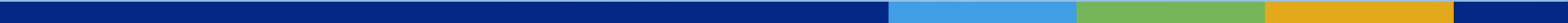


Geometric Dimensioning and Tolerancing



CAD 2204

Spring 2010

CAD 2204

- ❑ Introductions
- ❑ Syllabus
- ❑ My expectations for the class
- ❑ Your expectations

Introductions

- ❑ Instructor
- ❑ Introduce yourself and tell me one expectation you have for this class.

Syllabus

- The syllabus contains the written class expectations for learning and behavior. Not all situations are covered. If a question arises not covered then the student handbook and College rules will apply.

How We learn

- Hearing spoken text and looking at graphics ? 91%
- more learning,
- Looking at graphics alone ? 63% more,
- Reading printed text plus looking at graphics ? 56%
- more,
- Listening to spoken text, reading text, and looking at
- graphics ? 46% more,
- Hearing spoken text plus reading printed text ? 32%
- more,
- Reading printed text alone ? 12% more,
- Hearing spoken text alone ? 7% more.

Chapter One

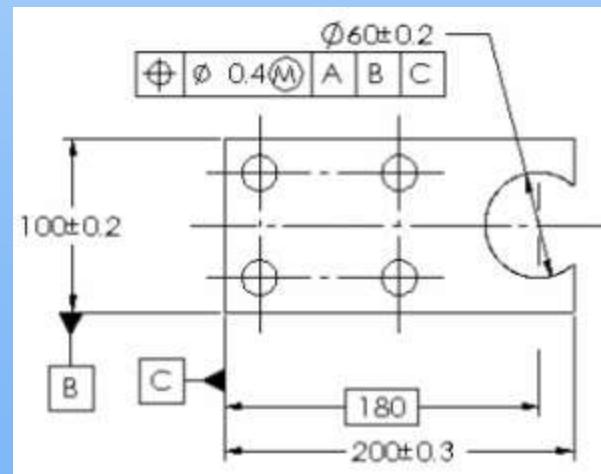
Engineering Drawings and Tolerancing

Chapter Goals

- ❑ Understand what an engineering drawing is
- ❑ Understand why geometric tolerancing is superior to coordinate tolerancing

What is an Engineering Drawing?

- A document that communicates a precise description of a part.
- The description will consist of pictures, words, numbers and symbols.

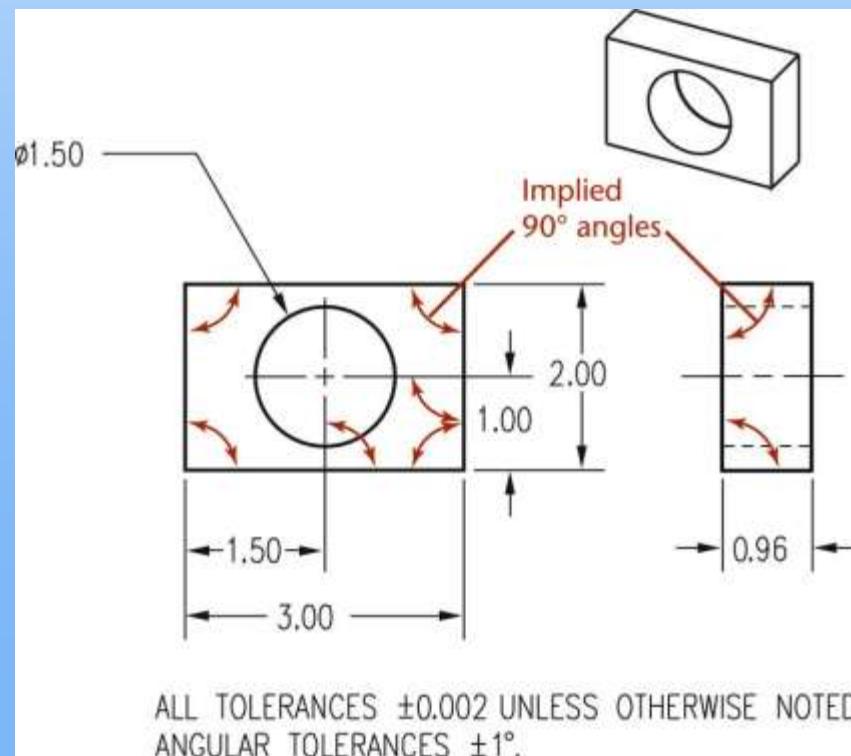


What is an Engineering Drawing?

- An engineering drawing may communicate the following:
 - Geometry of the part
 - Critical functional relationships
 - Tolerances
 - Material, heat treat, surface coatings
 - Part documentation such as part number and drawing revision number

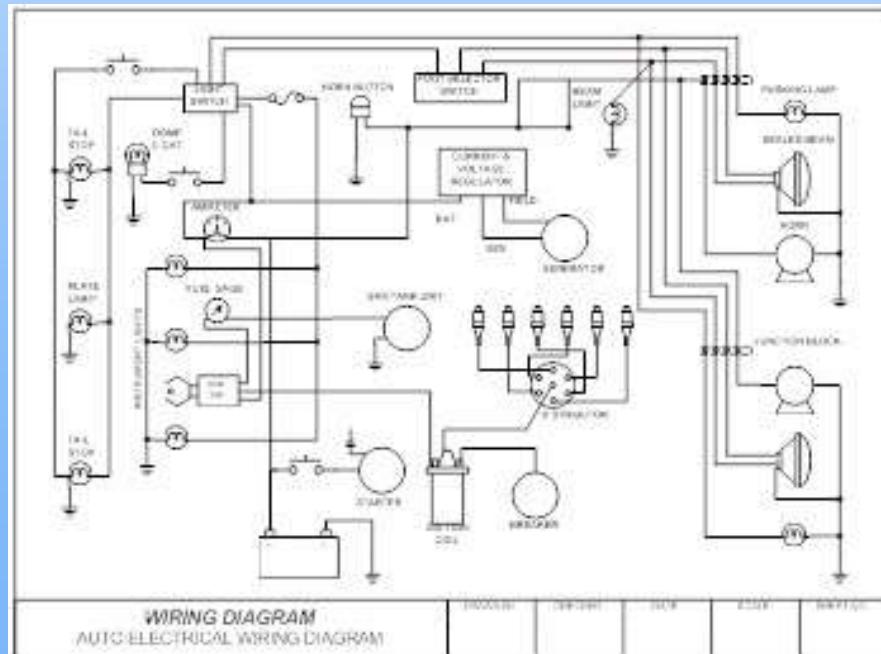
What is an Engineering Drawing?

Note the dimensions

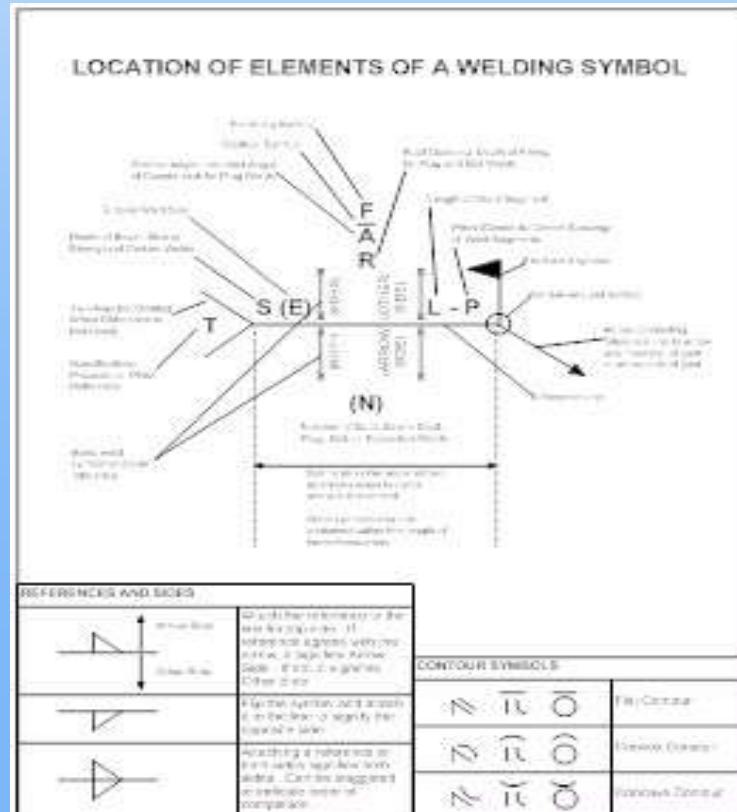


What is an Engineering Drawing?

□ Electrical



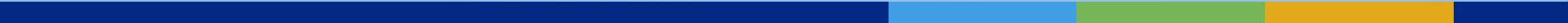
□ Quality/Descriptive



Brief History of Engineering Drawing.

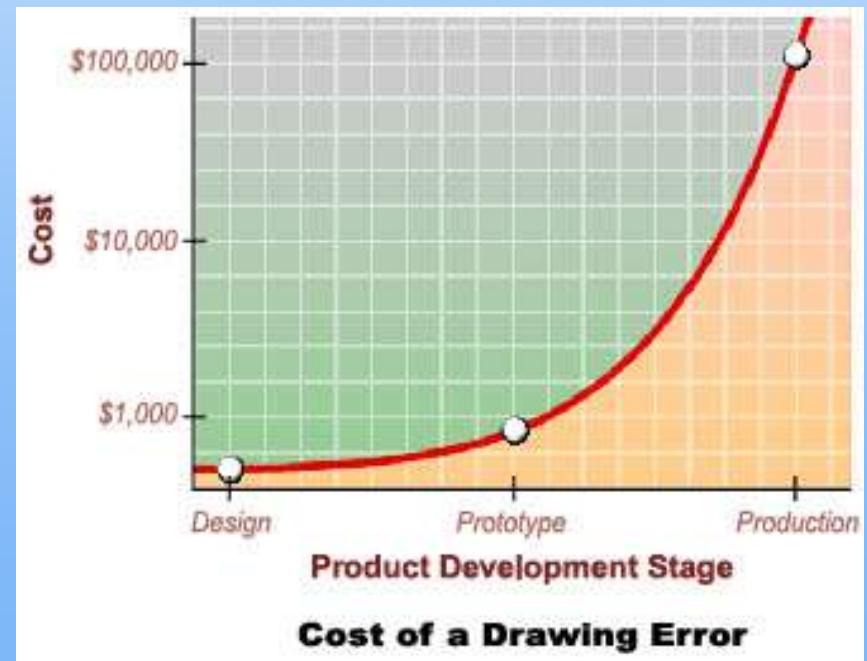
- Papyrus/Rock/Clay tablet
 - Sumerian, Egyptian and other early civilizations.
- Pen and Ink
- Pencil
- Mainframe
- PC
 - Several different software's
 - AutoCAD
 - Pro-E

What was wrong with the way we always made our drawings?

A horizontal bar consisting of four colored segments: dark blue, light blue, green, and yellow.

Cost of Drawing Errors!

- Drawing errors compound the cost of the error as you move further from initial design to production.
- Drawing errors cost:
 - Money
 - Time
 - Material
 - Customer satisfaction



What Else Was Wrong?

Disagreements over drawing interpretation.



Difficulty communicating drawing requirements.



Can not understand the designer's intent.



What Else Was Wrong?

- Bad parts passing inspections.



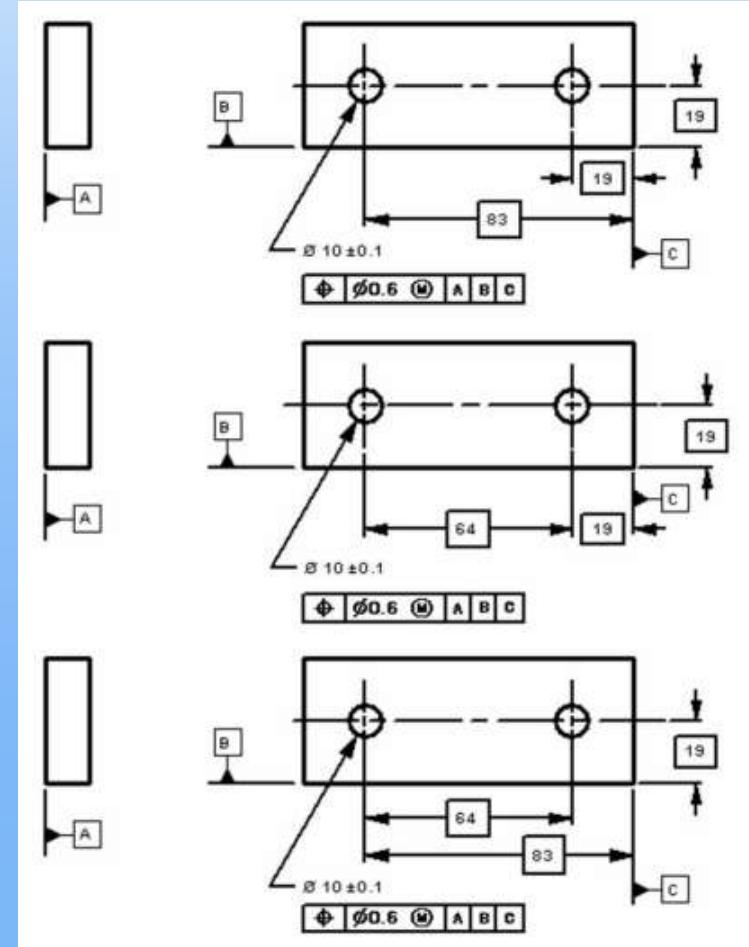
- Wasting money on parts that do not fit.



Back To Engineering Drawings!

■ Dimensions

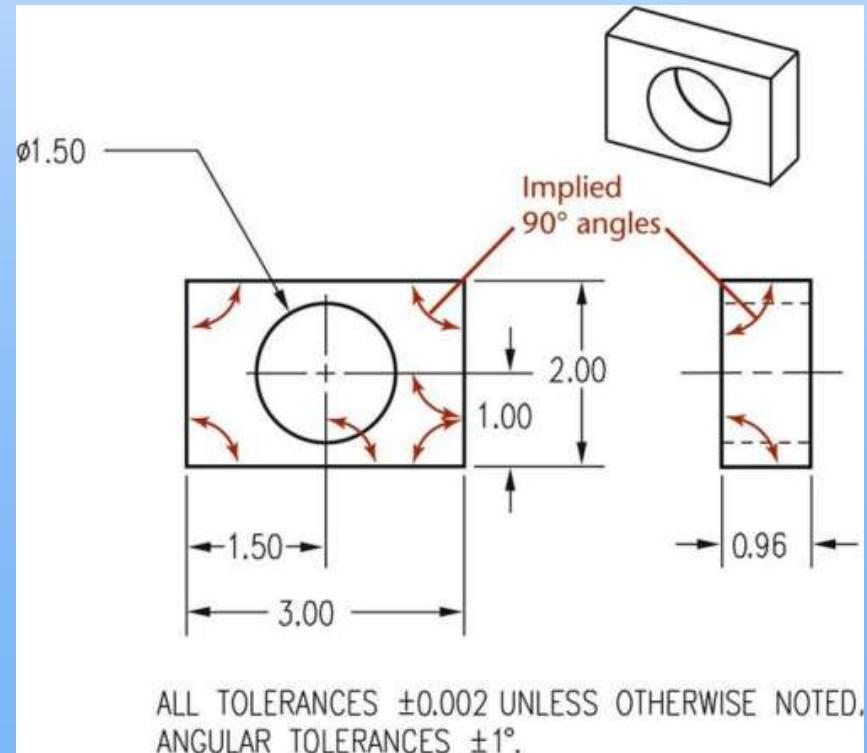
- A numerical value expressed in appropriate units of measure and used to define the size, location, orientation, form or other geometric characteristics of a part



Engineering Drawings

■ Tolerance

- Tolerance is the total amount a specific dimension is permitted to vary from the specified dimension. The tolerance is the difference between the maximum and minimum limits.



Types of Tolerances

□ Limit Tolerance:

- When a dimension has a high and low limit stated.
 - 1.250/1.150 is a limit tolerance.

□ Plus-Minus Tolerance

- The nominal or target value of the dimension is given first, followed by a plus-minus expression of tolerance.
 - 32 +/- .004 is a plus-minus tolerance.

Types of Tolerances

- **Bilateral Tolerance**

- A tolerance that allows the dimension to vary in both the plus and minus directions.

- **Equal Bilateral Tolerance**

- Variation from the nominal is the same in both directions.

Types of Tolerances

- **Unilateral Tolerance**

- Where allowable variation is only in one direction and zero in the other.

- **Unequal Bilateral Tolerance**

- Where the allowable variation is from the target value and the variation is not the same in both directions.

- **Figure 1-4, page 7 for examples.**

Metric Dimensions

- For this class we will be using metric dimensions.
 - Drawings must state the units used and certain other information used in the drawing.

GENERAL NOTES

1. ALL DIMENSIONS IN MILLIMETRES UNLESS STATED OTHERWISE
2. DRAWING IN ACCORDANCE WITH THE STANDARDS REFERENCED IN BS8888: 2004
3. COMPONENTS TO BE STAMPED WITH THEIR PLANT IDENT. NUMBER WHERE MARKED  IN LETTERS AT LEAST 3MM HIGH.

Metric Dimensions

- The primary ways to indicate tolerances in a drawing are:
 - A general tolerance note
 - A note providing a tolerance for a specific dimension
 - A reference on the drawing to another document that specifies the required tolerances
 - These are similar to specifying dimension units

MACHINED COMPONENTS

1. TOLERANCES UNLESS OTHERWISE STATED :

LINEAR DIMENSIONS 100 AND UNDER	$\pm 0.25\text{mm}$
LINEAR DIMENSIONS OVER 100 AND UNDER 1000	$\pm 0.5\text{mm}$
LINEAR DIMENSIONS OVER 1000	$\pm 1.0\text{mm}$
ANGULAR DIMENSIONS	$\pm 0^\circ - 15'$
HOLE POSITIONS	± 0.5

2. REMOVE ALL BURRS AND SHARP EDGES.

3. SURFACE TEXTURE INDICATED IN ACCORDANCE WITH BS EN ISO 1302:2002

SURFACE TEXTURE BASIC SYMBOLS
UNMACHINED ✓ MACHINED ✓ MACHINING NOT ALLOWED ✓

4. MACHINE SURFACES ✓ UNLESS OTHERWISE STATED

5. ALL METRIC THREADS TO BS 3643 PT 2:1981 (6H /6g FIT). ALL METRIC LENGTHS AND INTERNAL DEPTHS TO BE FULL THREAD UNLESS OTHERWISE STATED

Metric Dimension Rules

- ❑ When a metric dimension is a whole number, the decimal point and zero are omitted.
 - ❑ Ex. 56
- ❑ When a metric dimension is less than one millimeter, a zero precedes the decimal point.
 - ❑ Ex. 0.7
- ❑ When a metric dimension is not a whole number, a decimal point with the portion of a millimeter (10ths or 100ths) is specified.
 - ❑ Ex. 14.2

Dimensioning Limits

- All dimensioning limits are absolute
 - That is all dimensions are considered to have a zero after the last true digit.
 - Ex. 1.25 means 1.250
 - See page 8 for other examples
- This interpretation is important for gauging.
 - A part with a dimension and tolerance of 1.0 +/- .1 would fail inspection at what Measure?

Dimensioning History

■ History of GD&T

- Drawings before industrial evolution were elaborate pictures of a part that could not be made today. Why?
 - Parts made were unique one of a kind that were made as the craftsman saw the part.
- The industrial revolution made manufacturers realize that one of a kind was not economical.

Dimensioning History

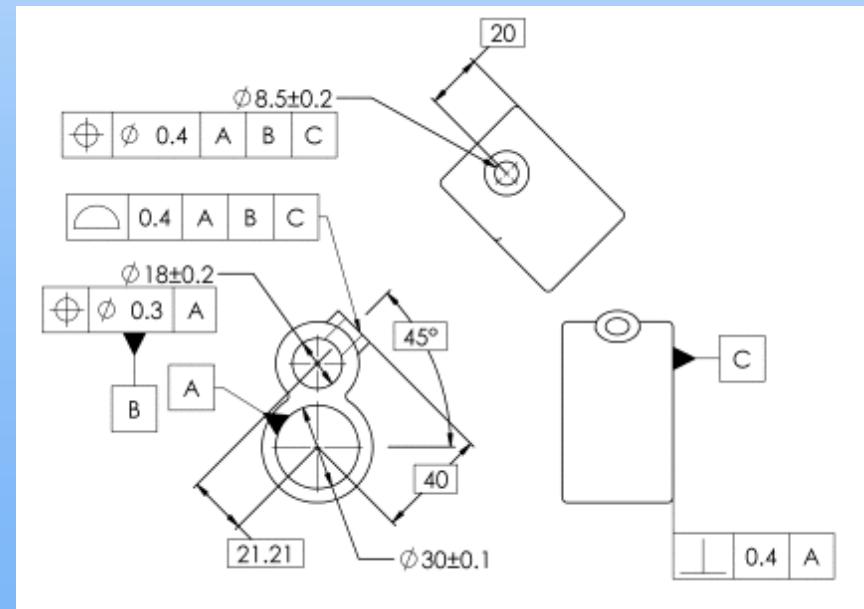
- Mass production created the need for interchangeable parts. This meant drawings that were simple and standardized.
- The first standard was coordinate based.
- 1966 saw the publication of the first GD&T standard
 - ANSI Y14.5
 - Updates have been published in 1982, 1994 and 2004.

Fundamental Dimensioning Rules

- Ten rules apply to our work. They are based on ANSI Y14.5M-1994.
 - Each dimension shall have a tolerance except those dimensions specifically identified as reference, maximum, minimum or stock size.
 - These terms are explained later in the text.
 - Dimensioning and tolerancing shall be complete so there is full definition of each part feature.

Fundamental Dimensioning Rules

- Dimensions shall be selected and arranged to suit the function and mating relationship of a part and not be subject to more than one interpretation.
- These three rules establish dimensioning conventions.

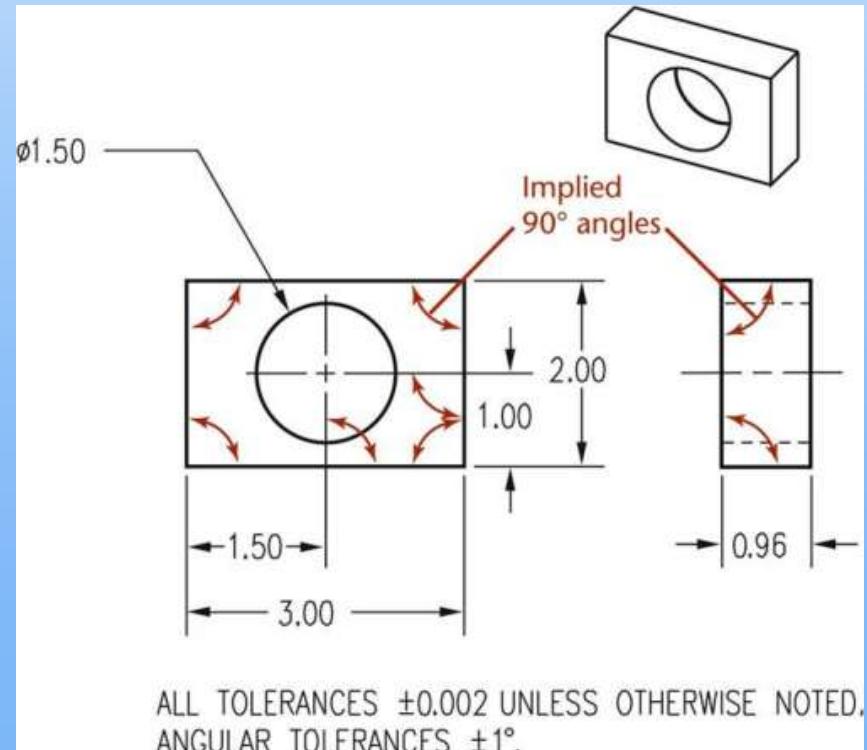


Fundamental Dimensioning Rules

- ❑ The drawing should define a part without specifying manufacturing methods.
- ❑ A 90 degree angle applies where centerlines and lines depicting features are shown on a drawing at right angles and no dimension is shown.

Fundamental Dimensioning Rules

- A 90 degree basic angle applies where centerlines of features in a pattern or surfaces shown at right angles on a drawing, are located and defined by basic dimensions and no angle is specified.
- These last two rules establish convention for implied 90 degree angles.

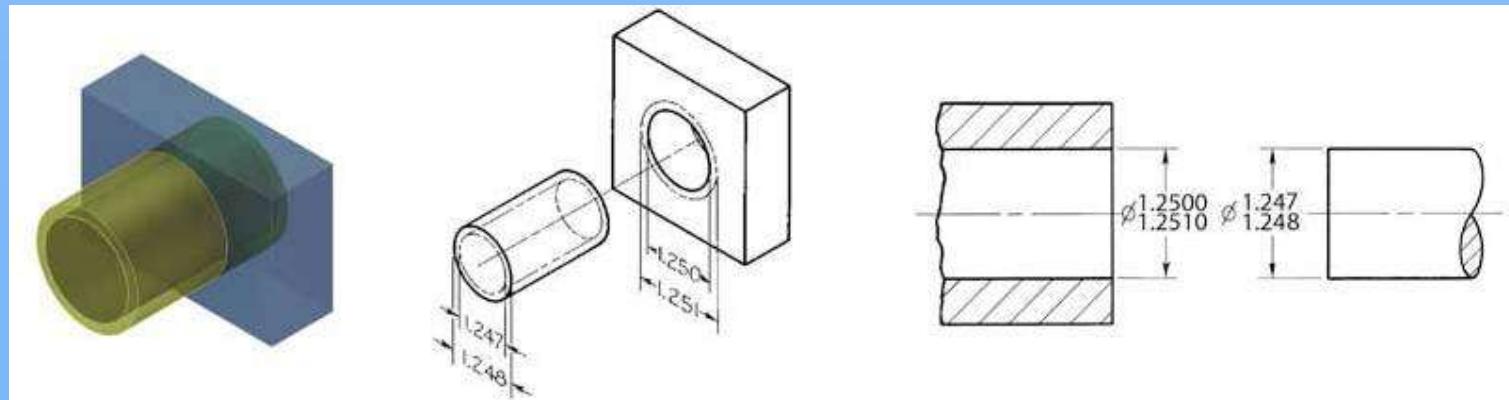


Fundamental Dimensioning Rules

- Unless otherwise specified, all dimensions are applicable at 20 degrees Centigrade (68 deg. F)
 - What happens in other types of weather?
- All dimensions and tolerances apply in the free-state condition. This condition does not apply to non-rigid parts.
- Unless otherwise specified, all geometric tolerances apply to the full depth, length and width of the feature.
- These three rules establish default conditions for dimensions and tolerance zones.

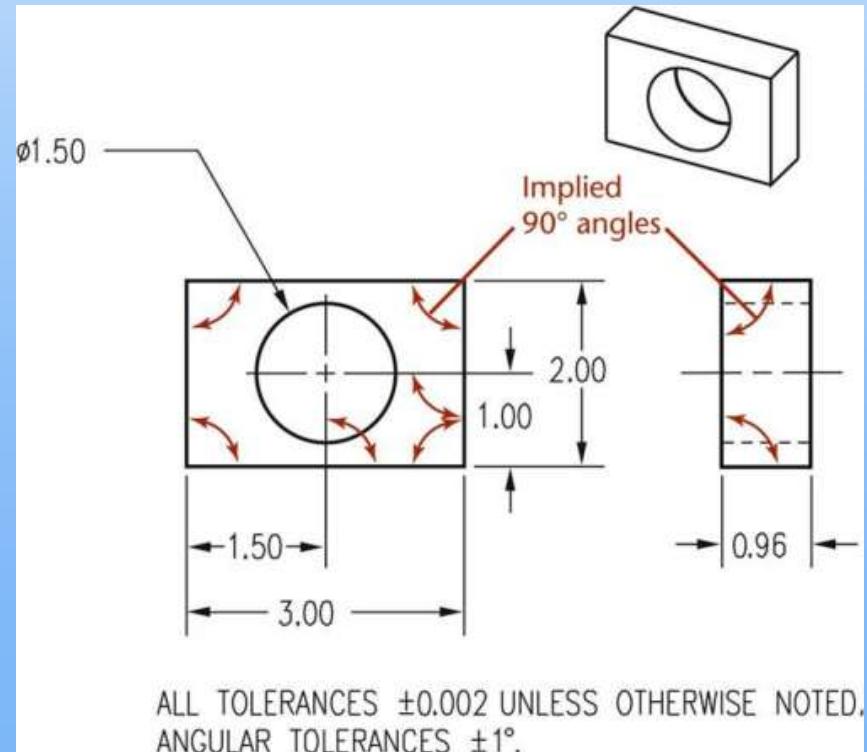
Fundamental Dimensioning Rules

- Dimensions and tolerances apply only at the drawing level where they are specified. A dimension specified on a detail drawing is not mandatory for that feature on the assembly drawing.



Lets Talk About Coordinate Tolerancing

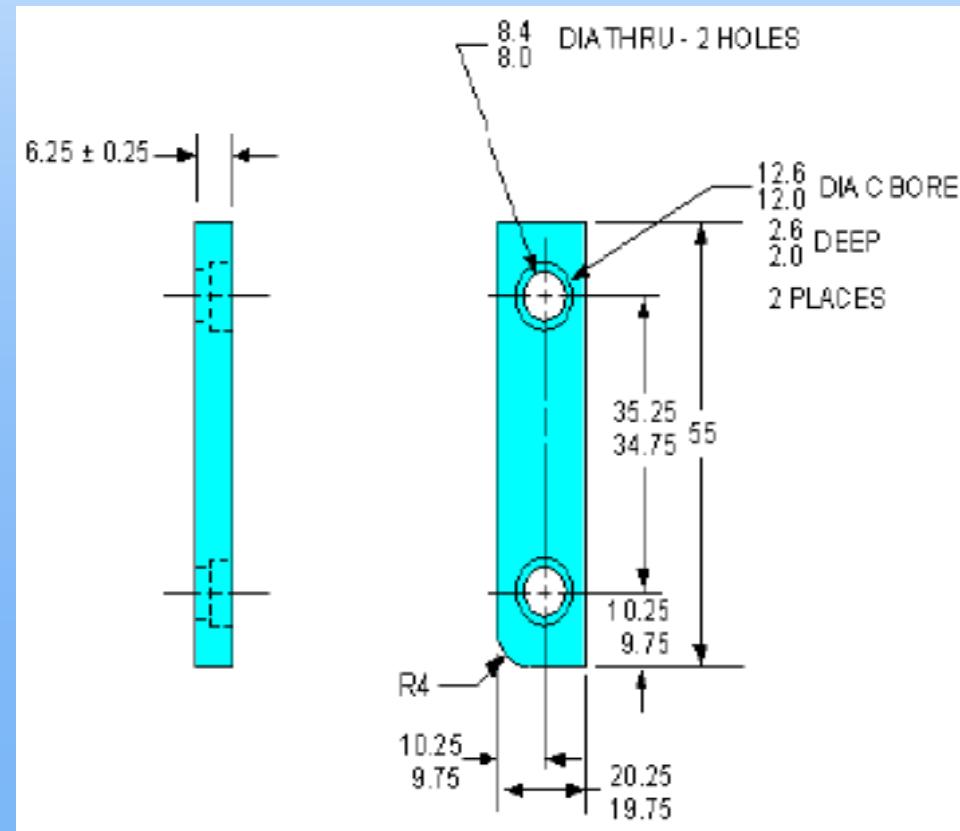
- Coordinate tolerancing is a dimensioning system where a part feature is located by a means of rectangular dimensions with given tolerances.



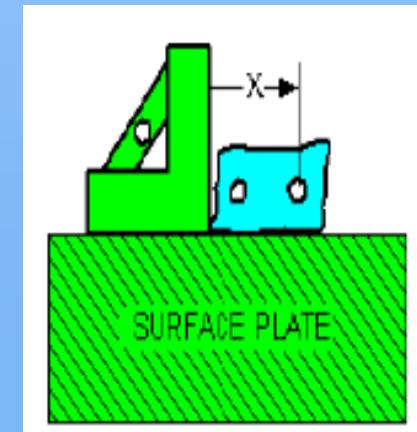
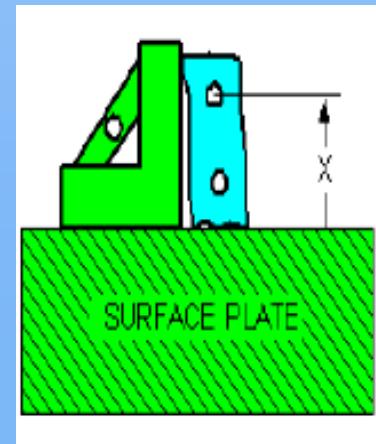
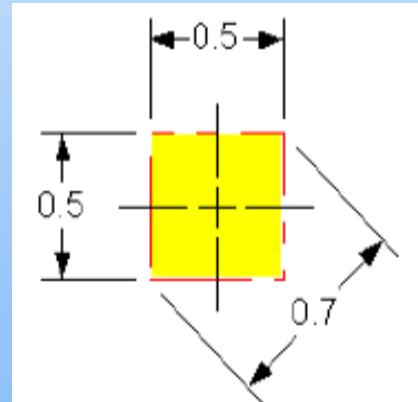
Coordinate Tolerancing

■ Shortcomings

- CT or coordinate tolerancing, does not have the completeness needed today to enable efficient and economical production of parts.



- CT shortcomings are:
 - Square or rectangular tolerance zones
 - Fixed size tolerance zones
 - Ambiguous instructions for inspection



Square Tolerance Zone

Ambiguous Inspection



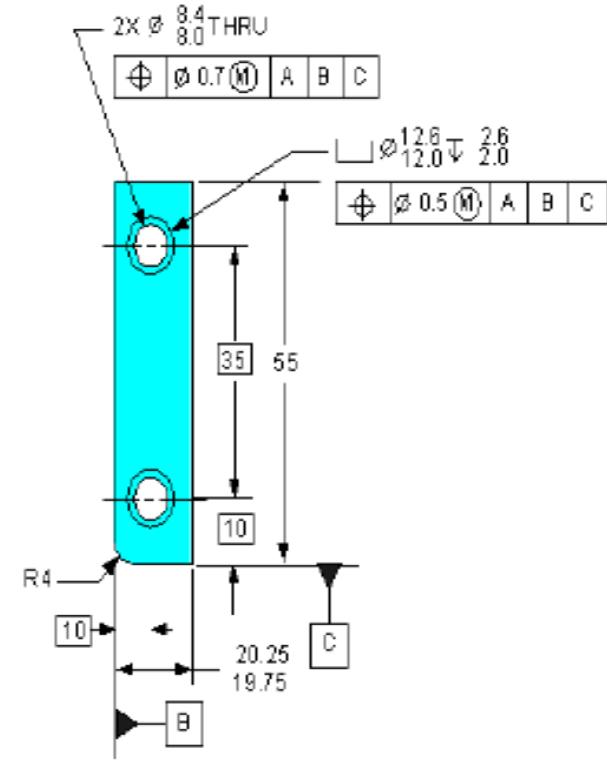
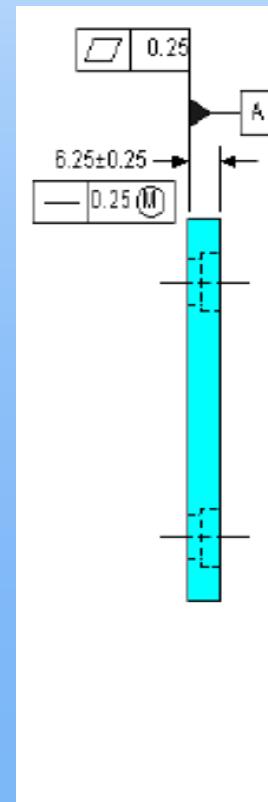
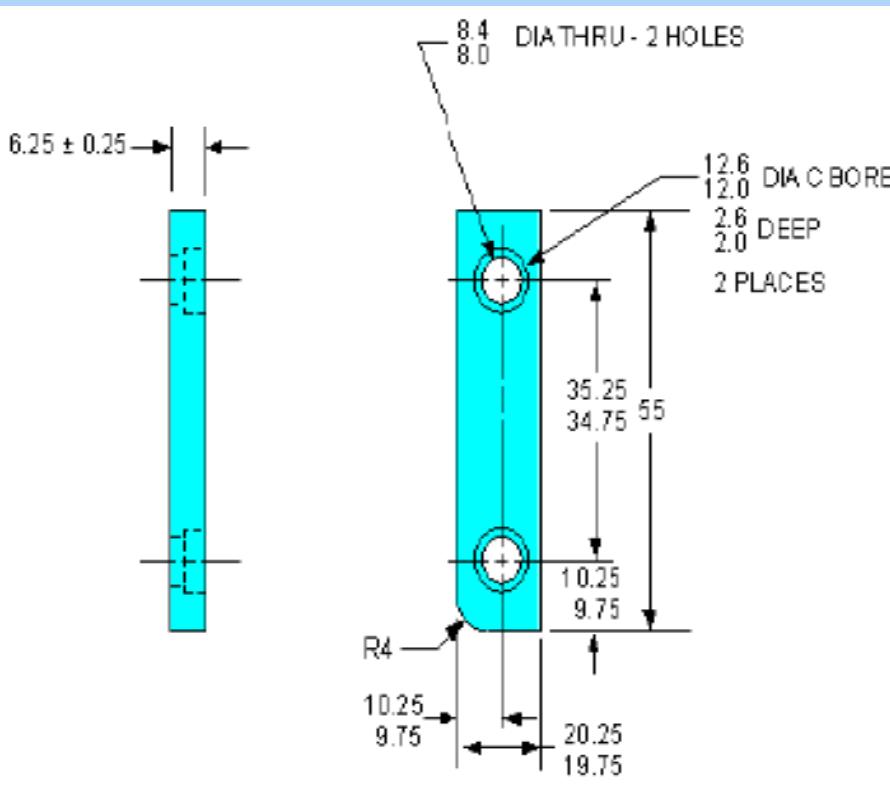
Good Use of Coordinate Dimensioning

Coordinate Dimension Usage		
Type of Dimension	Appropriate Use	Poor Use
Size		
Chamfer		
Radius		
Locating Part Features		
Controlling angular Relationships		
Defining the Form of Part Features		

Geometric Dimensioning and Tolerancing

- Geometric Dimensioning and Tolerancing (GD&T) is an international language that is used on engineering drawings to accurately describe a part.
- GD&T is a precise mathematical language that can be used to describe the size, form, orientation and location of part features.
- GD&T is also a design philosophy.

CT and GD&T Drawing Comparison



Philosophy of GD&T

❑ Functional Dimensioning

- ❑ Part is defined based upon function in final product
- ❑ Works well with Concurrent Product and Process Development (CPPD) also called Simultaneous Engineering)
 - ❑ Design is created with inputs from marketing, customers, manufacturing, purchasing and any other area that has impact on the final production and use of the product.

Ultimate Goal

- ❑ Provide the customer with what they want, when they want the product, how or the form they want the product in and at the price they want.
- ❑ The process of producing the product starts with the designer.

GD&T Benefits

- Improved Communications

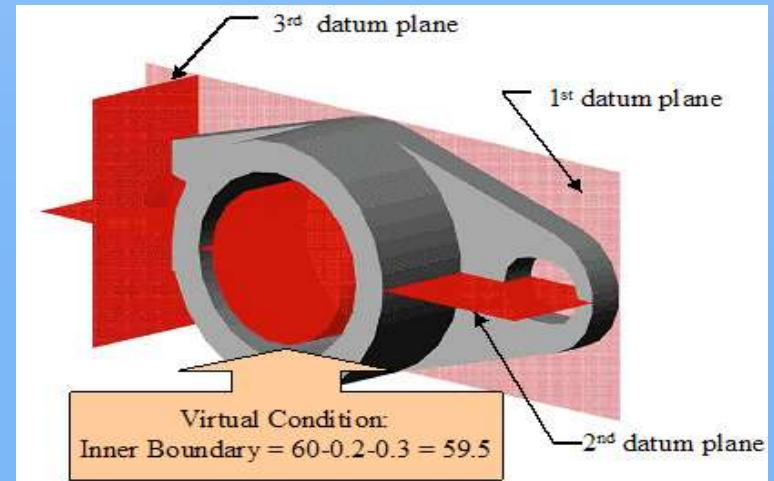
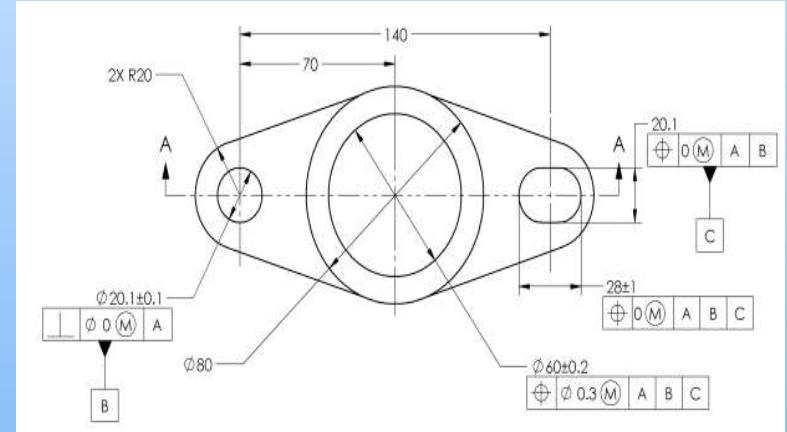
- Design uniformity allows design, production and inspection to all work from the same view. There is no argument over what to do

- Better Product Design

- Allows designers to say what they mean instead of having engineers explain what to do

Benefits of GD&T

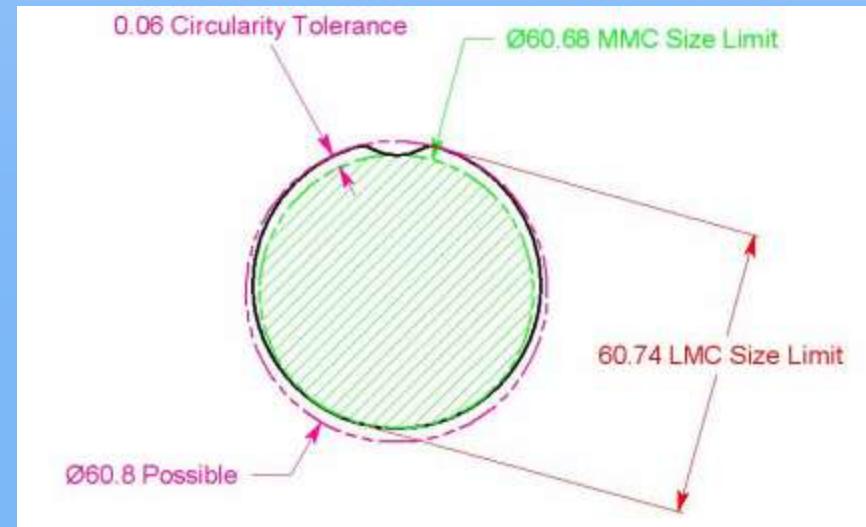
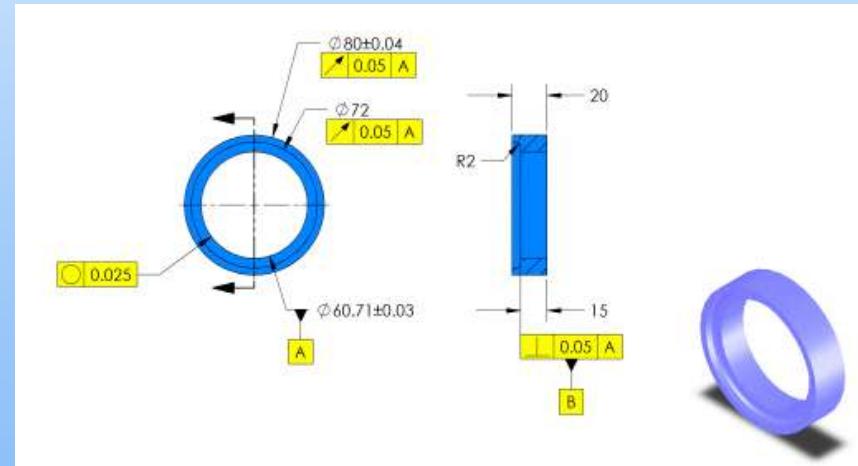
- Increased Production Tolerance
 - Bonus Tolerance: saves cost of manufacturing
 - Tolerance based upon part functional requirements. Technique for determining a tolerance is not part of this course.



GD&T and CT Comparison

□ Increased Tolerance Zone

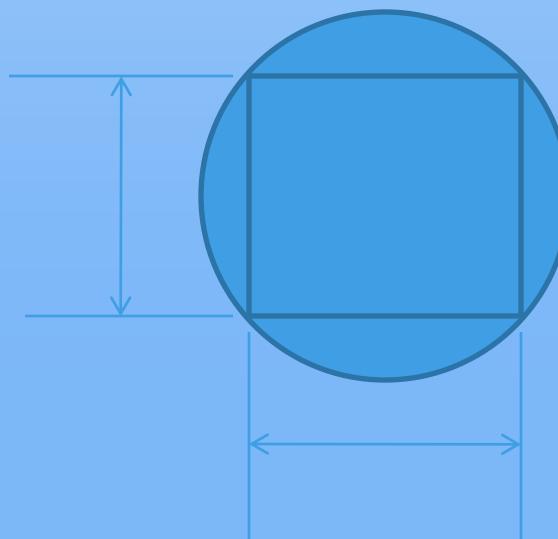
- Tolerance is no longer square but fits the geometry of the feature



GD&T and CT Comparison

■ Fixed Size and Square Tolerance Zones

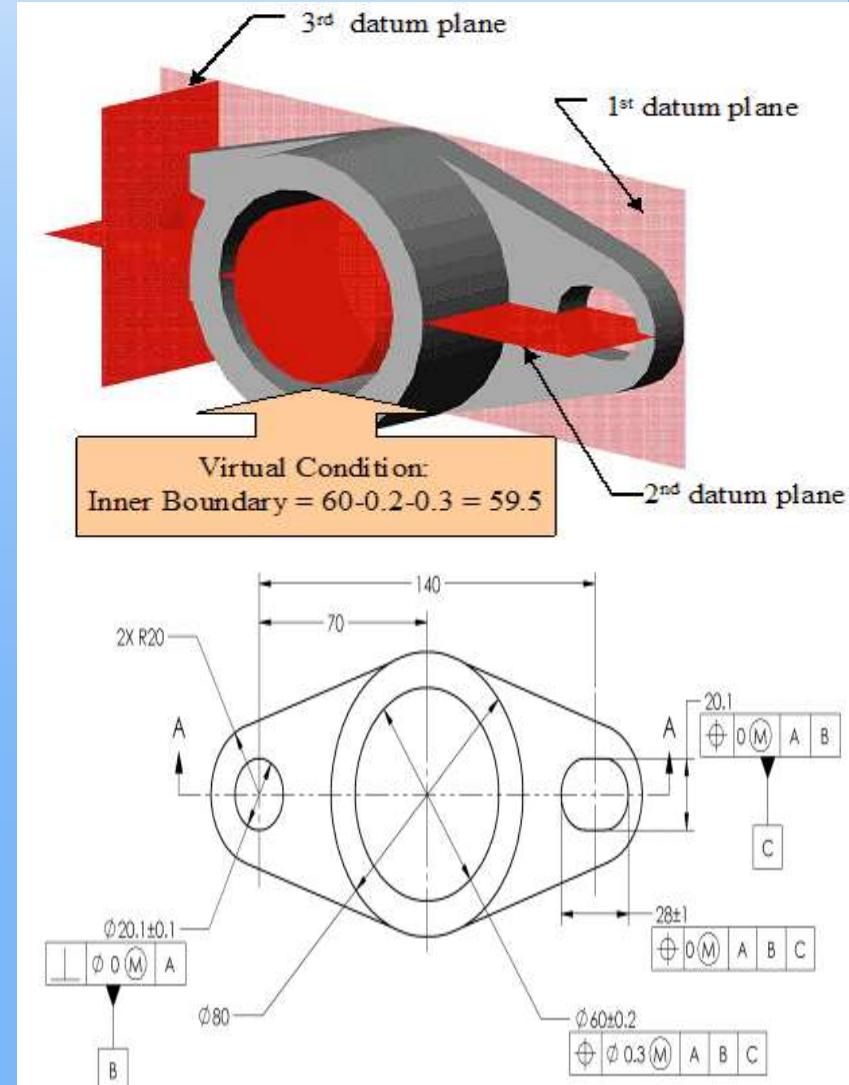
- Circle is 0.71 diameter
- Square is 0.5 each side



GD&T and CT Comparison

■ Ambiguous Instructions for Inspection

- CT allowed the inspector to “inspect” the part from whatever perspective the inspector deemed appropriate. GD&T gives a defined process for inspection.
- GD&T uses the datum system to eliminate inspection confusion.



Great Myths of GD&T

- ❑ GD&T Raises product cost.
- ❑ We do not need GD&T
- ❑ GD&T and Y14.5M-1994 are confusing
- ❑ GD&T drawings take too long to make
- ❑ Easier to use CT
- ❑ GD&T should only be used on critical parts
- ❑ I know GD&T
- ❑ Dimensioning and tolerancing are separate steps
- ❑ I can learn GD&T in two days.

Great Myth of GD&T

- ❑ The misconception that geometric tolerancing raises product cost is still considered to be the great myth of GD&T.

Summary

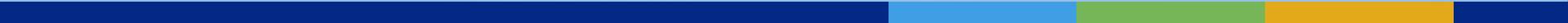
- We discussed engineer drawings and what they were
 - Engineer drawings do what?
- Cost of poor drawing
 - When should we catch mistakes?
 - What do errors cost?
- Dimensions and Tolerances
 - What are they?

Summary

- Fundamental Dimensioning Rules
- Coordinate Tolerancing
 - Do we use CT today?
 - Shortcomings of CT
- What is GD&T
 - Benefits
- Great Myth of GD&T

Questions?

Chapter Two



Introduction to Geometric Tolerancing Symbols and Terms

Chapter Goals

- ❑ Understand eight key terms and how they affect the interpretation of a drawing.
- ❑ Understand the modifiers and symbols used in Geometric Tolerancing.

Features

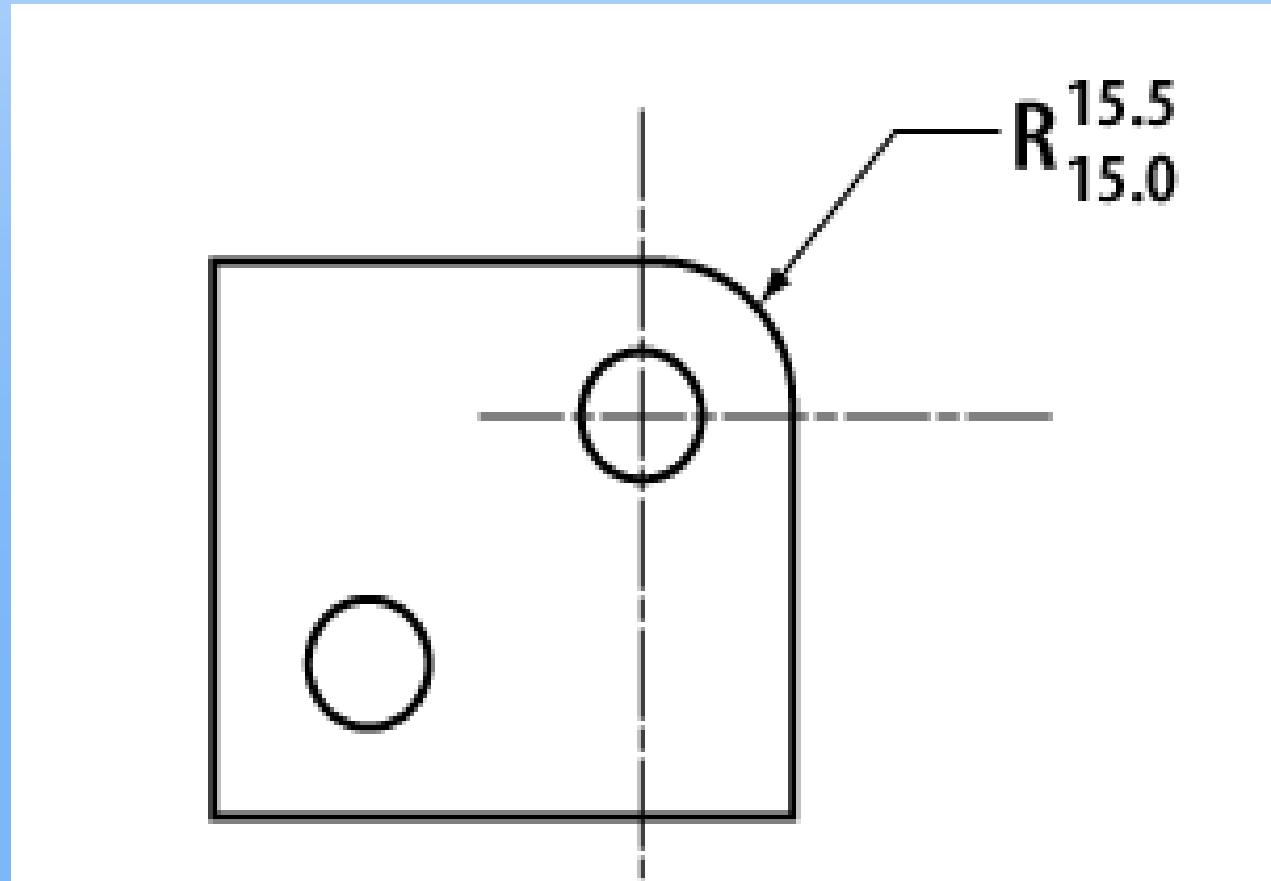
□ Feature

- A general term applied to a physical portion of a part such as a surface, hole or slot.
 - A feature may be considered a part surface.

Features

□ How many features for this view?

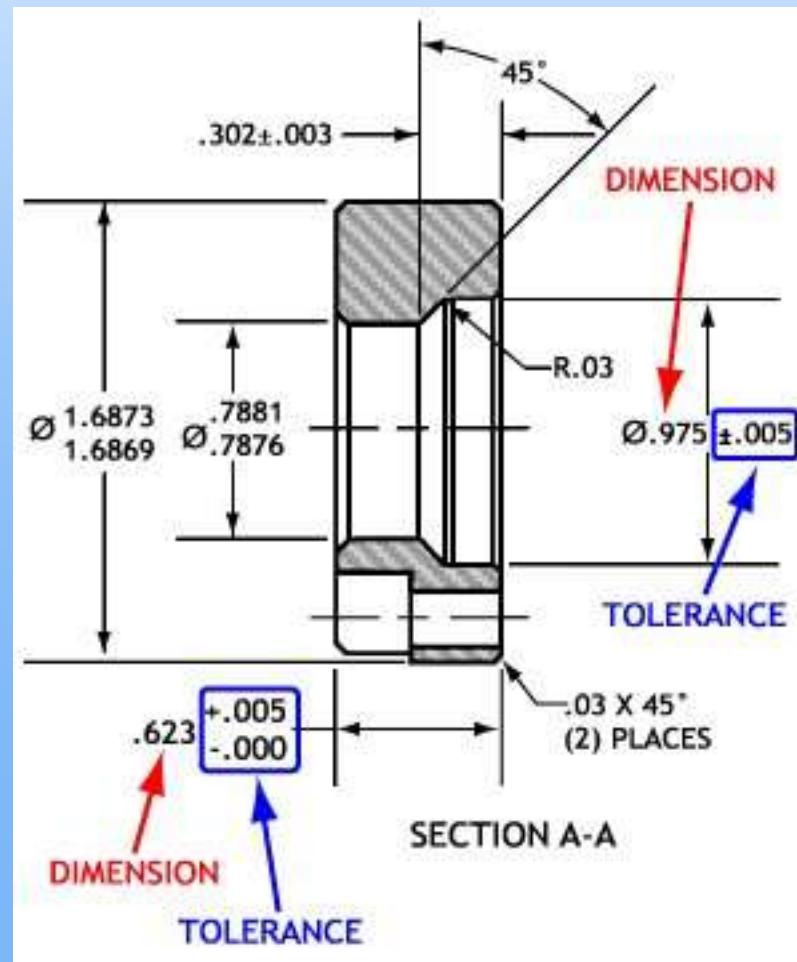
- 6
- 7
- 8
- 9
- NONE



Feature of Size

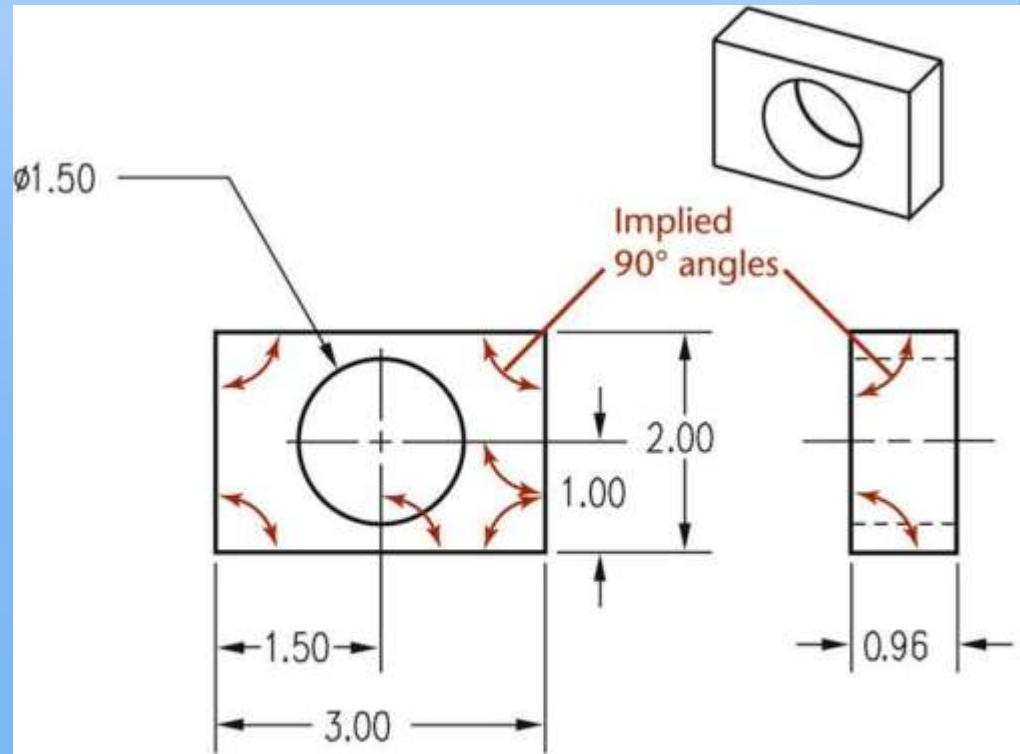
- ▣ Feature of Size
 - ▣ One cylindrical or spherical surface or a set of two opposed elements or opposed parallel surfaces associated with a size dimension.
- ▣ What does this really say?

Features of Size



Features of Size

- How many features of size in the drawing?
 - 2
 - 3
 - 4
 - 5
 - NONE



ALL TOLERANCES ± 0.002 UNLESS OTHERWISE NOTED.
ANGULAR TOLERANCES $\pm 1^\circ$.

Internal Features of Size

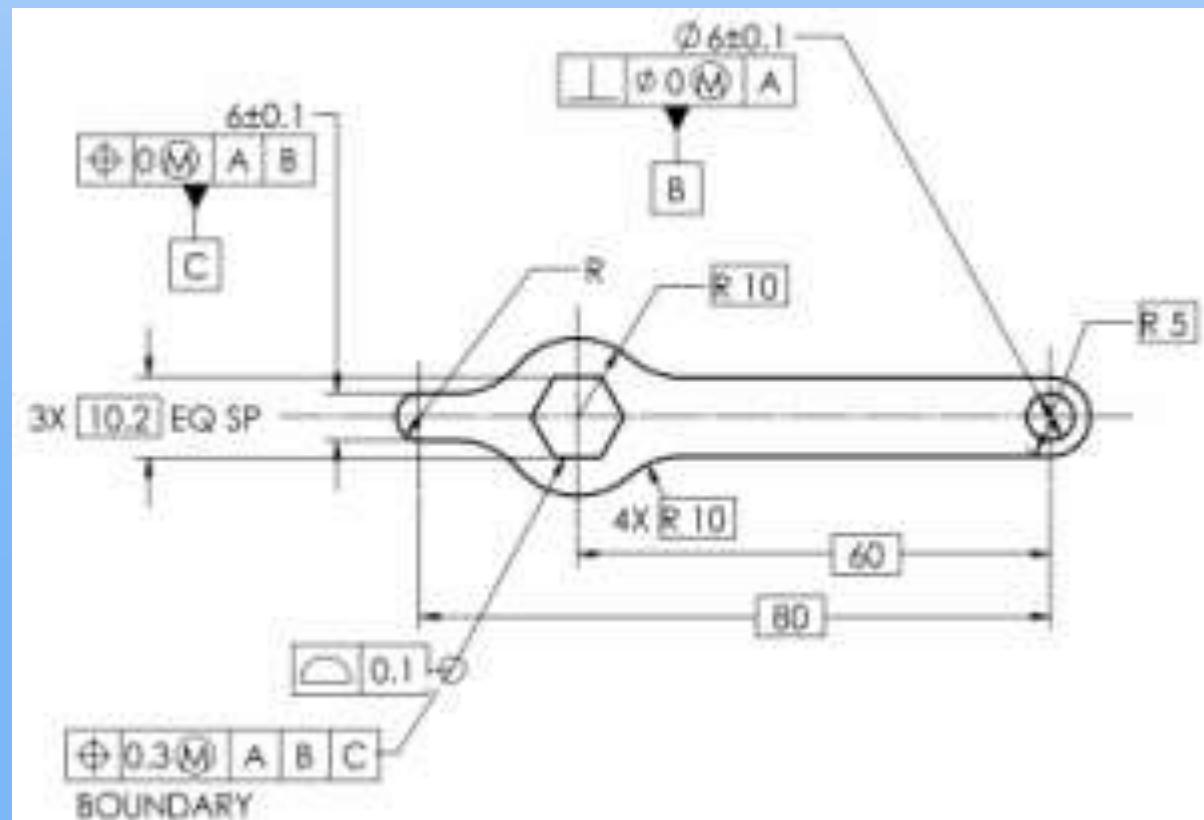
■ Internal FOS

- Comprised of part surfaces, or elements, that are internal part surfaces such as a hole diameter or the width of a slot.

Internal Features of Size

- How many Internal FOS are there in the drawing?

- 2
 - 3
 - 4
 - 5
 - NONE



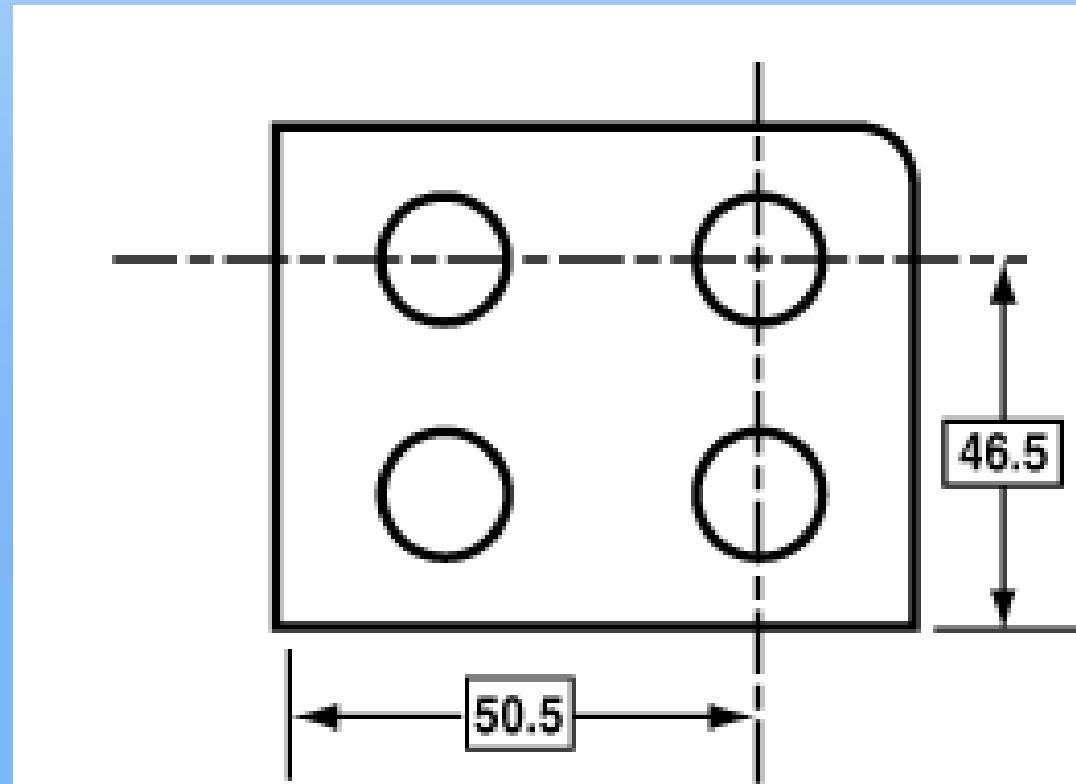
External Feature of Size

□ External FOS

- Comprised of part surfaces, or elements, that are external surfaces such as a shaft diameter or an overall width or height of a planar part.

External Features of Size

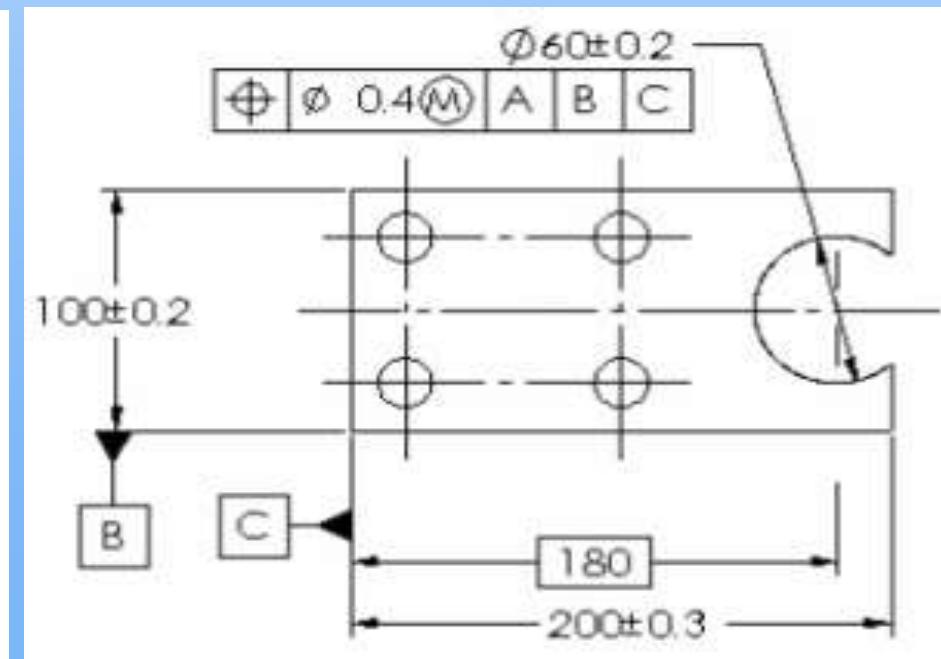
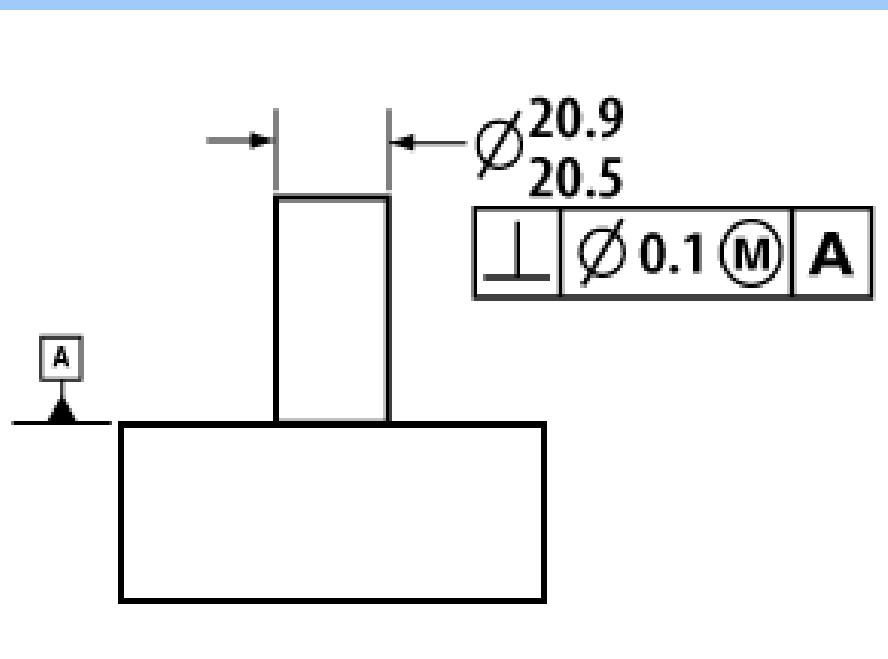
- How many external FOS are there in the drawing?
 - 2
 - 3
 - 4
 - 8
 - NONE



Feature of Size Dimensions

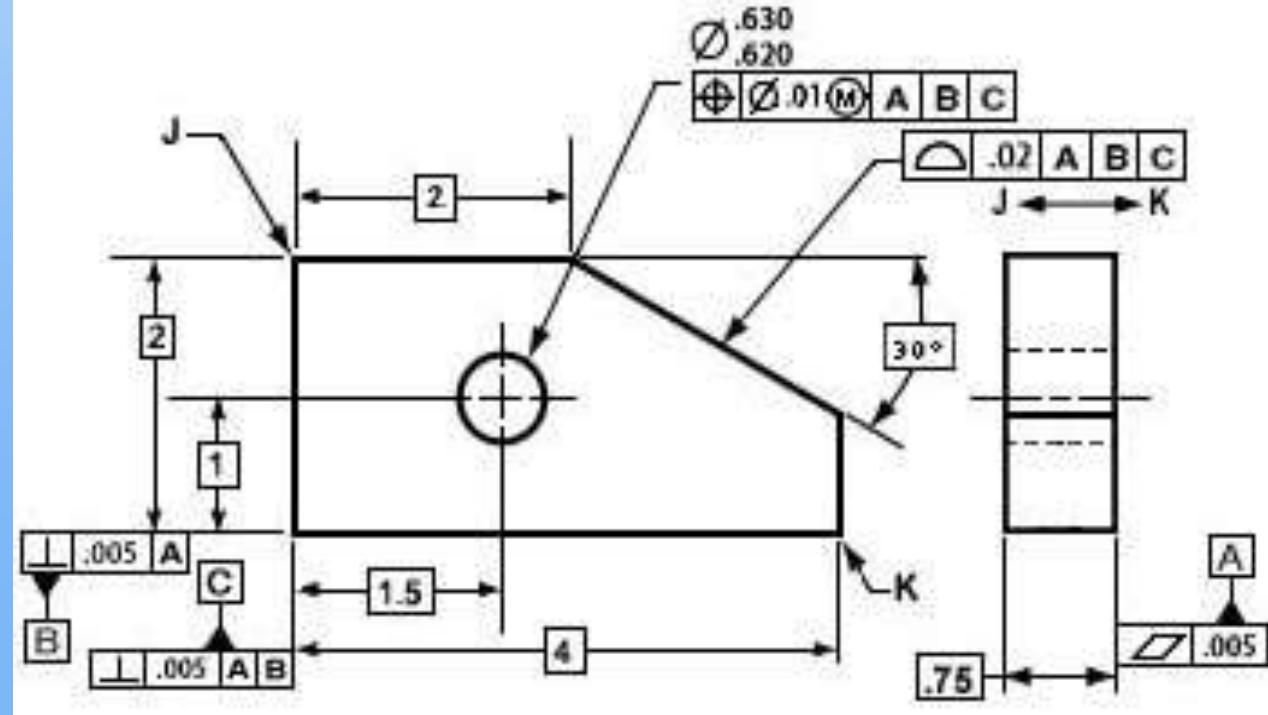
- ❑ A feature of size dimension is a dimension that is associated with a feature of size.
- ❑ A non-feature of size dimension is a dimension not associated with a feature of size.

Feature of Size Dimensions



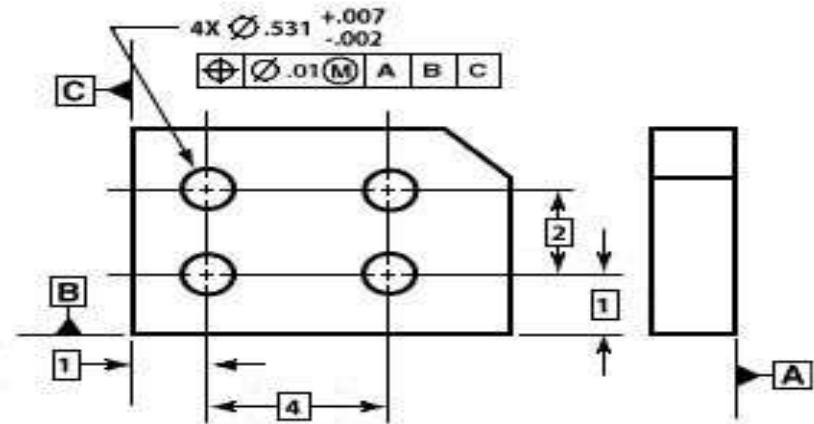
Feature of Size Dimensions

- How many feature of size dimensions are in this drawing?
 - 3
 - 4
 - 5
 - 6
 - 7
 - NONE



Non-Feature of Size Dimensions

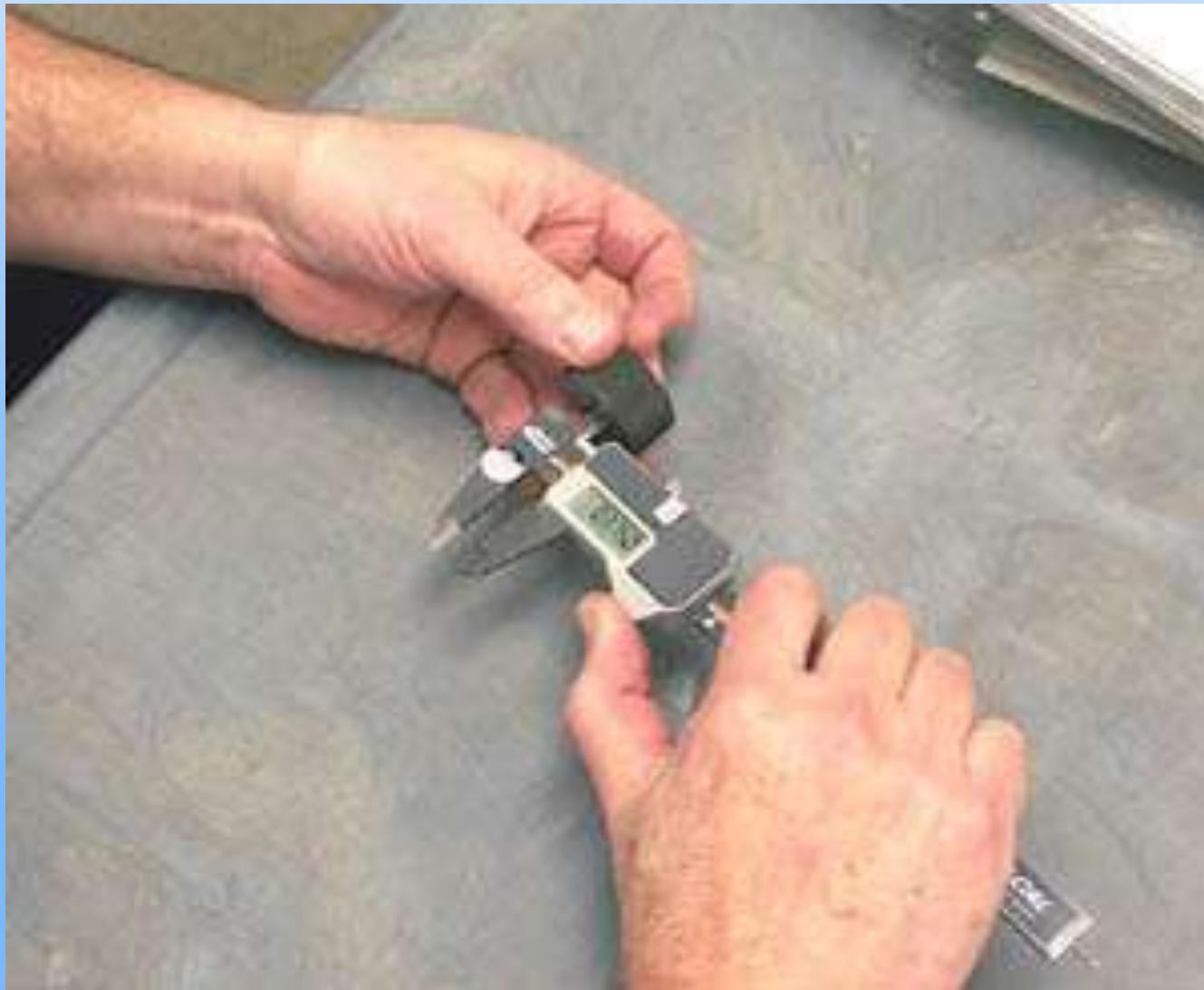
- How many non-FOS dimensions are there in the drawing?
 - 2
 - 3
 - 4
 - 5
 - NONE



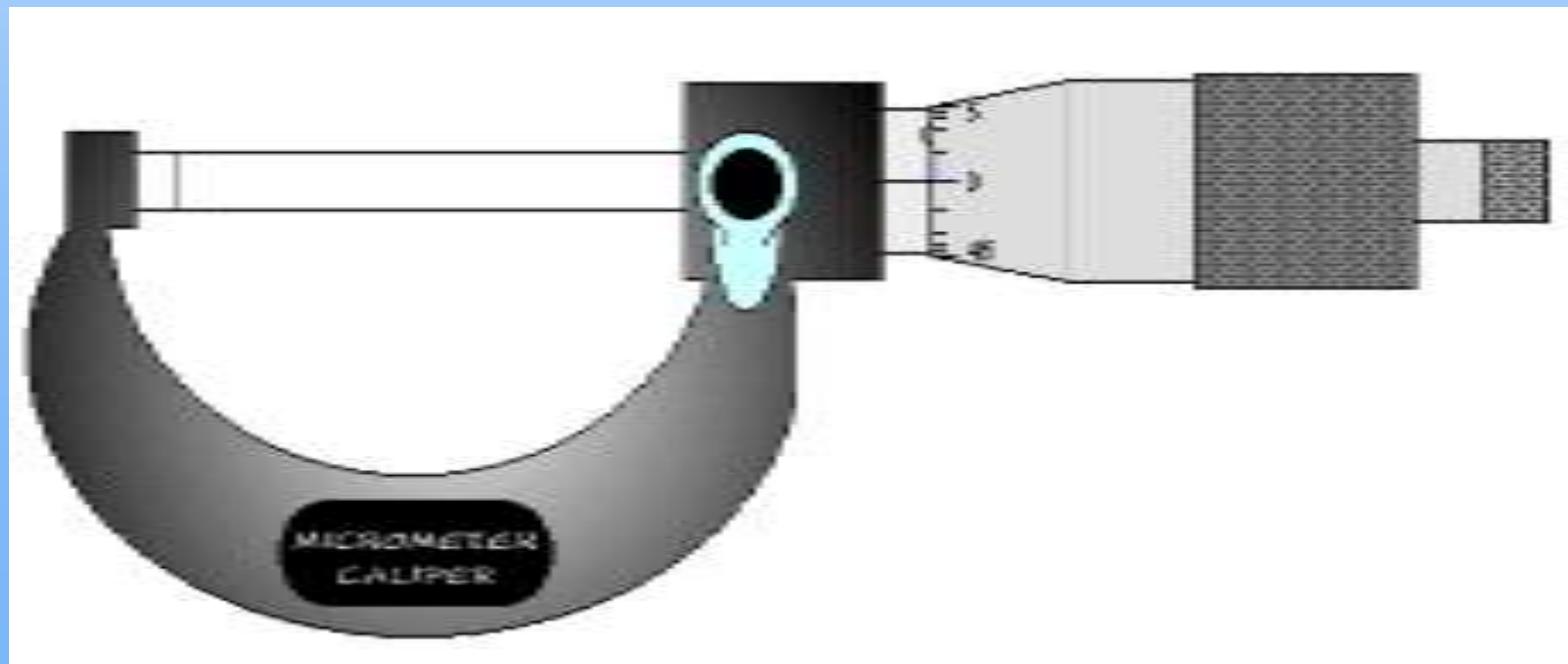
Actual Local Size

- ❑ The value of any individual distance at any cross section of a feature of size. The actual local size is a two-point measurement taken with an instrument like a caliper or micrometer that is checked at a point along the cross section of the part.
- ❑ A feature of size may have several different values of an actual local size.

Actual Local Size



Actual Local Size



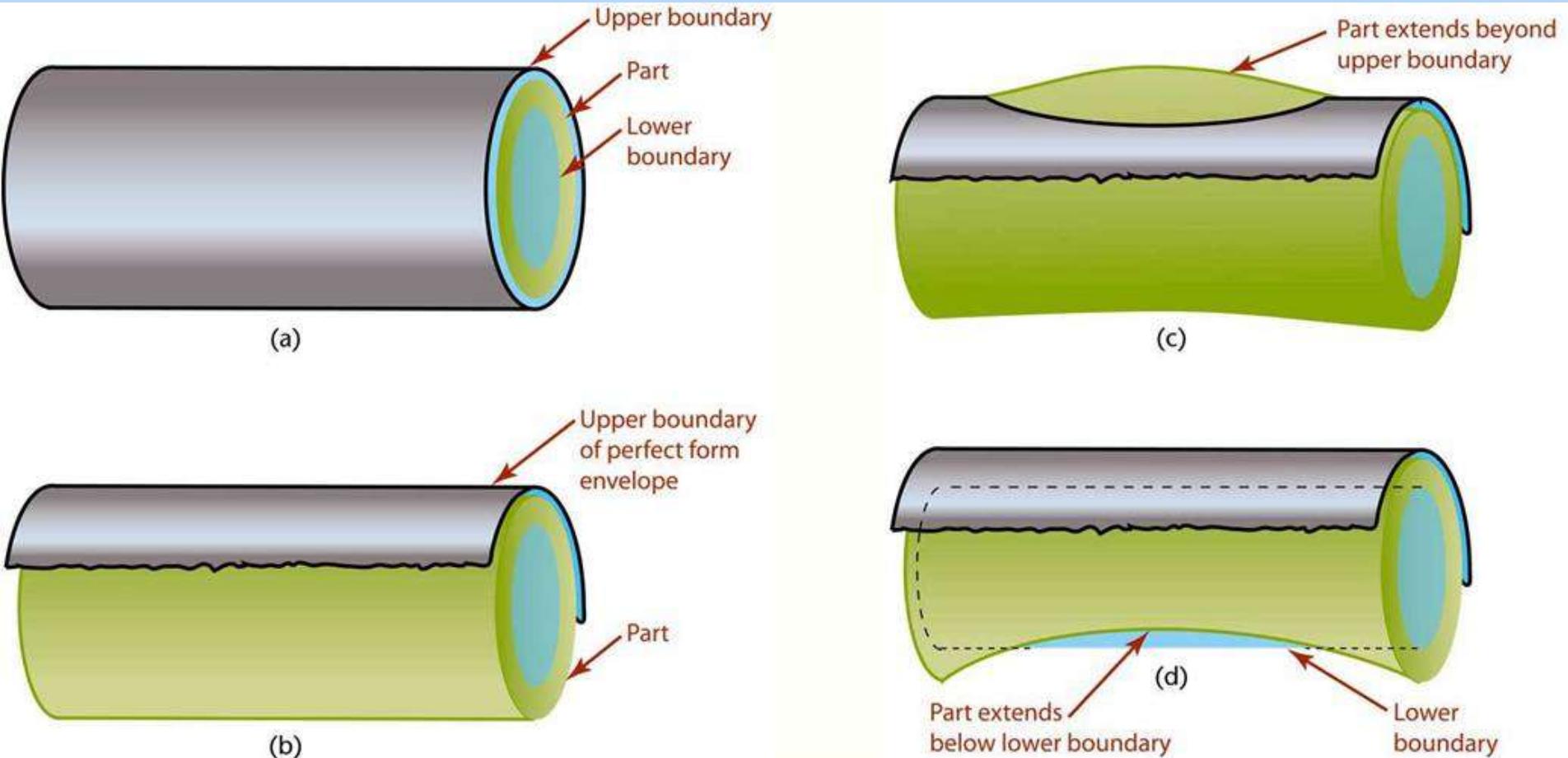
Actual Local Size

- ❑ If a hole has a dimension and tolerance of:
 - ❑ Diameter: 20 mm
 - ❑ Tolerance: $+\text{-} .5$ mm
- ❑ What would be three local readings that would allow the hole to be within specification?

Actual Mating Envelope

- ❑ An actual mating envelope is defined by the type of feature of size considered.
- ❑ The actual mating envelope of an external feature of size is a similar perfect feature counterpart of the smallest size that can be circumscribed about the feature so it just contacts the surfaces at the highest points.
- ❑ The AME will vary as this is based upon the actual part.

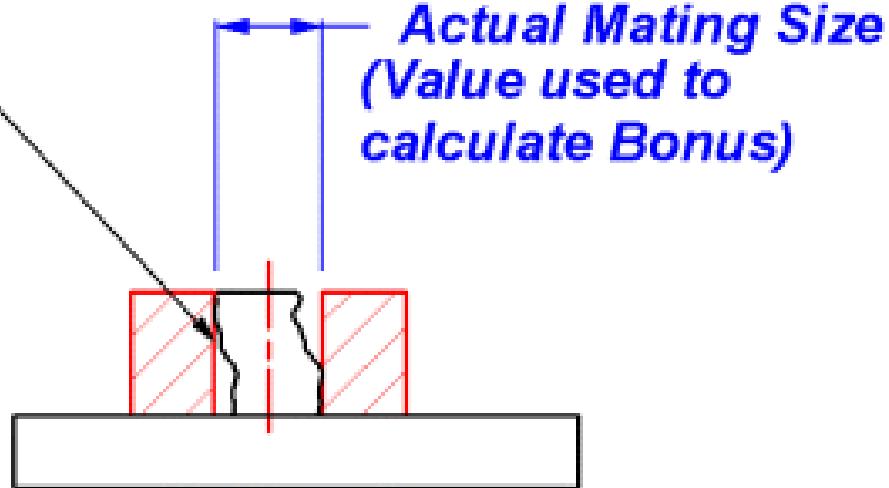
External Actual Mating Envelope



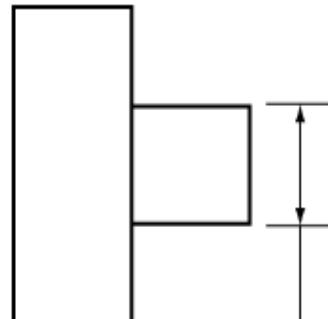
Actual Mating Envelope

Actual Mating Envelope

[Oriented with respect to the appropriate datum(s)]



A



The red hashed sections are the similar perfect surface At the smallest size which just touches the highest Points of the feature of size.

Internal Actual Mating Envelope

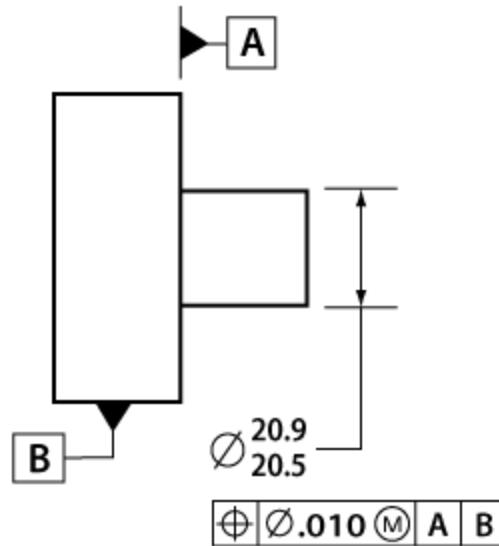
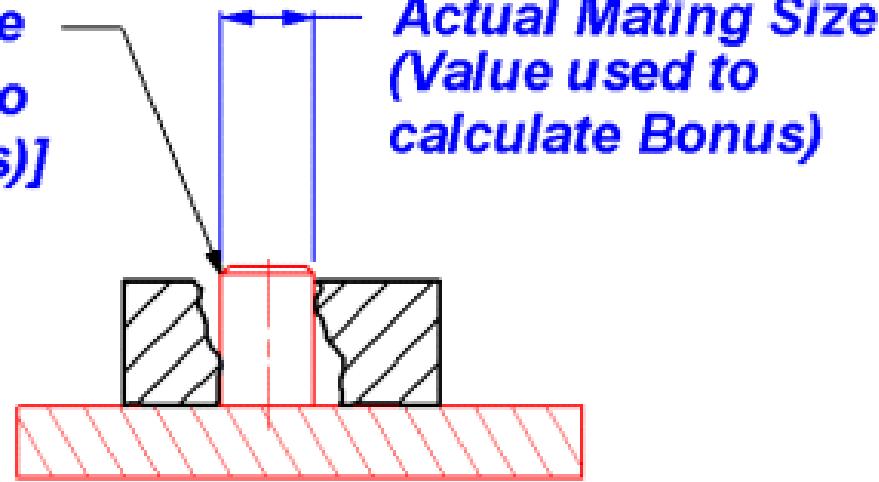
- The actual mating envelope of an internal feature of size is a similar perfect feature counterpart of the largest size that can be inscribed within the feature so that it just contacts the surface at their highest points.

Internal Actual Mating Envelope



Internal Actual Mating Envelope

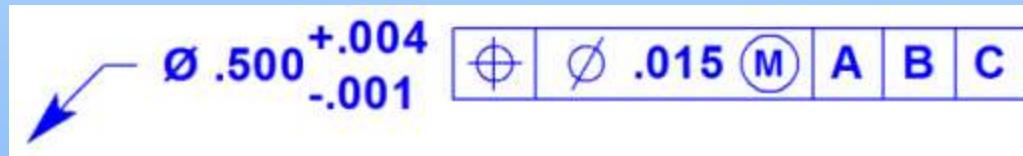
Actual Mating Envelope
[Oriented with respect to the appropriate datum(s)]



The red hashed area represents the similar perfect surface inscribed Within the feature of size.

Actual Mating Envelope

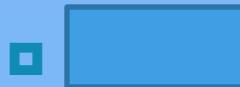
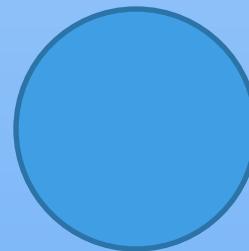
- **Note:** The Actual Mating Envelope must be oriented relative to the specified Datums. When an inspector merely uses the size of a feature to calculate the bonus tolerance, out of spec parts may be accepted.



- If a hole, for instance, has the following size and geometric control, and the hole measures .502. It would be incorrect to use a bonus tolerance of .003 (.502 - .499(MMC)) if the hole is not perfectly oriented to the Datums. If the hole is out of perpendicular to datum A by .002, for instance, the bonus that may be used is reduced by that amount. The bonus would be merely .001 and the allowable position tolerance = .016.

Actual Mating Envelopes

- A hole is defined as: Diam. 25 mm +1,-0.
 - There are four local measurements of 25.2, 25.6, 25.4, 25.0.
 - What is the AME size?



Material Conditions



Material Conditions

- ❑ GD&T allows for certain modifiers to be used in specifying tolerances at various part feature conditions. These conditions may be largest size, smallest size or actual size of that feature of size.
- ❑ Material conditions may only be used when referring to a feature of size.

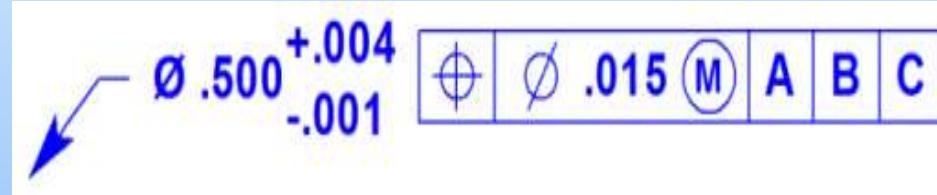
Material Conditions

Geometric Characteristic Symbols			Modifying Symbols		
	Type of Tolerance	Characteristic	Symbol	Term	Symbol
For individual features	Form	Straightness	—	At maximum material condition	(M)
		Flatness	□	At least material condition	(L)
		Circularity (roundness)	○	Projected tolerance zone	(P)
		Cylindricity	Ø	Free state	(F)
For individual or related features	Profile	Profile of a line	⌒	Tangent plane	(T)
		Profile of a surface	⌒	Diameter	Ø
For related features	Orientation	Angularity	∠	Spherical diameter	sØ
		Perpendicularity	⊥	Radius	R
		Parallelism	//	Spherical radius	SR
	Location	Position	⊕	Controlled radius	CR
		Concentricity	○	Reference	()
	Runout	Symmetry	≡	Arc length	⌒
		Circular runout *	↗	Statistical tolerance	(ST)
		Total runout *	↔	Between *	↔

Material Condition Modifiers

MATERIAL CONDITION MODIFIERS	
TYPE	SYMBOL
MAXIMUM MATERIAL CONDITION	(M)
LEAST MATERIAL CONDITION	(L)
REGARDLESS OF FEATURE SIZE	(S) OR NO SYMBOL

Maximum Material Condition



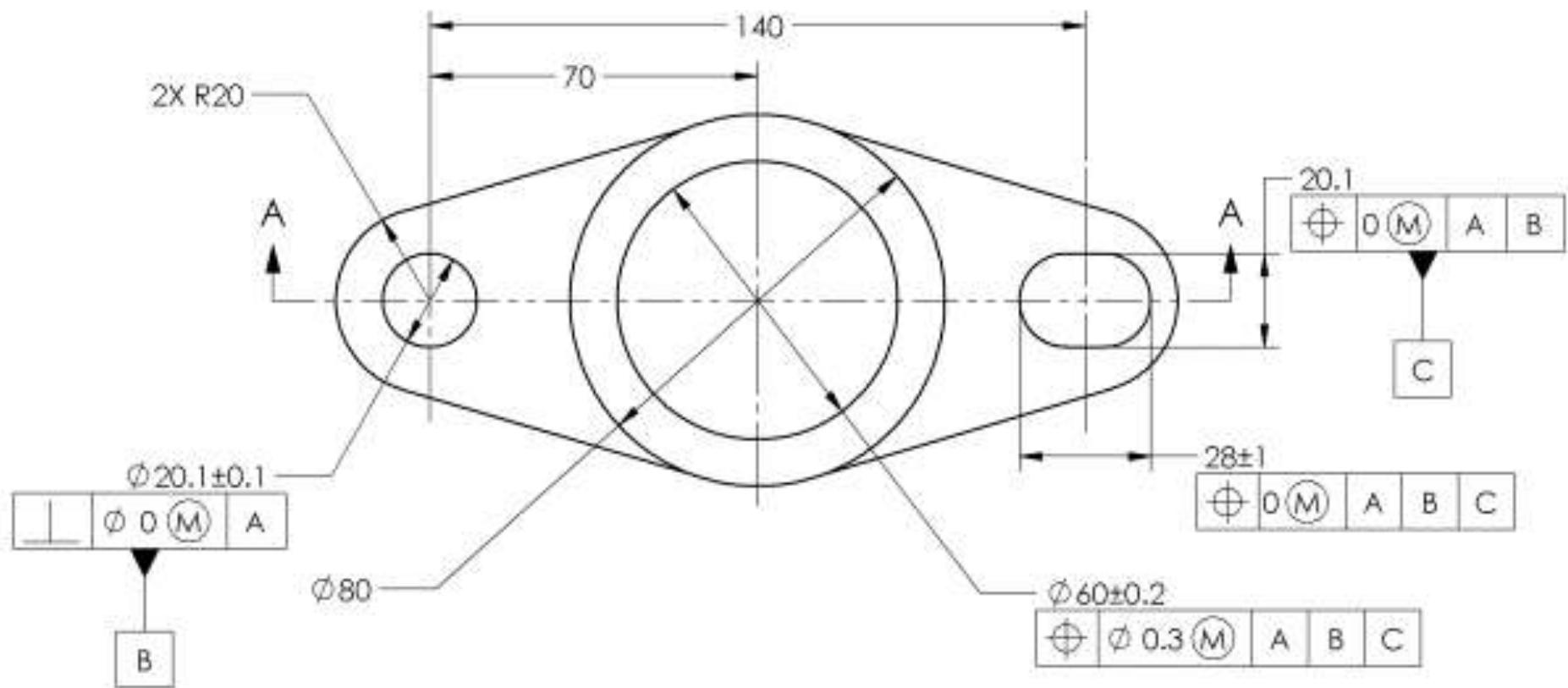
- Maximum material condition  is when a feature of size contains the maximum amount of material everywhere within the stated limits of size.
- This could be the largest shaft diameter or smallest hole.

Maximum Material Condition

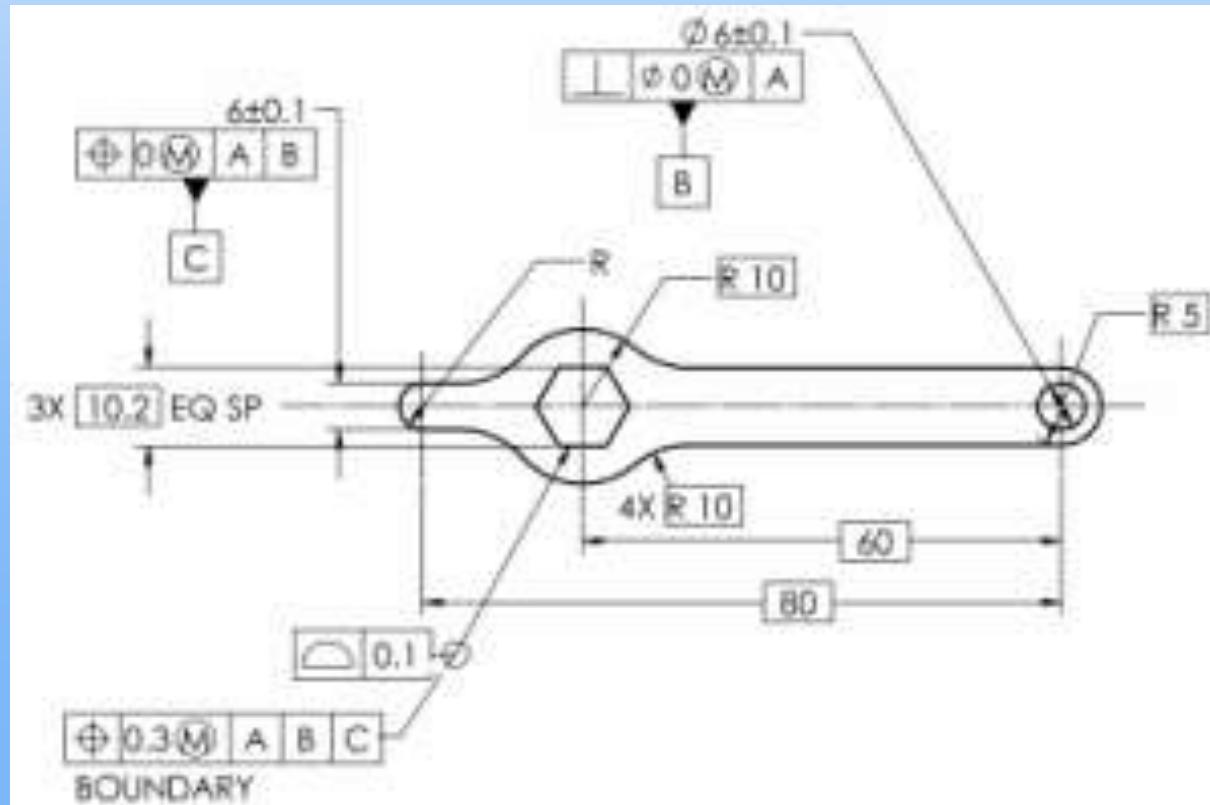
■ Points to Remember:

- Maximum material condition, MMC, of an external feature of size is the largest size limit.
- Maximum material condition, MMC, of an internal feature of size is the smallest size limit.

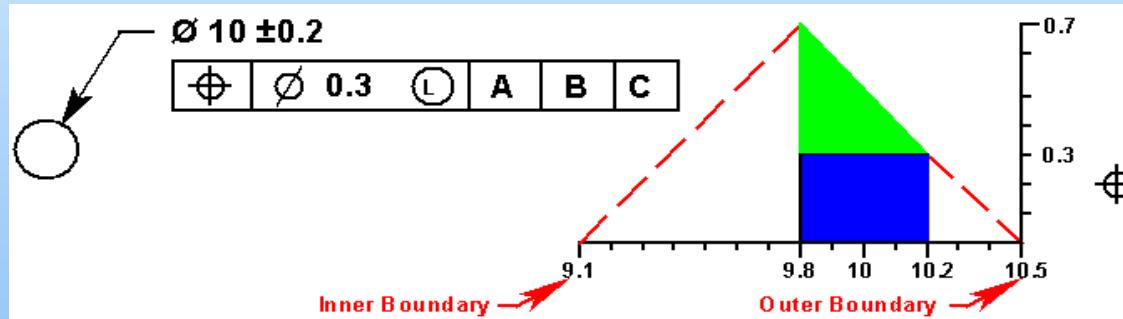
Maximum Material Condition



Maximum Material Condition



Least Material Condition



- Least Material Condition (LMC) is the condition in which a feature of size contains the least amount of material everywhere within the stated limits of size.
- This would be the smallest shaft diameter or the largest hole diameter.

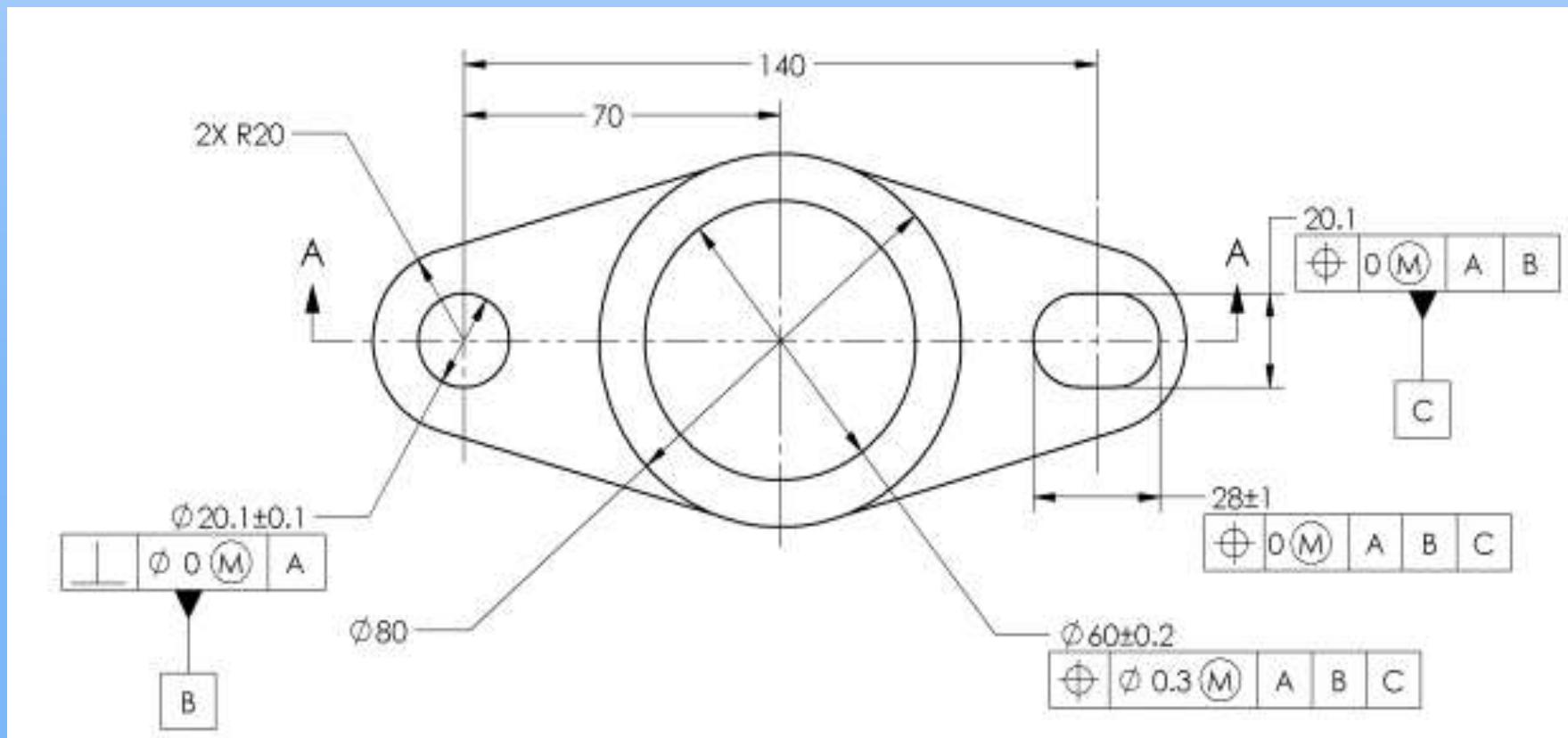
Least Material Condition

■ Points to Remember

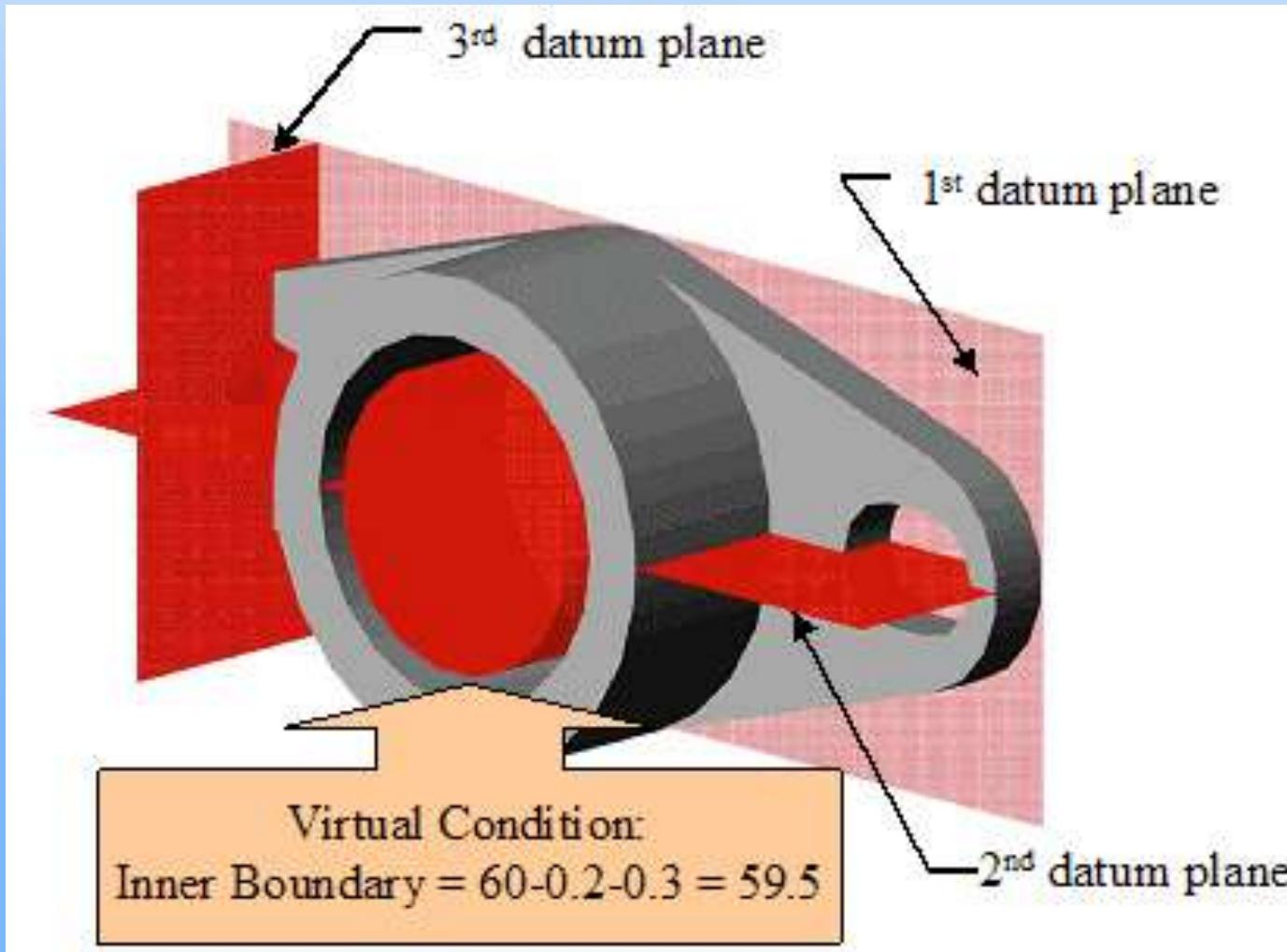
- The least material condition, (LMC), for an external feature of size is the smallest size limit.
- The least material condition, (LMC), for an internal feature of size is the largest size limit.

Least Material Condition

- What is the LMC of the hole with internal diameter of 60?



Least Material Condition

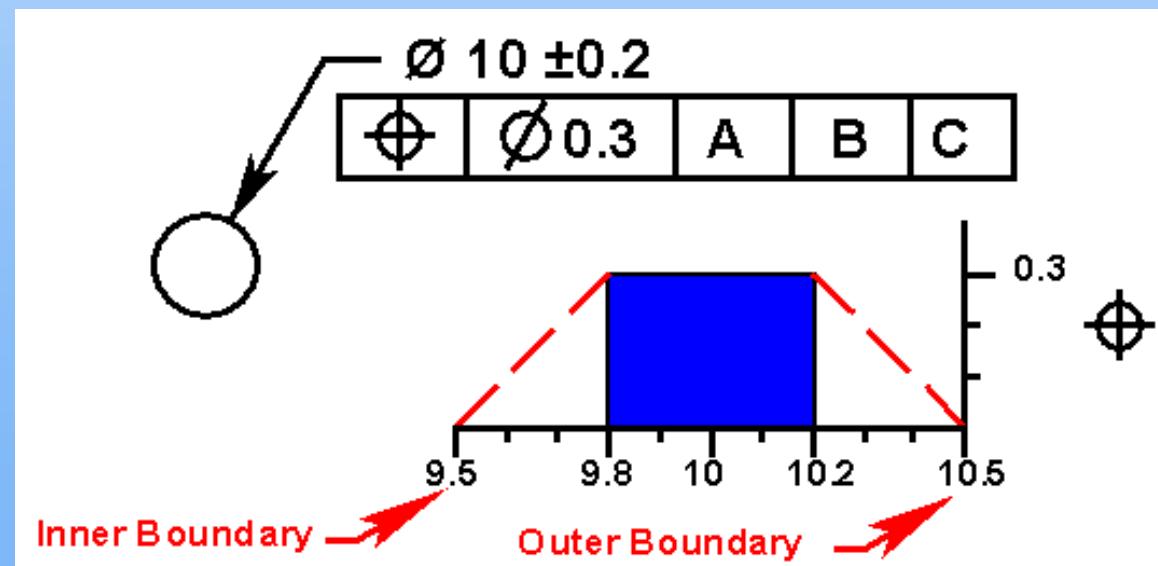


Regardless of Feature Size

- ❑ Regardless of feature size is the term that indicates a geometric tolerance applies at any increment of size of the feature within the size tolerance.
- ❑ There is no symbol for RFS.
- ❑ RFS is the default condition for all geometric tolerances.

Regardless of Feature Size

- In this example Regardless of Feature Size is implied. This is the most expensive control. Although the size may not be less than 9.8mm or greater than 10.2mm, the hole may act like 9.5mm to 10.5mm. Any holes made from 9.5mm to 9.8mm or from 10.2mm to 10.5mm should be rejected for size even though they may function.



Regardless of Feature Size

- An early look at rule two.

Rule #2

"RFS applies, with respect to the individual tolerance, datum reference, or both, where no modifying symbol is specified. MMC or LMC must be specified on the drawing where it is required."

Material Conditions and Part Dimensions

- ▣ Every feature of size has a maximum and a least material condition.
- ▣ RFS is the most expensive option available in geometric tolerancing.

Modifiers

Geometric Characteristic Symbols			Modifying Symbols		
	Type of Tolerance	Characteristic	Symbol	Term	Symbol
For individual features	Form	Straightness	—	At maximum material condition	(M)
		Flatness	□	At least material condition	(L)
		Circularity (roundness)	○	Projected tolerance zone	(P)
		Cylindricity	∅	Free state	(F)
For individual or related features	Profile	Profile of a line	⌞	Tangent plane	(T)
		Profile of a surface	⌞⌞	Diameter	∅
For related features	Orientation	Angularity	⌞⌞⌞	Spherical diameter	s∅
		Perpendicularity	⊥	Radius	R
		Parallelism	∥	Spherical radius	SR
	Location	Position	○○	Controlled radius	CR
		Concentricity	○○○	Reference	()
	Runout	Symmetry	—	Arc length	—
		Circular runout *	↗	Statistical tolerance	⟨ST⟩
		Total runout *	↗↗	Between *	↔

Modifiers

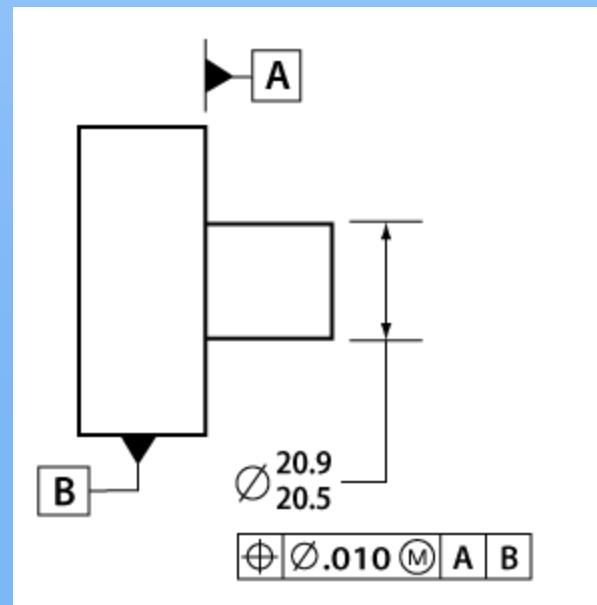
- ❑ GD&T modifiers communicate additional information about the part. This information is used to better understand the meaning of the drawing and how to manufacture that part.
- ❑ We have already discussed MMC, LMC and RFS.

Modifiers

- ▣ P: projected tolerance zone.
 - ▣ A tolerance zone that extends beyond a feature by a specified distance. Projected tolerance zones help ensure that mating parts fit during assembly.
- ▣ T: tangent plane.
 - ▣ Notes that only the tangent plane of the toleranced surface need be within the tolerance zone.

Modifiers

- ❑  :Diameter: used inside the feature control frame to denote the shape of the tolerance zone or outside the feature control frame to replace the word diameter.



Modifiers

- R: radius and CR: controlled radius
 - Used outside the feature control frame.
- (): reference
 - Denotes that information is for reference only. The information of the reference is enclosed in the parentheses.

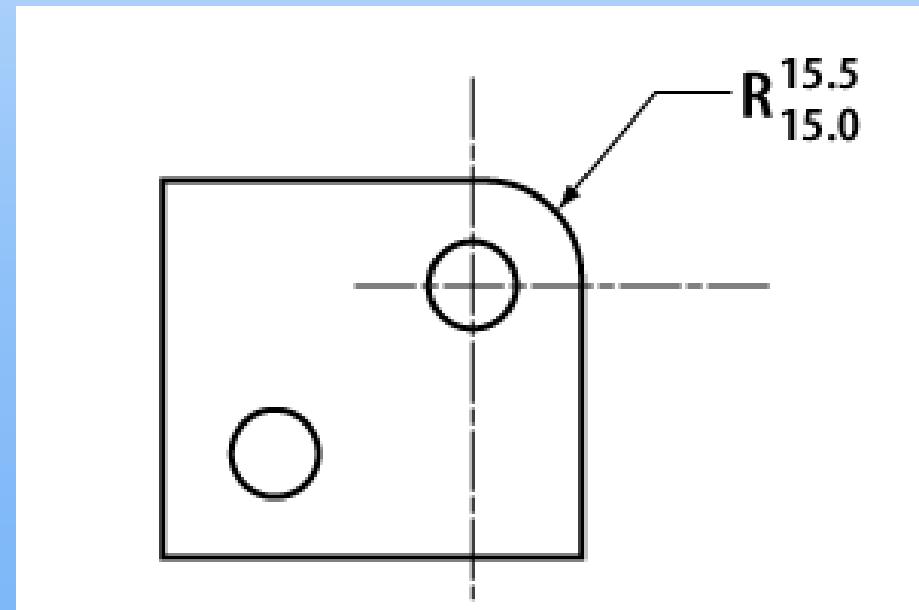
Radius

- Radius:

- A straight line extending from the center of an arc or circle to its surface.
- Symbol is “R”.
- Use of the Radius creates a zone defined by two arcs, which the actual surface must be contained within. The arcs are the minimum and maximum radii.

Radius

- This is a typical radius dimension.
 - The min and max radii are?

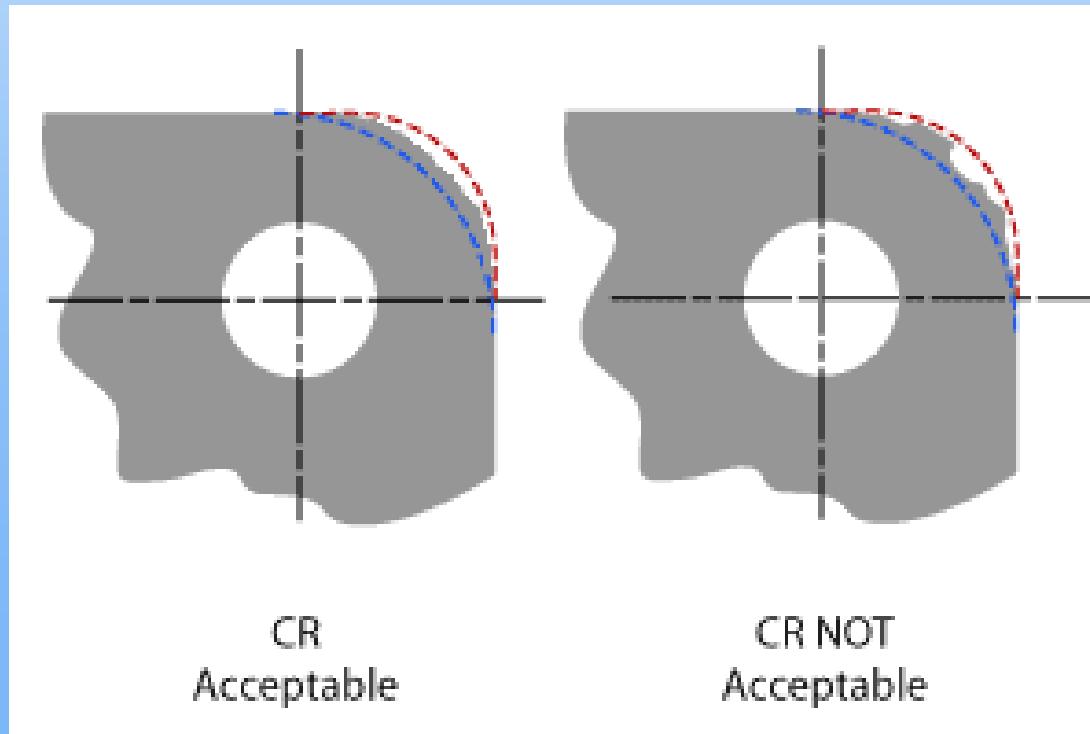


Controlled Radius

- CR: controlled radius

- A radius with no flats or reversals allowed. The drawing convention is the same but a CR must have no flats or reversals within the min and max radii.
 - The radius is marked as CR on the drawing as opposed to R for standard radius.

Controlled Radius



Geometric Tolerances

■ Geometric Characteristic Symbols

- Fourteen symbols used in GD&T to further define the characteristics of the part on the drawing.
- Five categories of characteristics exist:
 - Form
 - Profile
 - Orientation
 - Location
 - Runout

Geometric Characteristic Symbols

TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	DATUM REFERENCE
FORM	STRAIGHTNESS	—	INDIVIDUAL
	FLATNESS	/\	
	CIRCULARITY (ROUNDNESS)	○	
	CYLINDRICITY	ø	
PROFILE	PROFILE OF A LINE	⌒	INDIVIDUAL OR RELATED
	PROFILE OF A SURFACE	⌒	
ORIENTATION	ANGULARITY	∠	RELATED
	PERPENDICULARITY	⊥	
	PARALLELISM	//	
LOCATION	POSITION	⊕	
	CONCENTRICITY	○○	
	SYMMETRY	≡	
RUNOUT	CIRCULAR RUNOUT	↗	
	TOTAL RUNOUT	↗↗	

Geometric Characteristic Symbols

- ❑ Important as the first symbol in a feature control frame.
- ❑ Note that not all symbols use a datum reference frame
- ❑ .

Feature Control Frame

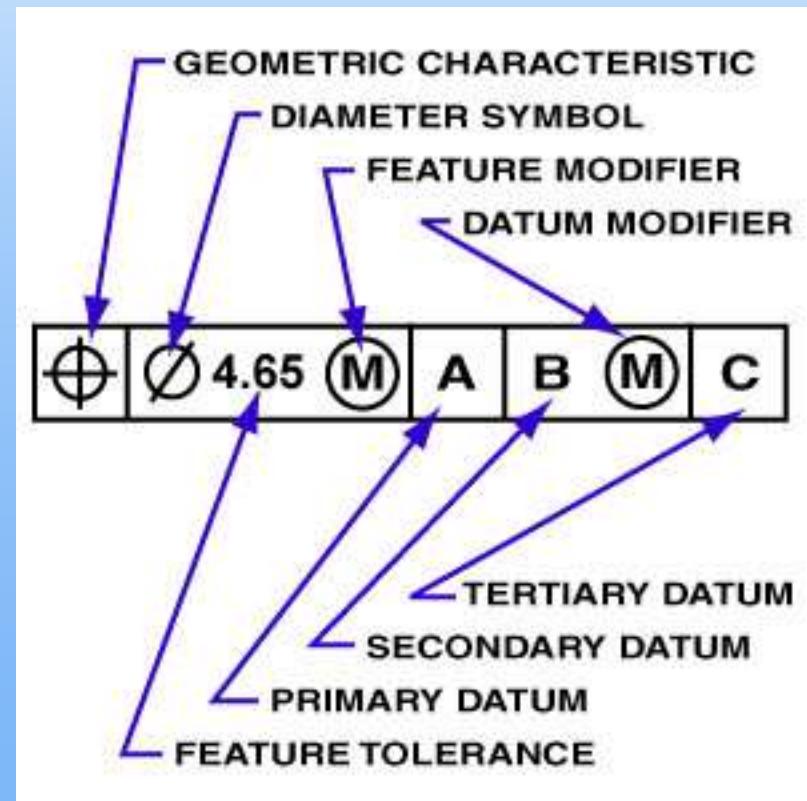
- ❑ The device by which geometric tolerances are specified on a drawing.
- ❑ A feature control frame is a rectangular box divided into compartments within which the geometric characteristic symbol, tolerance, modifiers, and datum references are used.

Feature Control Frame

- ❑ A symbol in the second compartment is not always needed. Second compartment will specify tolerance.
- ❑ Third, fourth and fifth compartments will contain datums if needed.
- ❑ A non-datum control will only have two compartments.

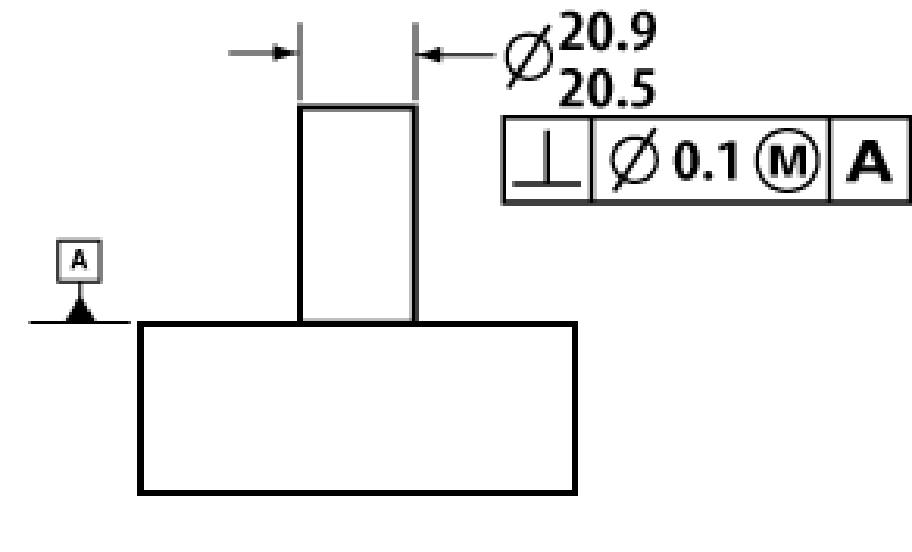
Feature Control Frame

- What does the M in the circle mean?
- What is the implied tolerance?
- What does the order of the datums represent?



Feature Control Frame

- What does the geometric characteristic symbol mean?
- Why only one datum?

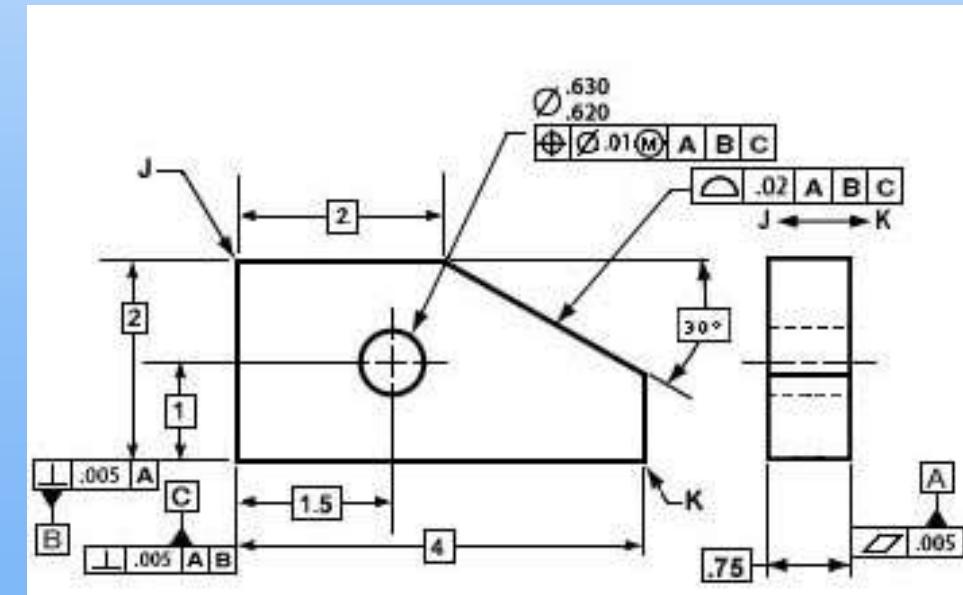


Feature Control Frame

Assume RFS if NO modifier present



Modifier MUST be present if Datum IS FOS



End of Chapter Two

- ❑ Questions?
- ❑ Time to do assessment. Remember that to get credit you must turn the assessment in before you leave class.
- ❑ Book and lab assignments are in the syllabus.
- ❑ I will be here to answer any questions.

Chapter Three

Rules and Concepts of GD&T

Chapter Goals

- ❑ Understand Rule One and Two
- ❑ Understand the concepts of basic dimensions, virtual condition, inner and outer boundary, worst case boundary and bonus tolerance.

Rule One

Rule #1

"Where only a tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its form, as well as its size, are allowed."

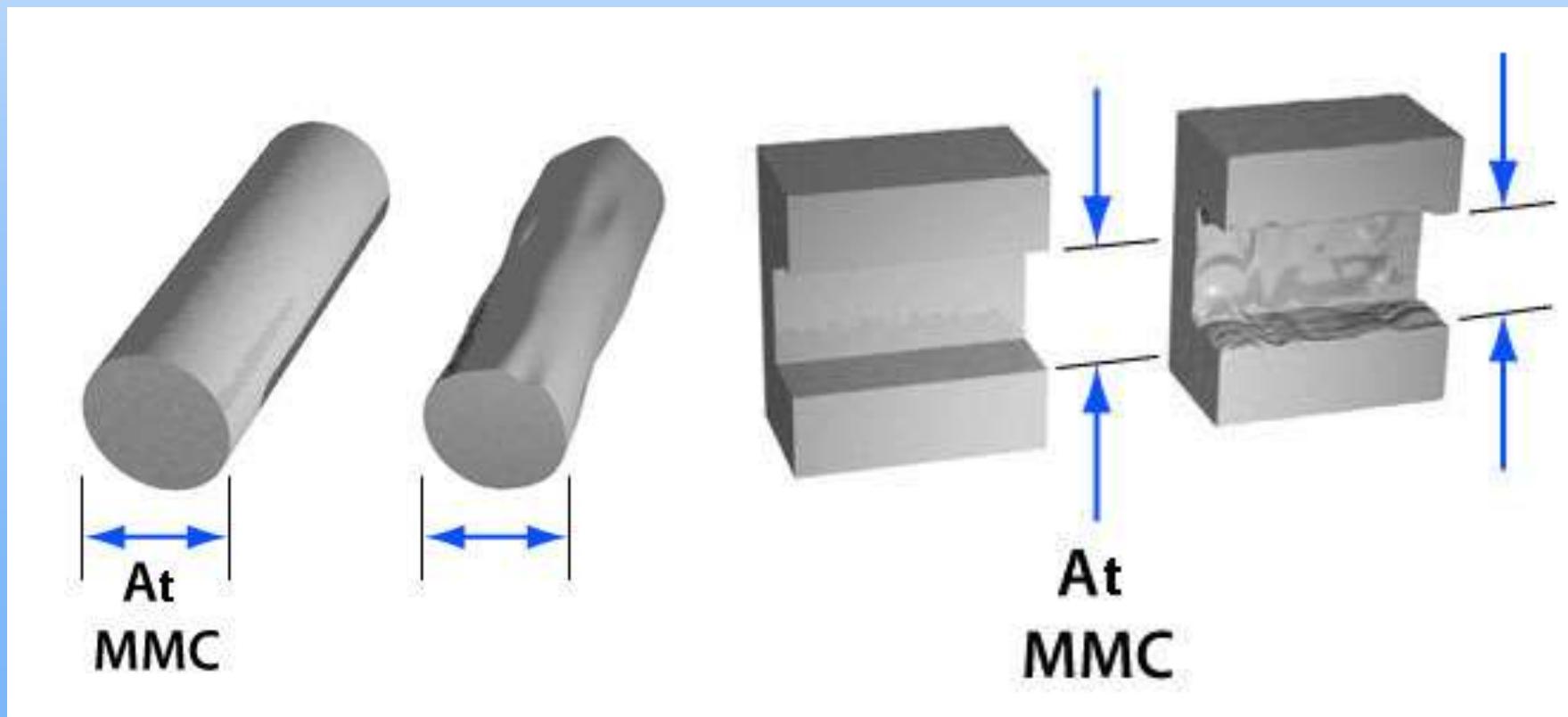
What Does Rule One mean?

- ❑ Rule states that when only a tolerance of size, i.e. ~~Ø~~ 10.9/10.5, the maximum boundary or envelope for an external FOS is at MMC and the minimum envelope for an internal FOS is at MMC.
- ❑ This rule allows for fit ups of shapes to make sure that the parts will fit together.

Rule Number One

- ❑ Rule number one is also called perfect form at MMC or the envelope rule.
- ❑ This means that if a FOS is at MMC, that FOS must have perfect form.
- ❑ What is perfect form?

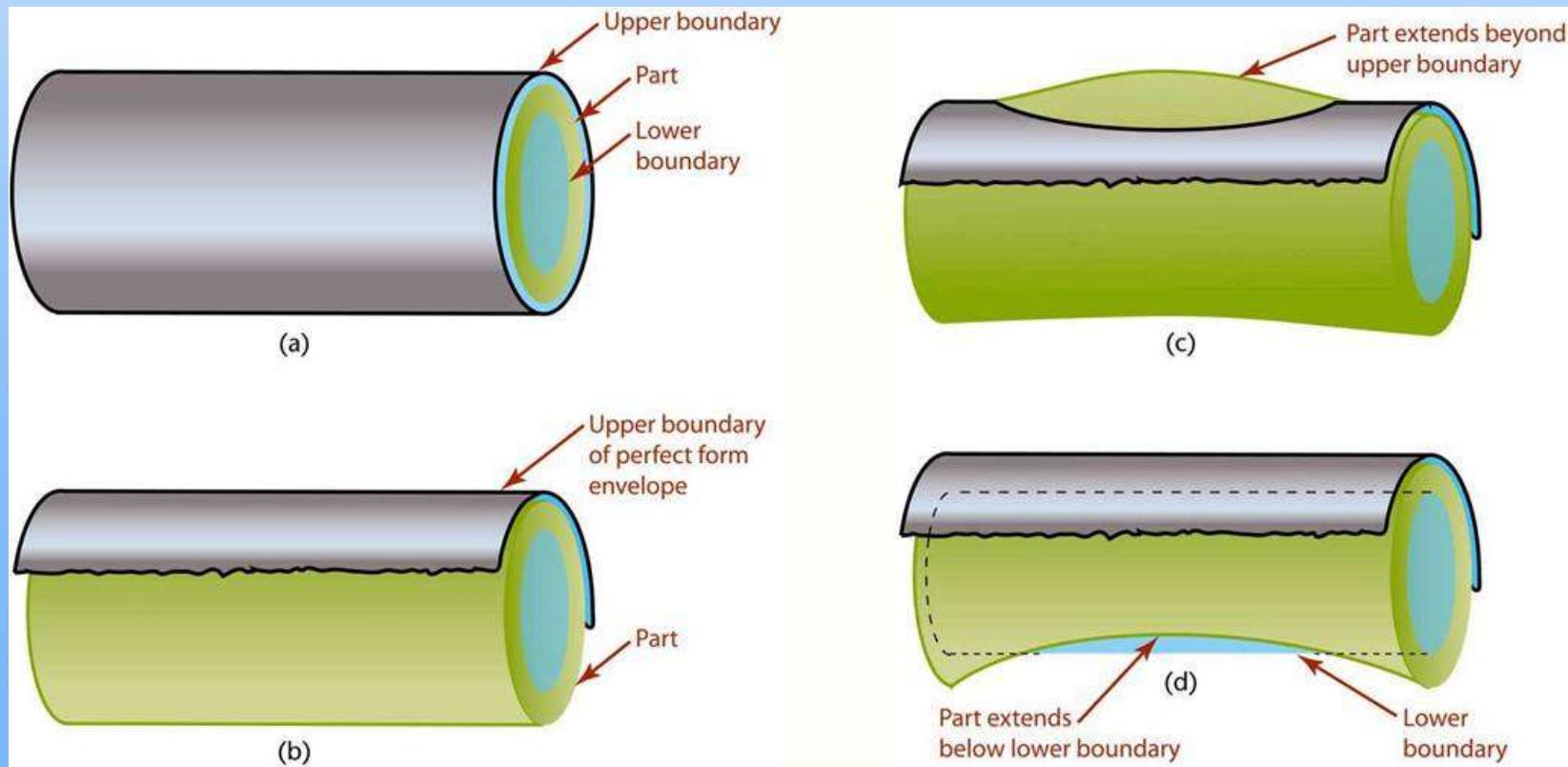
Perfect Form



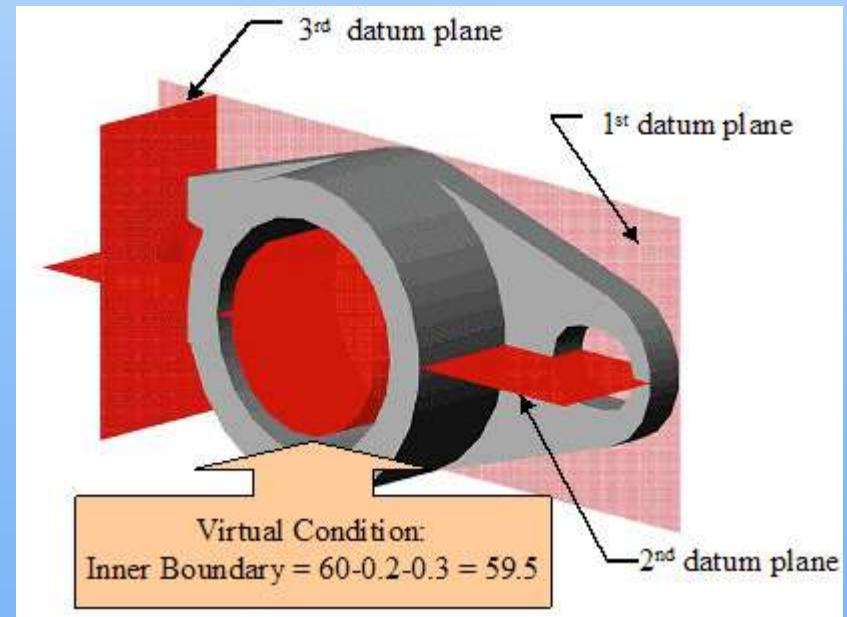
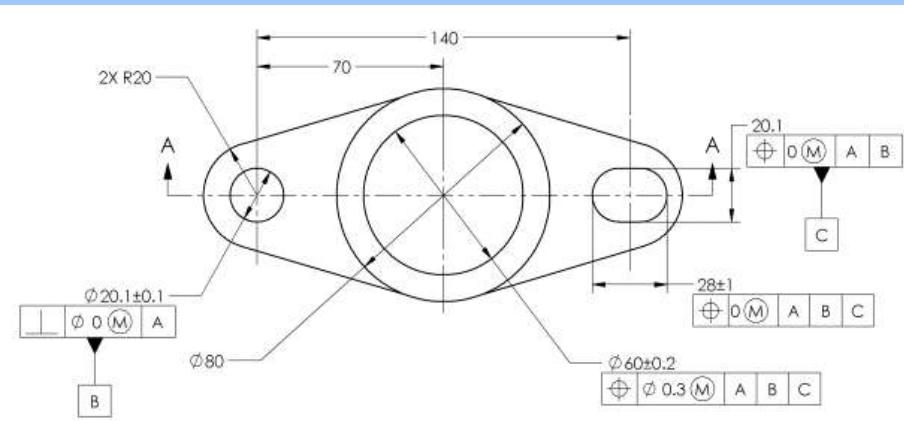
Rule Number One

- ❑ The perfect form only applies at MMC. If the FOS has a size less than MMC, then a form error is allowed equal to the amount the FOS departed from MMC.
- ❑ What does this mean?

Rule Number One



Perfect Form and Variation Allowed



Rules of Form

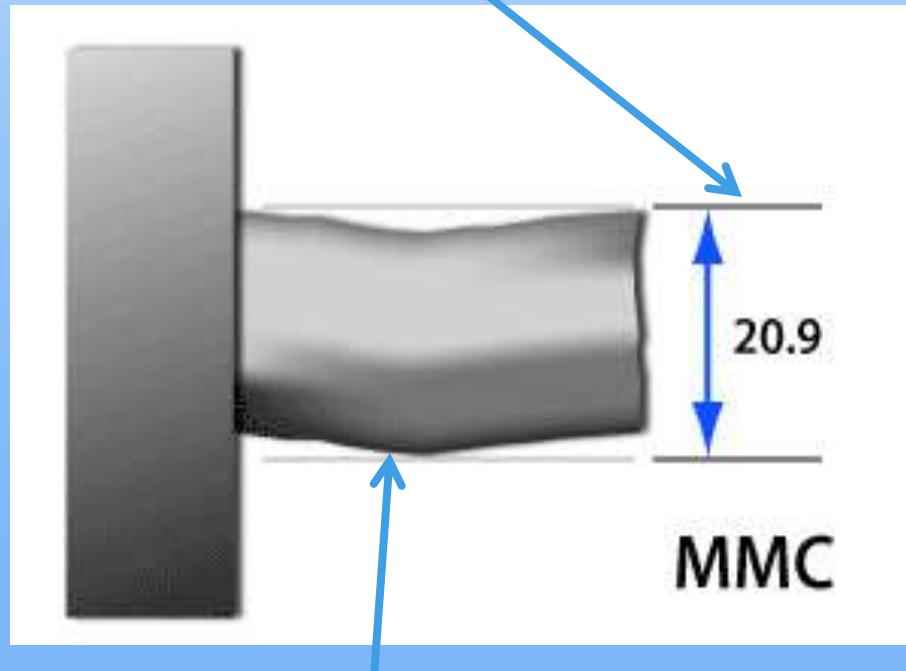
- Surfaces of an FOS shall not extend beyond a boundary or envelope of perfect form at MMC.
- If the actual local size deviates from MMC, towards LMC, the form may vary by the amount of deviation.
- The actual local size must be within the specified tolerance zone.
- There is no requirement for a boundary of perfect form at LMC. Variation allowed is the amount of deviation from MMC.
- What happens if the FOS goes below LMC?

How To Calculate Allowable Variation!

- ❑ Example on page 49.
 - ❑ Remember that this is for a FOS that only has a tolerance of size specified.
 - ❑ MMC = 10.8
 - ❑ LMC = 10.2
 - ❑ Maximum variation from MMC allowed is MMC – LMC or .6

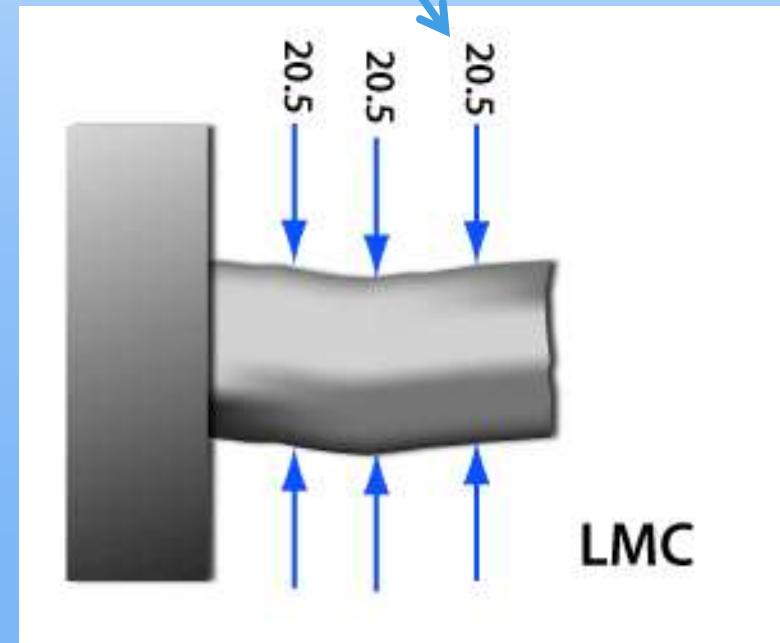
Another Example

Perfect Form at
MMC



Form Error

Local
Measurements



When to Override Rule One

- ▣ Two ways to override rule one
 - ▣ If a straightness control is applied
 - ▣ A special note applied to the FOS stating perfect form at MMC not required
- ▣ Note that rule one does not control location, orientation or relationship between Features of Size.

More Examples of Rule One

- ❑ Figure 3.3 page 51.

Other Exemptions to Rule One

- ❑ An FOS on a non-rigid part
- ❑ Stock sizes

Inspecting a Feature of Size

■ Important

- When inspecting a FOS that is controlled by rule one, both form and size must be verified. Both the MMC and envelope may be verified with a go gage.
- See page 53 Figure 3-4 for more examples.



Rule Number Two

Rule #2

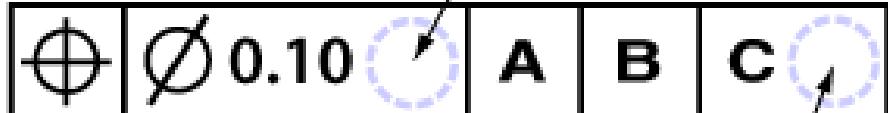
"RFS applies, with respect to the individual tolerance, datum reference, or both, where no modifying symbol is specified. MMC or LMC must be specified on the drawing where it is required."

Rule Number Two

- ❑ Rule two is for circumstances where rule number one does not apply. If there is no modifier, RFS applies and so does rule number two.
- ❑ Rule 2A is temporary and says the same thing as rule two except the S shows that RFS applies. The S is from earlier versions of YI4.5 and is temporally allowed to help in the transitions between standard versions.

RFS and Rule Number Two

Assume RFS if NO modifier present



Modifier MUST be present if Datum IS FOS

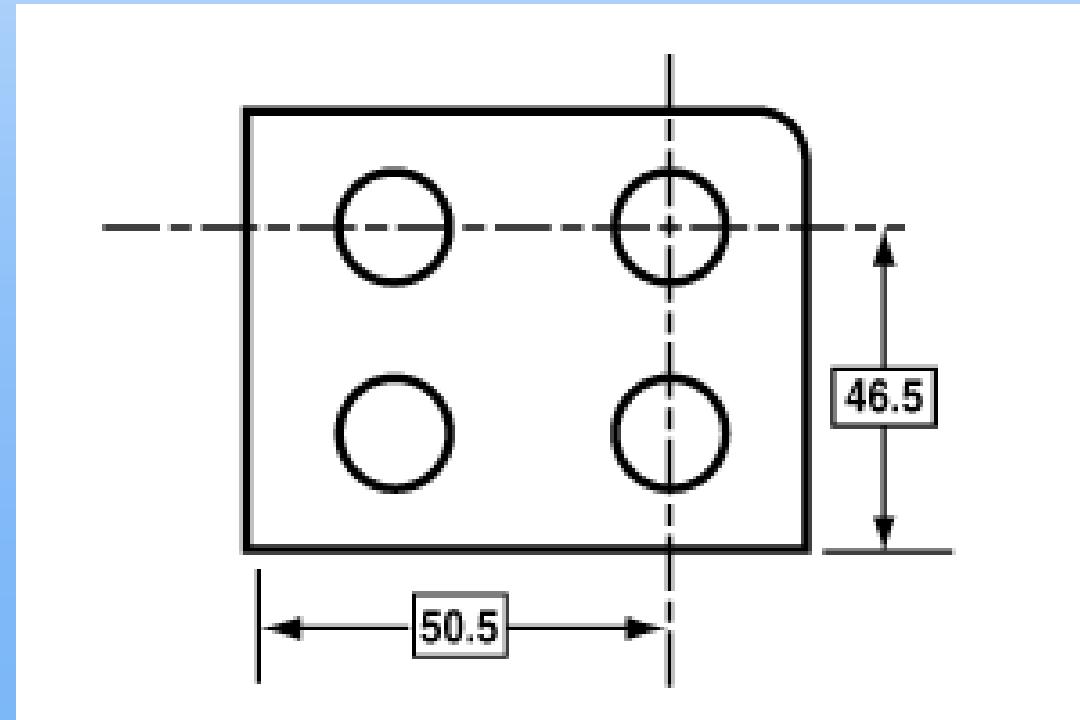
MATERIAL CONDITION MODIFIERS

TYPE	SYMBOL
MAXIMUM MATERIAL CONDITION	(M)
LEAST MATERIAL CONDITION	(L)
REGARDLESS OF FEATURE SIZE	(S) OR NO SYMBOL

Intro To Basic Dimensions

- Basic dimension:

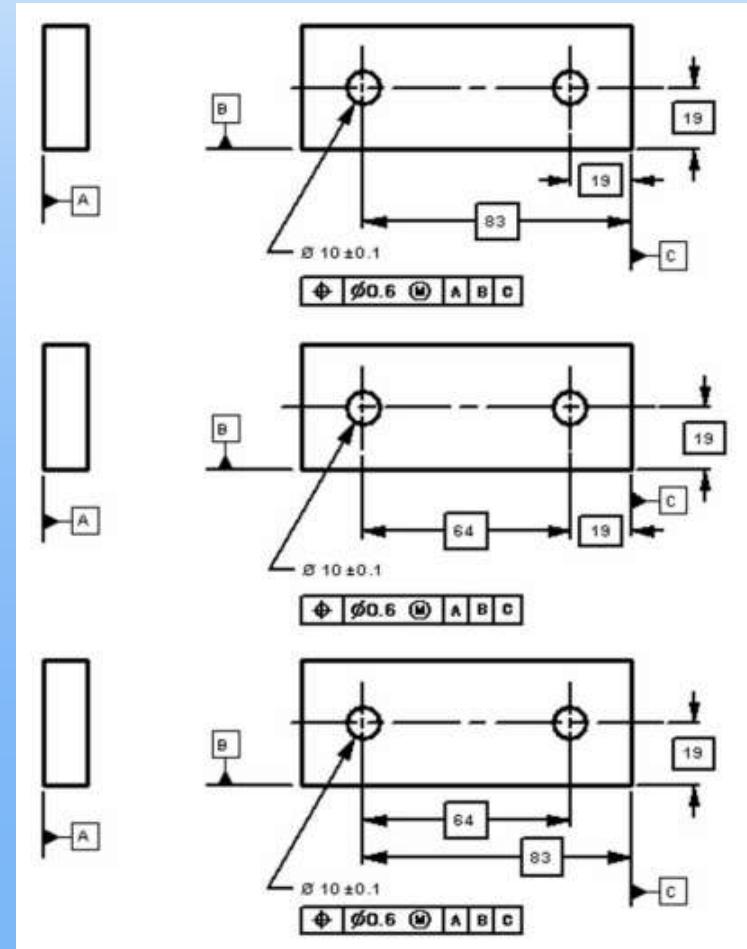
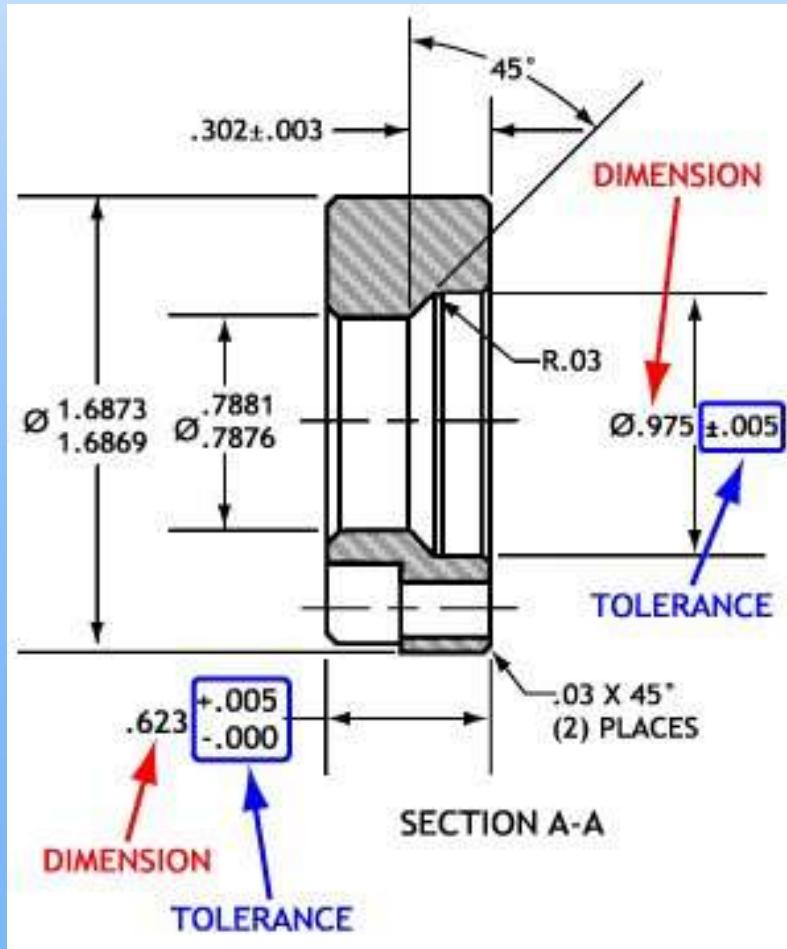
- A numerical value used to describe the theoretically exact size, true profile, orientation or location of a feature or gage (datum target) information.



Intro To Basic Dimensions

- ▣ Basic Dimensions are normally enclosed in a rectangle or specified on the drawing as a general note.
- ▣ A basic dimension only locates a geometric tolerance zone.
- ▣ If a basic dimension describes a part feature, a geometric tolerance must be applied.

Intro To Basic Dimensions



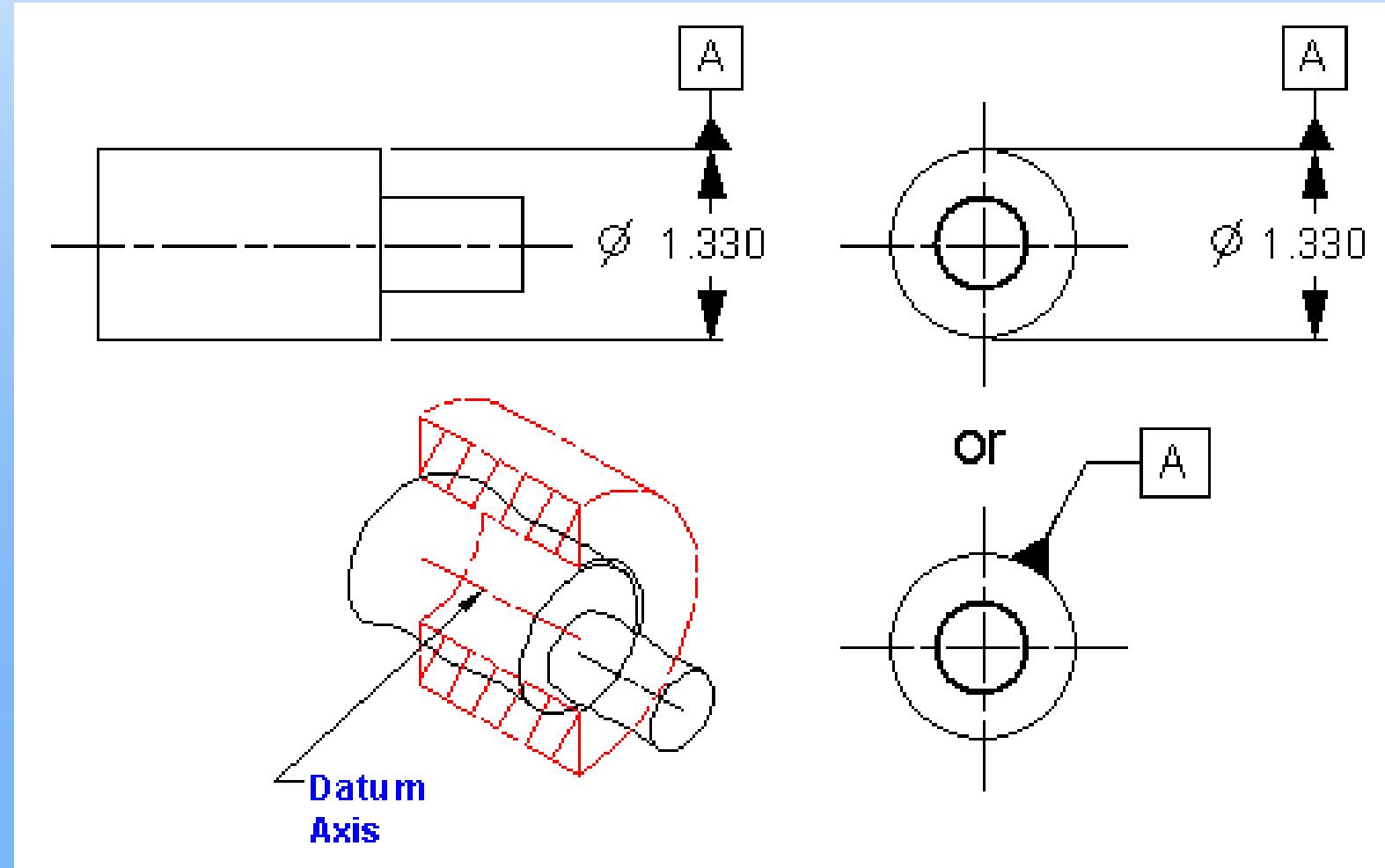
Intro To Basic Dimensions

- ▣ What would happen if there were no tolerances on a drawing?
 - ▣ A. Part cannot be made?
 - ▣ B. Part would be too costly to make?
 - ▣ C. Machinist should tell the designer to go back and study GD&T?
 - ▣ D. All of the above?

Intro To Basic Dimensions

- ▣ Basic dimensions may be used to specify datum targets.
 - ▣ These dimensions are considered to be gage targets. That is the tolerance of the gage applies to this dimension.
 - ▣ Gage tolerance is an extremely small number. Many gages have tolerances in the thousandth, ten thousandth or greater amount.

Intro To Basic Dimensions



Intro To Basic Dimensions

■ Basic Dimensions

- Used to define theoretically exact locations, orientation or true profile of part features or gage information.
- That define part features must be accompanied by a geometric tolerance.
- That define gage information do not have a tolerance on the drawing.
- Are theoretically exact. But gage makers tolerance does apply.

Knowledge Check

- ❑ A drawing shows a part with a center hole of dimension 5 and the following additional information:



- ❑ What of the following is true:
 - ❑ The dimension is a basic dimension respect to datum A.
 - ❑ The dimension is a gage dimension with a tolerance of 2.5.
 - ❑ This is a basic dimension that specifies a part feature.
 - ❑ All of the above.

Virtual and Boundary Conditions

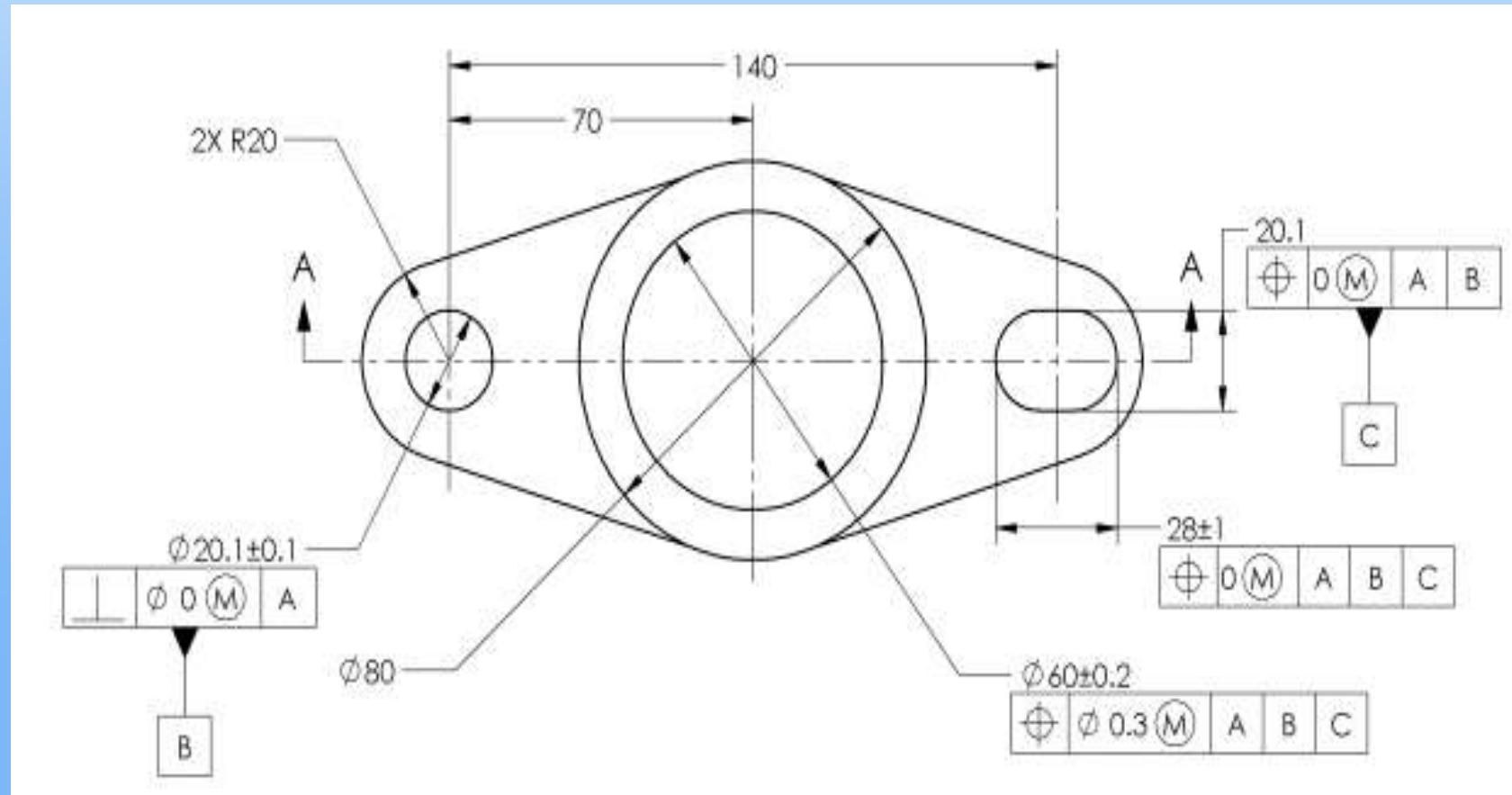
❑ Virtual Condition

- ❑ Worst case boundary generated by the collective effects of a FOS at MMC or LMC and the geometric tolerance for that material condition.
- ❑ Related to the datums that are referenced in the geometric tolerance used to determine VC.
- ❑ What does this mean?

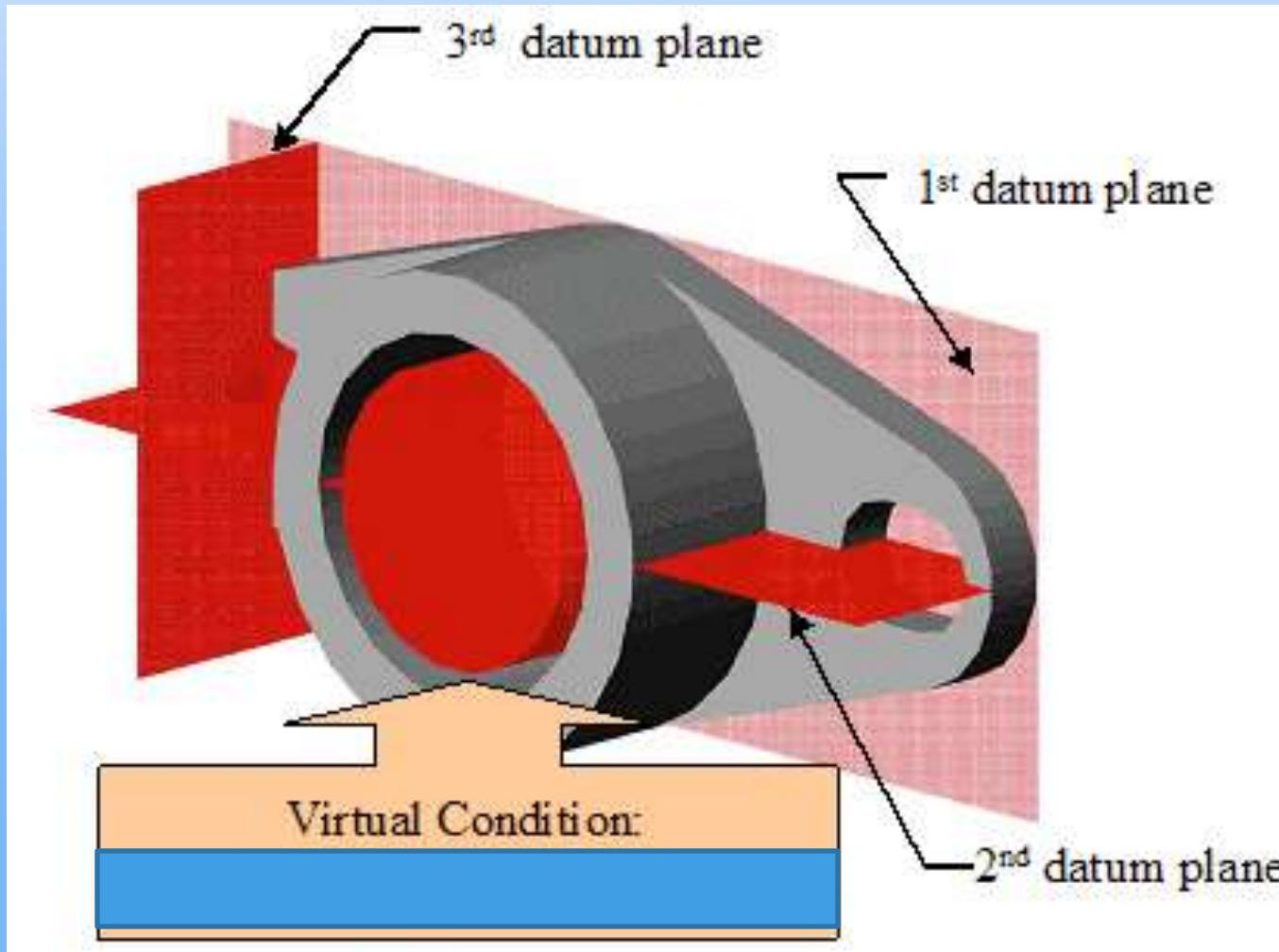
Virtual Condition

- A geometric tolerance modified at MMC or LMC becomes a single limit control. This single limit is known as the virtual condition. Features toleranced in this manner need only be inspected for size and to assure that none of the surface of the feature has violated the virtual condition.

Virtual Condition

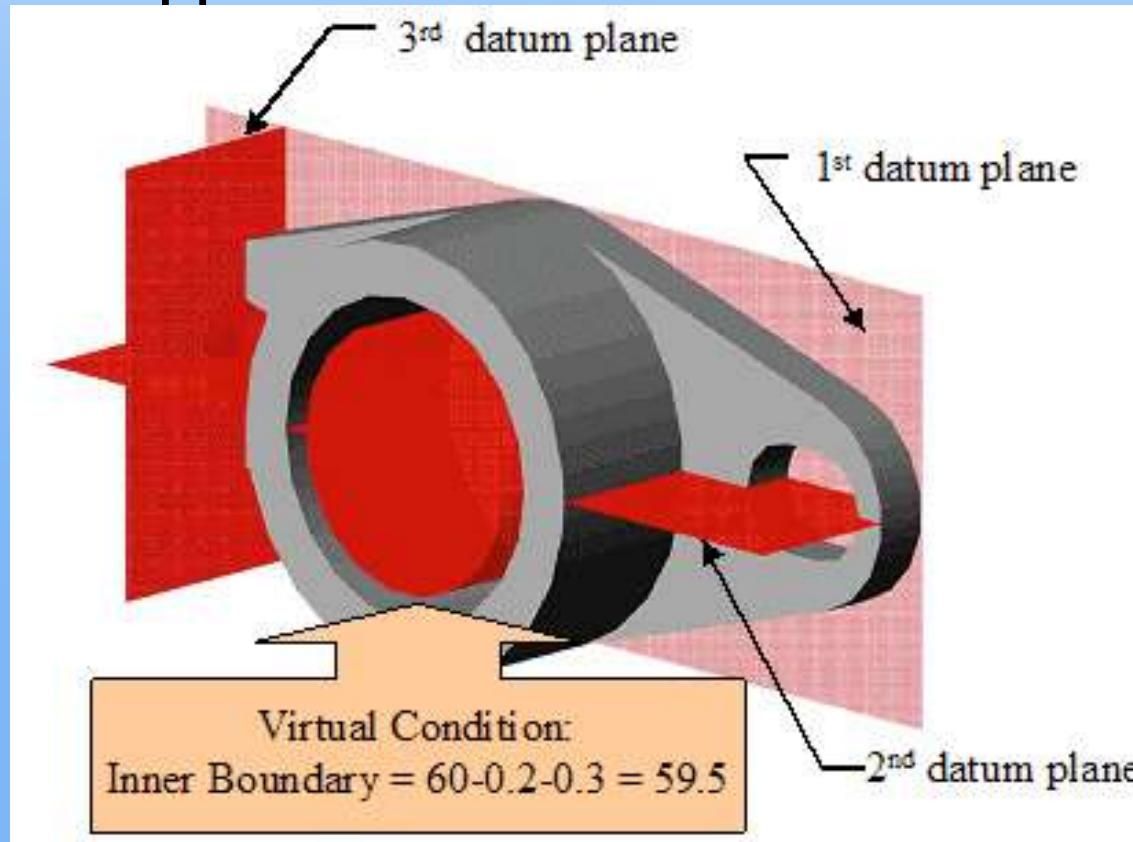


Virtual Condition



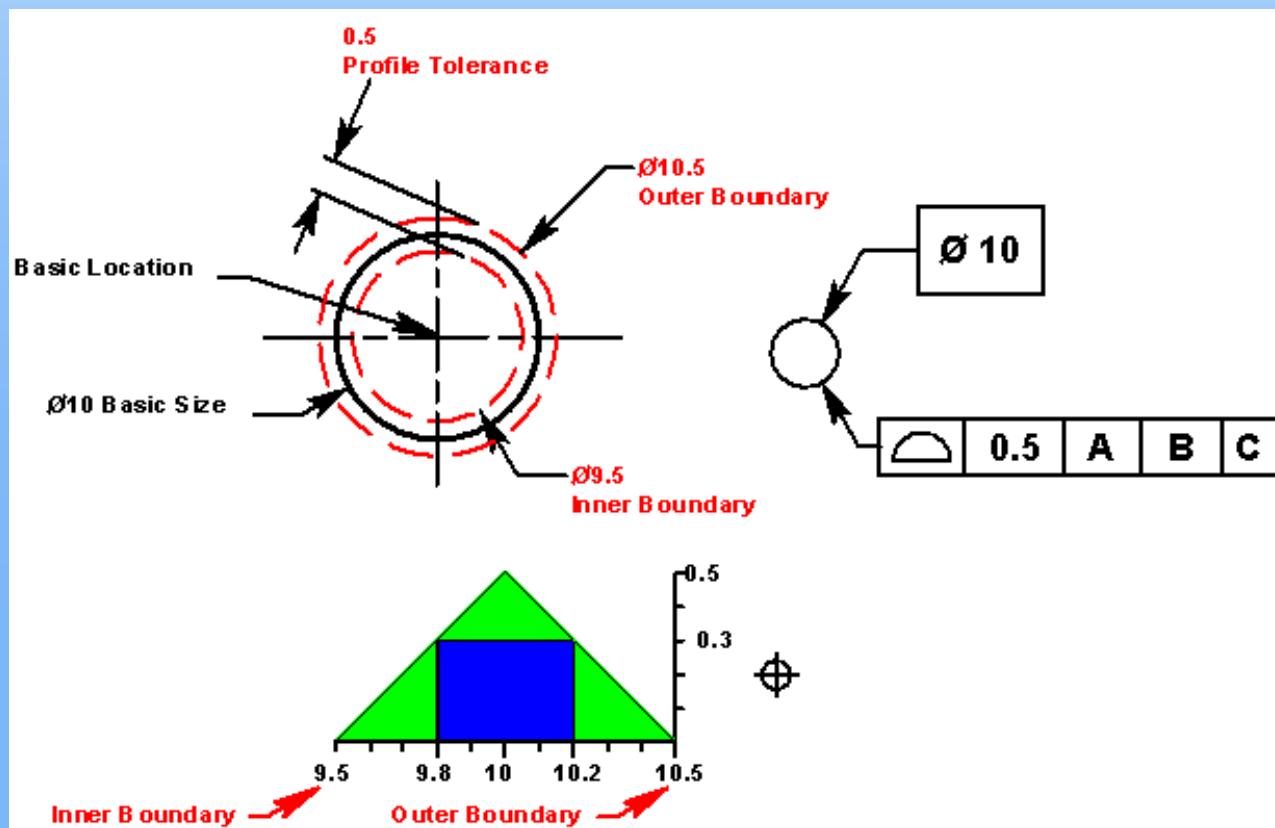
Inner Boundary

- Worst case boundary generated by the smallest FOS minus the stated geometric tolerance and any other tolerance if applicable.



Outer Boundary

- Worst case boundary generated by the largest FOS plus the stated geometric tolerance and any additional tolerance.



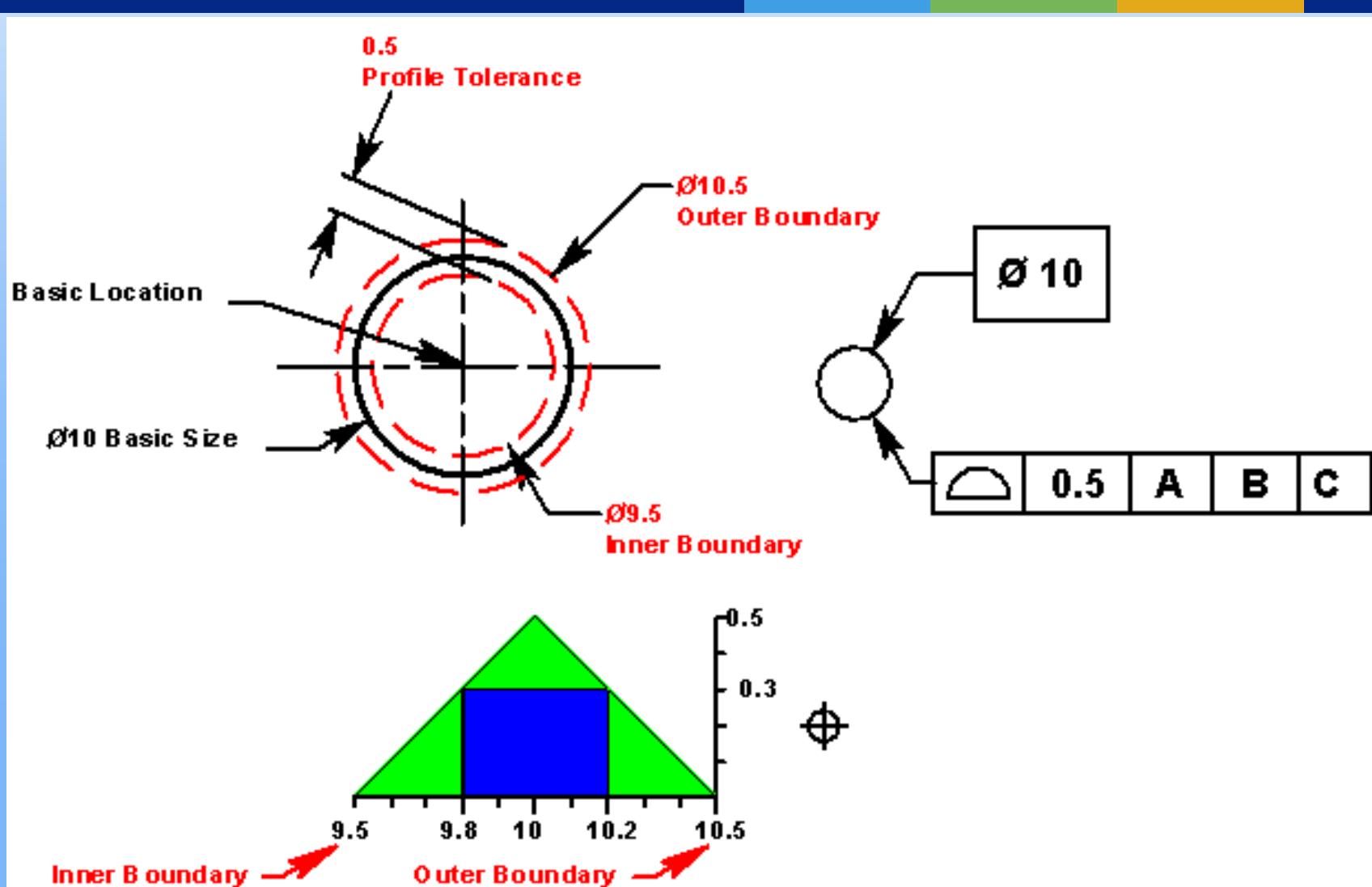
Worst Case Boundary

- ▣ A general term to refer to the extreme boundary of a FOS that is the worst case for assembly.
- ▣ Depending upon the part dimensioning, a WCB may be a VC, IB or an OB.

Feature of Size Boundary Conditions

- ❑ If there are no geometric controls applied to an FOS, the WCB is the inner or outer boundary.
- ❑ The outer or inner boundary is equal to the MMC boundary as defined by rule one. (figure 3-9).

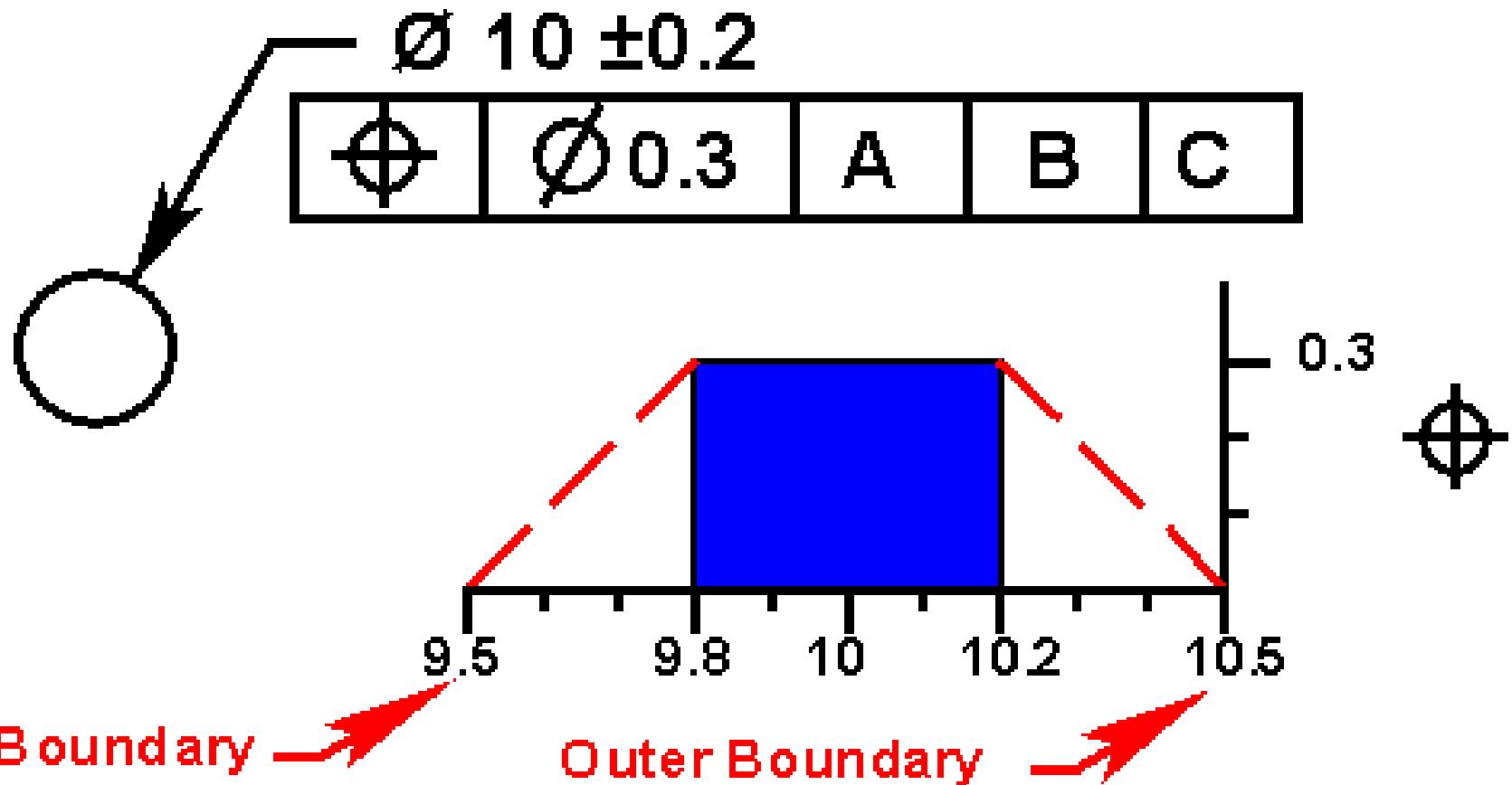
FOS Boundary Condition



FOS Boundary Condition

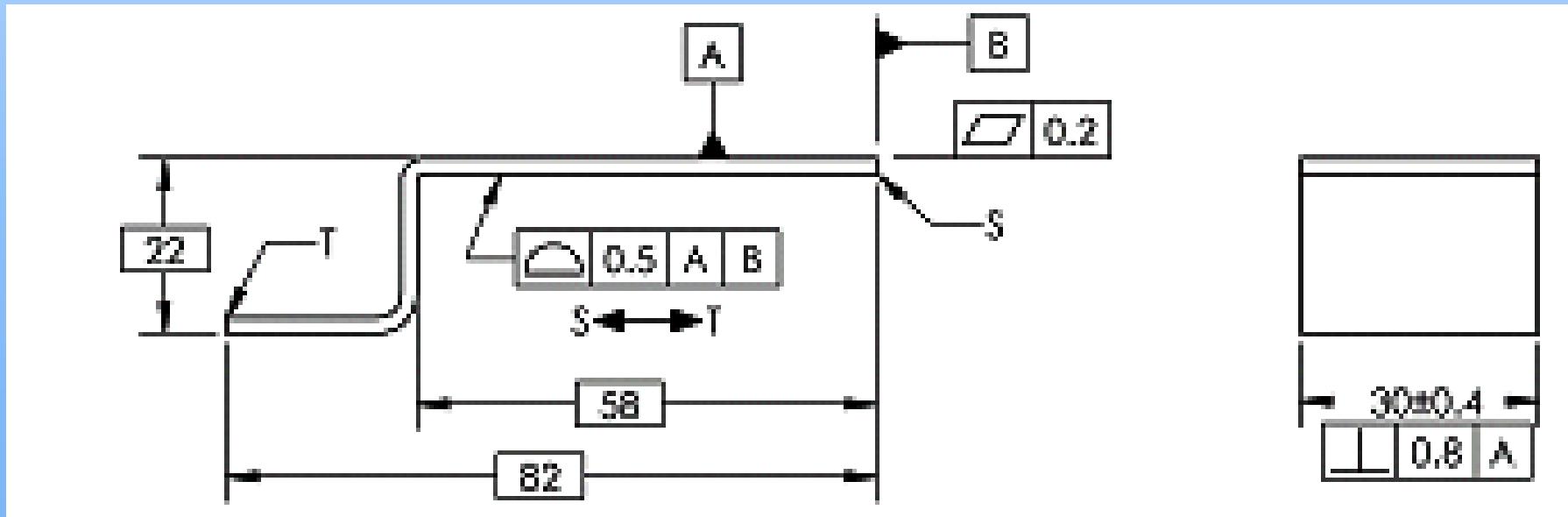
- ❑ If a geometric control is applied to the FOS, that is a feature control frame is associated with a FOS dimension, the WCB is affected.
- ❑ If the feature control frame is associated with a feature (surface) the WCB is not affected.

FOS Boundary Conditions



FOS Boundary Conditions

- Feature control frame applied to a feature or surface.



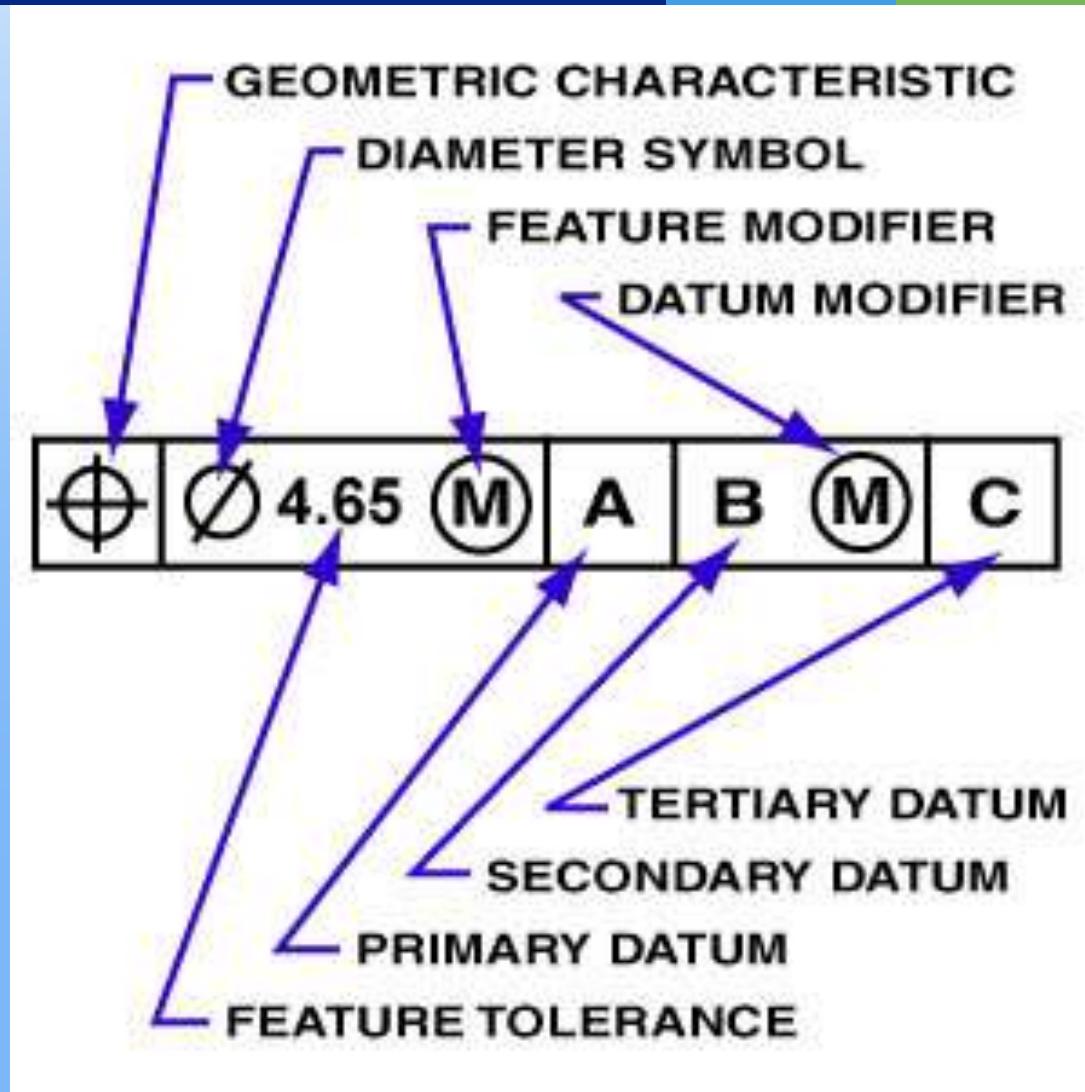
Important

- A feature control frame applied to a feature, surface, does not affect the WCB.
- A feature control frame applied to a FOS does affect the WCB.

MMC Virtual Condition

- ❑ If a geometric tolerance that contains an MMC modifier in the tolerance portion of the feature control frame is applied to a FOS, the virtual condition (WCB) of the FOS is affected.
- ❑ The virtual condition (WCB) is the extreme boundary that represents the worst case for functional requirements, such as clearance or assembly, with a mating part.

Feature Control Frame With Modifier



MMC Virtual Condition

External Shaft
with MMC Modifier

Internal Hole
with MMC Modifier

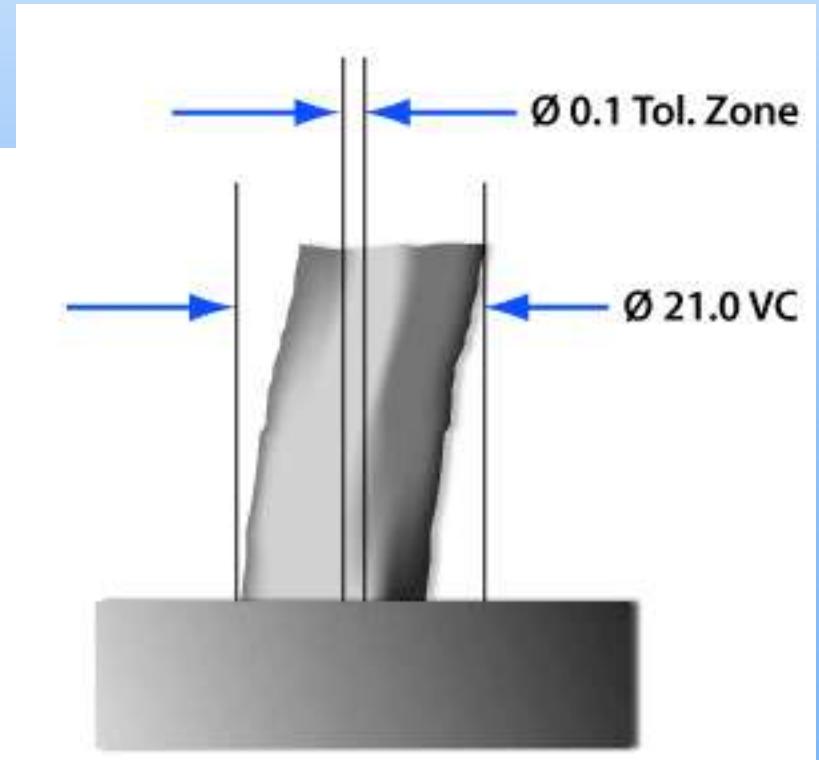
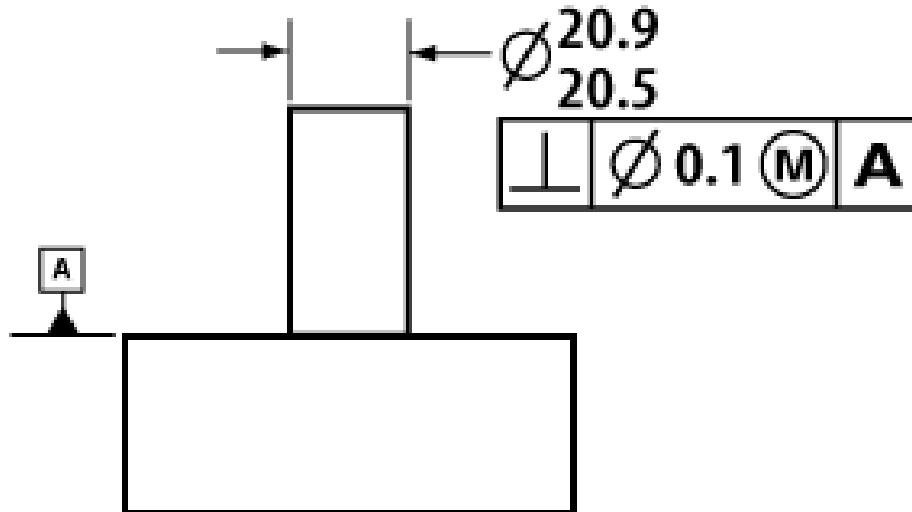
Outer Boundary
VC = MMC + Geometric Tol.

Inner Boundary
VC = MMC – Geometric Tol.

External FOS Virtual Condition

VC = MMC + Geometric Tolerance

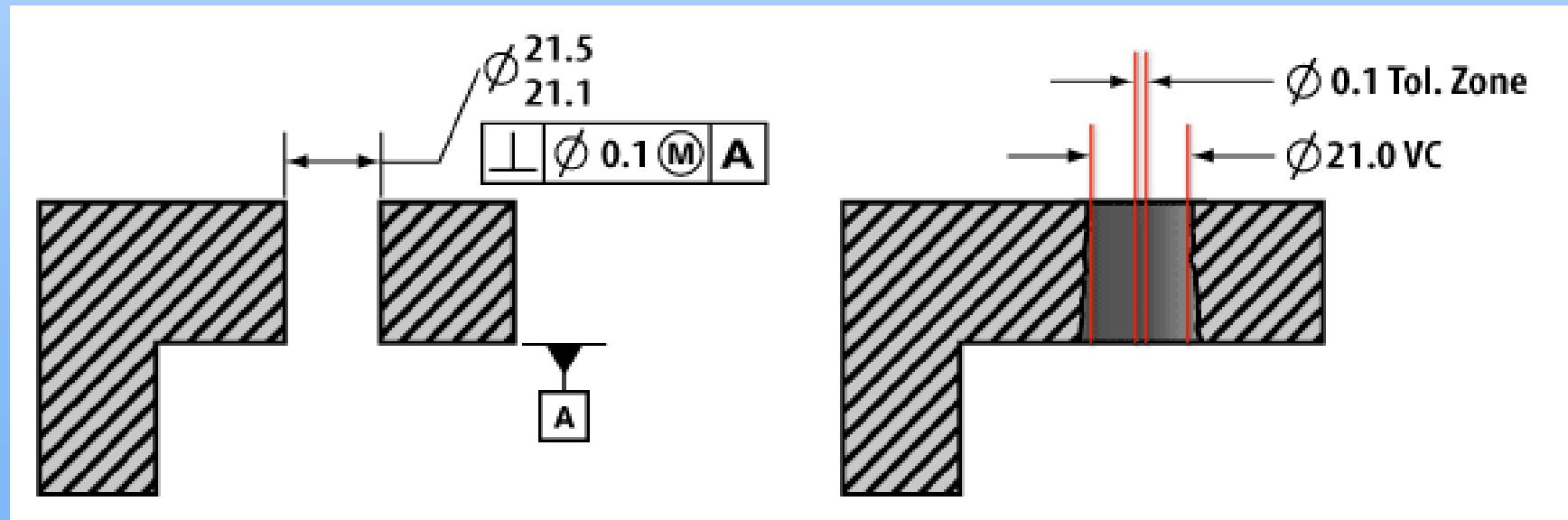
VC = 20.9 + .1



Internal FOS Virtual Condition

VC = MMC – Geometric Tolerance

VC = 21.1 - .1



Virtual Conditions

- ❑ The virtual condition of an external FOS, at MMC, is a constant value and is referred to as the outer boundary or WCB in assembly calculations.
- ❑ The virtual condition of an internal FOS, at MMC, is a constant value and is referred to as the inner boundary or worst case boundary in assembly calculations.

LMC Virtual Condition

- When a geometric tolerance that contains an LMC modifier in the tolerance portion of the feature control frame is applied to a FOS, the VC of the FOS is affected.
- The VC is the extreme boundary that represents the worst case for functional requirements such as wall thickness, alignment or minimum machine stock on a part.

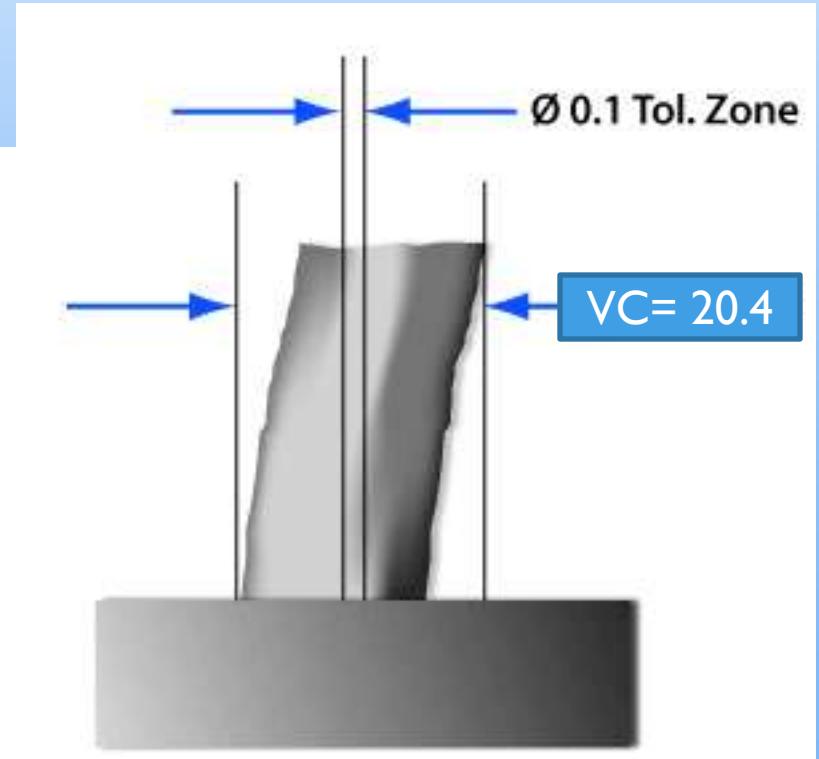
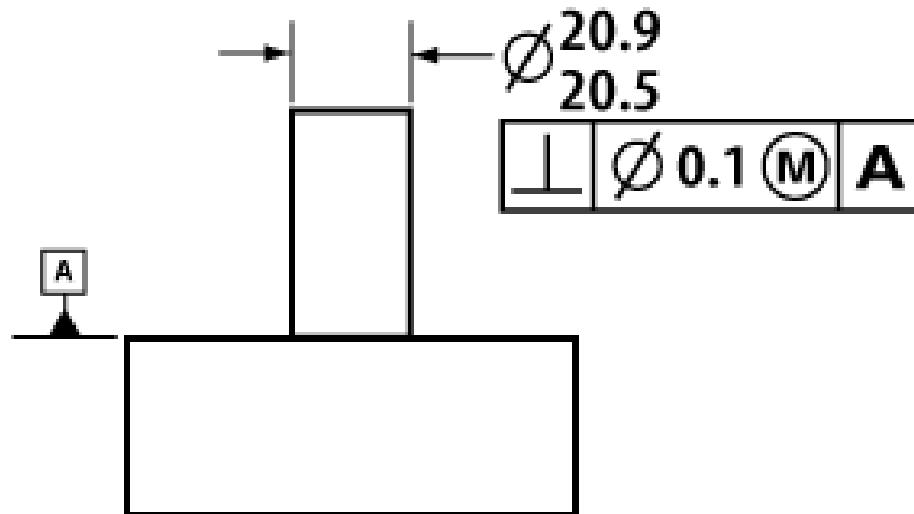
LMC Virtual Condition

- In the case of an external FOS such as a pin or shaft, the VC is found as follows:
 - $VC = LMC - \text{Geometric Tolerance}$
- In the case of an internal FOS such as a hole, the VC is found as follows:
 - $VC = LMC + \text{Geometric Tolerance}$

External FOS Virtual Condition

VC = LMC - Geometric Tolerance

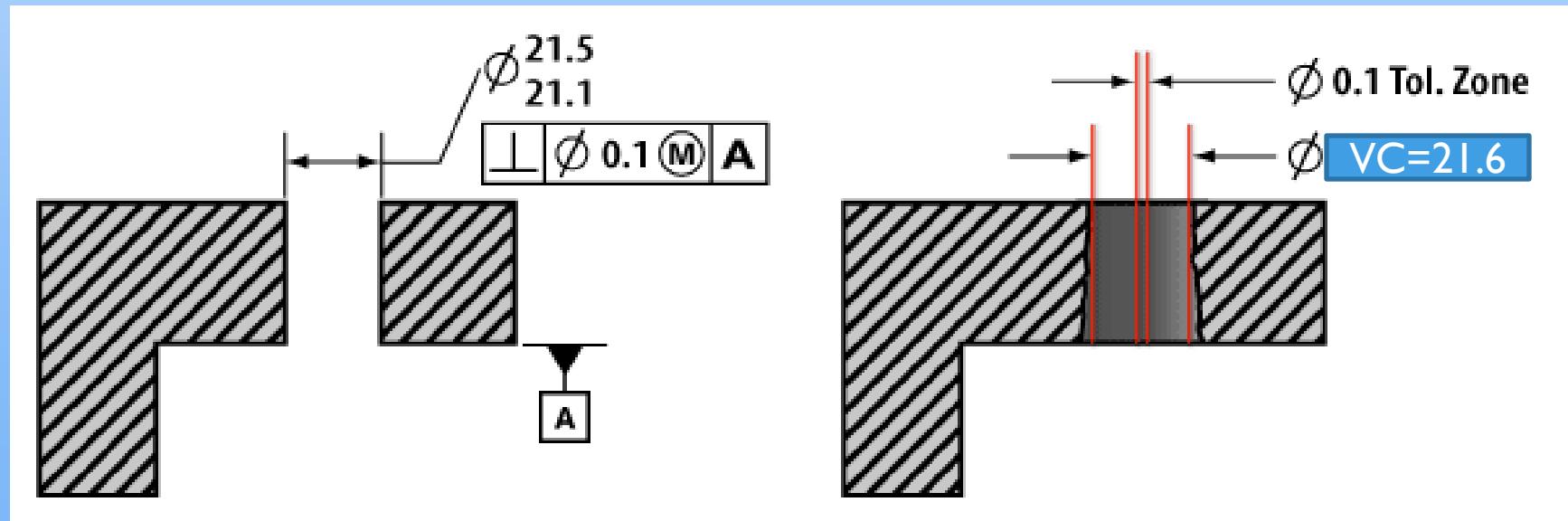
VC = 20.5 - .1



Internal FOS Virtual Condition

VC = LMC + Geometric Tolerance

VC = 21.5 + .1



RFS Inner and Outer Boundary

- When a geometric tolerance which contains no modifiers (default RFS) in the tolerance portion of the FCF is applied to a FOS, the inner or outer boundary (WCB) of that FOS is affected.
- Boundaries are determined as follows:
 - $OB = MMC + \text{Geometric Tolerance}$
 - $IB = MMC - \text{Geometric Tolerance}$.
- Is there a difference between RFS and MMCVC?
- The VC or WCB formulas are important to remember.

Multiple Virtual Conditions

- ▣ Complex drawings may have multiple geometric controls applied to a FOS.
 - ▣ Page 63 of text is our example.
- ▣ Review Rule One and positional tolerance.

Bonus Tolerance

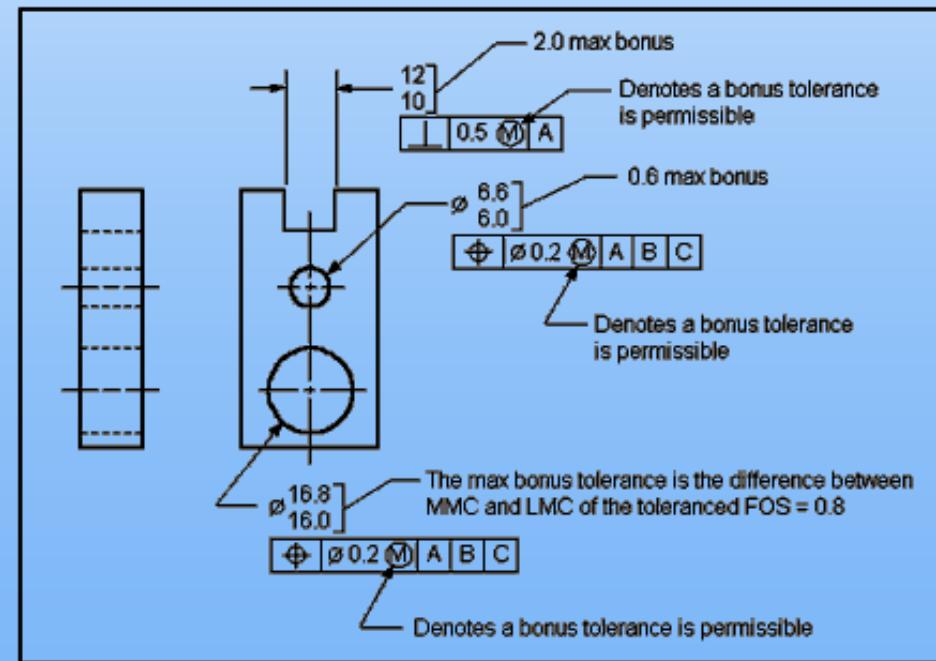
- ❑ A bonus tolerance is in addition to any geometric tolerance allowed when the geometric tolerance is applied to a FOS with an MMC or LMC modifier in the tolerance portion of the FCF.
- ❑ There are strict rules for when a bonus tolerance is allowed.

Bonus Tolerance

- Bonus tolerance is an additional tolerance for a geometric control. Whenever a geometric tolerance is applied to a feature of size, and it contains an MMC (or LMC) modifier in the tolerance portion of the feature control frame, a bonus tolerance is permissible.

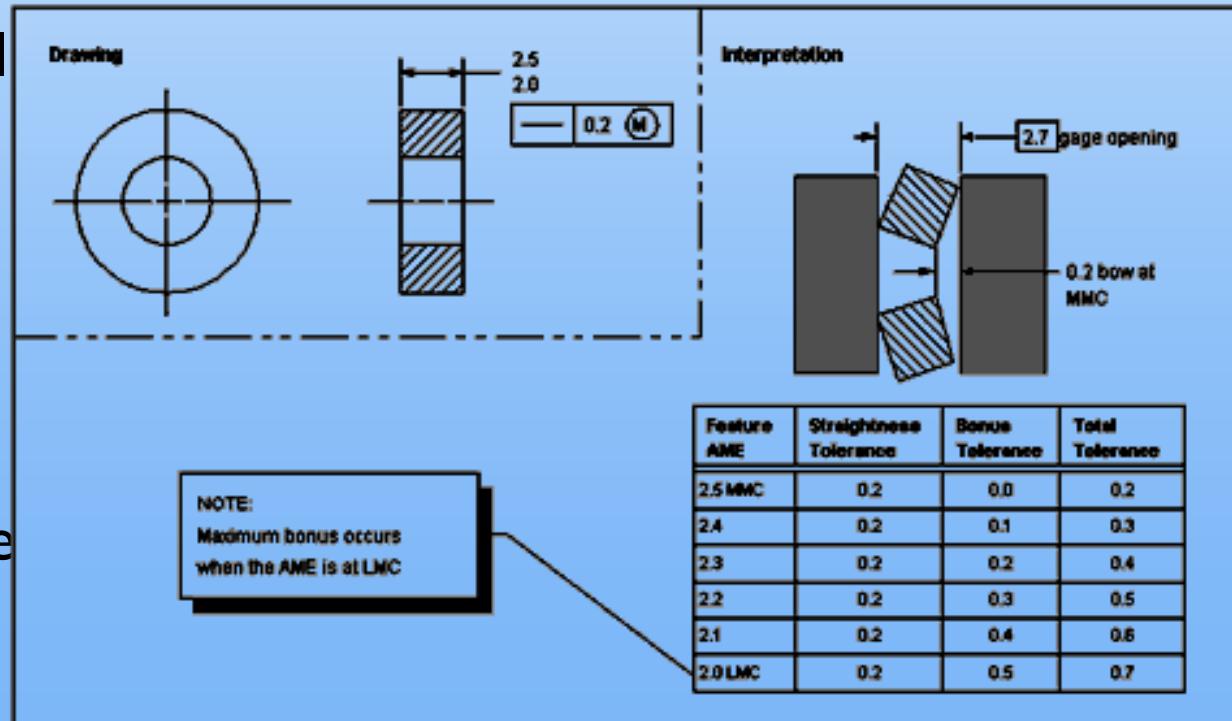
Bonus Tolerance

- This figure shows how to determine the amount of bonus tolerance permissible in an application. The MMC modifier in the tolerance portion of the feature control frame denotes that a bonus tolerance is permissible. The maximum amount of bonus tolerance permissible is equal to the difference between the MMC and the LMC of the toleranced feature of size



Bonus Tolerance

- In this figure, the functional gage is designed to the virtual condition (2.7) of the feature of size. Since the gage opening is constant, the thinner the washer becomes, the more straightness tolerance it could have and still pass through the gage.



Calculating A Bonus Tolerance

- The Bonus tolerance is the difference between MMC and LMC.
 - Bonus tolerance is only allowed if an MMC or LMC modifier is present.
 - The Bonus tolerance is that variation that the AME has from MMC or LMC.

Bonus Tolerance

□ Caution

- For an external FOS, there is no bonus tolerance if the AME is larger than the MMC.
- For an internal FOS, there is no bonus tolerance if the AME is smaller than the MMC.

Questions?



Chapter Four

Form Controls

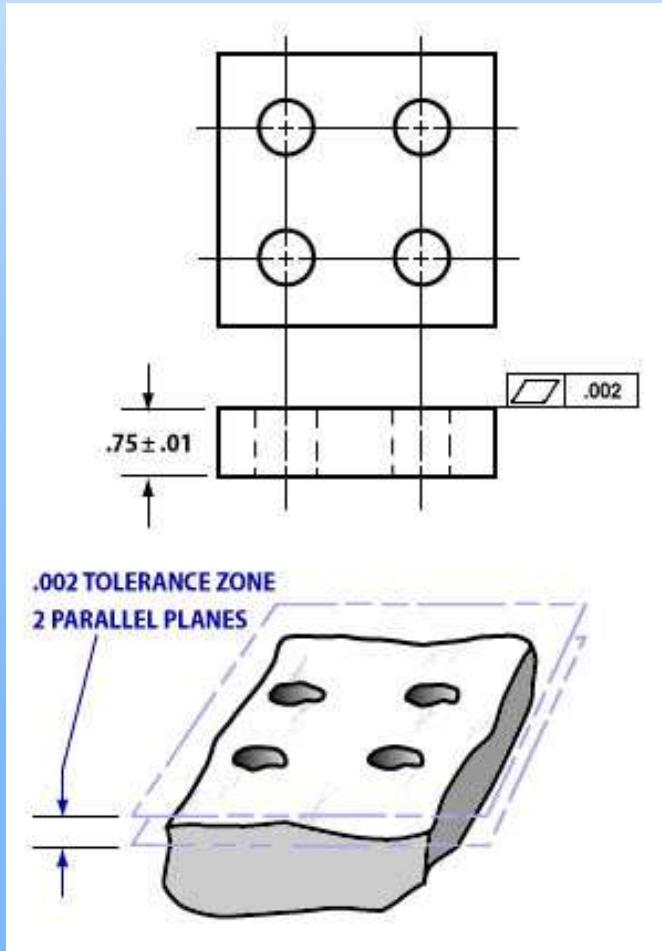
Chapter Goals

- ❑ Interpret the flatness control
- ❑ Interpret the straightness control
- ❑ Interpret the circularity control
- ❑ Interpret the cylindricity control

Flatness Control

- ❑ Flatness is the condition of a surface having all of its elements in one plane.
- ❑ A flatness control is a geometric tolerance that limits the amount of flatness error a surface is allowed.
- ❑ Flatness is measured by comparing the surface with its own true or perfect counterpart.

Flatness Example

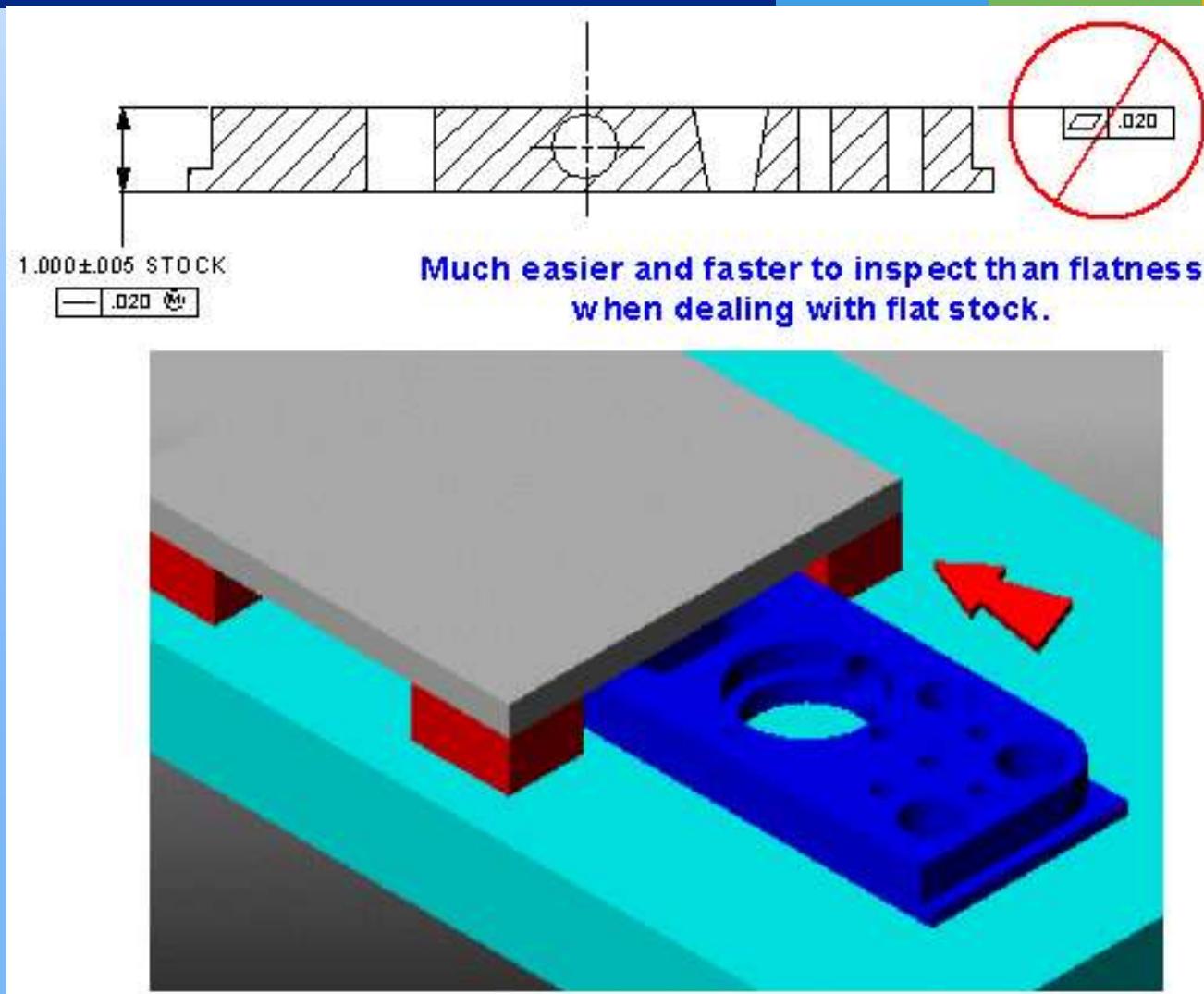


- Notice the tolerance zone. The surface of the feature of size needs to stay within the tolerance zone.

Flatness Control

- ❑ Is never applied to a non-planar surface.
 - ❑ MMC or LMC cannot be applied.
 - ❑ Remember that these modifiers are to be used with a geometric control applied to a FOS.
- ❑ Flatness is an independent requirement and verified separately from the size tolerance and rule one requirements.

Flatness Tolerance Zone



Flatness and Rule One

- ❑ If rule one applies to a FOS that consist of two parallel planes, an automatic indirect flatness control is applied. Why?
- ❑ For at MMC, the form must be perfect and that means flat.
- ❑ If you want the FOS to have a geometric tolerance and the MMC modifier, then you do not need a flatness control for the following reasons unless you want the flatness of the surface inspected.

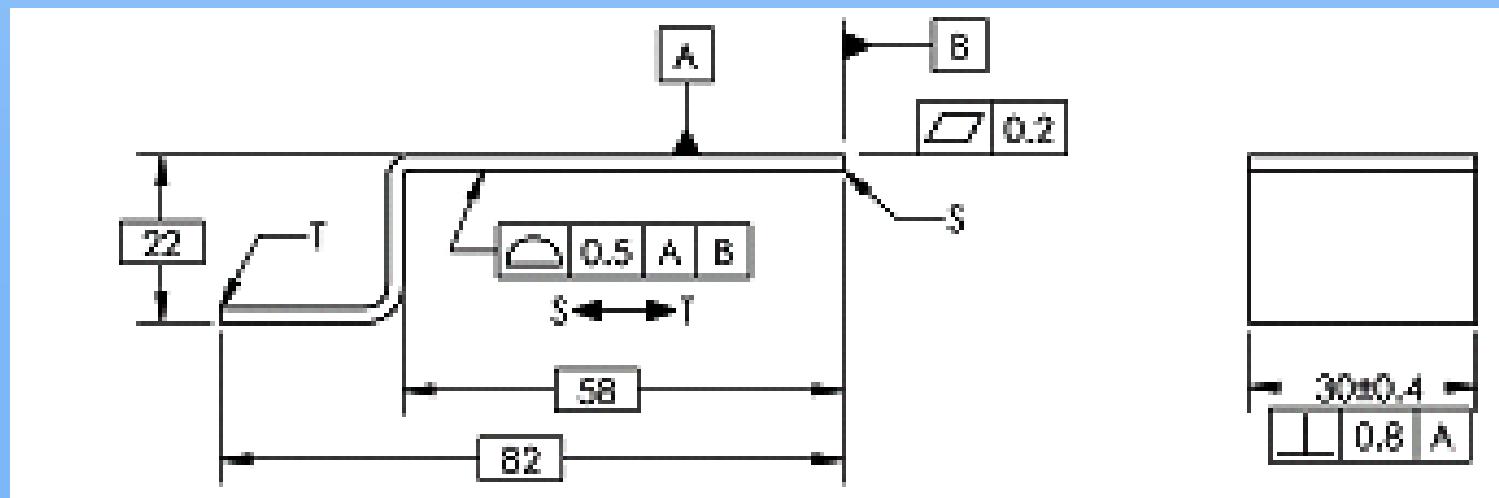
Flatness and Rule One

- At MMC both surfaces must be flat.

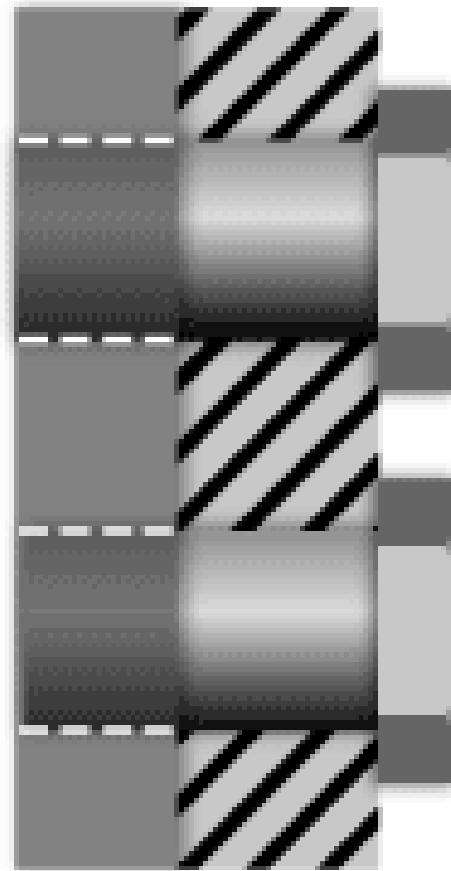
At LMC there is a tolerance zone that the surfaces may exist in that meets the flatness requirements.

Flatness Tolerance Zone

- Flatness is another geometric tolerance that is challenging to inspect. It requires isolating the feature from the rest of the part since there can never be a datum referenced with flatness. In this example, the flatness has been applied to datum feature A.



Flatness Control

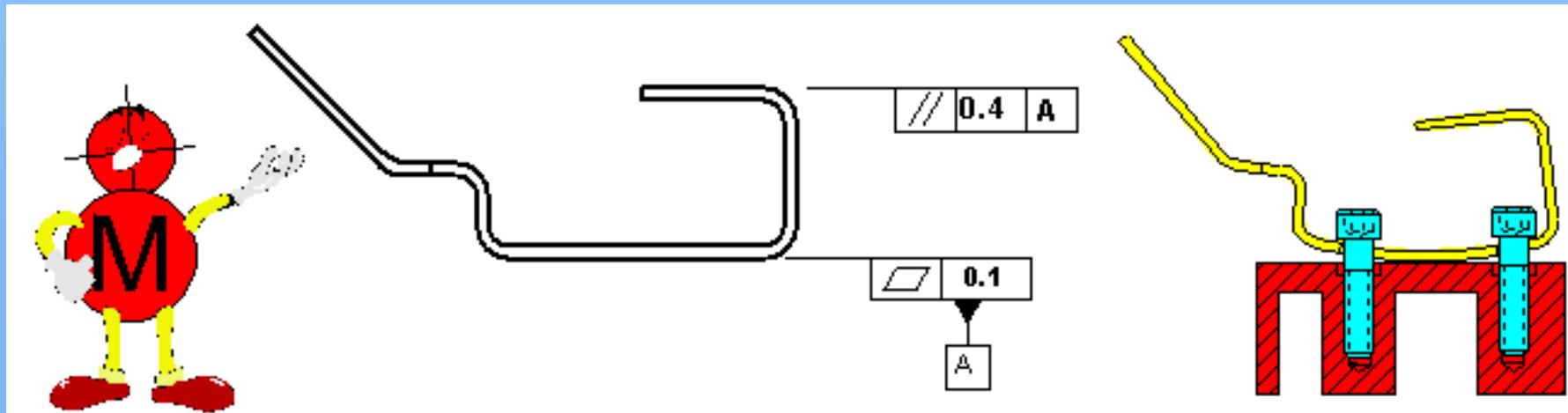


Additional Flatness Indirect Controls.

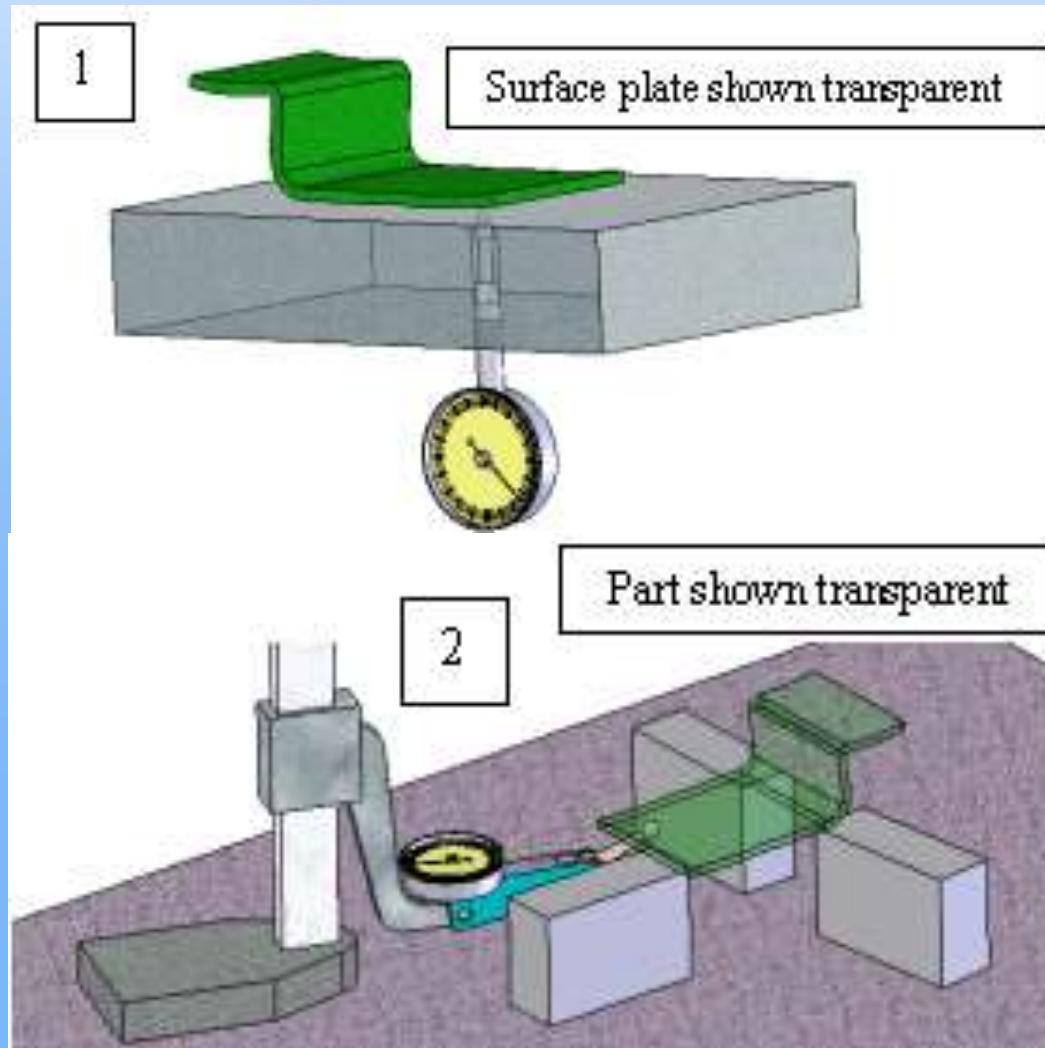
- ❑ Perpendicularity, parallelism, angularity, total runout and profile of a surface all indirectly control flatness of a surface.
- ❑ But flatness is not inspected on these controls. You must have a flatness control in order to inspect flatness.

Inspecting Flatness

- Flatness is another geometric tolerance that is challenging to inspect. It requires isolating the feature from the rest of the part since there can never be a datum referenced with flatness. In this example, the flatness has been applied to datum feature A.



How Do We Inspect Flatness?



Flatness Inspection

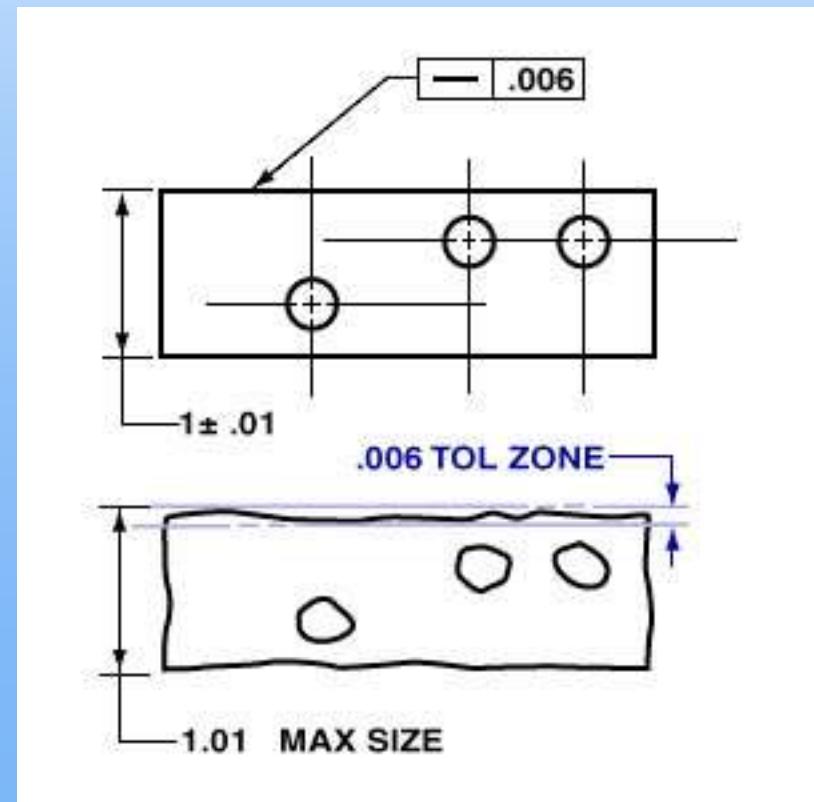
- Flatness is not easy to inspect. Time and patience is needed to do this task right. Rushing flatness inspection will lead to errors.

Straightness

- ❑ Straightness describes a condition where a line element of a surface, axis or center plane is a straight line.
- ❑ The straightness symbol is —
- ❑ A straightness control directed to a surface is a geometric tolerance that limits the amount of straightness error in each surface line element.

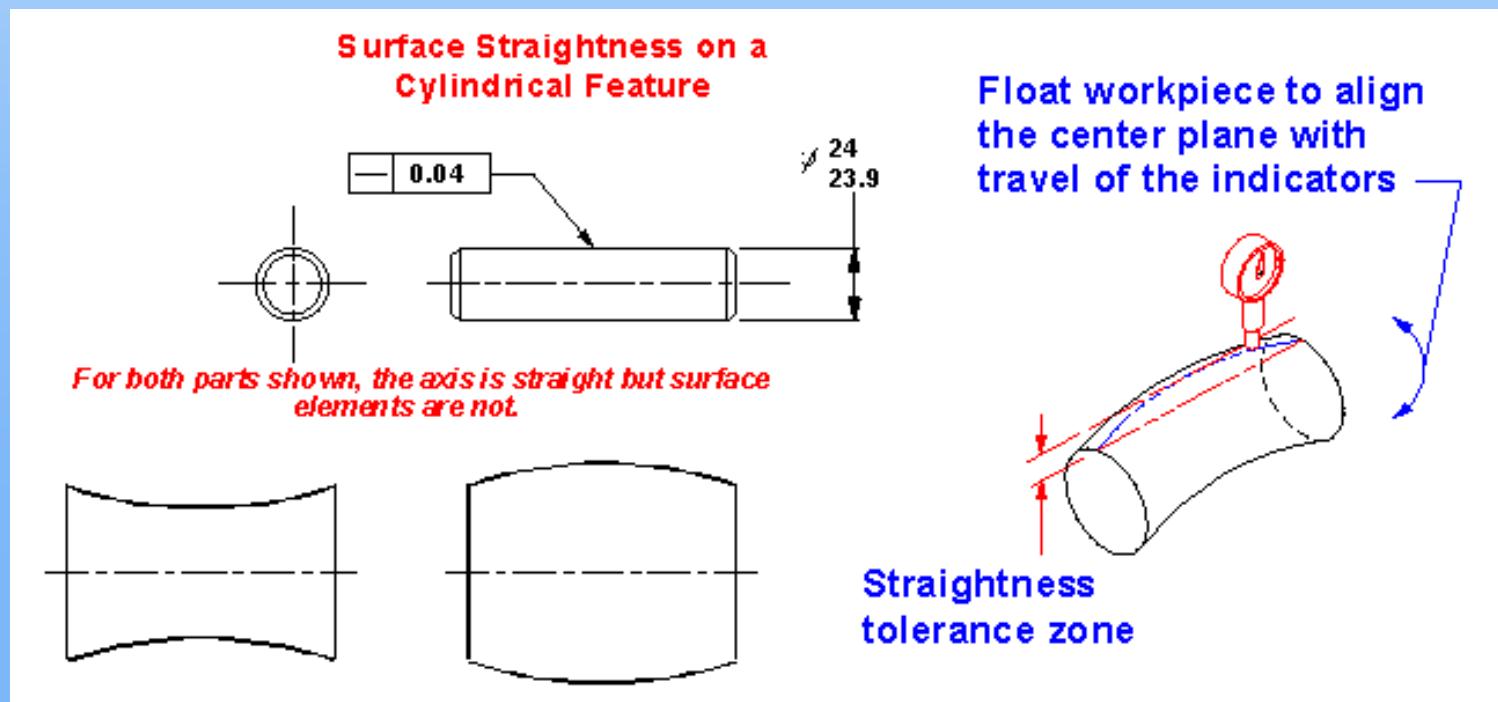
Straightness

- The tolerance zone for a straightness control, as a surface control, is two dimensional and consist of two parallel lines, for each line element with a separation equal to the straightness tolerance value.



Straightness

- When straightness is applied to surface elements, the MMC or LMC modifiers are not used.



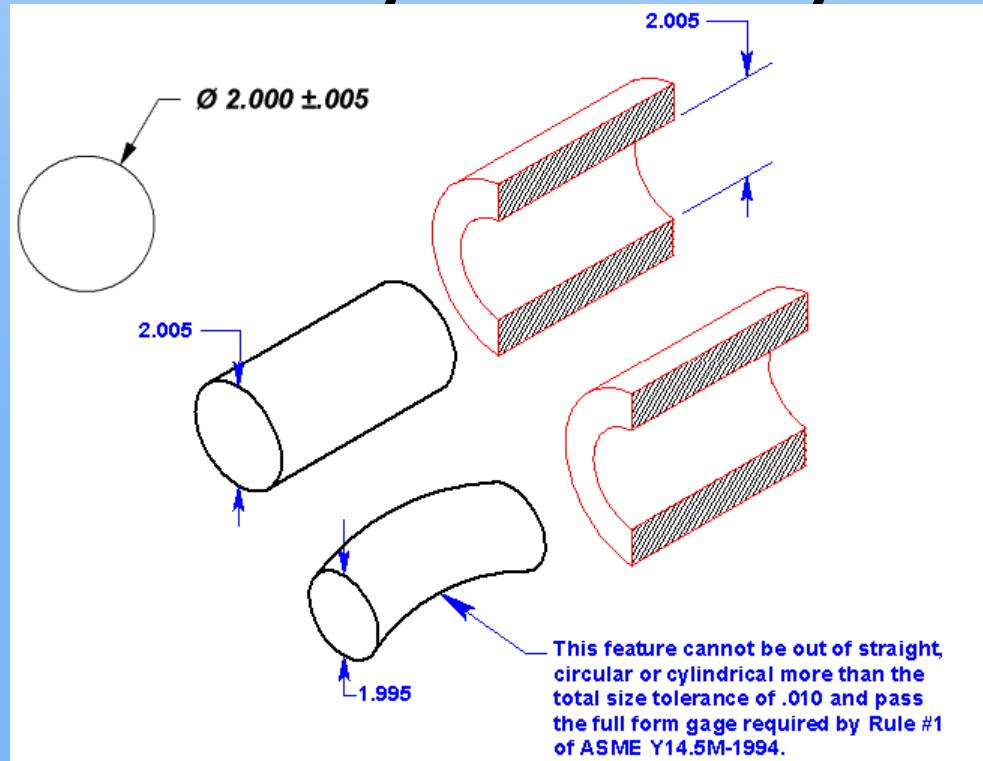
Straightness

□ Rule One and Straightness

- If rule one is in effect, that is the feature size is at MMC, then the line elements must be perfectly straight.
- A straightness error equal to the deviation from MMC will be allowed. To prevent size errors, no straightness control should be applied unless the straightness tolerance is less than the total deviation allowed.

What does this mean?

- If a feature has a FOS dimension with an MMC modifier, then at MMC the surface feature must be perfectly flat. If size is inspected correctly, the geometric form (straightness, flatness, circularity and cylindricity) of the feature must be within the total size tolerance. Therefore, **form controls are usually not necessary**.



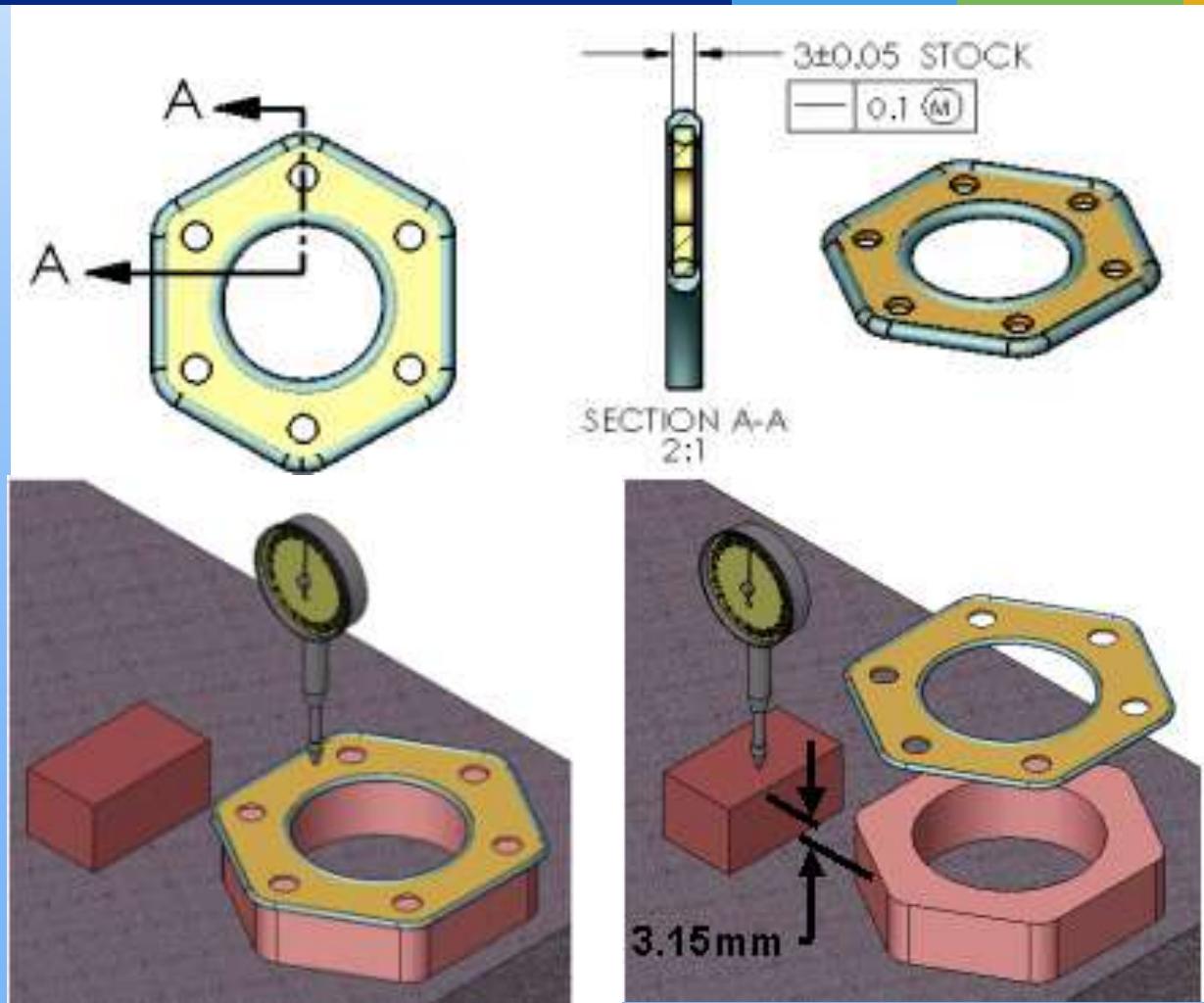
Legal Specification of Straightness Control

- Review flowchart on page 81.

Inspecting Straightness

- A size tolerance on a flat part, when checked correctly, will automatically control the flatness and straightness of the surfaces as well as the size. This is true unless the feature is stock, left in the "as furnished" condition. The gasket below is made of a metal plate with rubber seals molded in place. Since the plate is made of stock left in the "as furnished" condition, Rule #1 does not apply. The company wanted to assure that the plate was somewhat flat. A flatness control would work but may be time consuming to check. A better approach was to apply a straightness tolerance at MMC to the plate. In this case, the plate must fit through an envelope not greater than $3+0.05+0.1=3.15\text{mm}$. By setting one side of the plate on a flat surface and mastering an indicator 3.15mm above the plate, the straightness may be verified. This will restrict the amount the flatness may be out but is often easier to inspect.

Inspecting Straightness



Straightness as an Axis or Centerplane Control

- ❑ Important Note: A straightness control is the only form control that can be applied to either a surface or a feature of size.
- ❑ Interpretation of a straightness control applied to a FOS is different than when applied to a surface.

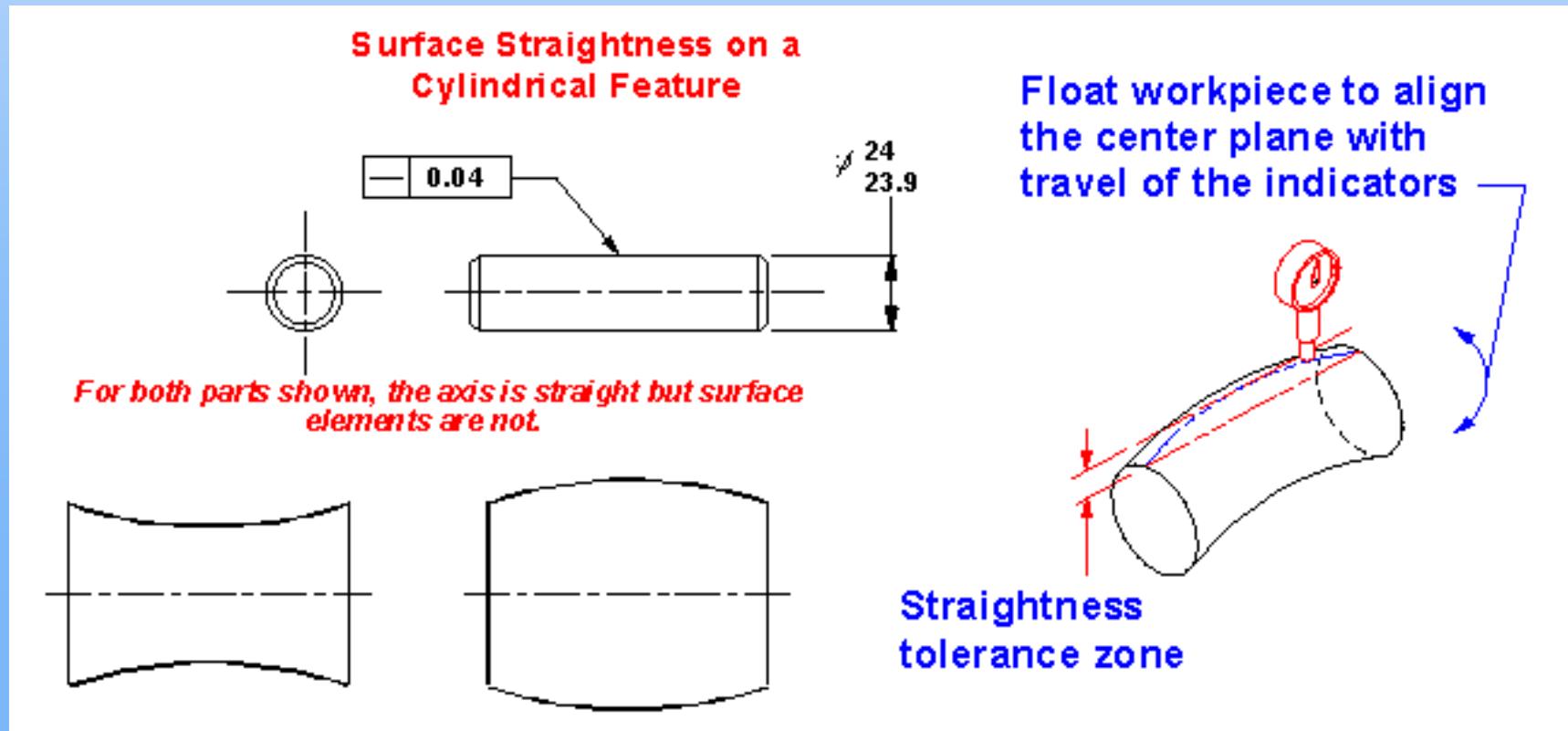
Straightness as an Axis or Centerplane Control

□ Why is this so?

- The tolerance zone applies to the axis or centerplane of the FOS.
- Rule one is overridden.
- The VC or inner/outer boundary of the FOS is affected.
- MMC or LMC may be used.
- The tolerance value specified may be greater than the size tolerance.

Straightness as an Axis or Centerplane Control

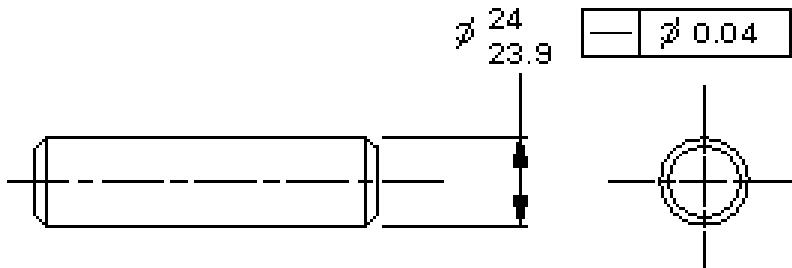
■ Straightness applied to a surface.



Straightness as an Axis or Centerplane Control

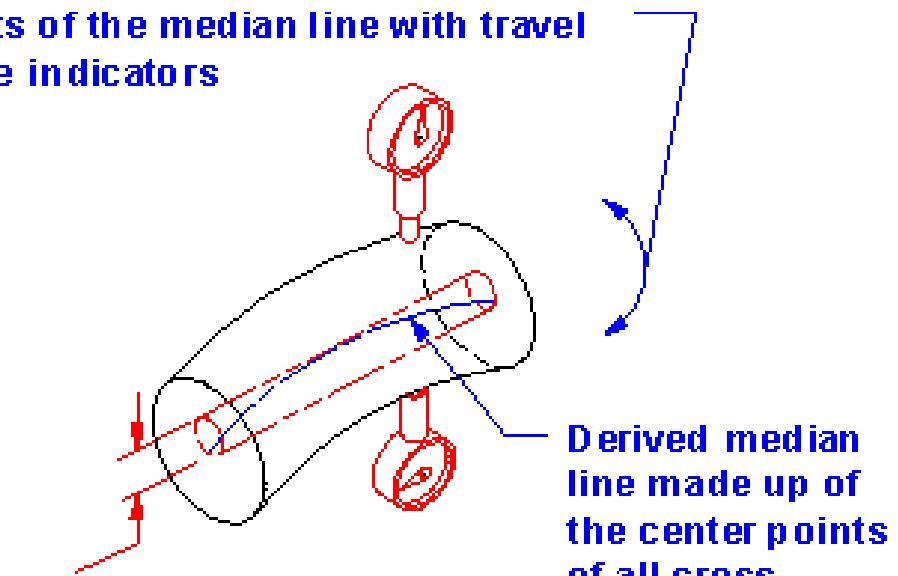
■ Straightness applied to a FOS.

Straightness of an Axis



Float workpiece to align the end points of the median line with travel of the indicators

Straightness tolerance zone



Straightness as an Axis or Centerplane Control

- Important Note: A straightness control applied to a FOS is a geometric tolerance that limits the amount of straightness error allowed in the axis or centerplane.

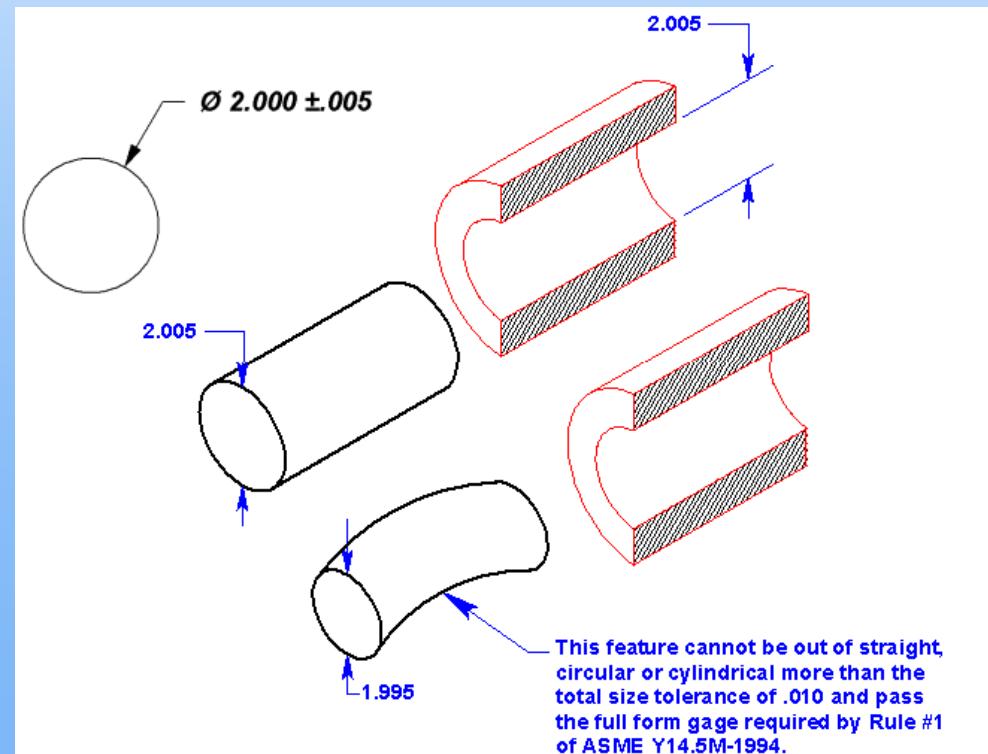
Rule One and Straightness

- ❑ Again, if rule one applies to the FOS then an automatic straightness control exist for the axis or centerplane of that FOS.
- ❑ Why?

- ❑

Straightness at MMC

- Applying a straightness control at MMC to a FOS helps ensure assembly of parts. If FOS departs from MMC, a bonus or extra tolerance is granted which is shown at right.



Indirect Straightness Controls

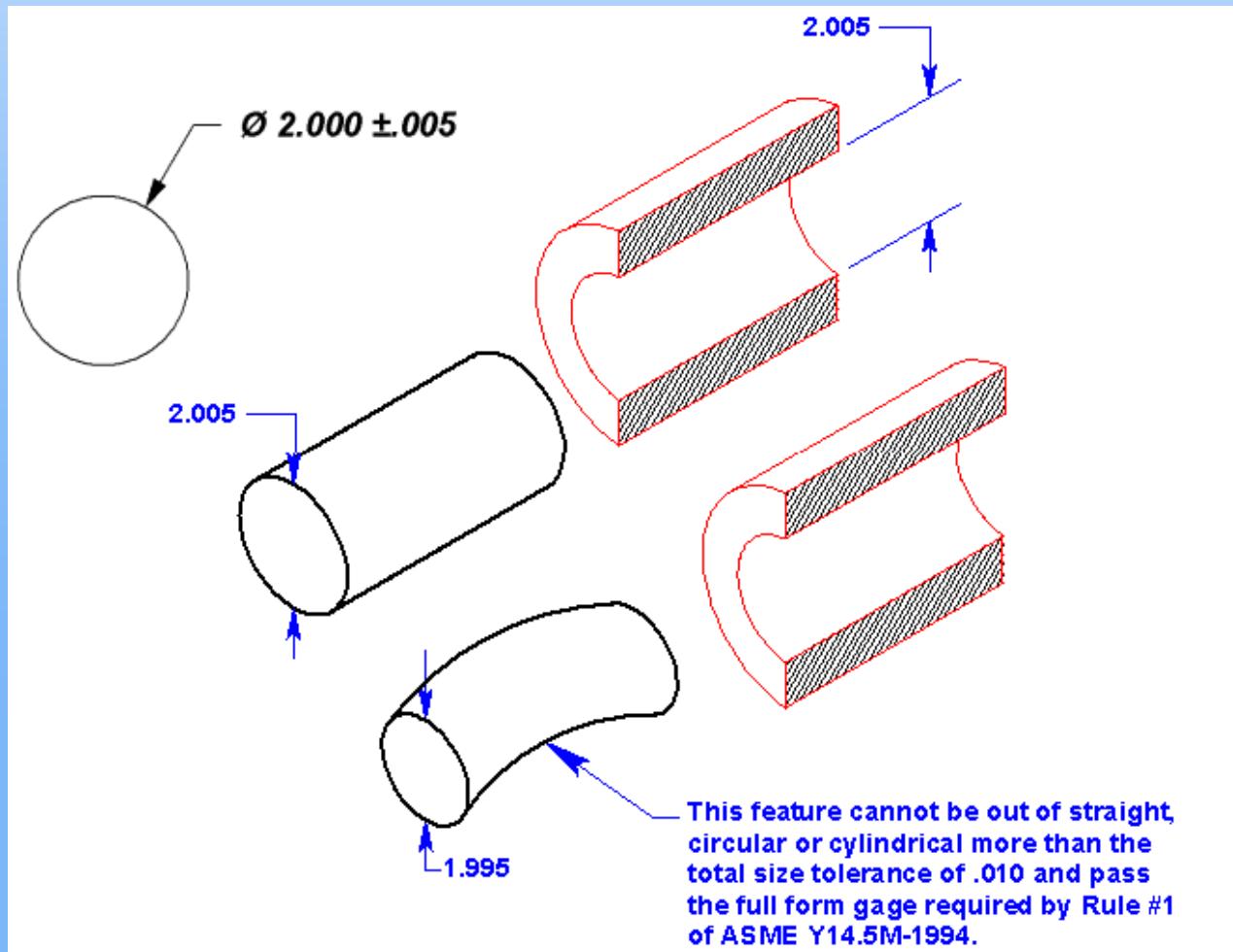
- Other indirect straightness controls are: profile of a surface, total runout and cylindricity.
- Important note: If size is inspected correctly, the geometric form (straightness, flatness, circularity and cylindricity) of the feature must be within the total size tolerance. Therefore, **form controls are usually not necessary.**

Legal Specification Test for Straightness Applied to a FOS

- Review flowchart on page 87.

Inspecting a Straightness Control

Multiple checks may be needed when inspecting a straightness control. Size of FOS and straightness are two. See also page 88.



Circularity

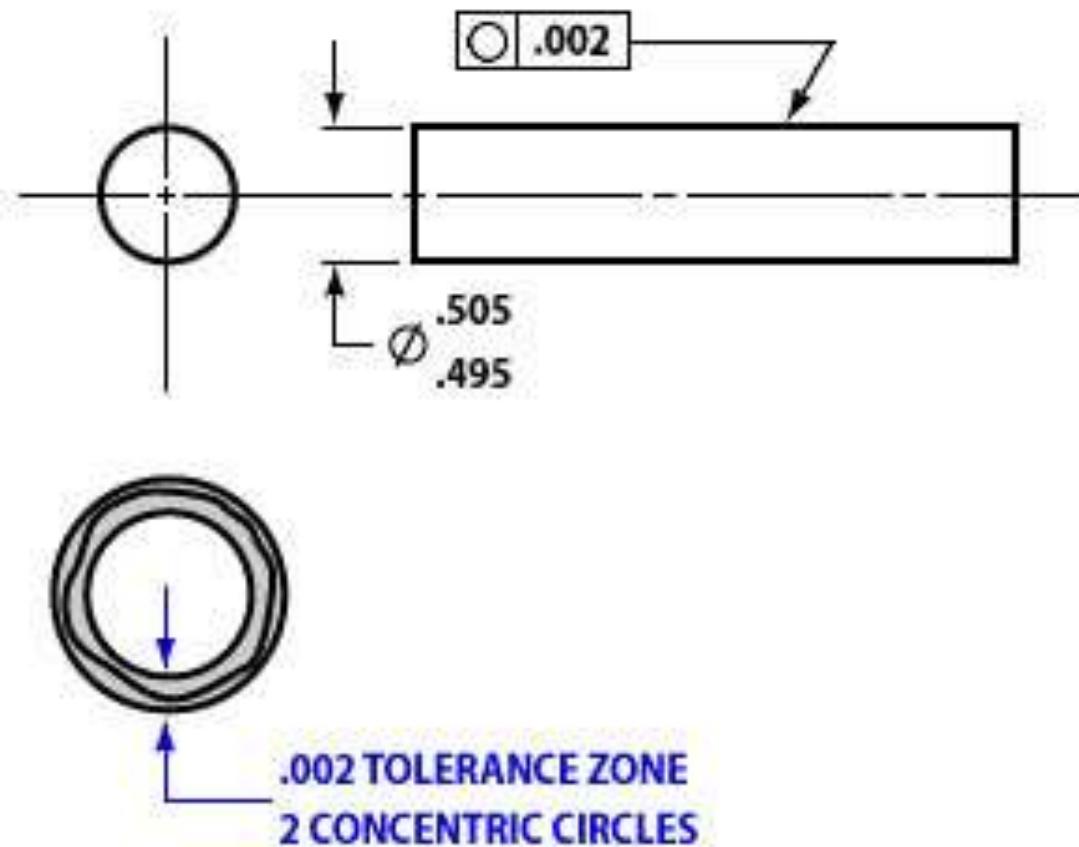
- ▣ Circularity is a condition where all points of revolution, at any section perpendicular to a common axis, are equidistant from that axis.
- ▣ Circularity may be applied to any feature with a round cross section.

Circularity Control

□ Circularity Control:

- A geometric tolerance that limits the amount of circularity on a part surface.
- Specifies that each circular element must lie between the tolerance zone of two coaxial circles.
- May only be applied to a surface. Therefore MMC, LMC, diameter, projected tolerance zone or tangent plane modifiers are not used.

Circularity



Rule One and Circularity

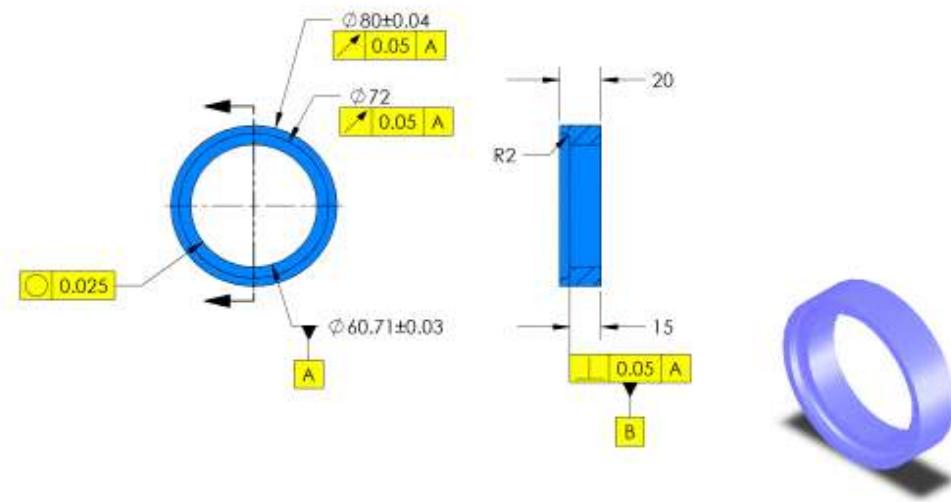
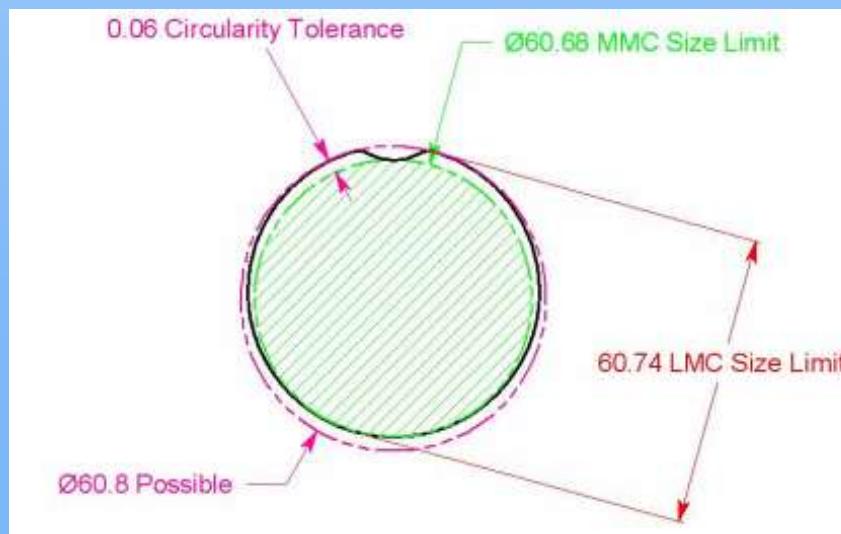
- ❑ Again, rule one states that at MMC, the FOS must be perfect in form and size. Thus at MMC, if MMC is specified, the FOS must fit within a perfect circle circumscribed about all elements. If the FOS deviates from MMC towards LMC, a bonus tolerance in form is allowed.
- ❑ This was seen in the slide before.

Circularity Applications

- ❑ Controls roundness along the length of a feature.
 - ❑ Shaft
 - ❑ Pin
 - ❑ Ring
 - ❑ Hole

Circularity Applications

The circularity error of this particular feature is 0.06, the total size tolerance, since the circularity tolerance zone is comprised of two concentric circles. The size of the circles defining the circularity error could be 60.68 and 60.8 as illustrated. If this isn't desirable, a circularity tolerance with a value less than 0.6 Could be added to refine the circularity control provided by the limits of size. In this case, a 0.025 circularity tolerance was added.



Indirect Circularity Controls

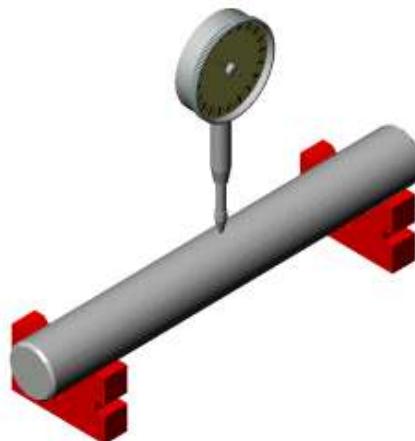
- Remember that other geometric controls will affect circularity of a diameter. These are rule one, cylindricity, profile and runout. If any of these controls are used, circularity error will be controlled.
- Note that indirect circularity controls are not inspected. If you want to inspect circularity, then apply a circularity control.
- If a circularity control is applied, that tolerance should be less than the tolerance of any indirect control that affects the diameter.

Legal Specification Test for a Circularity Control

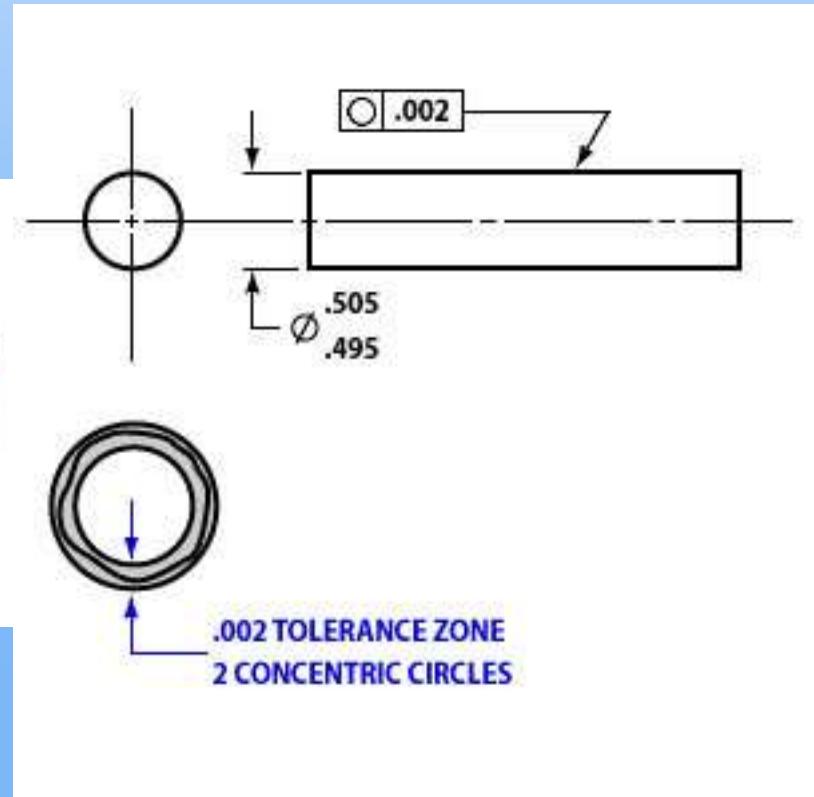
- Review flowchart on page 92.

Inspecting Circularity

Circularity of a shaft



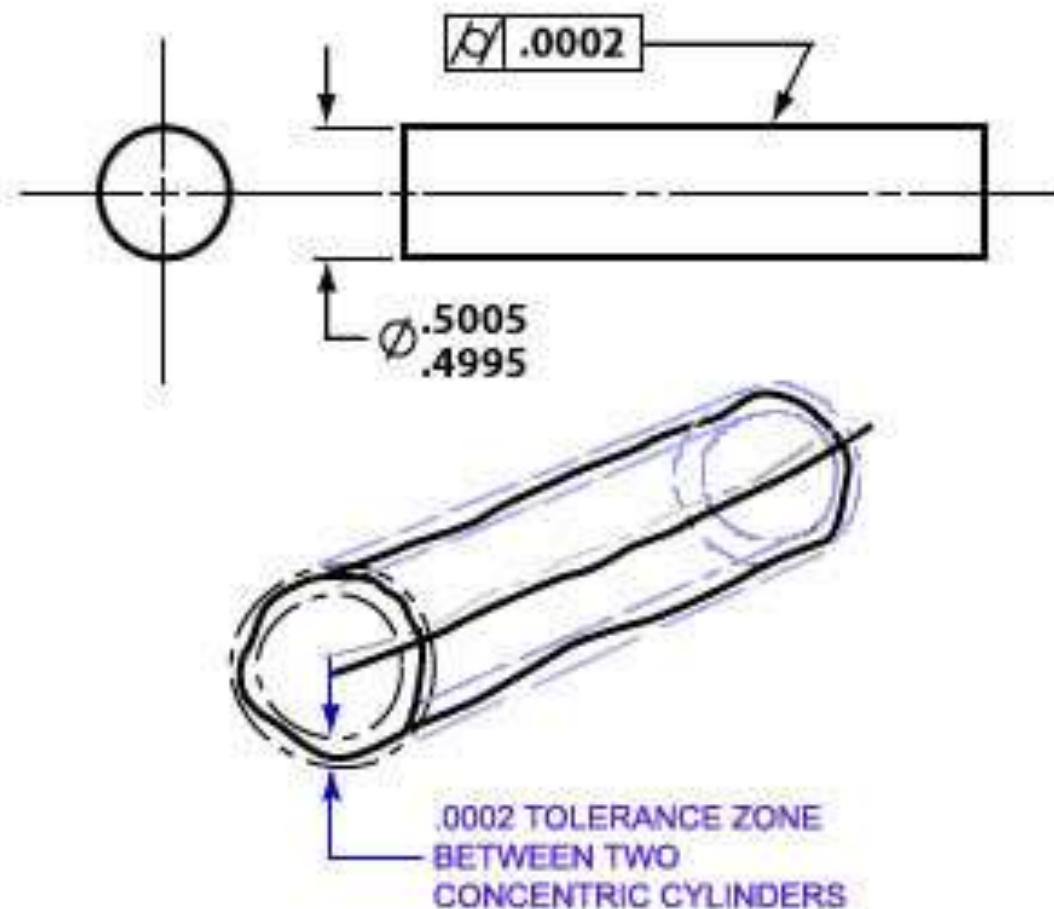
Circularity Control



Cylindricity

- A condition of a surface of revolution in which all points of the surface are equidistant from a common axis.
- Cylindricity Control:
 - A geometric tolerance that limits the amount of cylindricity error on a part surface.
 - Specifies a tolerance zone of two coaxial cylinders within which all points must lie.

Cylindricity Control



Rule One and Cylindricity

- ❑ If rule one applies, to a cylindrical FOS, an automatic indirect cylindricity control exists for the FOS surface.
- ❑ At MMC, the surface must be a perfect cylinder. If the size varies from MMC towards LMC, a cylindricity error is allowed. As shown on the slide before.
- ❑ Note TECHNOTE's for all sections.

Cylindricity Applications

- Cylindricity controls are used to limit surface conditions such as out of round, taper and straightness. When inspecting cylindricity, there is no datum.
- Note the following:
 - Diameter must be within size tolerance.
 - Cylindricity control does not overrule rule one.
 - Cylindricity control tolerance must be less than the total size tolerance.
 - Cylindricity control does not affect the outer boundary of the FOS.

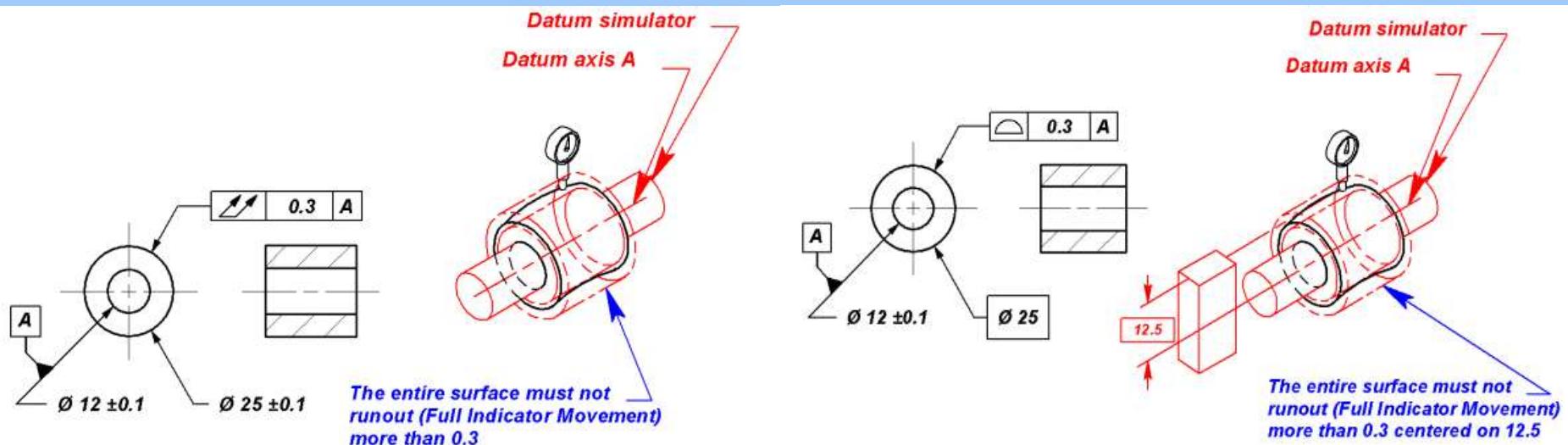
Indirect Cylindricity Controls

- As before, there are several geometric controls that indirectly control cylindricity. These are rule one, profile of a surface and total runout. As stated before, indirect controls are not inspected. If you need to inspect cylindricity, then a cylindricity control is needed. Note that a cylindricity control tolerance should be less than the tolerance value of any indirect control that affects diameter.

Example of an Indirect Control

Total Runout controls form, orientation and location

Surface Profile



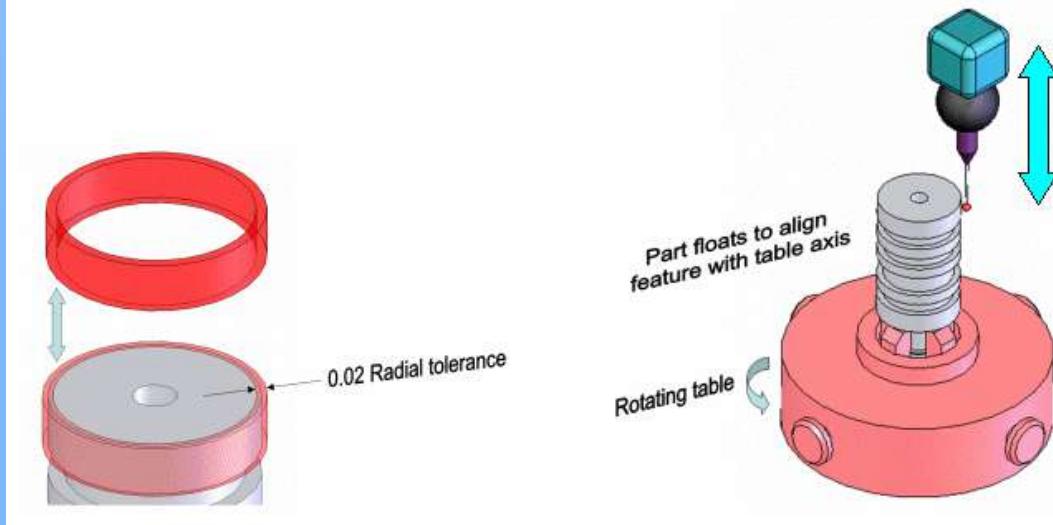
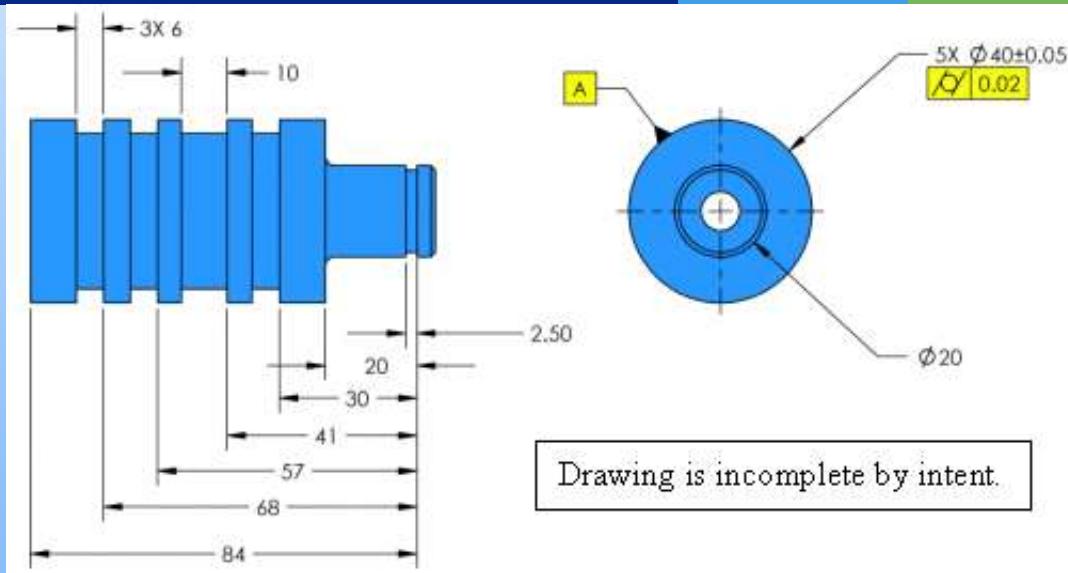
Legal Specification Test for a Cylindricity Control

- Review flowchart on page 97.

Inspecting Cylindricity

- ❑ Inspecting cylindricity requires several separate parameters be checked. These are size of FOS, rule one boundary and the cylindricity requirement.
- ❑ Cylindricity inspection requires the entire surface under the cylindricity control be inspected.

Inspecting a Cylinder



Summary of Chapter Four

- ❑ There are four geometric controls that are designated as form controls. These are:
 - ❑ Flatness
 - ❑ Straightness
 - ❑ Circularity
 - ❑ Cylindricity
- ❑ Figure 4-25, on page 98, summarizes data on these controls.

QUESTIONS?

Chapter Five

Datums (Planar)

Chapter Goals

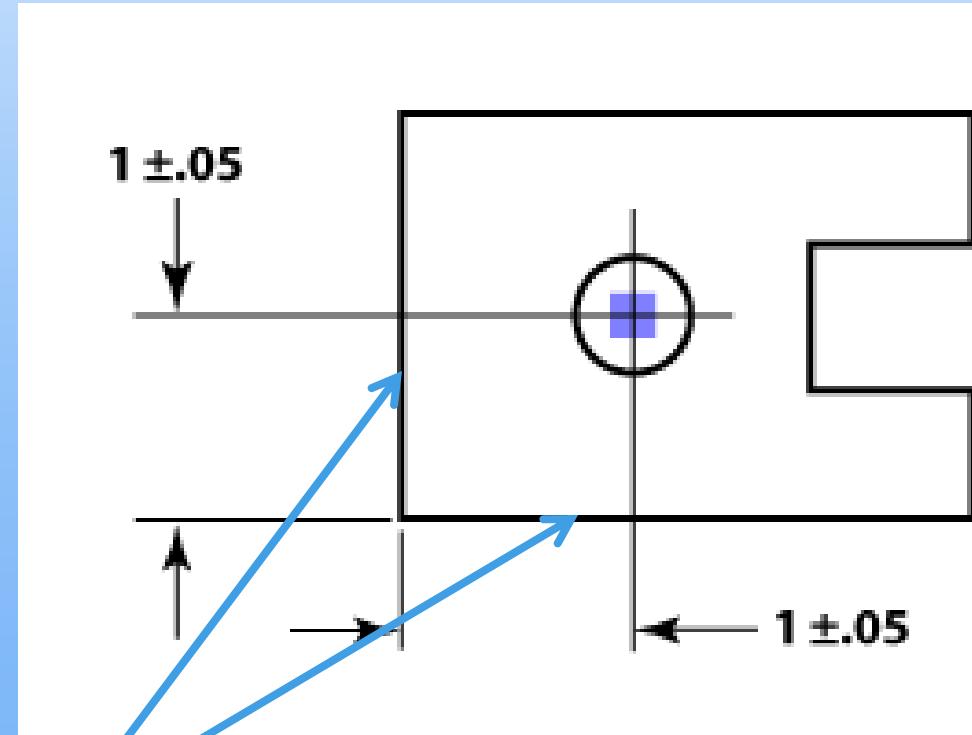
- ❑ Understand the datum system (Planar datums).
- ❑ Interpret Datum Targets.

Datums

■ Implied Datums

- An assumed plane, axis or point from which a measurement is made. Implied datums are leftovers from CT.

Implied
Datums



Datums

- Problems with Implied Datums
 - Do not clearly communicate to drawing user which surfaces should contact the inspection equipment.
 - Do not communicate which surface should contact the inspection equipment first.
- Thus the inspector has to guess what to do.
- Results may be:
 - Good parts rejected
 - Bad parts accepted

Datums

❑ Datums

- ❑ A theoretically exact plane, point or axis from which a dimensional measurement is made.

❑ Datum Feature

- ❑ A part feature that contacts a datum.

❑ Planar Datum

- ❑ True geometric counterpart of a planar datum feature where the true geometric counterpart is the theoretical perfect boundary or best fit tangent plane of a specified datum feature.

Datums

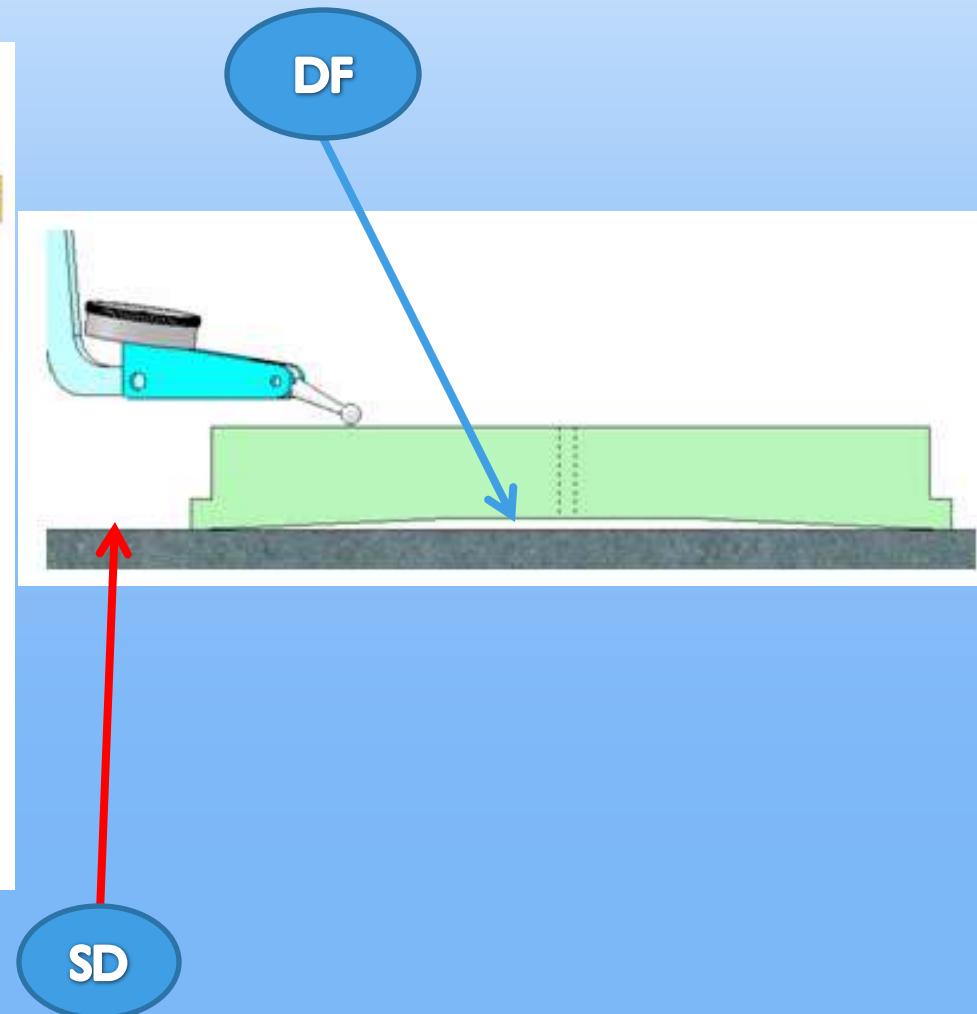
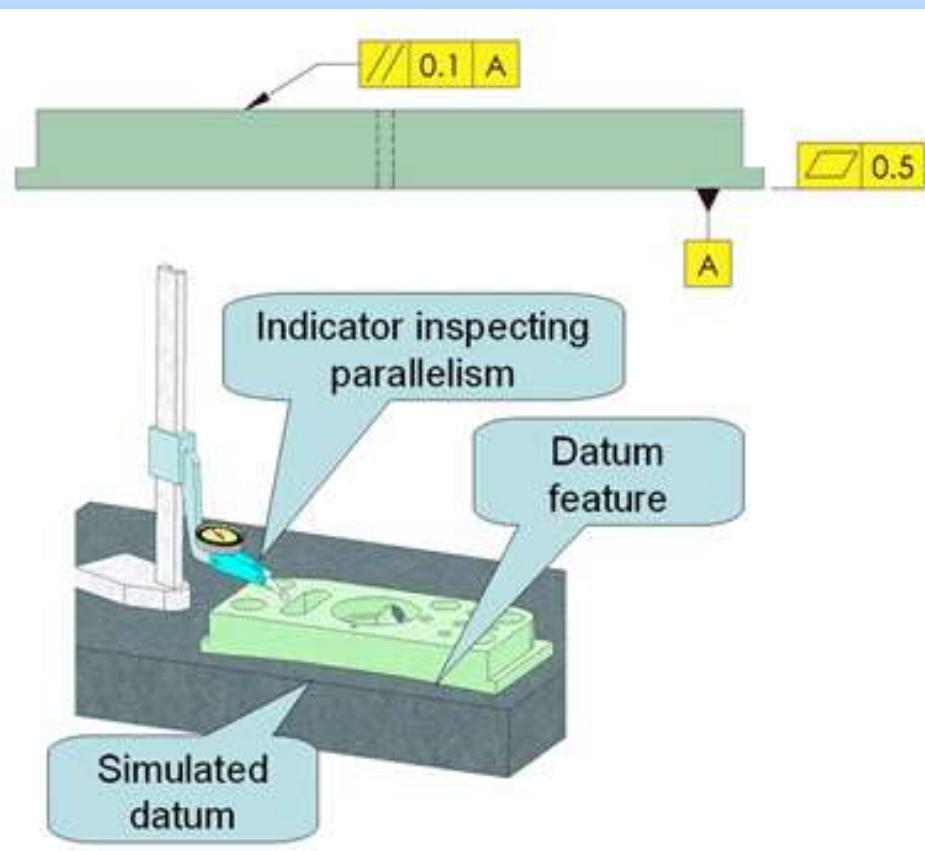
- ❑ Depending upon the type of datum feature, a true geometric counterpart may be:
 - ❑ A tangent plane contacting the high points of a surface.
 - ❑ A MMC or LMC boundary
 - ❑ AVC boundary
 - ❑ An AME
 - ❑ A worst case boundary
 - ❑ A mathematically defined contour

Datums

□ Datum Feature Simulator

- The true geometric counterpart is a theoretical feature which is assumed to exist and be simulated by the associated inspection equipment. The equipment used to establish the TGC surface is called the datum feature simulator.
- The surface established by the datum feature simulator is called the simulated datum.

Datums



Datums

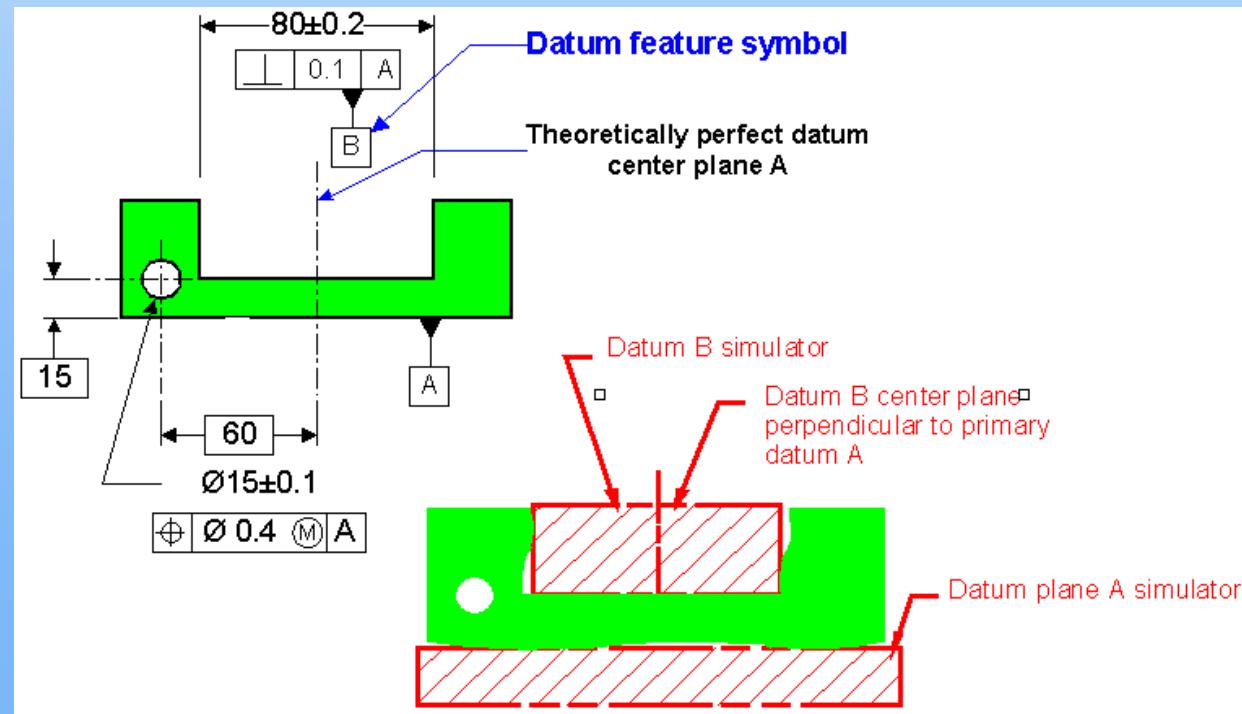
❑ Important to Remember

- ❑ Datum features are part features and exist on the part.
- ❑ A datum feature simulator is the inspection equipment used to establish a simulated datum.
- ❑ Datums are theoretical.
- ❑ A simulated datum is considered to be a datum.

Datum Feature Symbol

- This symbol is used to specify a datum feature. How the datum symbol is attached to the part feature determines if it designates a planar datum or a FOS datum. There are four ways to display the datum feature symbol.

Figure 5-3 pg. 117.

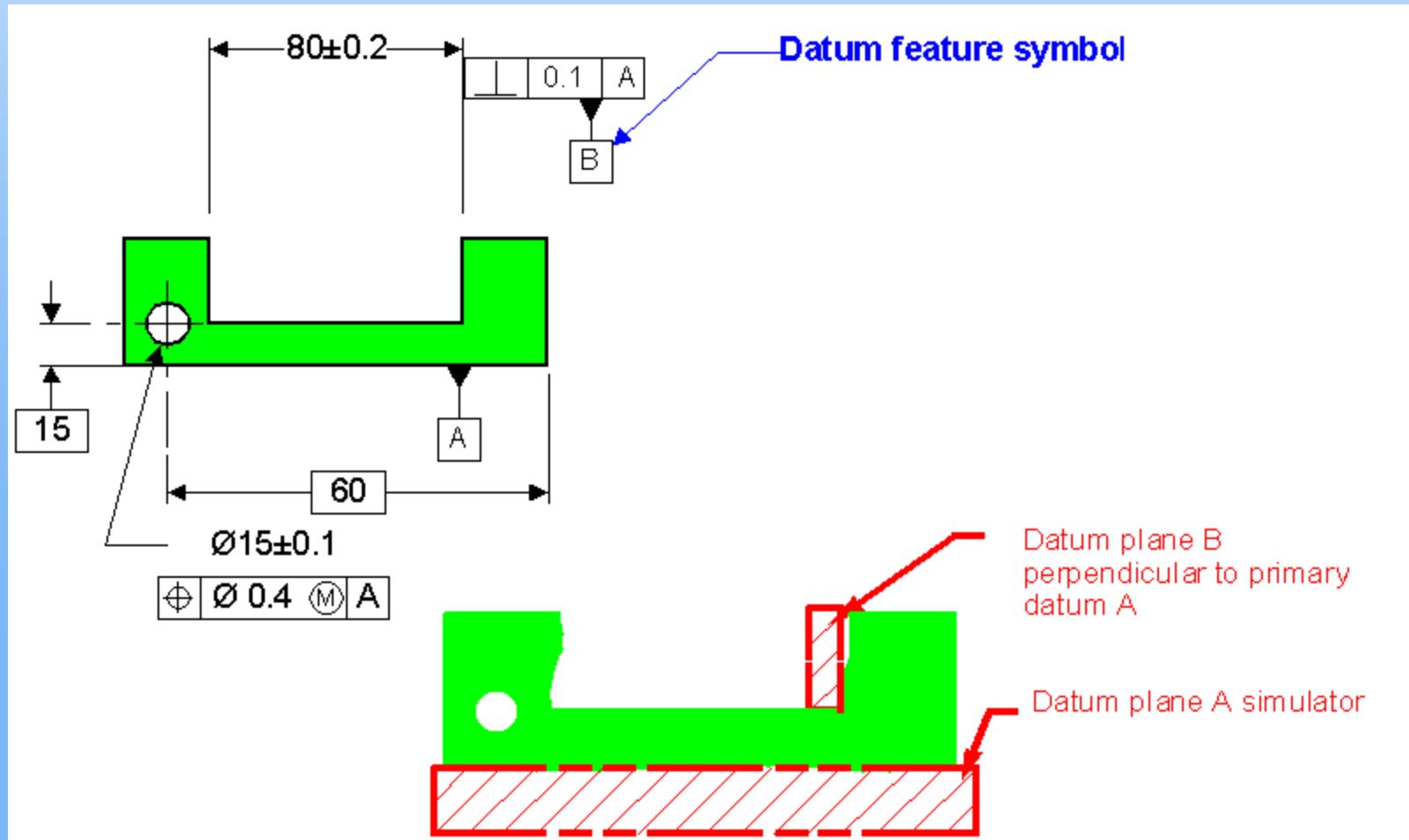


Datum Selection

- ❑ Datum features are not just selected any way the designer wants. You do not flip a coin. Datum features are selected by what is important to maintain, or the critical characteristics, on the part or assembly.
- ❑ Go to pg. 118 and we will discuss fig. 5-4.

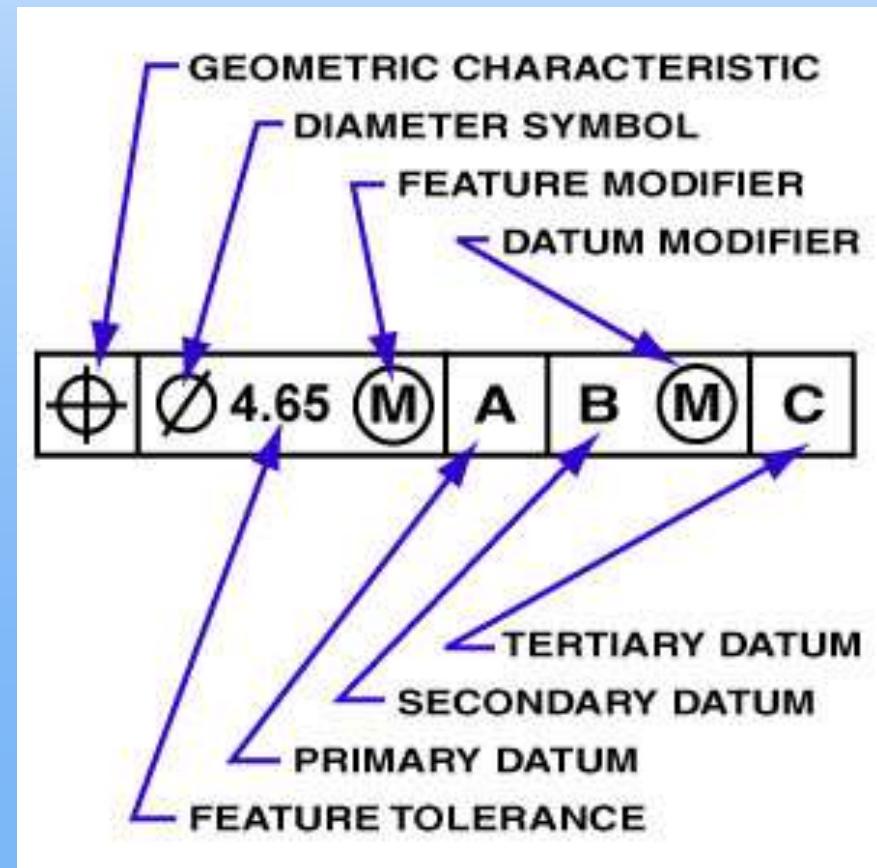
Datum Selection

- both the datum feature symbol and the perpendicularity apply to the left surface of the slot.



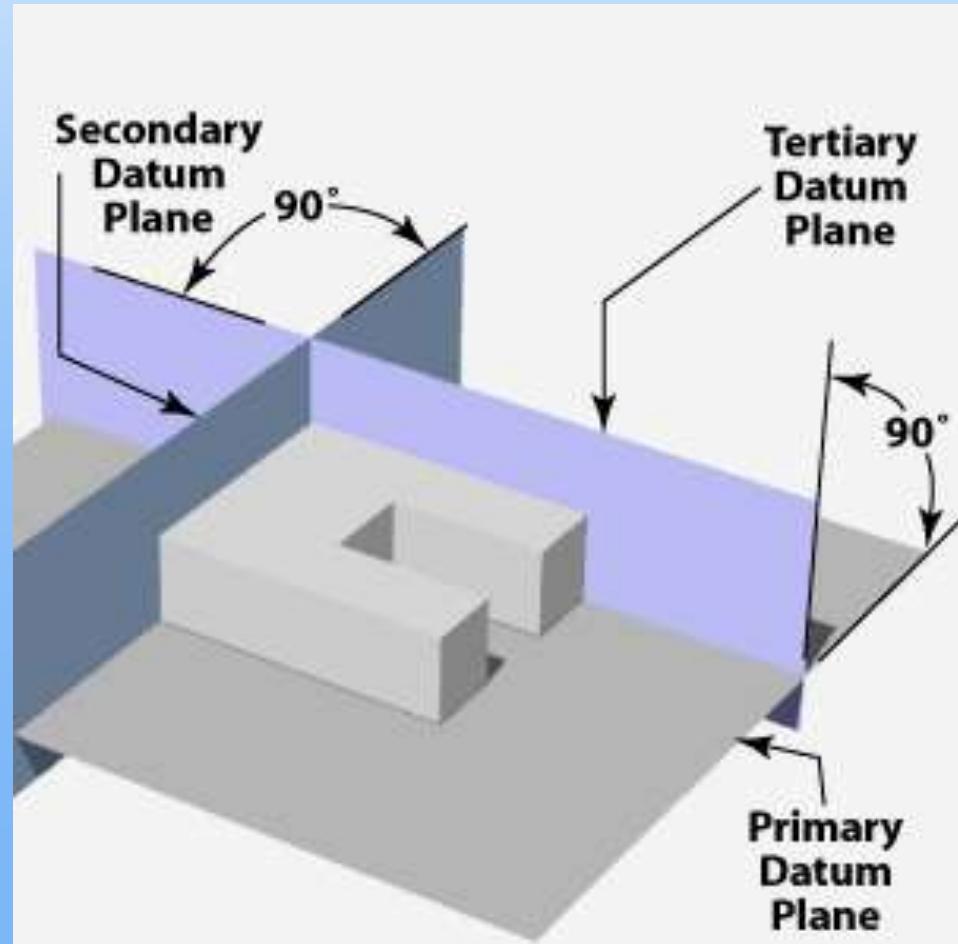
Datums and the Feature Control Frame

- As we have previously discussed, if datums are specified, the designer must tell how the datums are to be used. This is typically done by specifying the datums in a feature control frame.
- The information specified is as shown on the right.



Datum Reference Frame

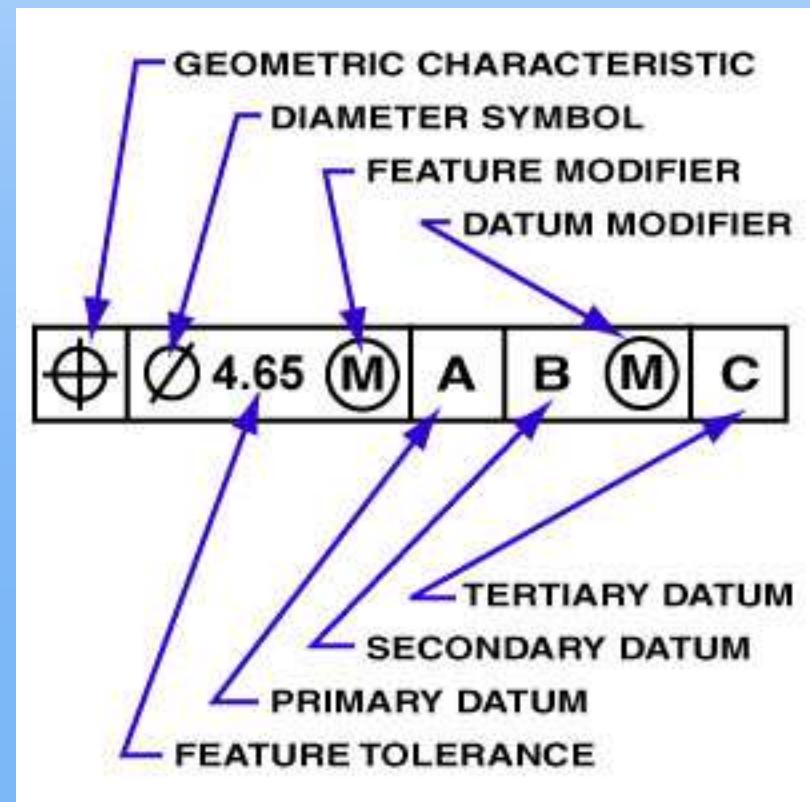
- Measurements cannot be repeated with any degree of confidence if the part being measured is loosely constrained. Thus the three datum planes, comprising the Datum Reference Frame, provide to constrain the 6 degrees of freedom.
- The frame also provides for direction and an origin for dimensional measurements.



Datum Reference Frame

- ❑ Note the table in fig. 5-6 on pg. 120.
- ❑ Again note the feature control frame:

- ❑ The order of the datums is fixed.



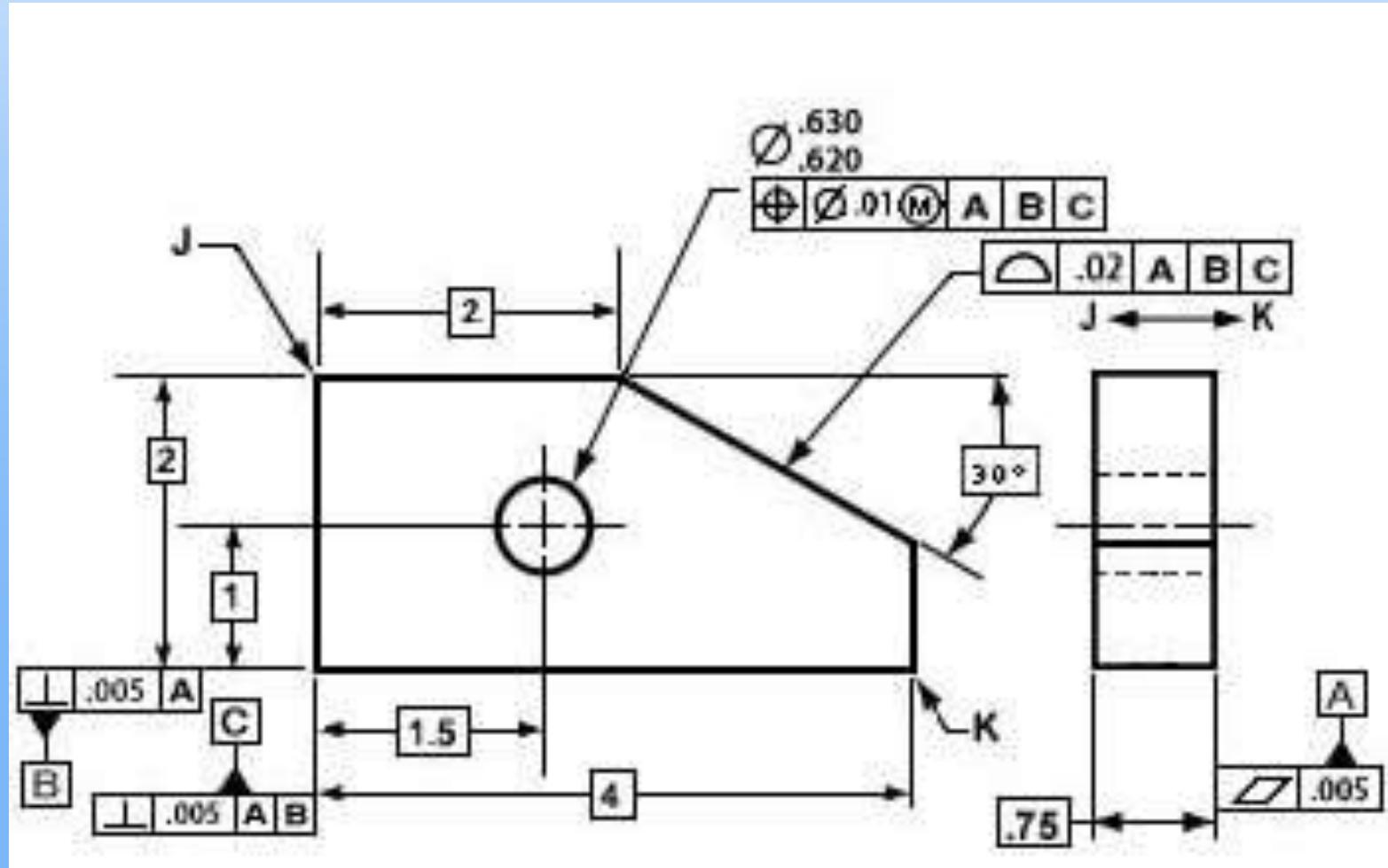
3-2-1 Rule

- ▣ This rule defines the minimum number of points of contact required for a part datum feature with the primary, secondary and tertiary datum planes.
- ▣ 3-2-1- rule only applies on a part with all planar datums and only to planar datum features.
- ▣ The rule is explained in fig. 5-9 pg. 123.

Datum Related Vs. FOS Dimensions

- ❑ Only dimensions that are related to a datum reference frame through geometric tolerances should be measured in a datum reference frame.
- ❑ If a dimension is not associated with a datum reference frame then there is no reference to measure from.

Datum Related Dimensions

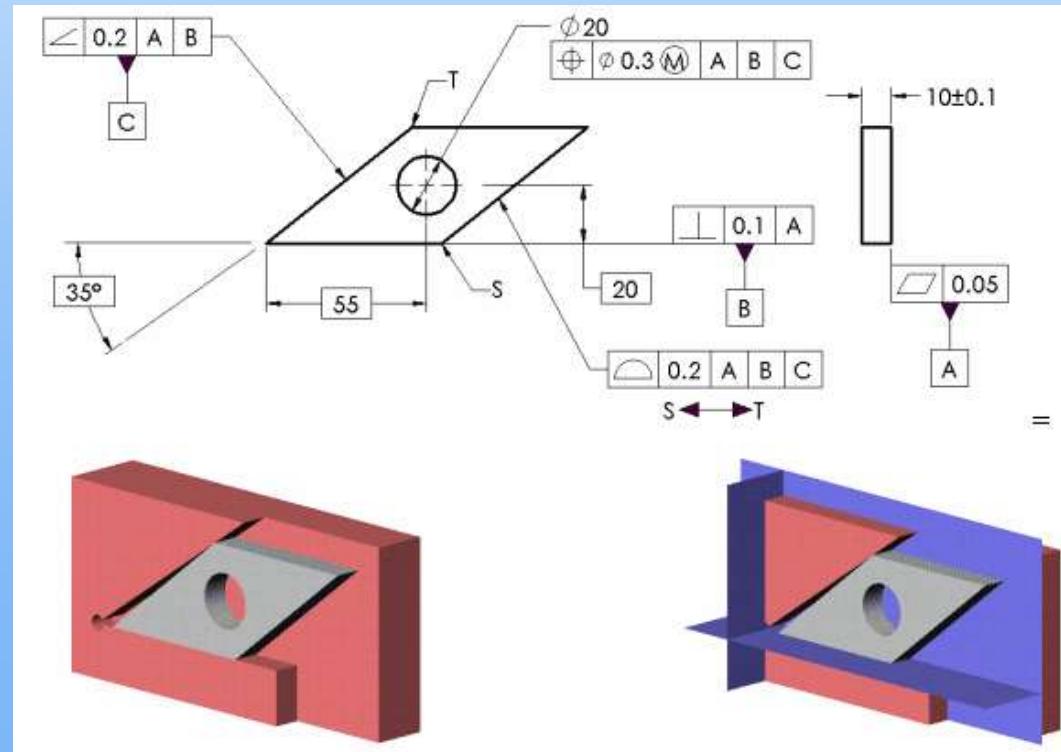


Inclined Datum Feature

- A datum feature that is at an angle other than 90 degrees, relative to the other datum features. Yet the datum reference frame will have the three planes at the basic angles. However a datum feature would be at the angle of the incline.

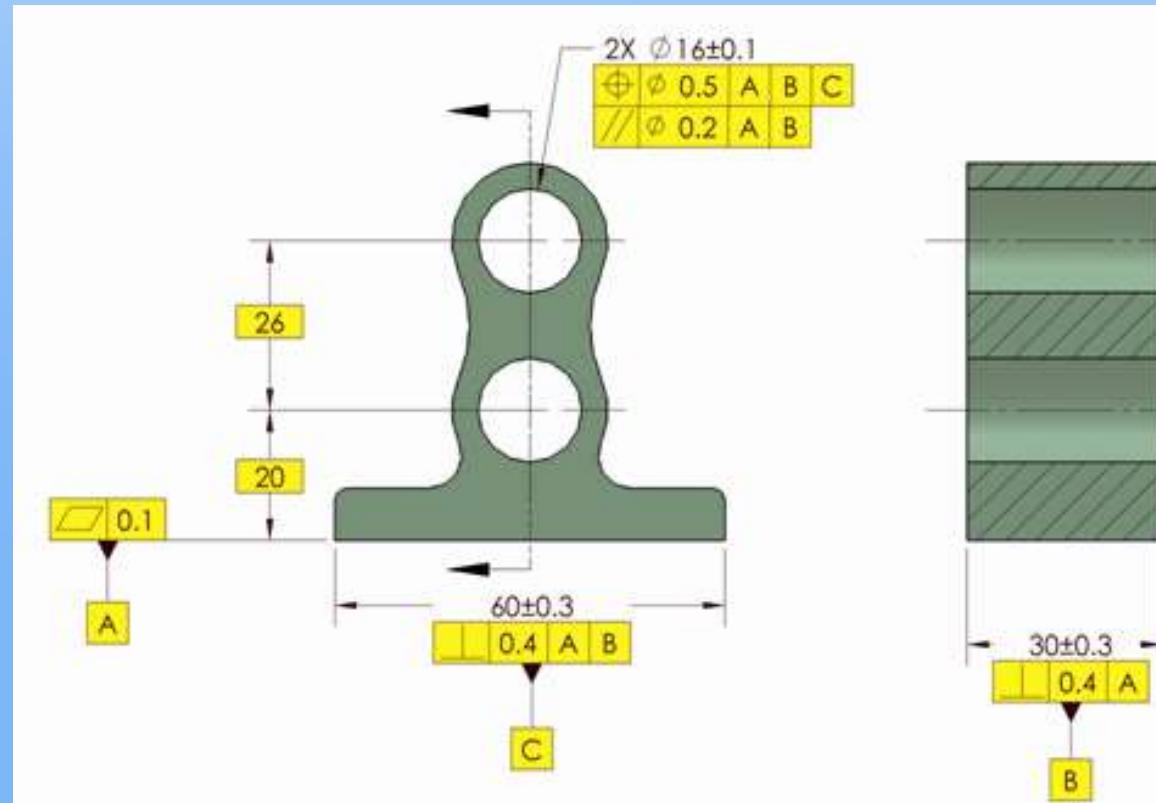
Inclined Datum Feature

- It isn't always possible or practical to select datum features that are mutually perpendicular to one another when establishing a datum reference frame. Notice that datum feature C is not nominally perpendicular to datum feature B. The datum feature simulator for C would be made at 35° to the datum feature simulator for B (shown here in red). The actual datum planes (shown in blue), which comprise the datum reference framework, would however be mutually perpendicular to one another as is illustrated in the last figure. The deviation of the hole from the 55mm BASIC location would be measured from the third datum plane-not from the sharp point on the actual part.



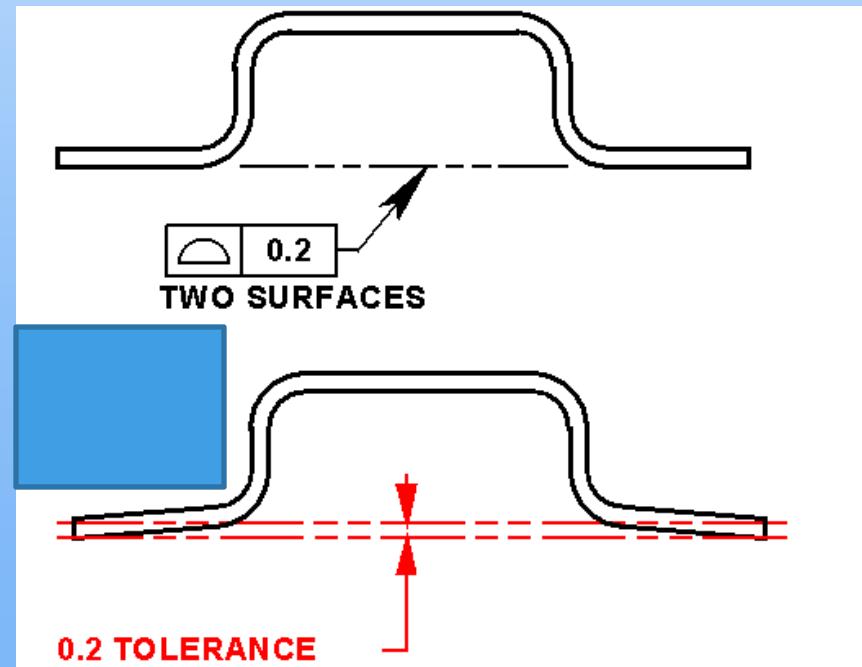
Multiple Datum Reference Frames

- If a part needs to have more than one datum reference frame to define functional relationships then as many as needed may be added.



Coplanar Surface or Datum Features

- Coplanar surfaces are two or more surfaces that are on the same plane.
- Coplanar datum features are two or more datum features that are on the same plane.

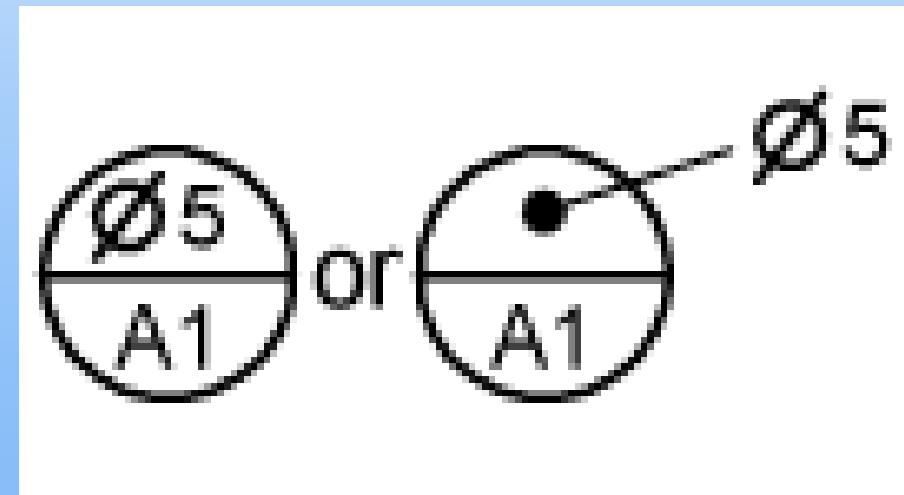


Datum Targets

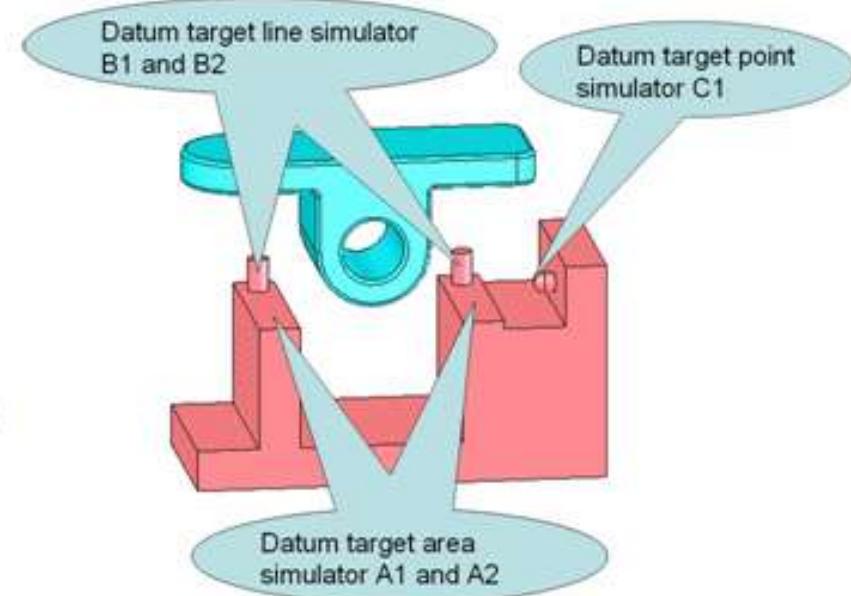
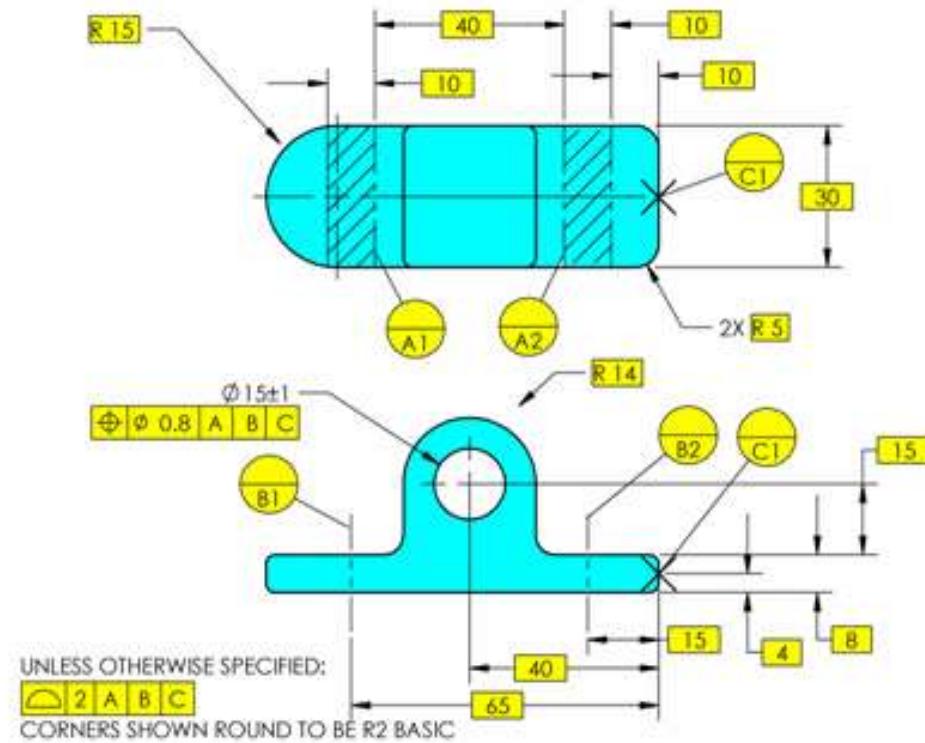
- ❑ Datum targets are symbols that describe the shape, size and location of gage elements that are used to establish datum planes or axes.
- ❑ Datum gage elements that are shown on the part surfaces on a drawing do not exist on the part.
- ❑ Datum targets describe gage elements which only contact a portion of the part surface.

Datum Target Symbol

- The datum target symbol has two parts. The bottom denotes the datum letter and target number associated with that datum. The top half has gage element size if applicable. The leader tells you if target exist on surface or is hidden.



Datum Targets On A Drawing



Datum target point ,
Datum target line,
Datum target area

See Page 128
and 129.

Datum Target Applications

- Datum targets are used to create efficient inspections of the part. By establishing datum targets the designer is letting the inspector know that only partial inspections are critical to the part being acceptable.
- Datum targets are also used to create the reference frame where one has irregular surfaces.
- Review pages 131, 132 and 133.

QUESTIONS?

Chapter Six

Datums: Axis and Centerplane

Chapter Goals

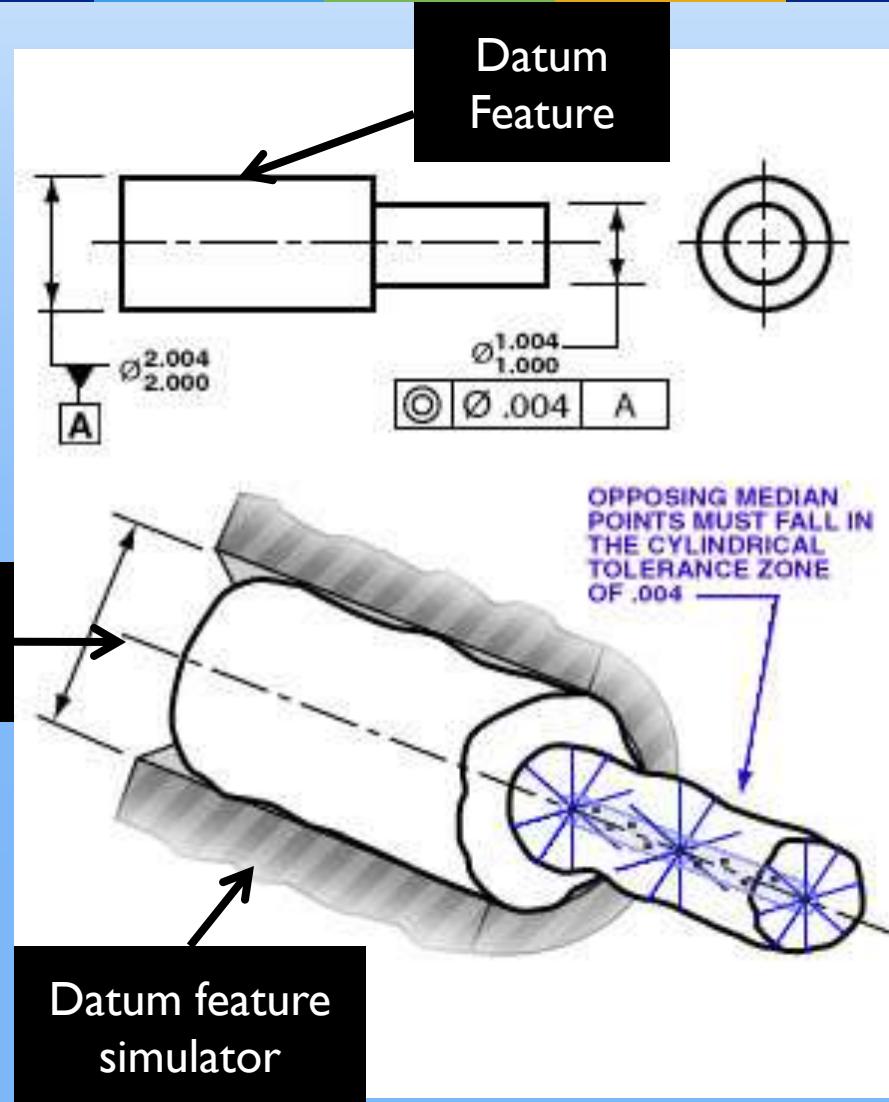
- ❑ Interpret FOS datum specifications. (RFS)
- ❑ Interpret FOS datum specifications. (MMC)

FOS Datum Features

- ❑ Axis or centerplane as a datum
 - ❑ A FOS is specified as a datum feature by associating the datum identification symbol with the FOS.
 - ❑ If a FOS is specified as a datum feature, the result is an axis or centerplane as a datum.
 - ❑ Review fig. 6-1, pg. 146 for five ways to specify an axis or centerplane as a datum.

Datum terminology

- If a diameter is designated as a datum feature, as shown in the drawing on the left, the datum feature is the part surface. The datum axis is the simulated axis and is the origin of all measurements.



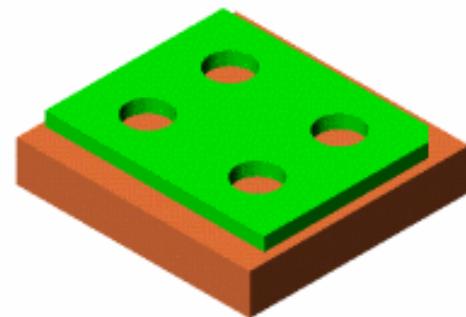
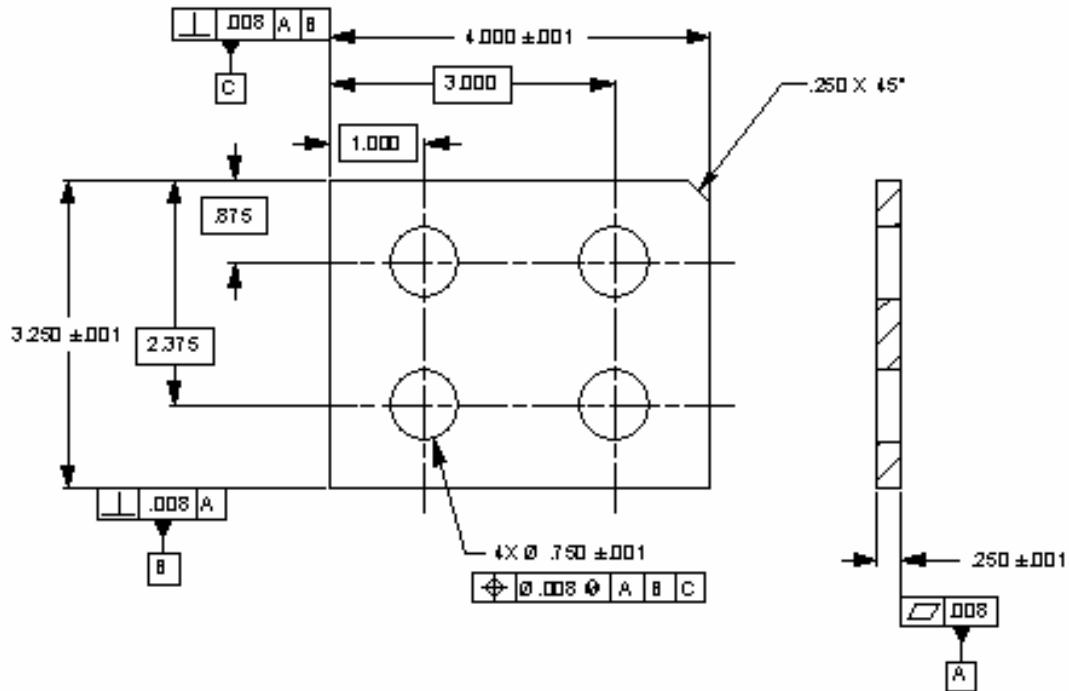
FOS Datum Features

- If a FOS datum feature is referenced on a drawing, the material condition for establishing the datum axis must be communicated. Remember that a feature control frame does not need a modifier to establish the datum axis. See rule two.
- Review pg. 147, fig 6-2.

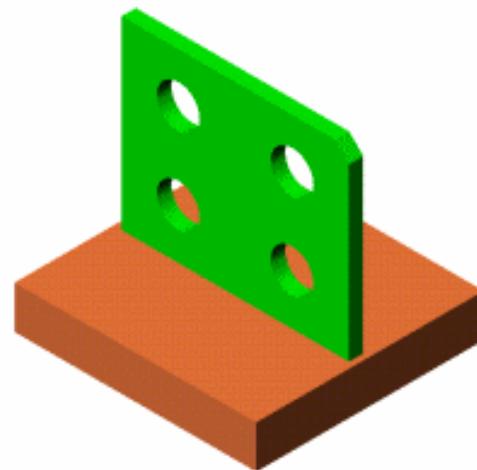
Rule #2

"RFS applies, with respect to the individual tolerance, datum reference, or both, where no modifying symbol is specified. MMC or LMC must be specified on the drawing where it is required."

FOS Datum Features



A is a stable primary datum.

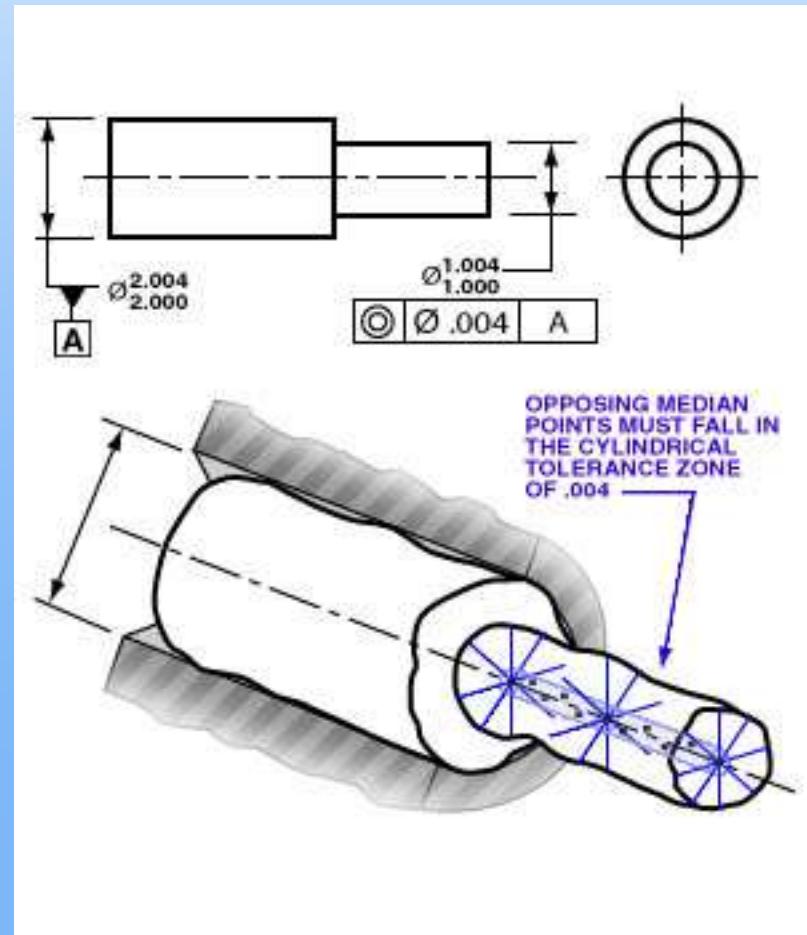
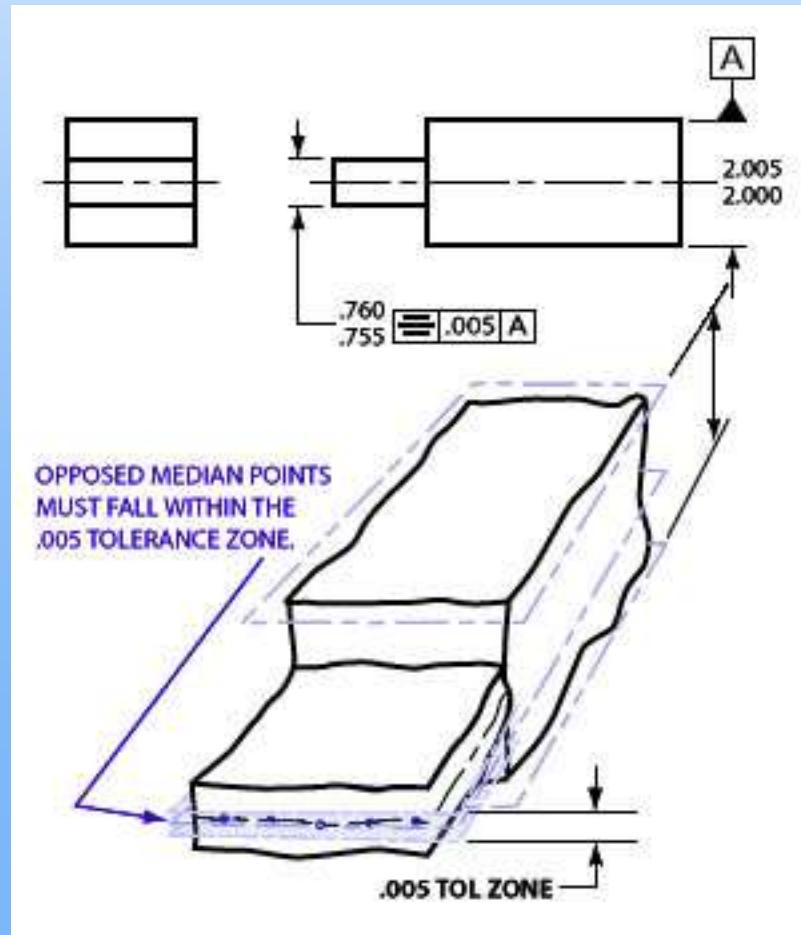


B is too thin to be primary.

FOS Datum Feature Applications: RFS

- Diameter designated as a datum feature and referenced in a FCF @ RFS, a datum axis is established. This axis is established through contact with the feature simulator.
 - Ref. fig 6-3, pg. 148.
- Parallel plane FOS designated as a datum feature and referenced in a FCF @ RFS establishes a datum centerplane. This plane is established through physical contact with the inspection equipment.
 - Ref. fig 6-4, pg. 149.

Datum Axis/Centerplane RFS Primary



Datum Axis RFS Secondary

- ❑ If a part is oriented by a surface and located by a diameter, the surface and diameter are usually designated datum features. Figure 6-5, pg. 150 shows such an example.
- ❑ You may specify the secondary datum as RFS and then the conditions on pg. 150 apply.
- ❑ Note that the surface datum feature A is the primary datum.

Datum Axis RFS Secondary and Datum Centerplane RFS Tertiary

- If a part is oriented by a surface, located by a diameter and has an angular relationship relative to a FOS, it is usual to have the surface, diameter and FOS designated as datum features. See Fig. 6-6 pg. 151.
- You may specify the diameter as secondary and the slot tertiary, both RFS, and the rules on pg. 151 apply.
- Note that the surface datum feature A is the primary datum.

Datum Axis, Coaxial Diameters, RFS

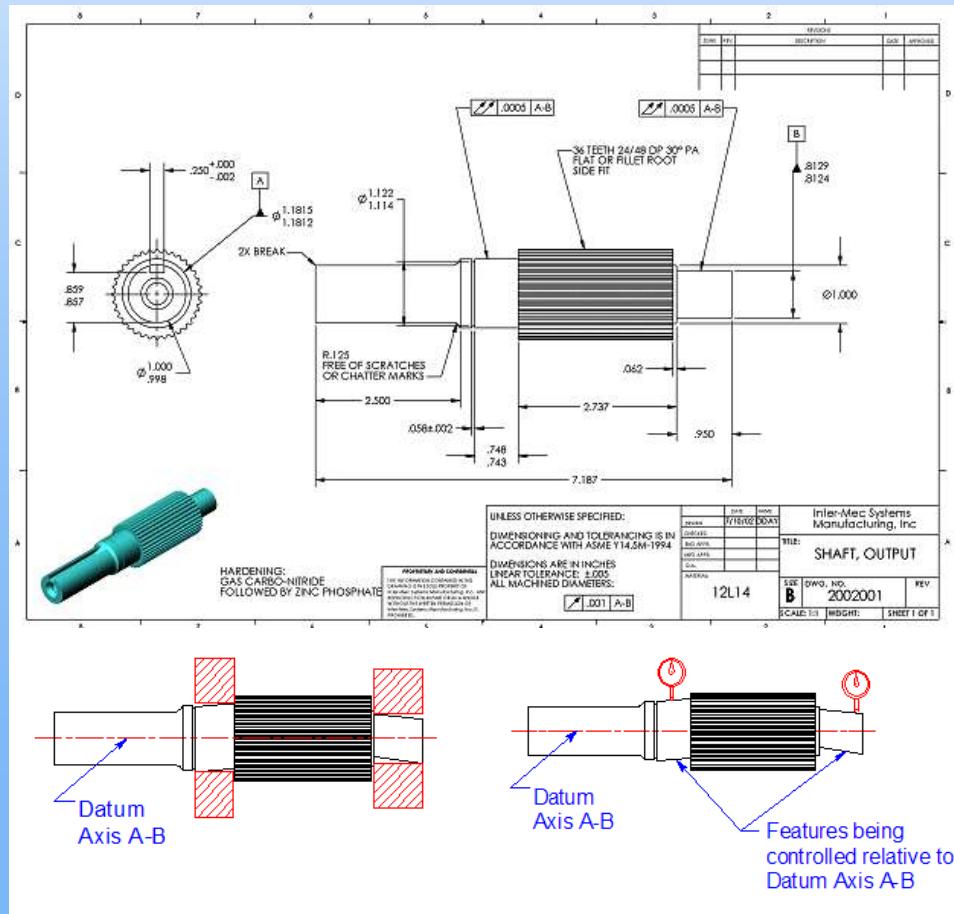
Primary

- One may use two coaxial diameters to establish a single datum axis.
 - Each diameter is designated a datum feature.
 - There must be a connecting part designed as referencing each datum feature and having some type of related feature control.
 - You must create the datum axis by contacting the highest point of each datum feature simultaneously.
 - This single datum axis is the primary datum.

Datum Referencing a Datum

- ❑ **It Is Referencing the Datum - Not Itself**
- ❑ "How can a feature reference itself in a feature control frame?" The answer is, "It can't." Datum features referenced in a feature control frame are establishing a datum reference framework which serves as the origin of measurement. The shaft shown in the next slide has two total runout controls that appear to be referencing themselves. The total runout tolerance applies to the surface of the features. The datum is an axis which is established using the two datum features A and B.

Datum Referencing a Datum



FOS Datum Feature Referenced at MMC

- ❑ If a FOS datum is referenced at MMC (note?) then the gaging equipment that is the datum feature simulator (note?) will be a fixed size (Why?).
- ❑ The datum axis or centerplane is the axis or centerplane of the gage element.
- ❑ The size of the true counterpart of the datum feature is determined by the specified MMC or the MMC VC.
- ❑ Note that the gage is fixed in size and the part may be loose in the gage.

Special Case FOS Datum

- ❑ Discuss fig. 6-9 pg. 154.
- ❑ Note that case one is for an external FOS while case two is for an internal.

Datum Shift

- ❑ Remember that if a FOS datum feature is referenced at MMC, the gage element (datum feature simulator) will be fixed in size.
- ❑ But if the part varies there is an allowance called datum shift.
 - ❑ $DS = MMC$ (gage size) – actual local size. (Almost bonus tol.)
- ❑ You need to change location of datum axis is what this means.
- ❑ See fig. 6-10, pg. 155.

More datum Shift

- ❑ Datum shift is only allowed when the MMC is in the datum portion of the FCF.
- ❑ Datum shift occurs when the AME of the datum feature departs from MMC.
- ❑ Max datum shift is gage – LMC for the datum feature.
- ❑ Datum shift is additional tolerance allowed.
- ❑ Fig. 6-11, pg. 156 and fig 6-12, pg. 157.

FOS Datum Feature Applications

■ Datum Axis MMC Primary

- If a diameter is a datum feature and referenced in a FCF as primary at MMC, a fixed gage element may be used as the datum feature simulator. The fixed gage size is equal to the MMC or WCB for some cases.
- The datum axis is the axis of the datum feature and datum shift may be available.
- Fig. 6-13 shows an external and internal FOS as a datum feature.

Datum Centerplane MMC Primary

- If an FOS that consist of parallel planes is datum feature and referenced in a FCF as primary at MMC, a fixed gage element may be used as the datum feature simulator. The fixed gage size is equal to the MMC or WCB for some cases.
- A datum shift may be available depending upon the AME.
- The datum centerplane is the axis of the datum feature simulator.
- Review fig. 6-14, pg. 159.

Datum Axis MMC Secondary

- ❑ If a part is oriented by a surface and located by a diameter, usually the surface and diameter is designated as datum features.
- ❑ The rules that apply are on pg. 160.
- ❑ Review fig. 6-15, pg. 160.

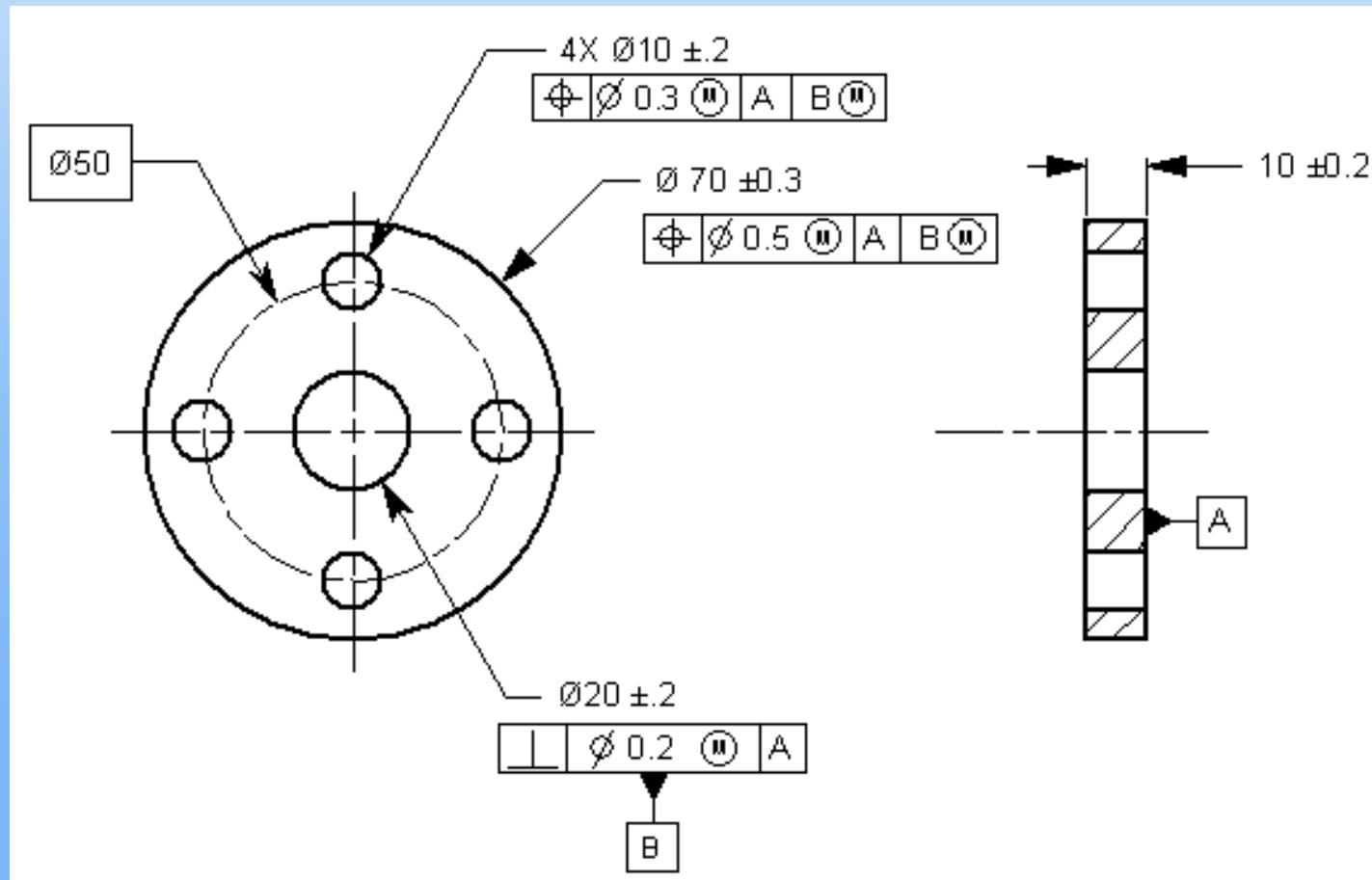
Datum Axis Secondary and datum Centerplane Tertiary

- ❑ If a part is oriented by a surface, located by a diameter and has an angular relationship to a FOS, usually the surface, diameter and FOS are designated as datum features.
- ❑ The rules for this condition are on pg. 161.
- ❑ Review fig. 6-16, pg. 161.

Datum Axis from a Pattern of Holes and MMC Secondary

- ❑ If a part is oriented by a surface and located by a hole pattern, usually the surface and hole pattern is designated as datum features. With the face primary and hole pattern secondary, the rules on pg. 162 apply.
- ❑ Review fig 6-17, pg. 163.

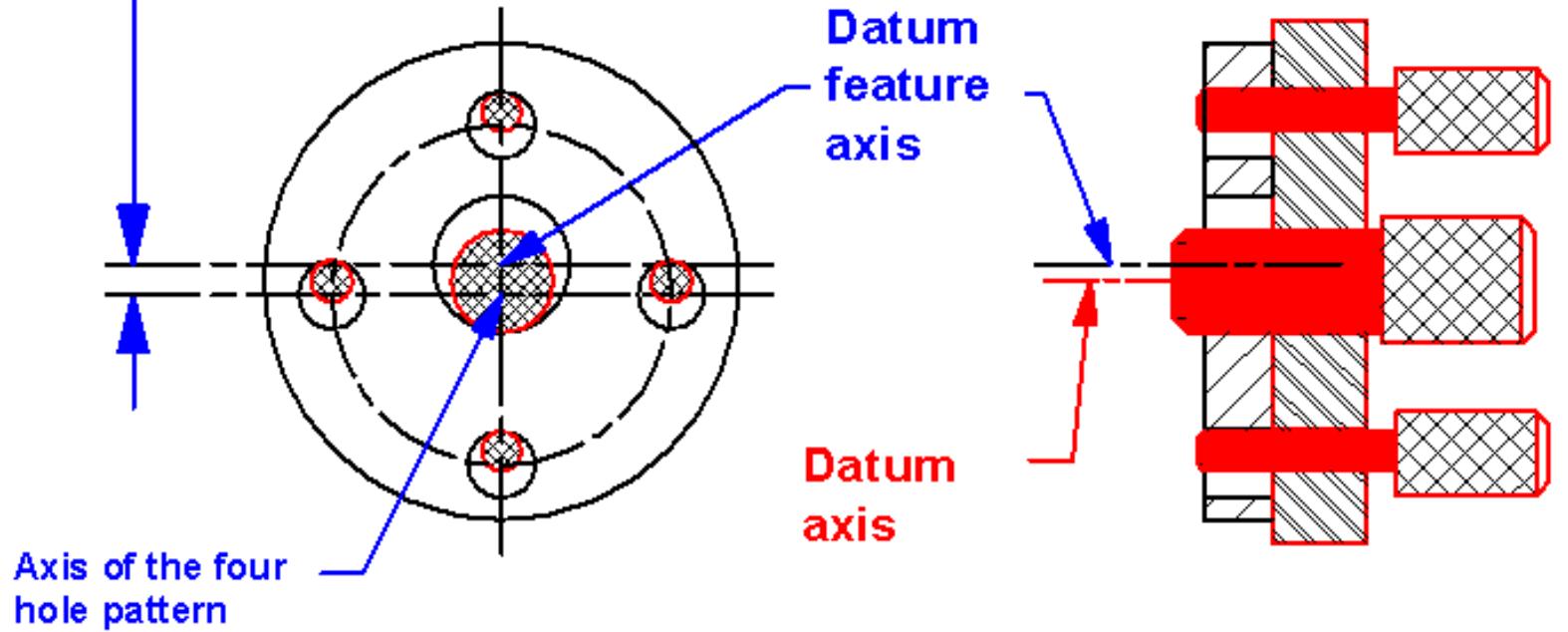
Datum Axis and Hole Pattern



More on Datum Shift

Allowing the datum feature axis to shift may cause an unacceptable shift in the position of the hole pattern if the datum shift is viewed as a bonus tolerance on the position of the features being controlled.

Shift between the datum feature axis and the axis of the pattern of four holes



Datum Sequence

- ❑ In interpreting datum reference frames that involve FOS datum features and planar datum features, the datum sequence is critical to part final tolerances.
- ❑ Review fig. 6-18, pg. 164.

Summary Chart

Concept	Can be applied to a			Is applicable when		
	Surface	FOS	MMC	datum is referenced at		
				LMC	RFS (rule 2)	
Datum Shift	No	Yes	Yes	Yes	No	
Datum Targets	Yes	Yes	No	No	Yes	
MMC	No	Yes	Yes			