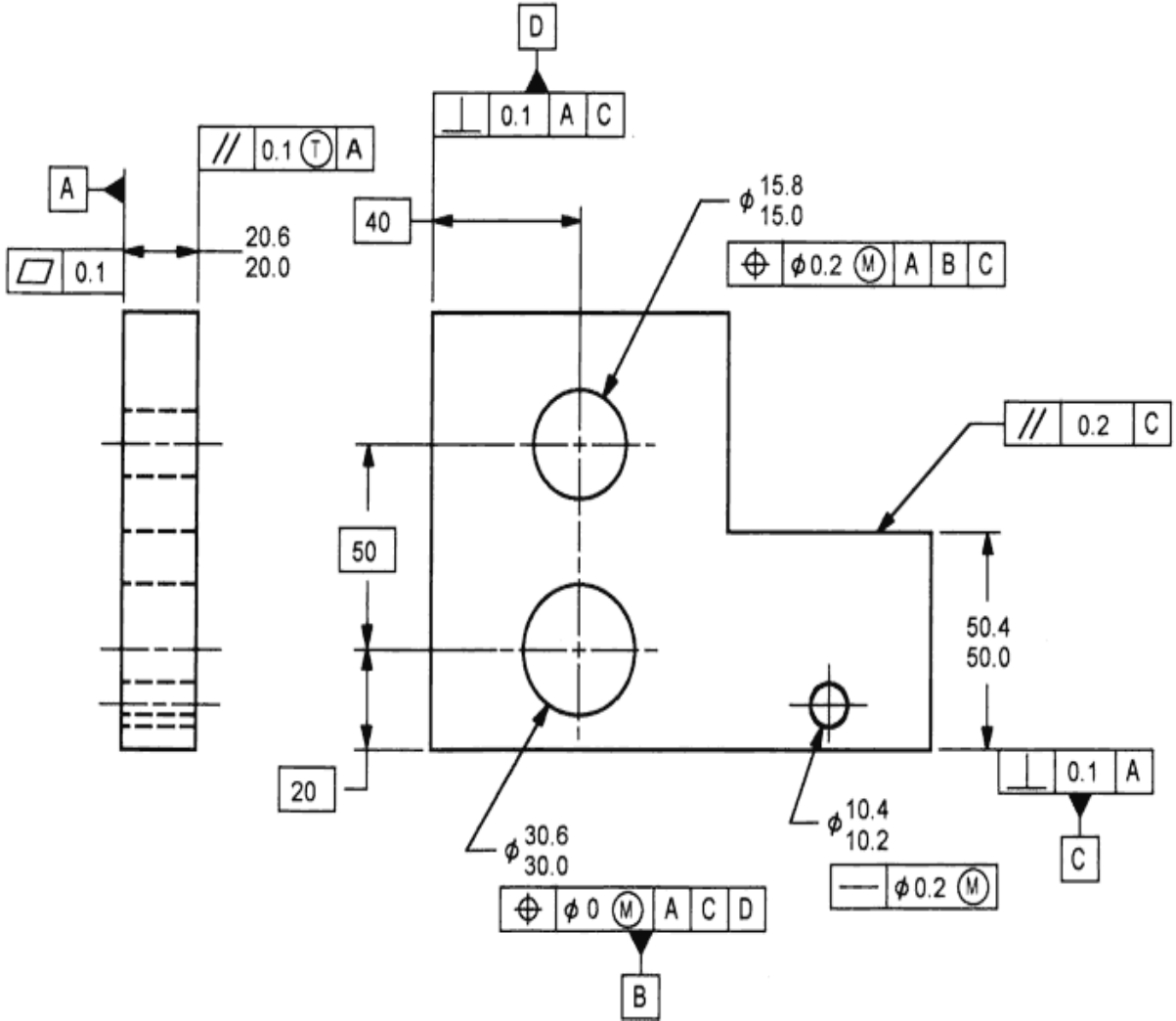




ZERO POSITION TOLERANCE



EXAMPLE



3DEXPERIENCE®
WORLD

Dr. Thomas Allsup
Assistant Professor of Instruction
University of Texas at Arlington
thomas.allsup@uta.edu

DASSAULT SYSTEMES

Discover more on 3DS.Com

DASSAULT SYSTEMES