GD&T - Geometric Dimensioning and Tolerancing

The Language of Tolerancing on Prints

325 Eisenhower Parkway Suite 4
Ann Arbor, Michigan 48108
USA
734-761-4940
Fax: 734-761-4966

Third Edition

July 2015

Omnex provides training, consulting and software solutions to the international market with offices in the USA, Canada, Mexico, China (PRC), Germany, India, the Middle East, and SE Asia. Omnex offers over 400 standard and customized training courses in business, quality, environmental, food safety, laboratory and health & safety management systems worldwide.

Email: **info@omnex.com**

Web: www.omnex.com

Course Objectives

- Describe the tolerance zone for each GD&T symbol.
- Determine when to use Rule #1 to control form or when other form controls are appropriate.
- Calculate the virtual condition for a given feature.
- Recognize correct syntax for feature control frames.
- Define datum and datum feature, and correctly interpret the relationship of datums to a geometric tolerance.
- Correctly apply and interpret the MMC modifier.
- Describe how various geometric tolerances should be measured.

Agenda

- Chapter 1 – Need for GD&T
- Chapter 2 – Definitions and Rules
- Chapter 3 – Form
- Chapter 4 – Datums
- Chapter 5 – Profile Tolerances
- Chapter 6 – Orientation
- Chapter 7 – Position
- Chapter 8 – Other Types of Location

A BRIEF INTRODUCTION TO OMNEX

Omnex Introduction

- International consulting, training and software development organization founded in 1985.
- Specialties:
  - Integrated management system solutions.
  - Elevating the performance of client organizations.
  - Consulting and training services in:
    - Quality Management Systems, e.g. ISO 9001, ISO/TS 16949, AS9100, QOS
    - Environmental Management Systems, e.g. ISO 14001
    - Health and Safety Management Systems, e.g. OHSAS 18001
- Leader in Lean, Six Sigma and other breakthrough systems and performance enhancement.
  - Provider of Lean Six Sigma services to Automotive Industry via AIAG alliance.

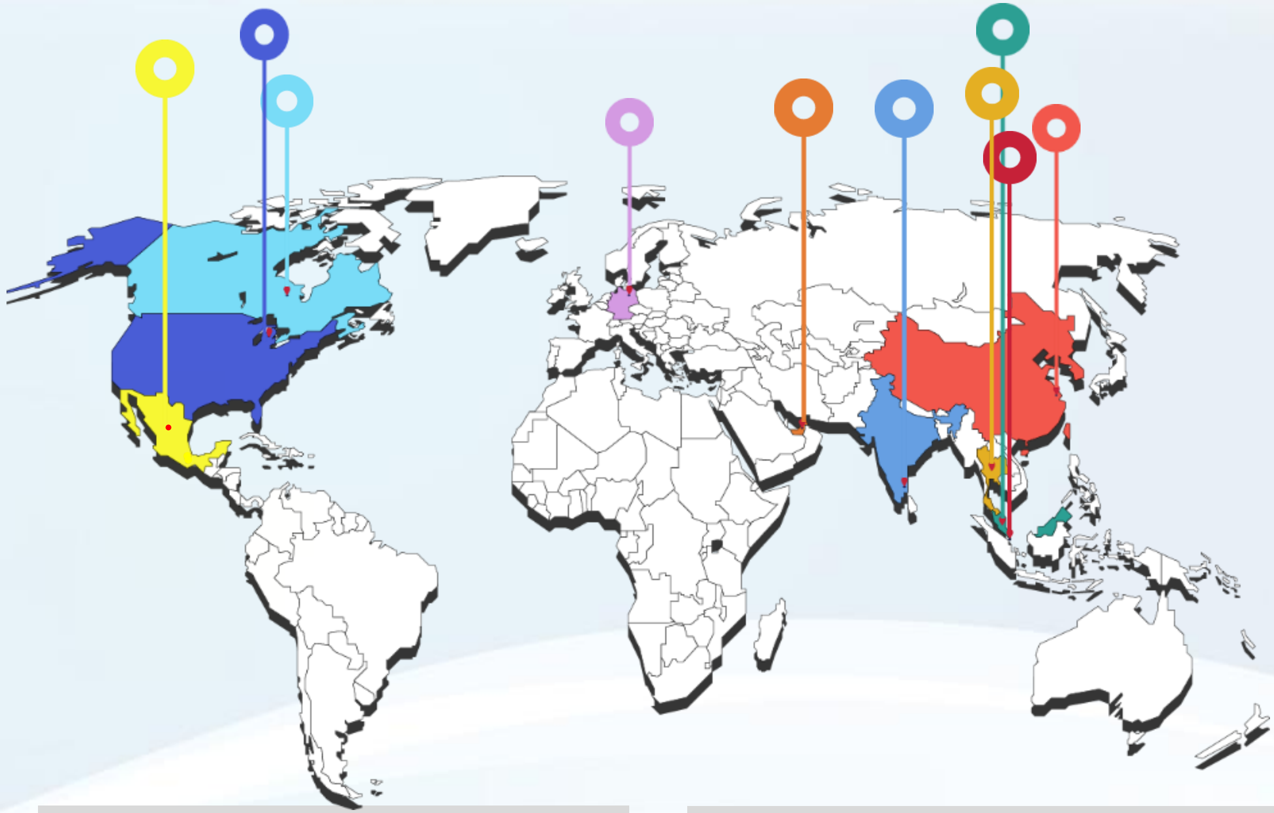
About Omnex

- Headquartered in Ann Arbor, Michigan with offices in major global markets.
- In 1995-97 provided global roll out supplier training and development for Ford Motor Company.
- Trained more than 100,000 individuals in over 30 countries.
- Workforce of over 300 professionals, speaking over a dozen languages.
- Former Delegation Leader of the International Automotive Task Force (IATF) responsible for ISO/TS16949.
- Served on committees that wrote QOS, ISO 9001:2000, QS-9000 and its Semiconductor Supplement, and ISO IWA 1 (ISO 9000 for healthcare).
- Member of AIAG manual writing committees for FMEA, SPC, MSA, Sub-tier Supplier Development, Error Proofing, and Effective Problem Solving (EPS).

Omnex Worldwide Offices

Omnex is headquartered and operates from the United States through offices in Michigan.

The company maintains international operations in many countries to provide comprehensive services to clients throughout Western Europe, Latin America and the Pacific Rim.



- **Omnex Global Head Quarters** (Michigan, USA)
  West Coast Operations (San Jose, CA)
- **Asia Pacific HQ** (Chennai, Pune, Delhi, Bangalore)
- **China** (Shanghai, Guangzhou, Wuhan, Chengdu)
- **Canada** (Mississauga)
- **Europe** (Berlin, Germany)
- **Middle East** (Dubai, Saudi Arabia, Bahrain)
- **Thailand** (Bangkok)
- **Mexico** (Monterrey)
- **Singapore**
- **Malaysia** (Kuala Lumpur)

www.omnex.com
info@omnex.com

OMNEX

Rules of the Classroom

- ✓ Start and end on time
- ✓ Return from breaks and lunch on time
- ✓ All questions welcome
- ✓ Your input is valuable and is encouraged . OMNEX
- ✓ Don’t interrupt others
- ✓ One meeting at a time
- ✓ Listen – and respect others’ ideas
- ✓ No “buts” – keep an open mind
- ✓ Cell phones & pagers off or silent mode
- ✓ No e-mails, texting or tweeting during class
- ✓ If you must take a phone call or answer a text please leave the room for as short a period as possible

Icebreaker

- Instructor Information:
  - Name
  - Background

- Student Introductions:
  - Name
  - Position / Responsibilities
  - What is your involvement in the new product development process?
  - What are your experiences with reviewing and interpreting engineering drawings?
  - Please share something unique and/or interesting about yourself.

Chapter 1

The Need for GD&T

Chapter 1: The Need for GD&T – What We Will Cover

Learning Objectives

At the end of this chapter, you will be able to:

- Identify GD&T callouts on a drawing
- Explain at least five benefits using GD&T
- Distinguish between radius and controlled radius, and why they are different
- Name the four main types of tolerance
- Identify the five main categories of GD&T symbols

Chapter Agenda

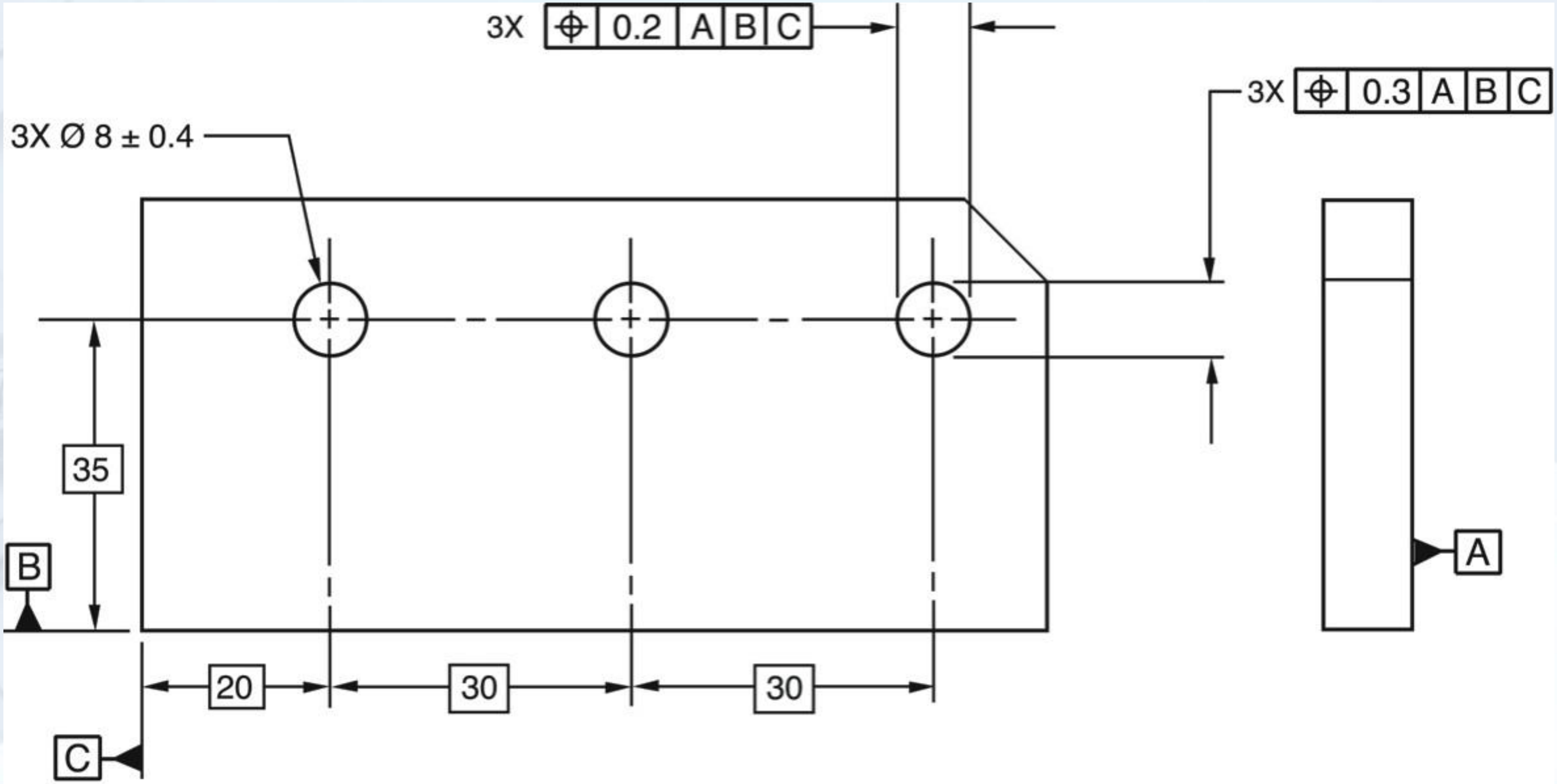
- What is GD&T?
- GD&T Controls

What Is GD&T?

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 symbolic language that can be used to describe the **size** , **form** , **orientation**, and **location** of part features.
- GD&T is a design philosophy **(functional dimensioning)** on how to design and dimension parts based on the FUNCTION of a part or assembly.

What's It Look Like?



Things to Notice on the Previous Slide

Three terms that you'll need to understand when looking at a print with GD&T:

- Feature Control Frame
- Datum Feature Symbol
- Basic Dimension

If any of these items appear on a drawing, you know that it is using the language of GD&T.

Benefits and Results

- Allows maximum tolerances
- Eliminates assumptions
- Based on fit and function of the part or assembly
- Allows bonus and shift
- Flexible tolerance shape
- Enables Functional gaging
- International understanding
- Replaces notes
- Saves money!!

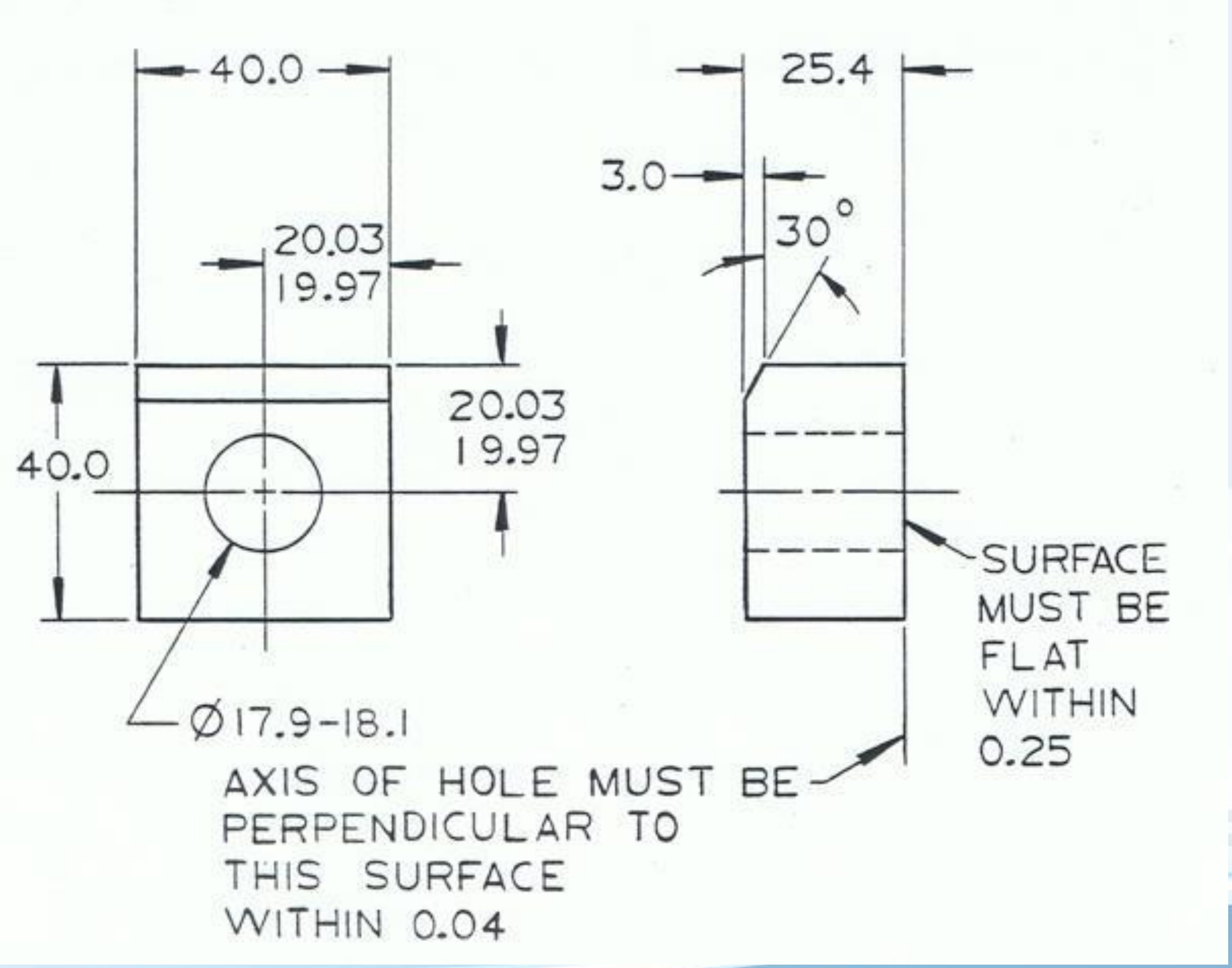
The ASME Standard

- Y14.5-2009 is published by the American Society of Mechanical Engineers; this is the focus of this course
- Previous edition was 1994

Other related standards from ASME and ISO

- International GPS Standard: ISO 1101-2012
  - China: GB/T 1182-2008 equivalent to ISO 1101
  - Germany: DIN ISO 1101 equivalent to ISO 1101
  - Japan: JIS B0021 equivalent to ISO 1101
  - Britain: BS ISO 1101 equivalent to ISO 1101

A Print Without GD&T



Possible Problems?

- Ambiguous datums for measurement
- No priority to the datums
- Square tolerance zones
- Notes can be Constant-size tolerance zones, confusing. o.m.n.ex
- no bonus tolerance allowed

QUALITY

OMNEX

Class Exercise

Using the manual, try to fill in as much of each of the following charts as you can…

FORM		
SYM.	Name	FCF Example

OMNEX

Profile

PROFILE		
SYM.	Name	FCF Example

Orientation

ORIENTATION		
SYM.	Name	FCF Example

Location

LOCATION		
SYM.	Name	FCF Example

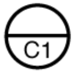
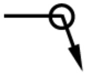
Runout

RUNOUT		
SYM.	Name	FCF Example

Other Symbols…

FEATURE CONTROL FRAME	⊥ | 0.15 | B
MAXIMUM MATERIAL CONDITION	Ⓜ
LEAST MATERIAL CONDITION	Ⓛ
PROJECTED TOLERANCE ZONE	Ⓟ
FREE STATE	Ⓕ
TANGENT PLANE	Ⓣ
INDEPENDENCY RULE	Ⓘ
UNEQUAL TOLERANCE ZONE	Ⓤ
CONTINUOUS FEATURE	⟨CF⟩
STATISTICAL TOLERANCE	⟨ST⟩
DATUM FEATURE	▶ A

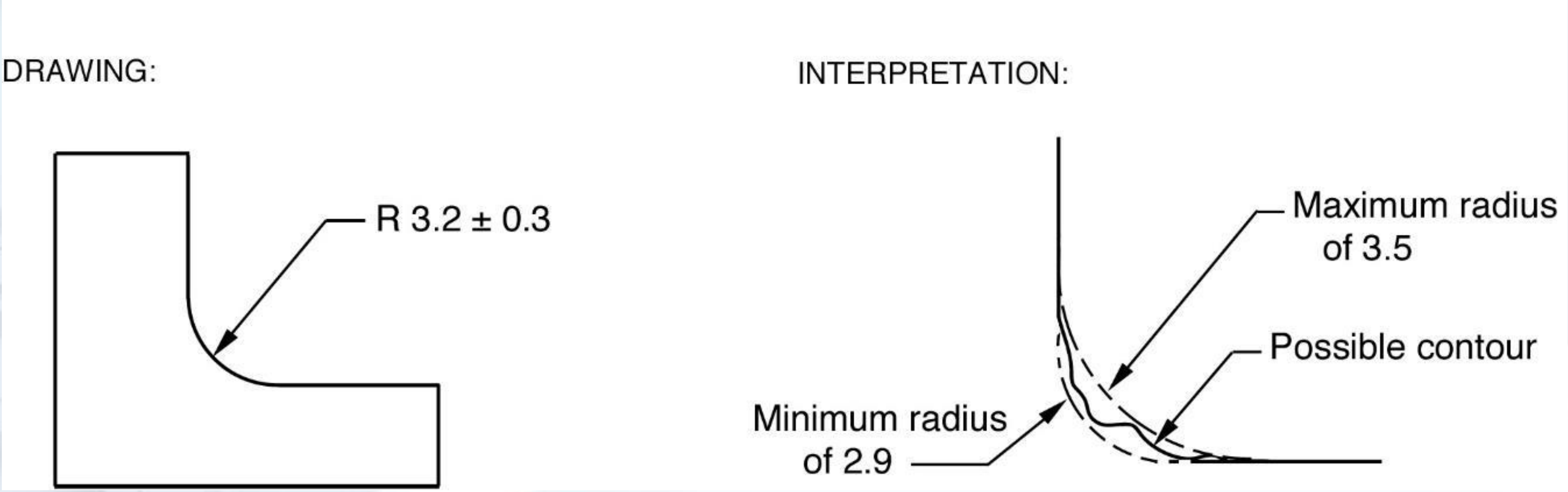
Other Symbols…

DATUM TARGET	
MOVABLE DATUM TARGET	
DATUM TARGET POINT	×
TRANSLATION	▷
BASIC DIMENSION	32.5
ALL AROUND	
ALL OVER	
DIMENSION ORIGIN	
DIAMETER	Ø
SPHERICAL DIAMETER	SØ
RADIUS	R

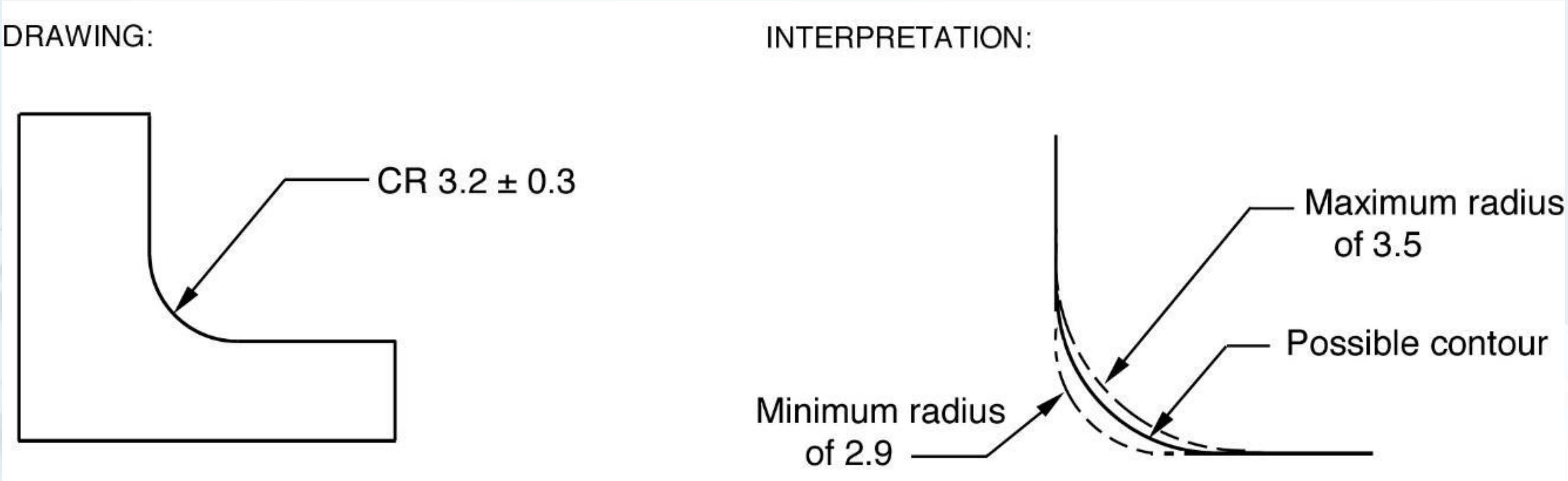
Other Symbols…

CONTROLLED RADIUS	CR
SPHERICAL RADIUS	SR
REFERENCE DIMENSION	(65)
NUMBER OF PLACES	2X
BETWEEN	⟷
DIMENSION NOT TO SCALE	$\underline{130}$
COUNTERBORE	⌴
SPOTFACE	⌊SF⌋
COUNTERSINK	⌵
DEPTH	↧
SQUARE	□

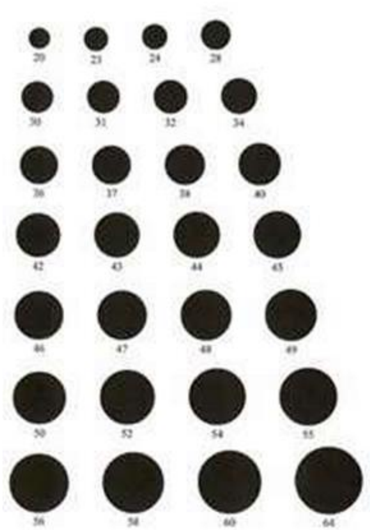
Radius: Regular or Controlled



Radius: Regular or Controlled



Four Types of Tolerance: Big Four, “SLOF”

(Size)	(Location)	(Orientation)	(Form)
	 		 

Things to control / GD&T Tools	Size	Location	Orientation	Form
Size				
Location				
Orientation				
Form				
Runout				
Profile				

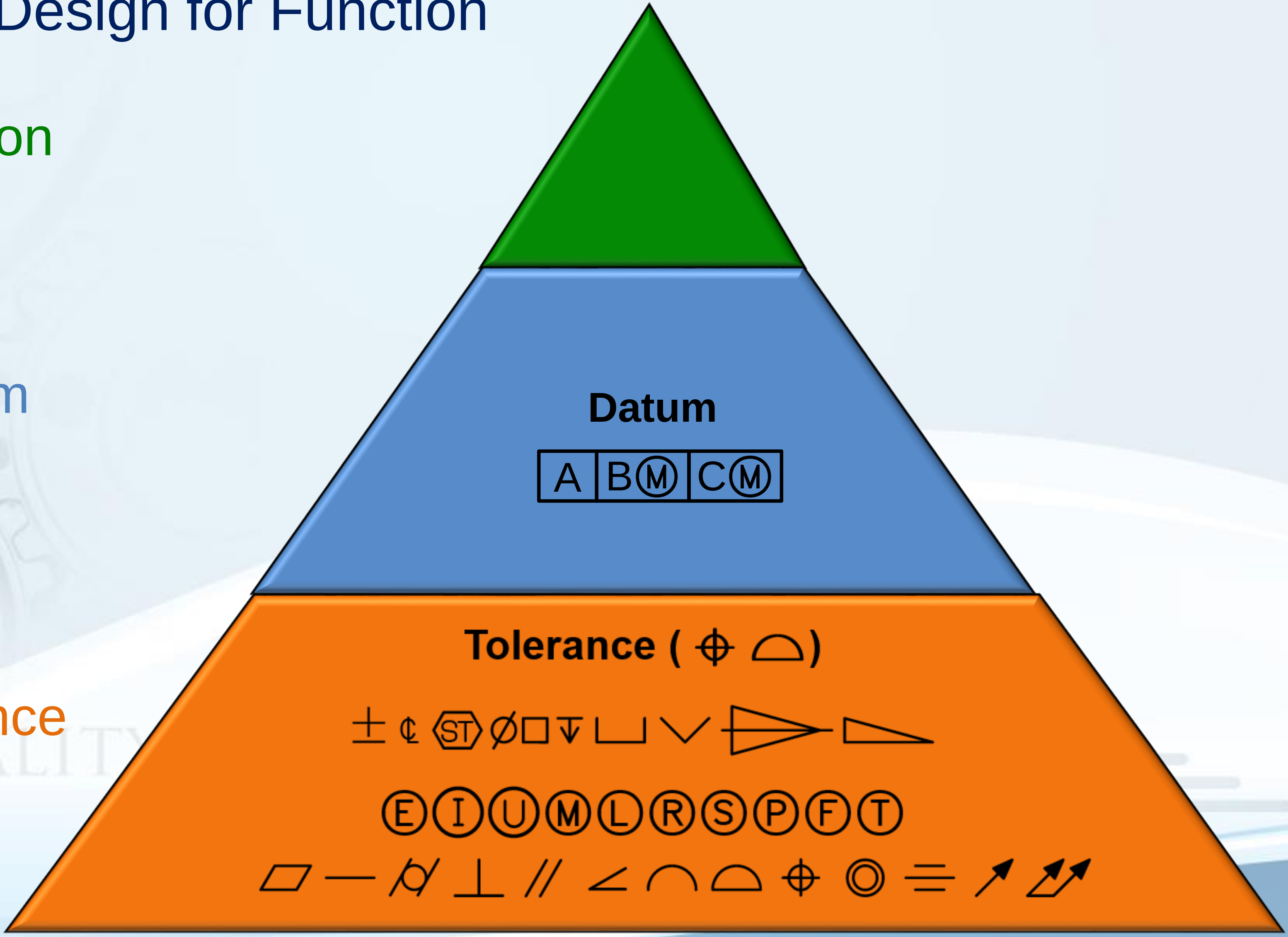
Omnex GD&T Pyramid

Design for Function

Function

Datum

Tolerance



Chapter 1: The Need for GD&T – What We Covered

Learning Objectives

You should now be able to:

- Identify GD&T callouts on a drawing.
- Explain at least five benefits using GD&T.
- Distinguish between radius and controlled radius, and why they are different.
- Name the four main types of tolerance.
- Identify the five main categories of GD&T symbols.

Chapter Agenda

- What is GD&T?
- GD&T Controls

Chapter 2

Definitions and Rules

Chapter 2: Definitions and Rules – What We Will Cover

Learning Objectives

At the end of this chapter, you will be able to:

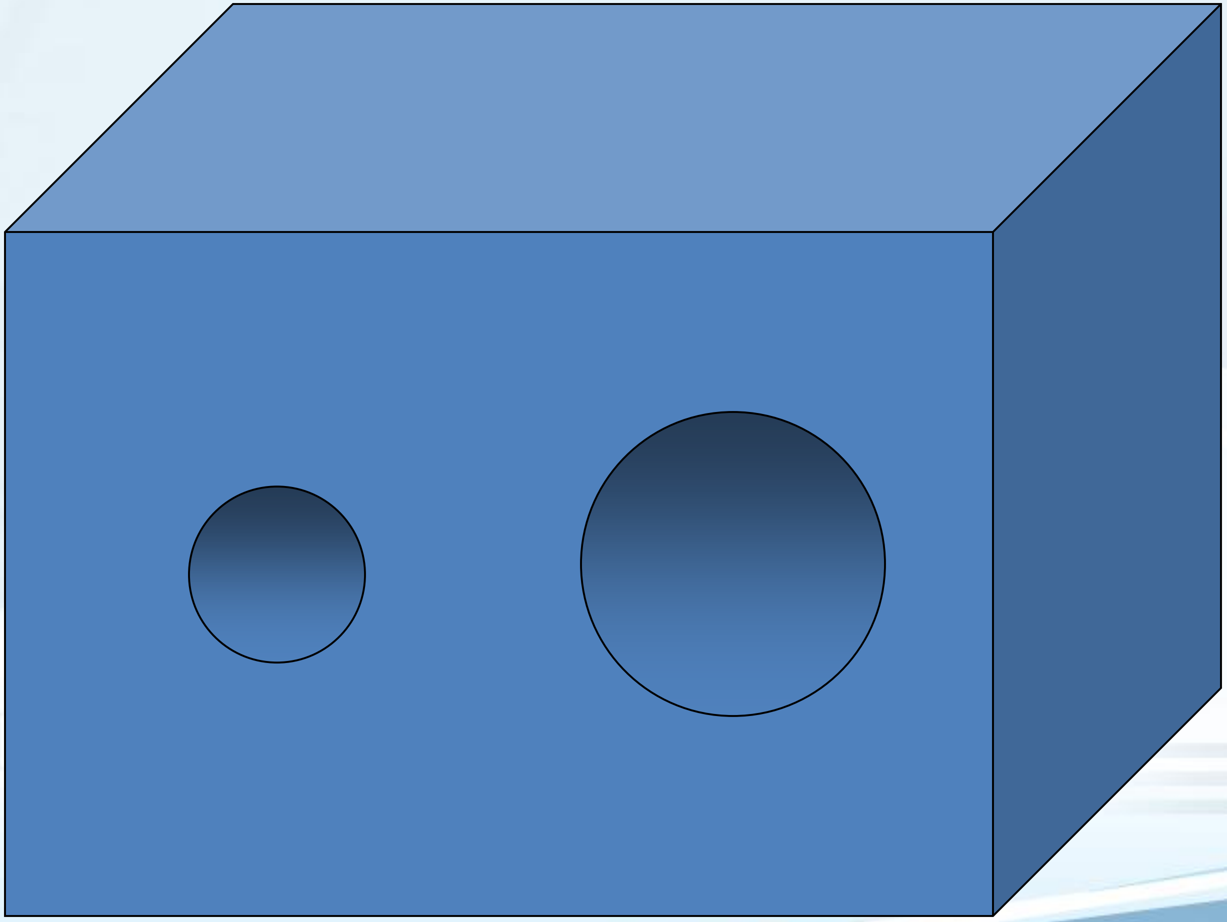
- Define feature, feature of size, actual local size, and actual mating envelope
- Determine MMC and LMC values from a given size range
- Explain Rule #1 and Rule #2
- Define virtual condition and identify VC formulas for internal and external features

Chapter Agenda

- Feature
- Feature of Size
- Actual Local Size
- Actual Mating Envelope
- Basic Dimensions
- Material Conditions
- Rule #1
- Rule #2

Definitions: Feature

- Any physical portion of a part (surface, hole, pin, etc.)



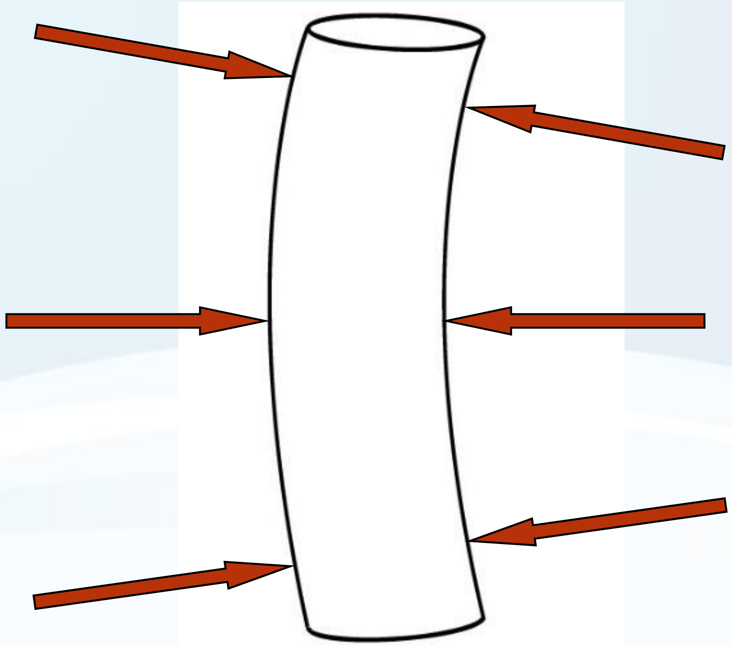
Definitions: Feature of Size (FOS):

Feature of Size (FOS): is one cylindrical or spherical, or a set of two opposed elements or opposed parallel surfaces, associated with a size dimension.

- Contains opposing elements or surfaces, can be used to establish an axis, median plane, or centerpoint
- Think of measuring with calipers / micrometer
- Typically a hole, pin, slot, etc.

Definitions: Actual Local Size

- Two-point opposed measurements
- Is the value of any individual distance at any cross section of a FOS
- Two-point measurement, measured with instrument, like a caliper or micrometer
- A FOS may have several different values of actual local size

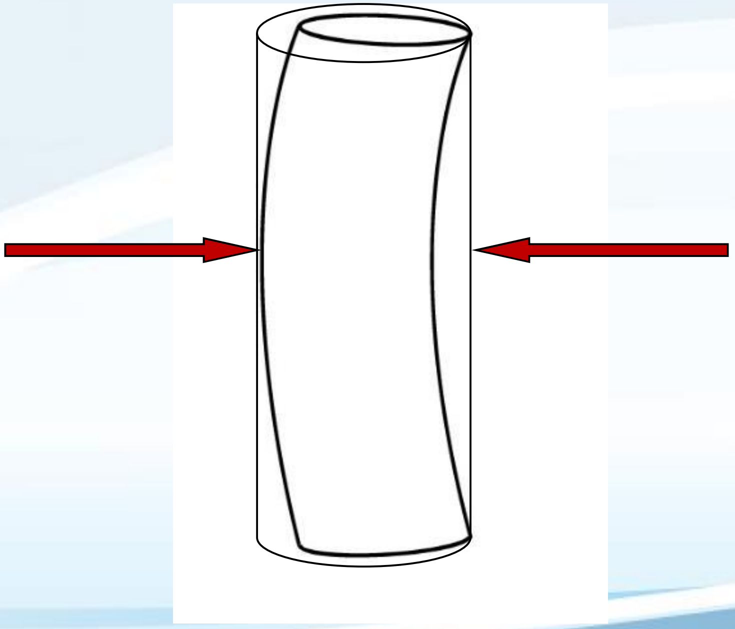


Definitions: Actual Mating Envelope

- Smallest circumscribed cylinder

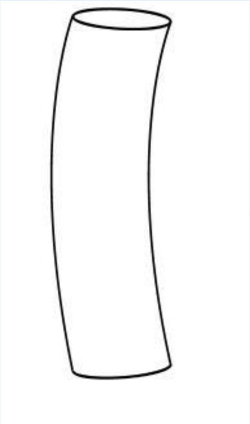
  (for an external, round feature of size)

  - 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 .
  - AME can be unrelated, as shown here, or related to appropriate datums



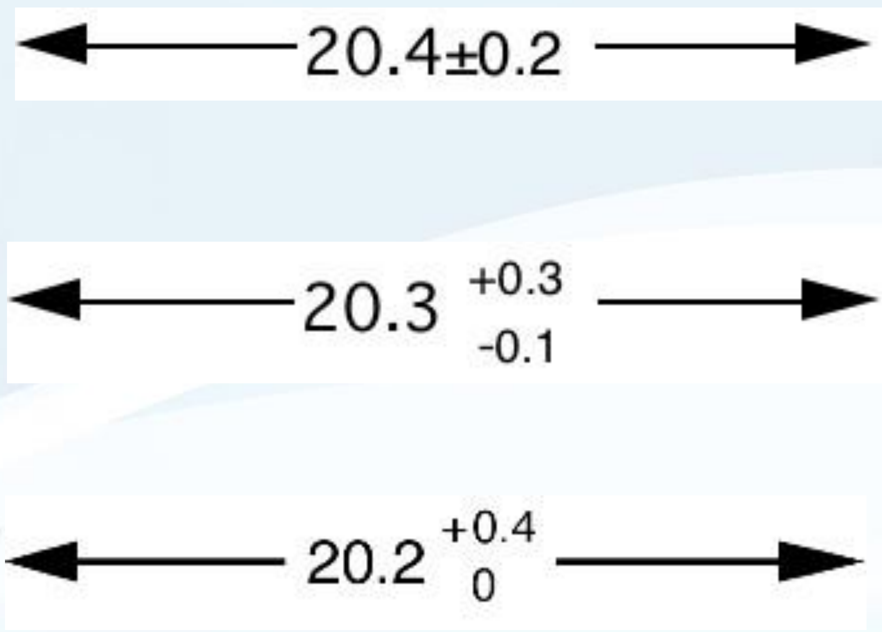
What Is Size?

- Actual Local Size
- Actual Mating Envelope
- **ASME standard: Must check both!**



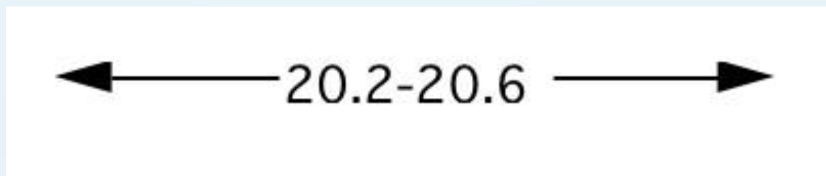
Review

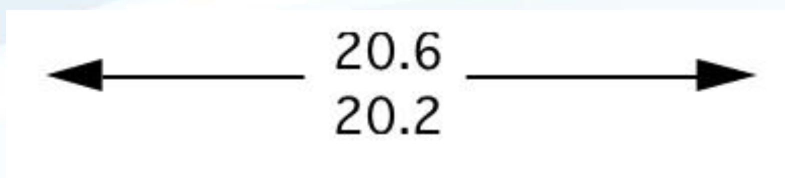
Plus/Minus Tolerancing



Review

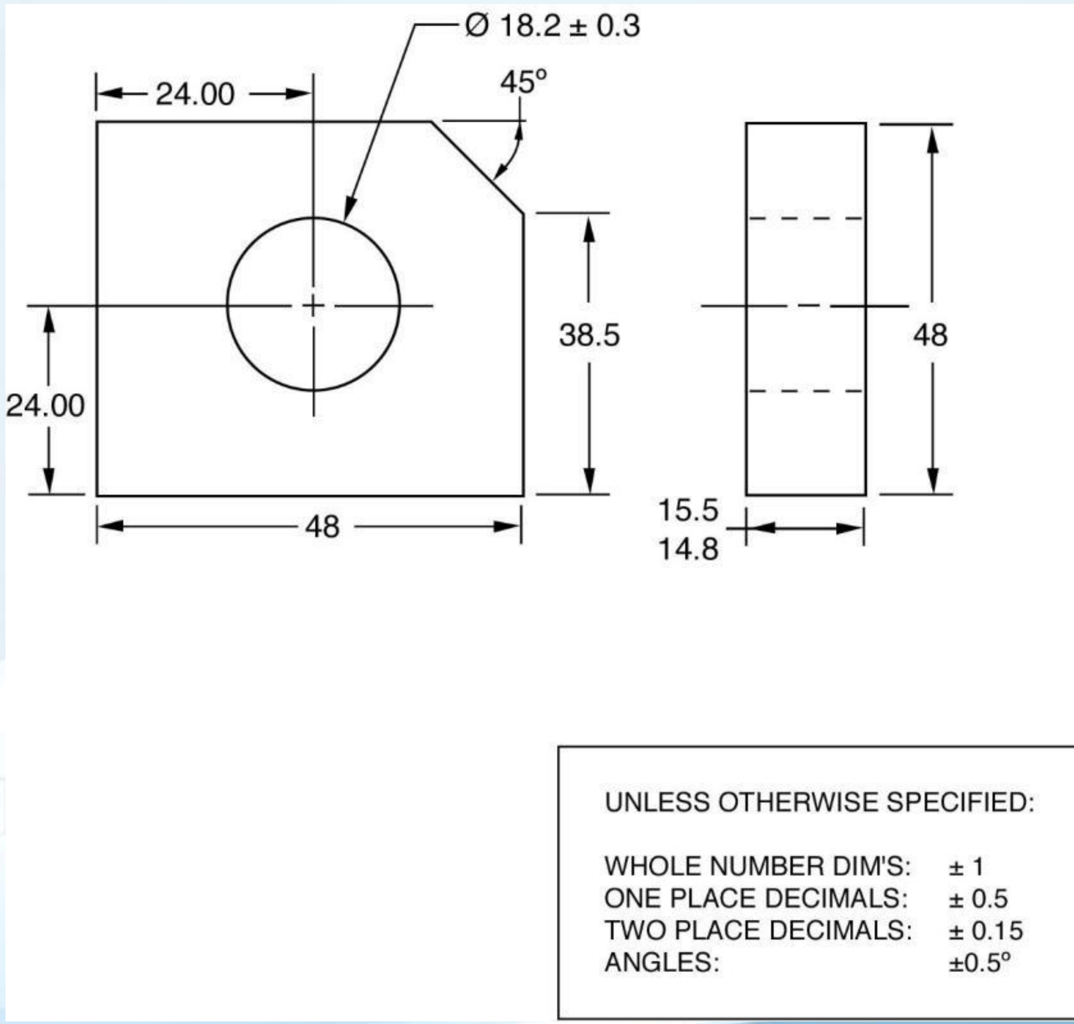
Limit Dimensioning





Review

General Print Tolerances (title block tolerance)



Rule #1

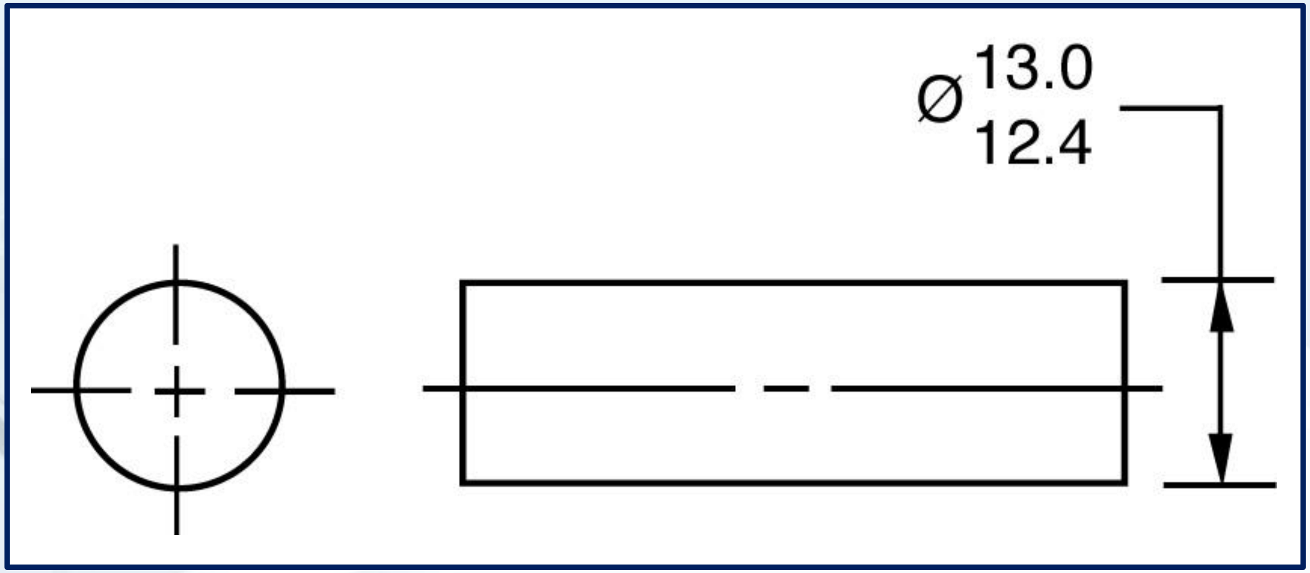
The limits of size of a feature prescribe the extent within which variations in geometric form, as well as size, are allowed.

Or, rephrased…

- Size dimensions also control form

QUALITY

Rule #1: Effect on Form



How Much Form Error?

Actual Local Size	Implied Tol. (Straightness)	Rule #1 Boundary
13.0	0	13.0
12.9	0.1	13.0
12.8	0.2	13.0
12.7	0.3	13.0
12.6	0.4	13.0
12.5	0.5	13.0
12.4	0.6	13.0

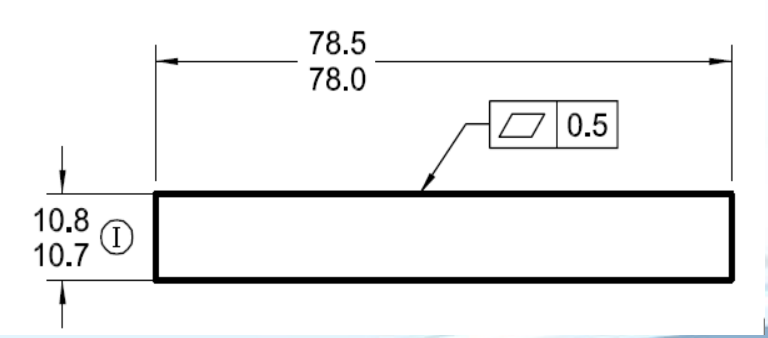
How to Override Rule #1

1. A straightness control applied to a FOS
   (such as axis and center line)

2. A flatness control applied to a FOS (median plane)
   * from ASME Y14.5M-2009

3. A special note applied to a FOS
   "PERFECT FORM AT MMC NOT REQUIRED"
   * from ASME Y14.5M-1994

4. Using the independency modifier
   * from ASME Y14.5M-2009



Rule #1 Limitations

- Does not control the location, orientation, or relationship between features of sizes.
- Does not apply to flexible parts that are not restrained.
- Does not apply to stock sizes, such as bar stock, tubing, sheet metal, or structure shapes.

“standards for these items govern the surfaces that remain in the as-furnished condition on the finished part”

Inspection and Gaging of Geometric Tolerances

- Open set-up
  - Advantages:
  - Disadvantages:

- CMM
  - Advantages:
  - Disadvantages:

- Functional gages
  - Advantages:
  - Disadvantages:

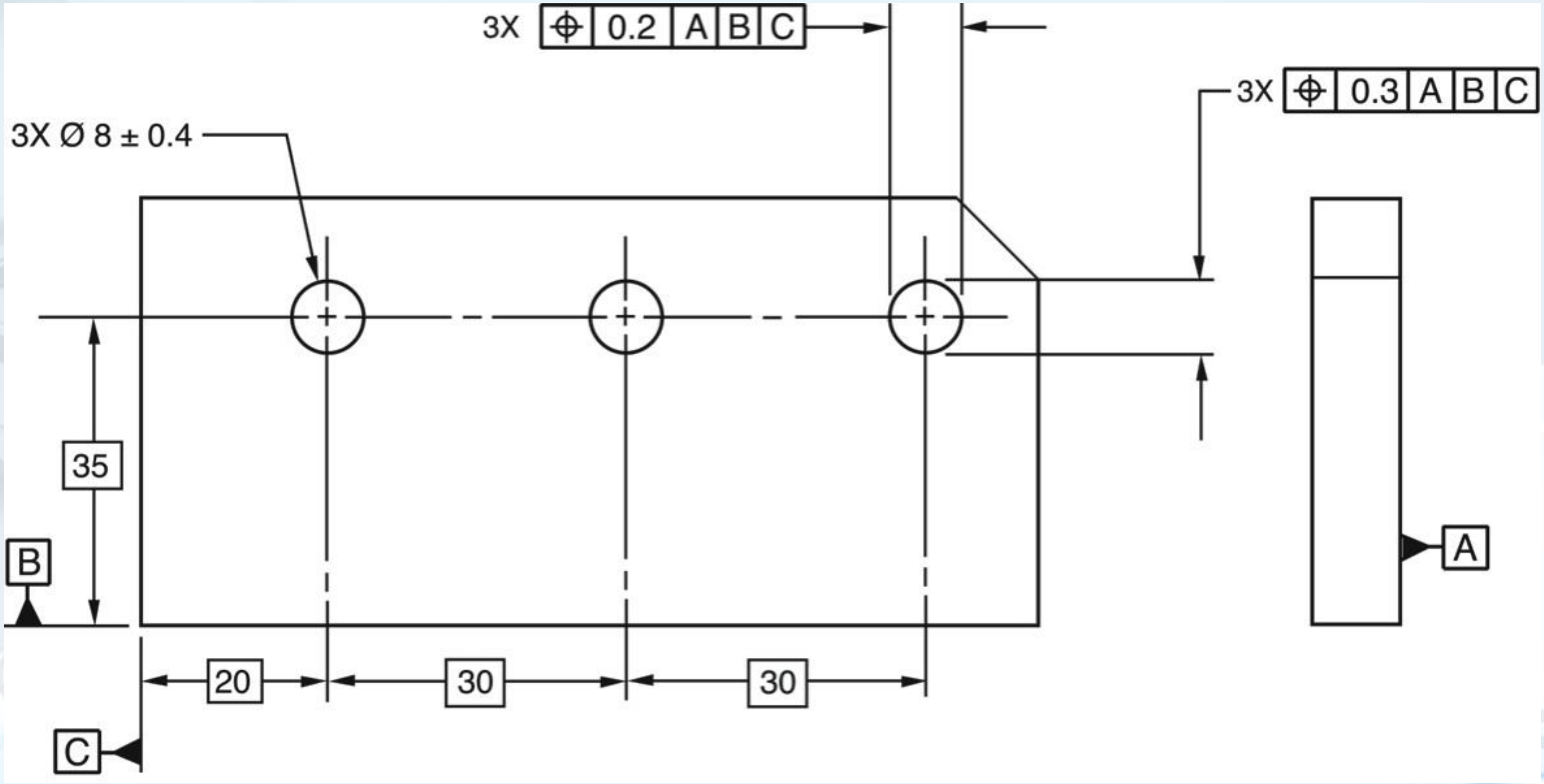
Definitions: Basic Dimensions

- Theoretically exact locations, angles and profiles

No direct tolerance, but look for a feature control frame!

Basic Dimensions

- They are the “basis” for a GD&T zone



Basic Dimensions

- Define part features (must be accompanied by geometric tolerance)
- Define gage information (do not have a tolerance shown on the print, but gage-makers' tolerances do apply)

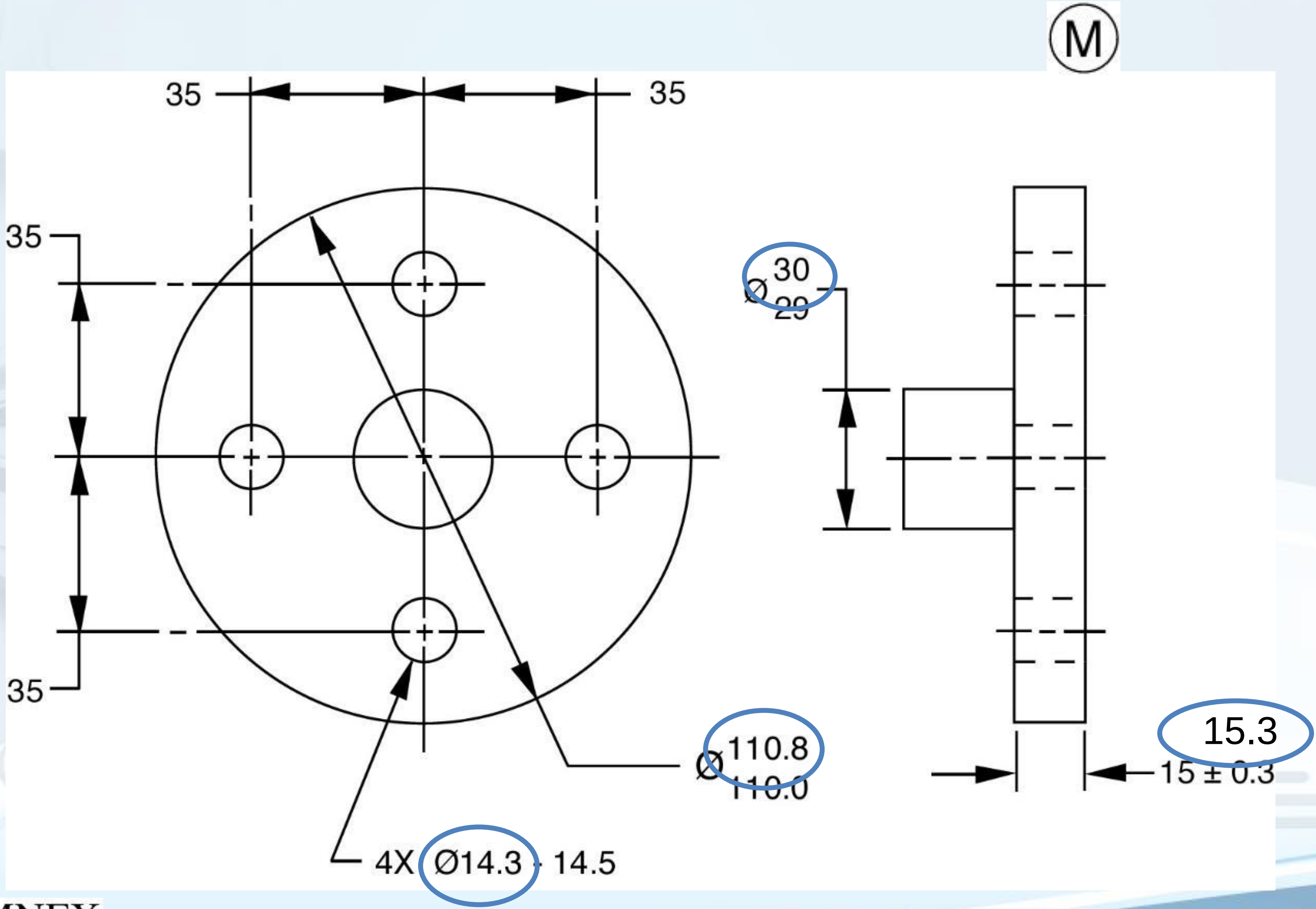
Title block tolerances never apply to basic dimensions

Material Conditions

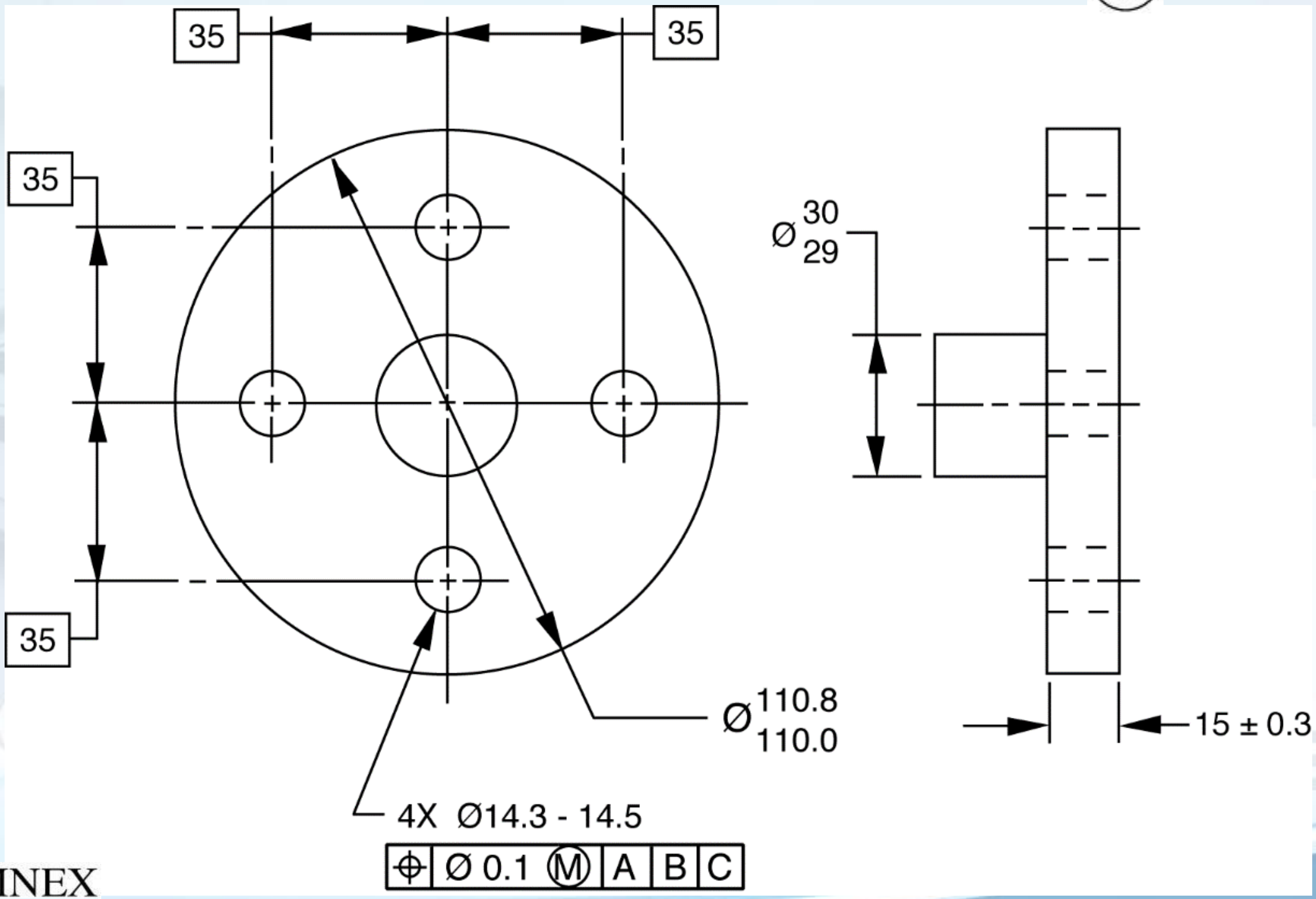
- **Maximum Material Condition (MMC):** MMC is the condition in which a feature of size contains the maximum amount of material everywhere within the stated limits of size, such as largest shaft diameter or smallest hole diameter.

- **Least Material Condition (LMC):** LMC is the condition in which a feature of size contains the least amount of material everywhere within the stated limits of size, such as smallest shaft diameter or largest hole diameter.

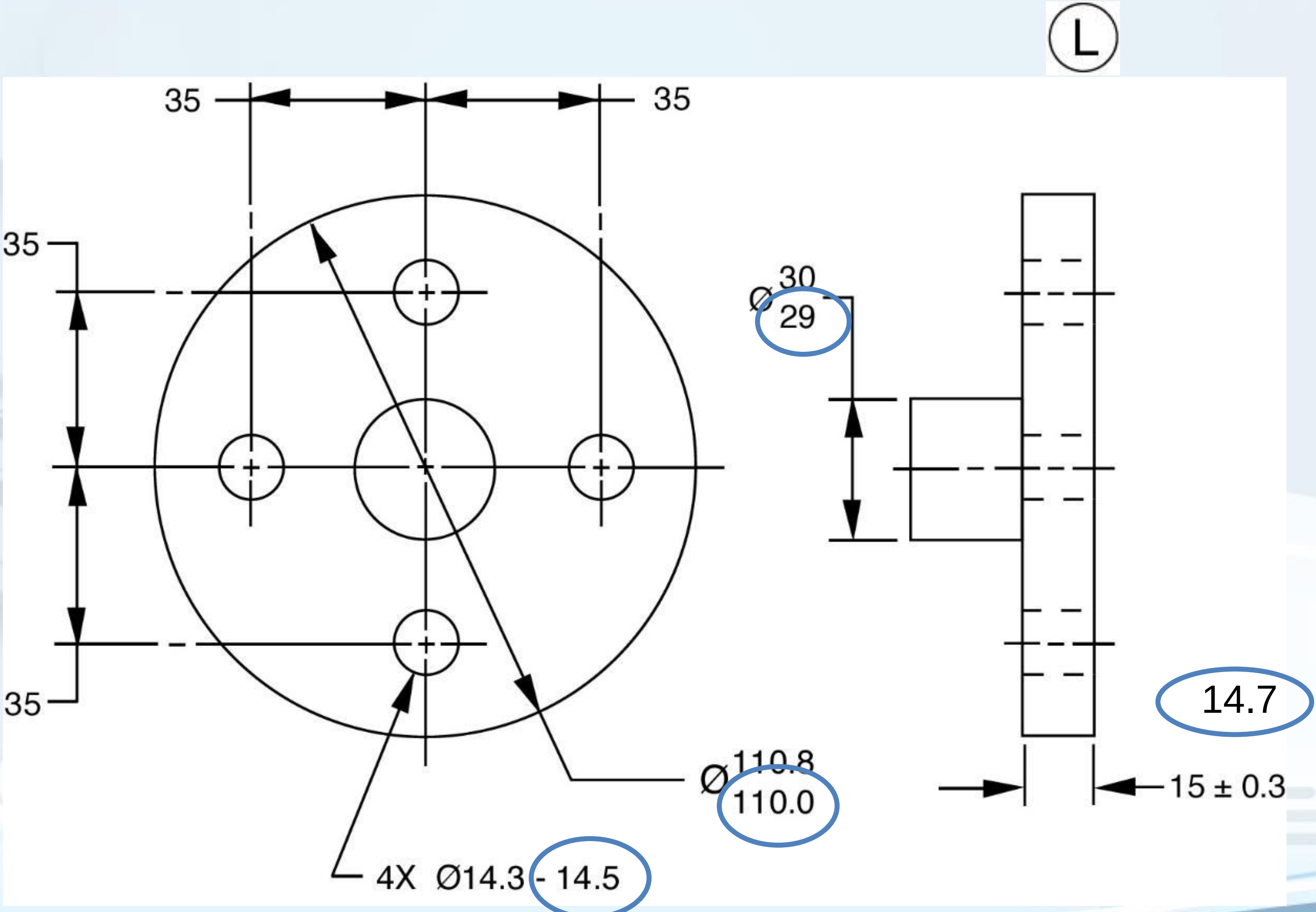
Maximum Material Condition (MMC)



Maximum Material Condition (MMC)



Least Material Condition (LMC)



Bonus Tolerances

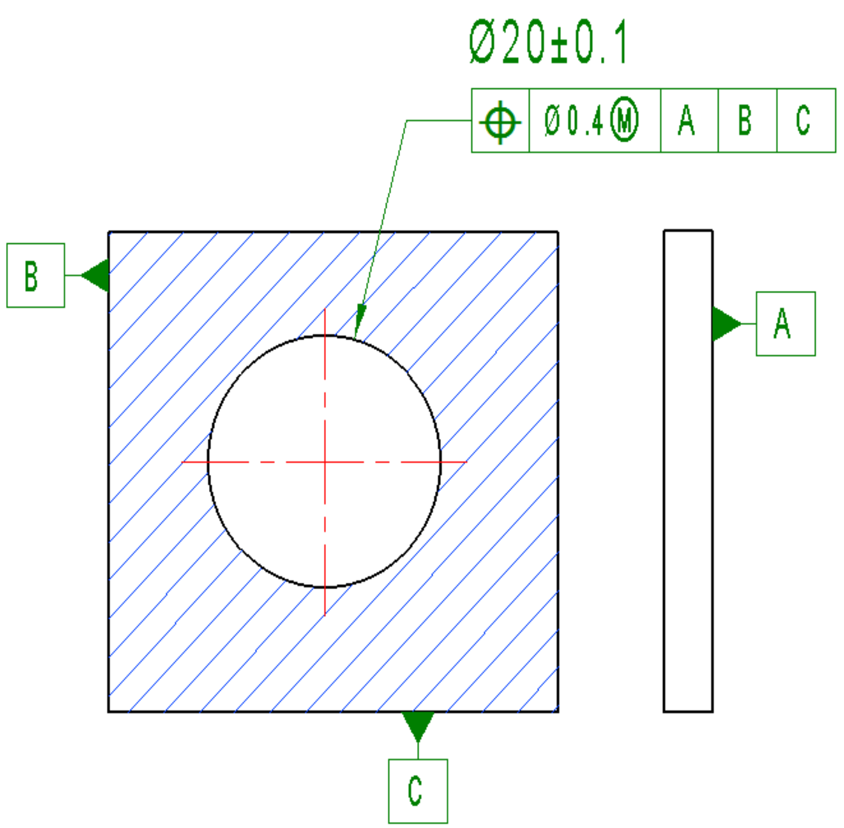
Bonus Tolerance is an additional tolerance for a geometric control

- A bonus tolerance is permissible if the MMC/LMC modifier is applied in a geometric tolerance to a FOS.
- If MMC is applied in a geometric tolerance, it means the geometric tolerance applies when the FOS is at its maximum material condition. If the FOS departure from MMC to LMC, there is an increase in geometric tolerance equal to the same amount of departure.
- When the MMC modifier is used, it means that the geometric characteristic can be verified with a fixed, functional gage.
- A functional gage is a gage that is built to a fixed dimension (the virtual condition) of a part feature.

Bonus Tolerances

Hole Bonus Tolerance Calculation (MMC)

Bonus tolerance will NOT affect assembly

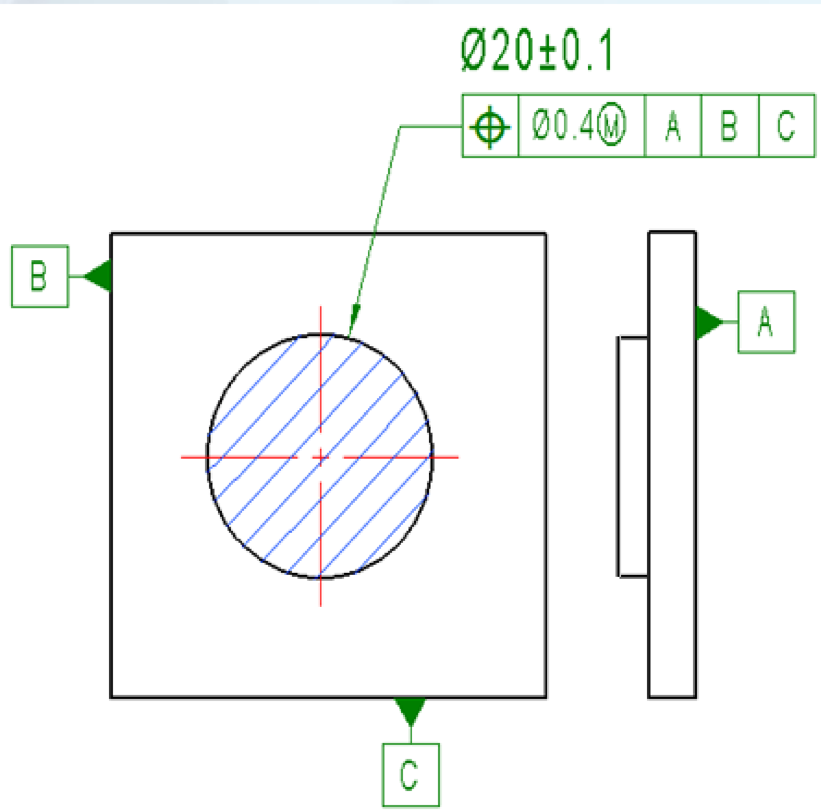


Feature AME	Stated Tolerance	Bonus Tolerance	Total Geo. Tolerance
Ø19.9 MMC	0.4	0	0.4
Ø20.0	0.4	0.1	0.5
Ø20.1 LMC	0.4	0.2	0.6

Bonus Tolerances

Shaft Bonus Tolerance Calculation (MMC)

Bonus tolerance will NOT affect assembly



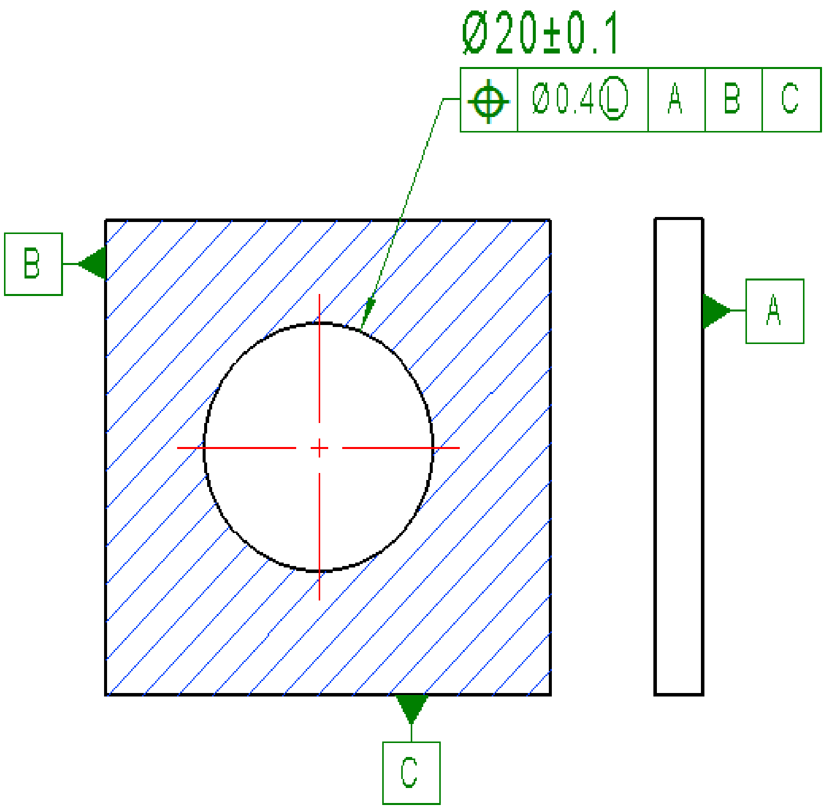
Feature AME	Stated Tolerance	Bonus Tolerance	Total Geo. Tolerance
Ø20.1 MMC	0.4	0	0.4
Ø20.0	0.4	0.1	0.5
Ø19.9 LMC	0.4	0.2	0.6

OMNEX

Bonus Tolerances

Hole Bonus Tolerance Calculation (LMC)

Bonus tolerance will NOT affect wall thickness



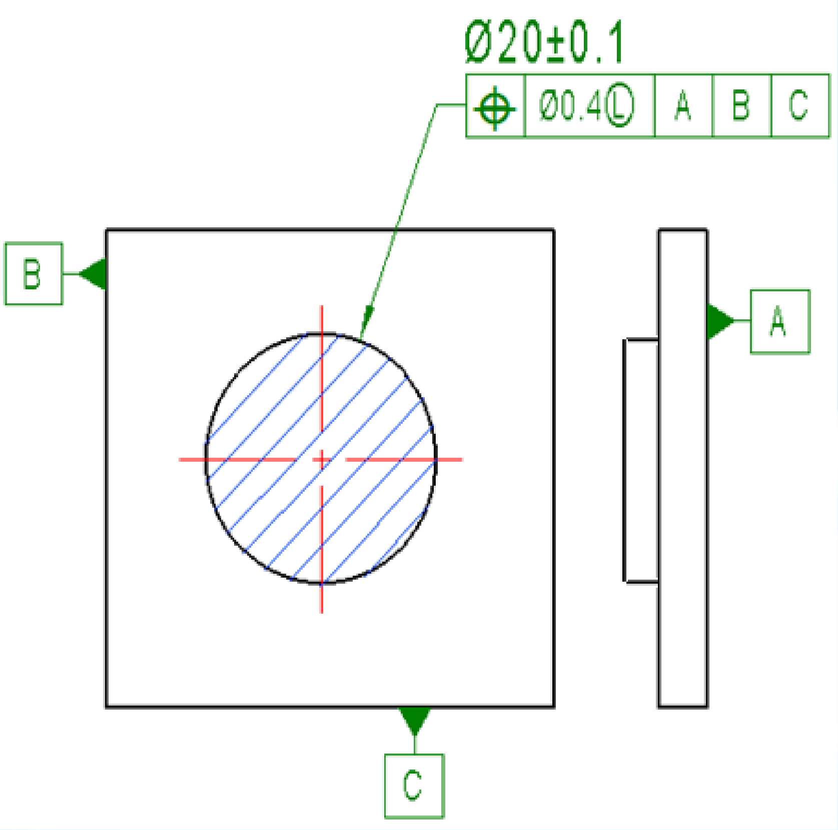
Feature AME	Stated Tolerance	Bonus Tolerance	Total Geo. Tolerance
Ø20.1 LMC	0.4	0	0.4
Ø20.0	0.4	0.1	0.5
Ø19.9 MMC	0.4	0.2	0.6

OMNEX

Bonus Tolerances

Shaft Bonus Tolerance Calculation (LMC)

Bonus tolerance will NOT affect wall thickness



Feature AME	Stated Tolerance	Bonus Tolerance	Total Geo. Tolerance
Ø19.9 LMC	0.4	0	0.4
Ø20.0	0.4	0.1	0.5
Ø20.1 MMC	0.4	0.2	0.6

Rule #2

Without an MMC or LMC modifier, always assume RFS by default.

- An older practice was to use a modifier for RFS when applied to position tolerances Ⓢ

Always One of These Three

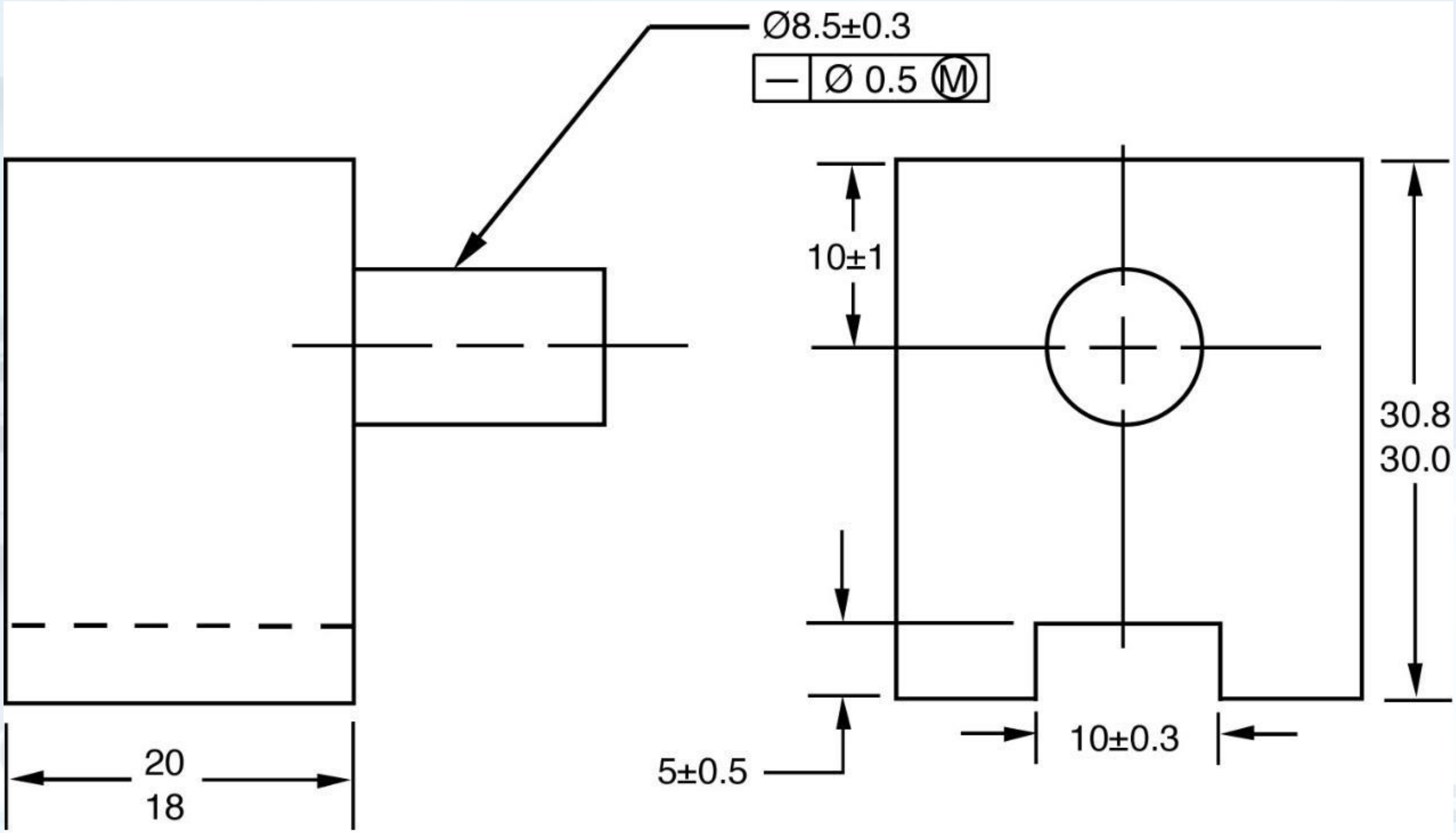
Material Condition	Common Usage	Comments
Ⓜ	Assembly (clearance fit)	• Very common modifier • Allows bonus tolerance • Always ensures clearance • Allows functional gaging
Ⓛ	Maintain minimum wall thickness or machine stock	• Least common modifier • Allows bonus tolerance • Opposite effect of MMC • Requires variable gaging
RFS (No modifier)	Centering/alignment; symmetrical relationships	• Most expensive condition • No bonus tolerance • Implied by Rule #2 • Requires variable gaging

Virtual Condition

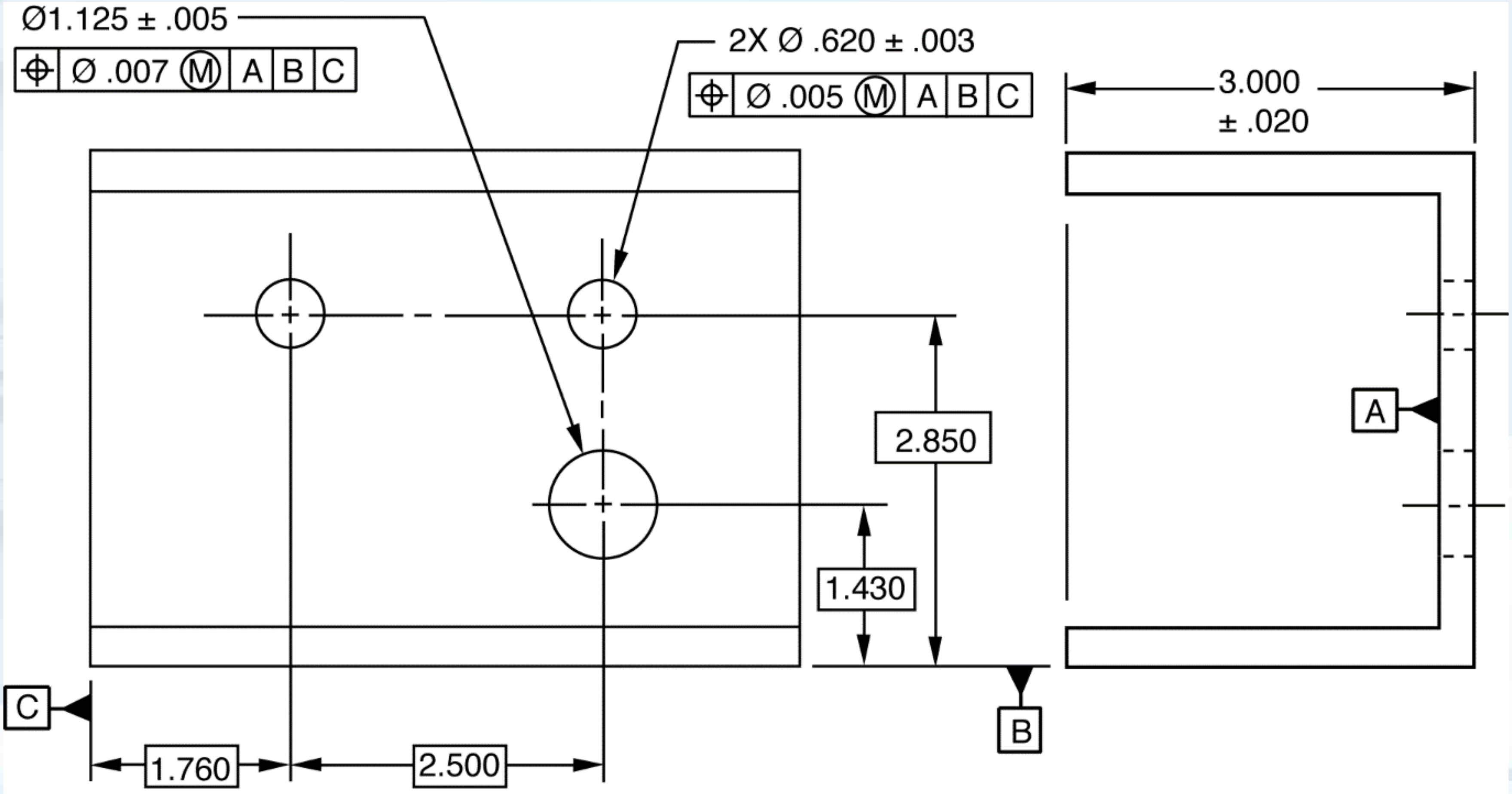
Virtual Condition (VC): A constant boundary generated by the collective effects of a feature of size at MMC (or LMC) and geometric tolerance for that material condition. VC includes effects of the size, form, orientation, and location applied to the FOS. The VC boundary is related to any datums that are referenced in the geometric tolerance.

- For an Internal Feature:
  - A constant value equal to its MMC size MINUS its applicable geometric tolerance
- For an External Feature:
  - A constant value equal to its MMC size PLUS its applicable geometric tolerance

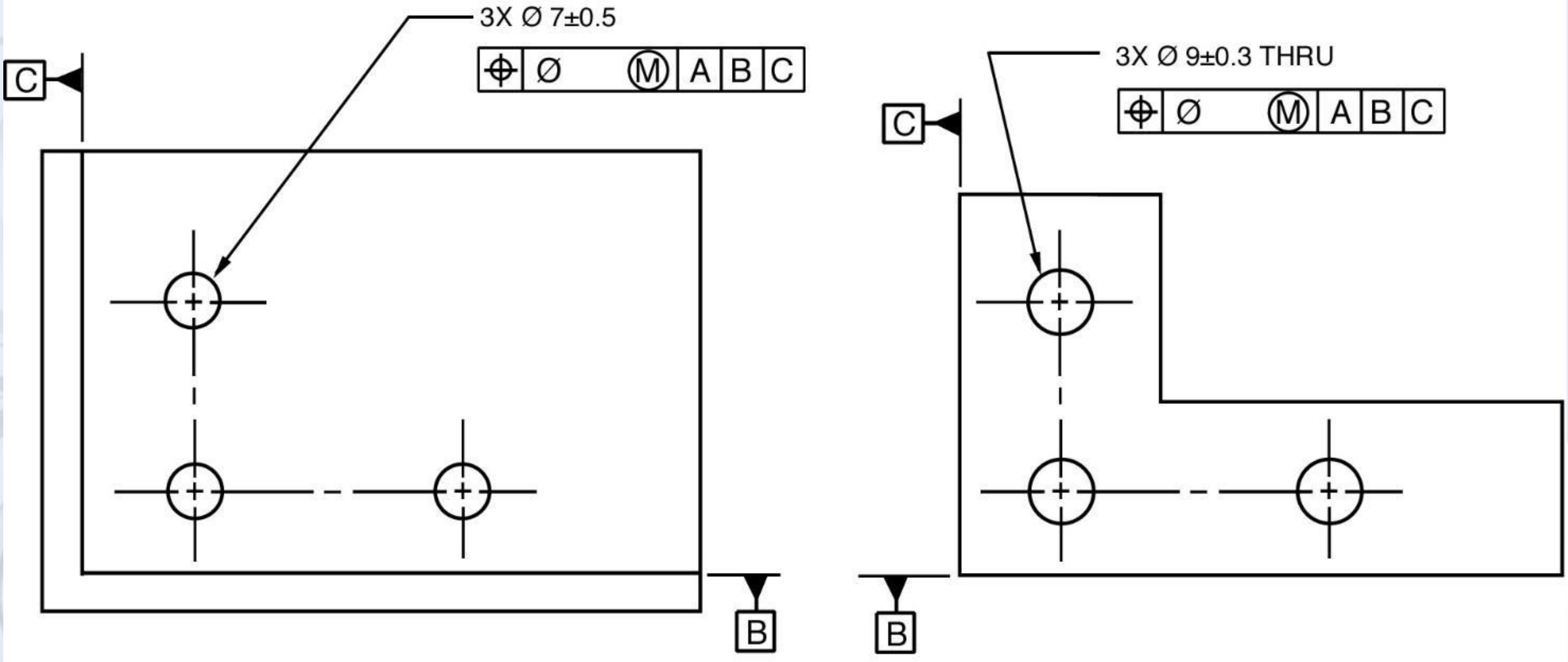
Calculate: Virtual Condition (pin)



Calculate: Virtual Condition (of the large hole)



Virtual Condition – Calculate Tolerances for Each Part



14 GD&T Symbols

CATEGORY	*CHARACTERISTIC*	*SYMBOL*
FORM	STRAIGHTNESS	⏤
	FLATNESS	⏥
	CIRCULARITY	○
	CYLINDRICITY	⌭
PROFILE	PROFILE OF A LINE	⌒
	PROFILE OF A SURFACE	⌓
ORIENTATION	PERPENDICULARITY	⊥
	PARALLELISM	//
	ANGULARITY	∠
LOCATION	POSITION	⌖
	CONCENTRICITY	◎
	SYMMETRY	⌯
RUNOUT	CIRCULAR RUNOUT	↗
	TOTAL RUNOUT	⌰

Chapter 2: Definitions and Rules – What We Covered

Learning Objectives

You should now be able to:

- Define feature, feature of size, actual local size, and actual mating envelope .
- Determine MMC and LMC values from a given size range
- Explain Rule #1 and Rule #2
- Define virtual condition and identify VC formulas for internal and external features

Chapter Agenda

- Feature
- Feature of Size
- Actual Local Size
- Actual Mating Envelope
- Basic Dimensions
- Material Conditions
- Rule #1
- Rule #2

Chapter 3

Form

Flatness
Straightness
Circularity
Cylindricity

Chapter 3: Form – What We Will Cover

Learning Objectives

At the end of this chapter, you will be able to:

- Describe the tolerance zone for flatness, and how it is inspected
- Describe the tolerance zone for straightness, and how it is inspected
- Describe the tolerance zone for circularity, and how it is inspected
- Describe the tolerance zone for cylindricity, and how it is inspected
- Name the two form tolerances that may be used on features of size
- Explain how bonus tolerance is calculated

Chapter Agenda

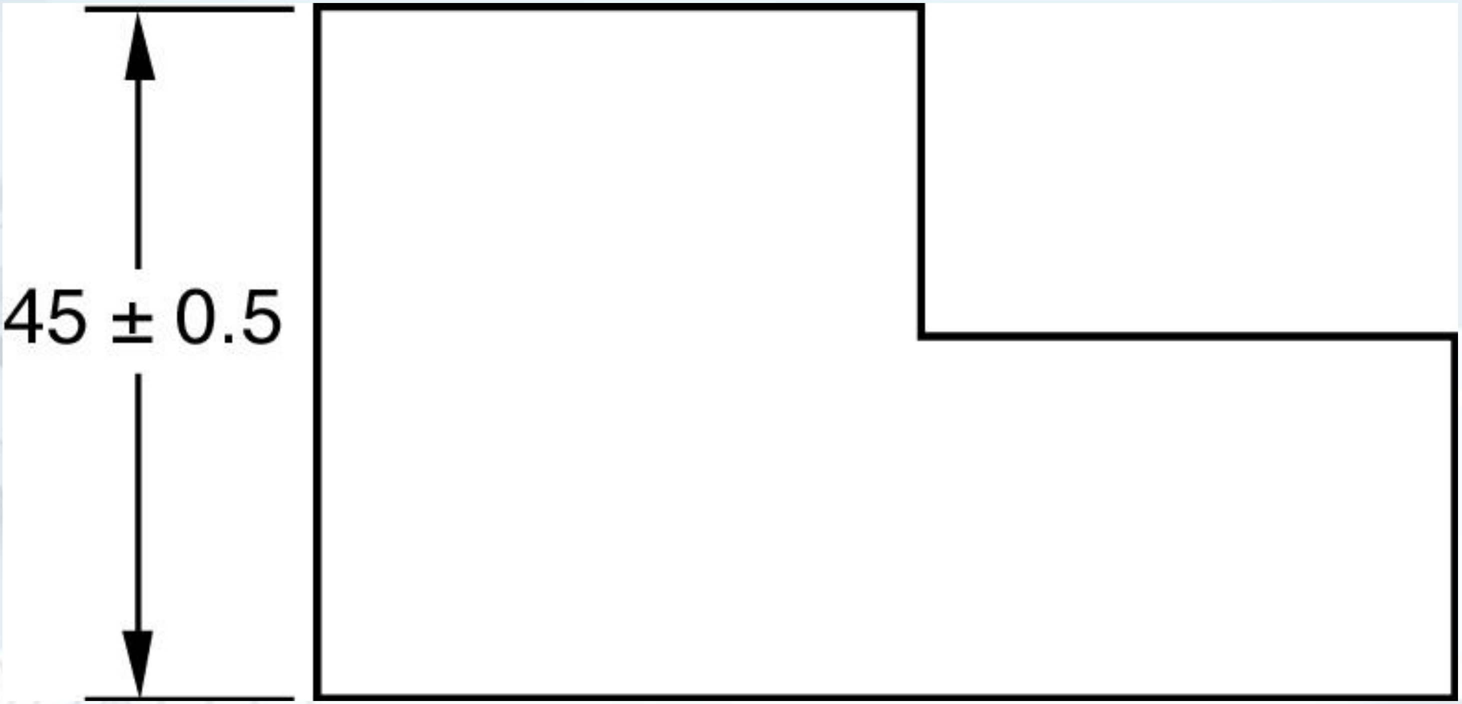
- Flatness
- Straightness
- Circularity
- Cylindricity

Flatness Definition

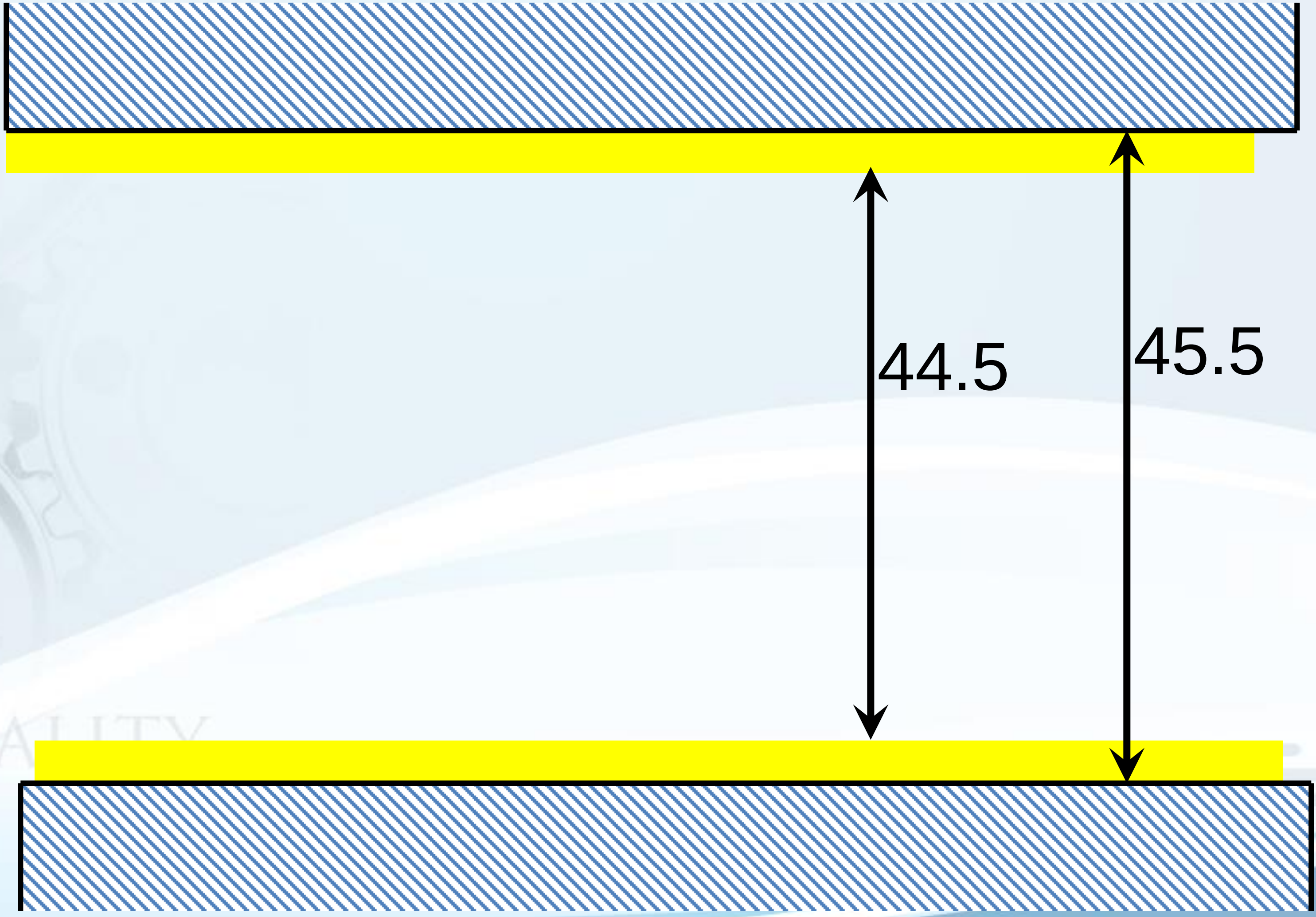
- Flatness is the condition of a surface or derived median plane having all elements within the same plane.

- By default, it is controlled by Rule #1, but it can be separated out to a feature control frame using the flatness symbol.

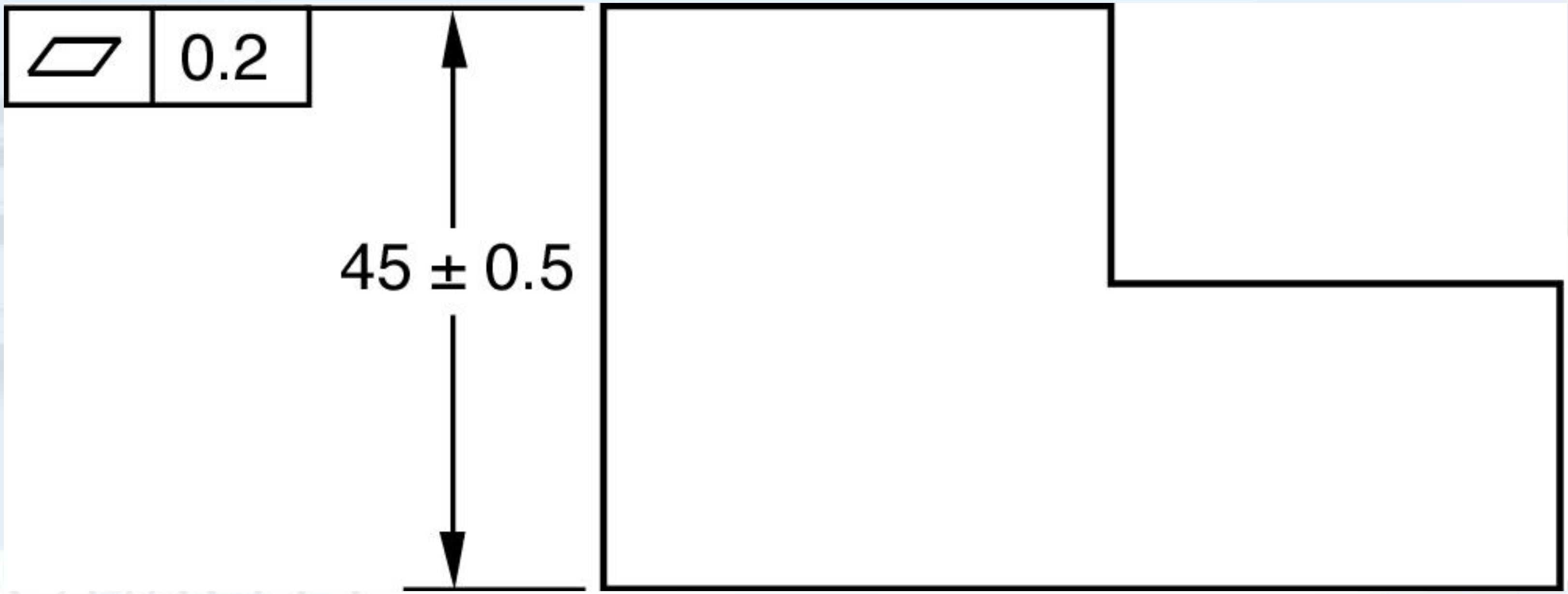
Flatness:
Rule #1 Size Boundary



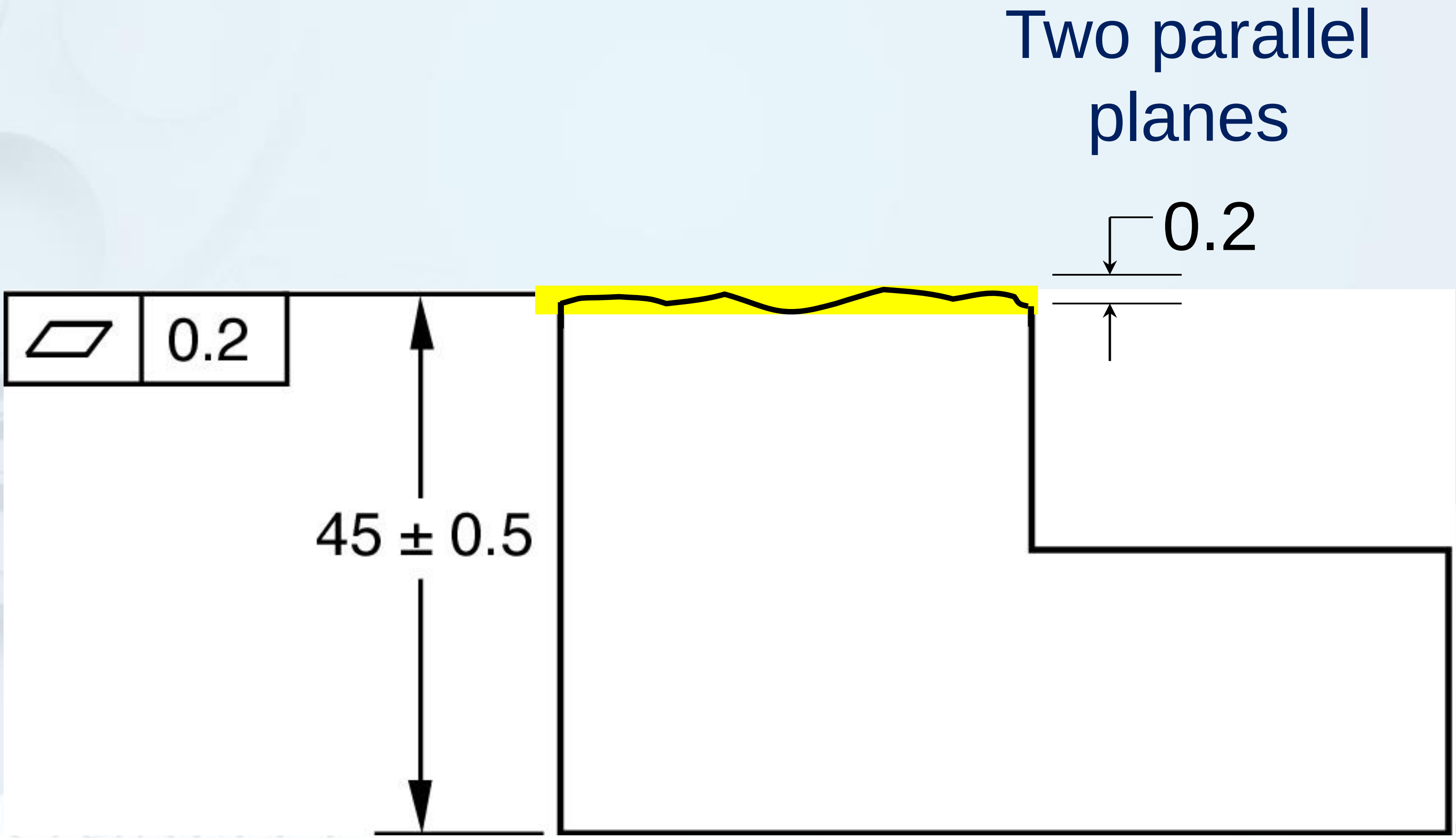
Flatness:
Rule #1 Size Boundary



Flatness

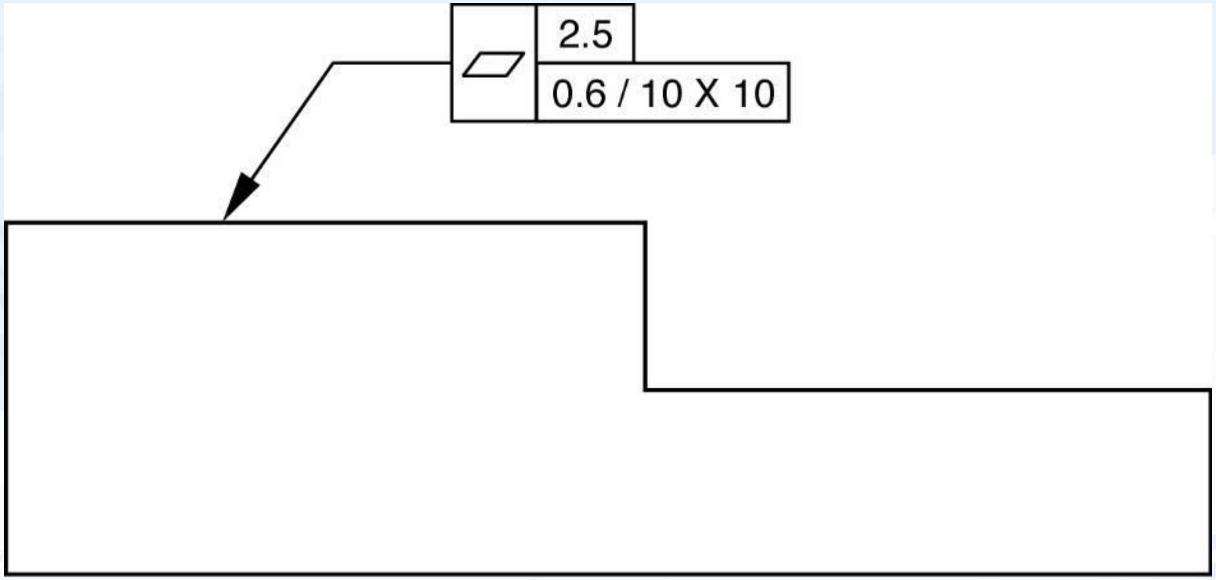


Flatness



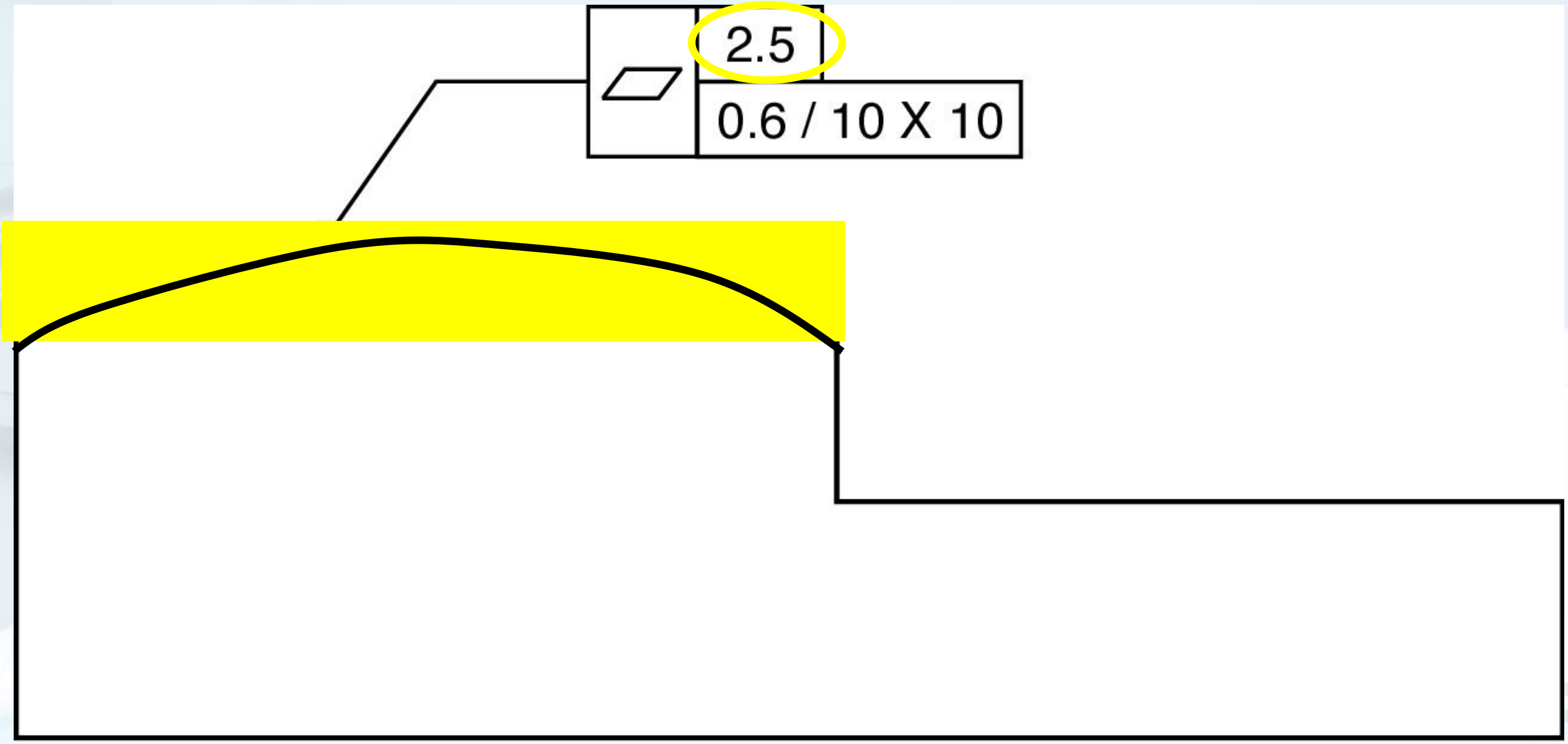
Flatness: Unit Basis

- 2.5 over entire surface
- 0.6 over any 10X10 area



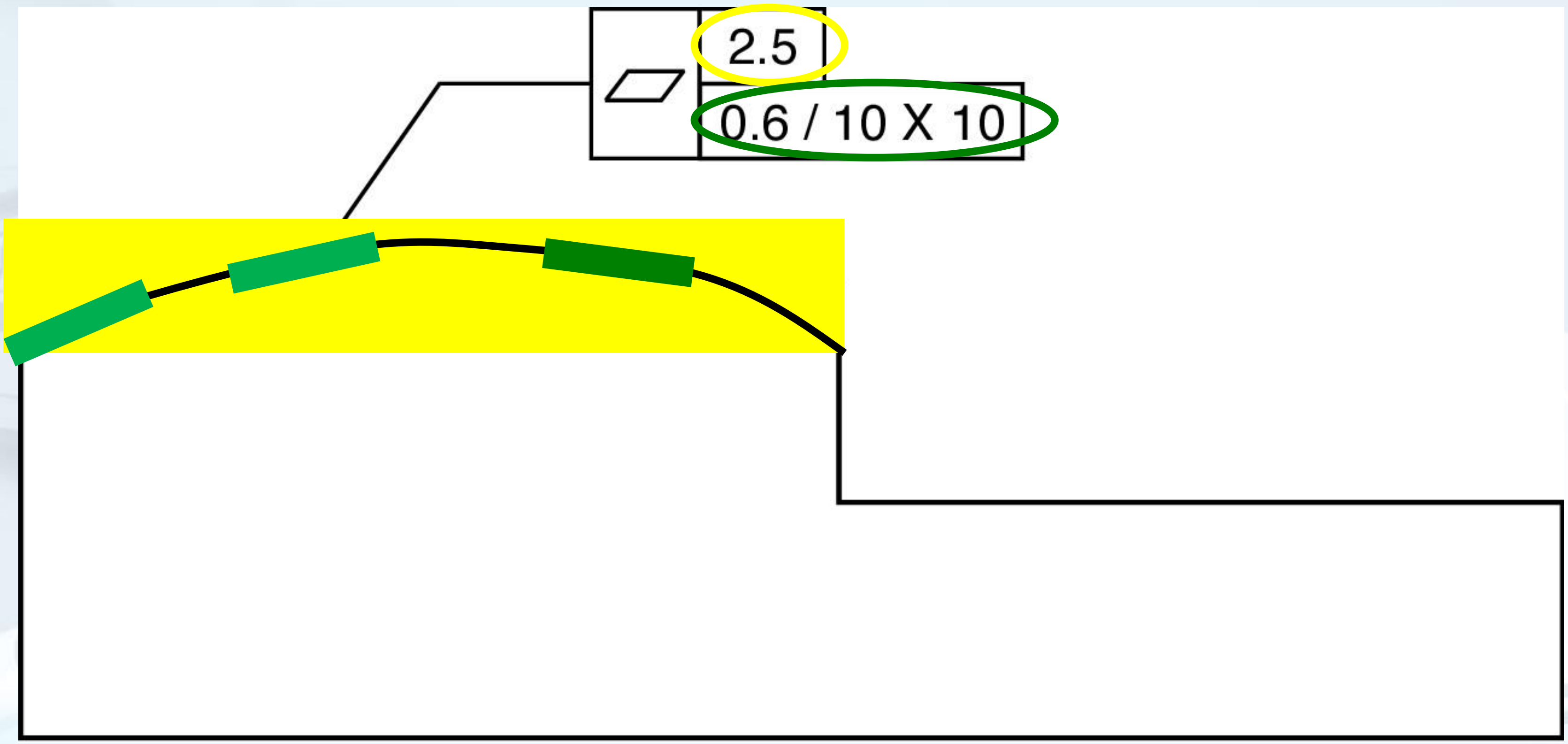
Flatness: Per Unit Basis

- 2.5 over entire surface limits overall flatness



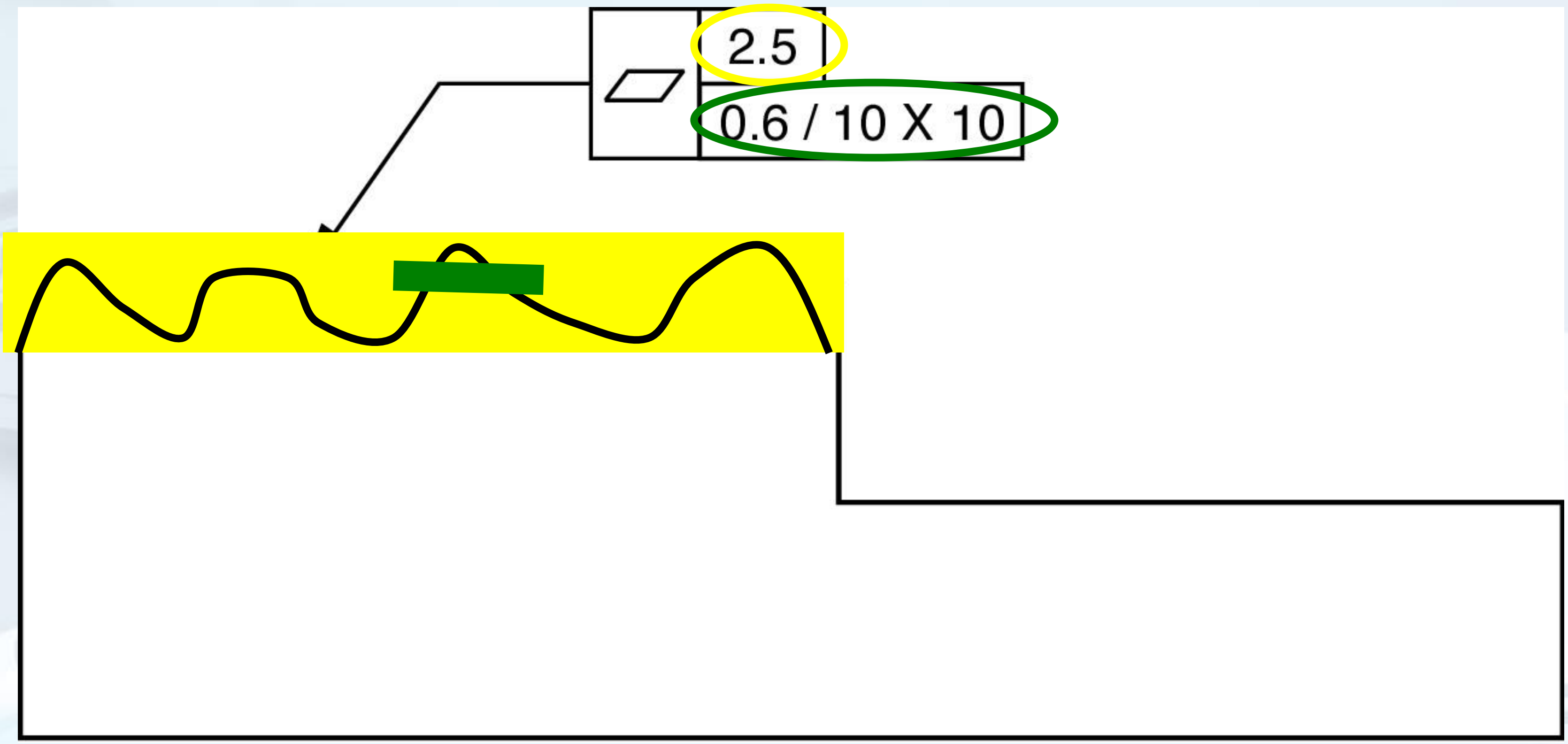
Flatness: Per Unit Basis

- 0.6 over each 10X10 area allows gradual surface changes



Flatness: Per Unit Basis

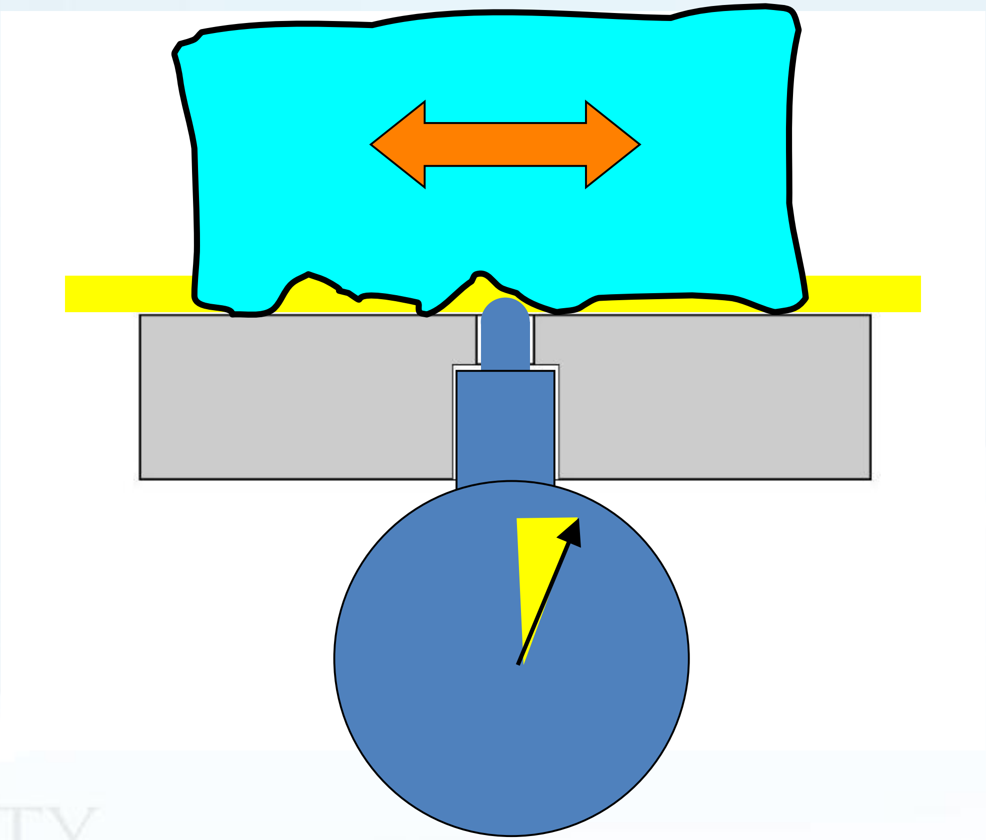
- Prevents abrupt surface changes



Things to Remember About Flatness

- Never references a datum
- Tolerance is not a plus/minus, it's the total distance between two parallel planes
- Surface flatness tolerance must be less than the size tolerance
- Usually applied to a surface, but in 2009 is it also allowed on a FOS (i.e., a center plane)
- No size modifiers are to be used for surface flatness
- It is different than surface roughness/surface finish
- Can be applied on a per-unit basis

Measuring Flatness

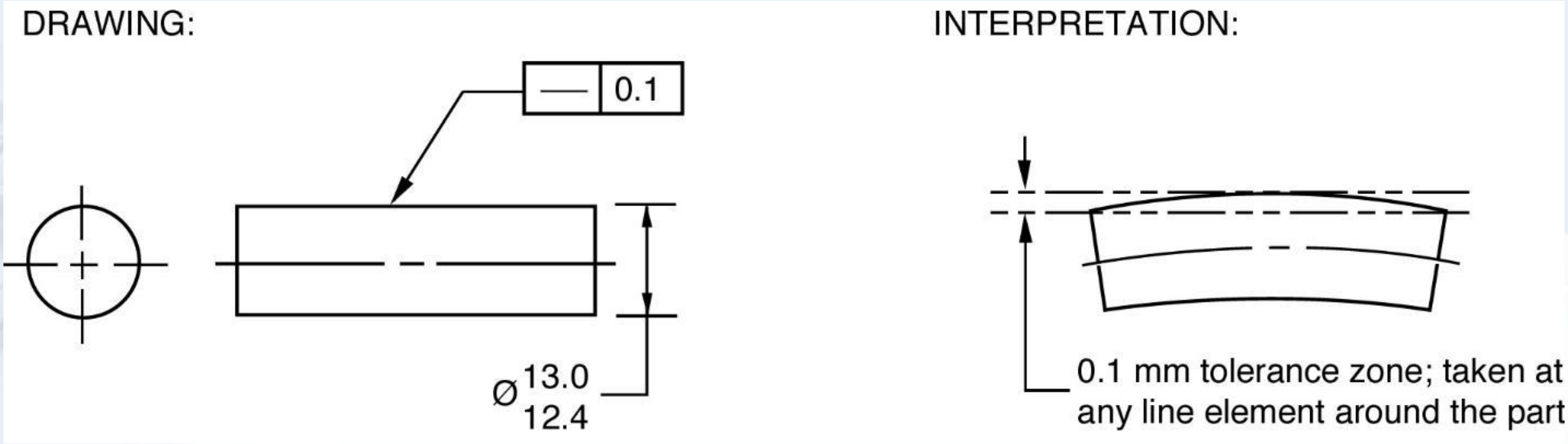


Straightness Definition

- Straightness is the condition of a surface or axis being a straight line.

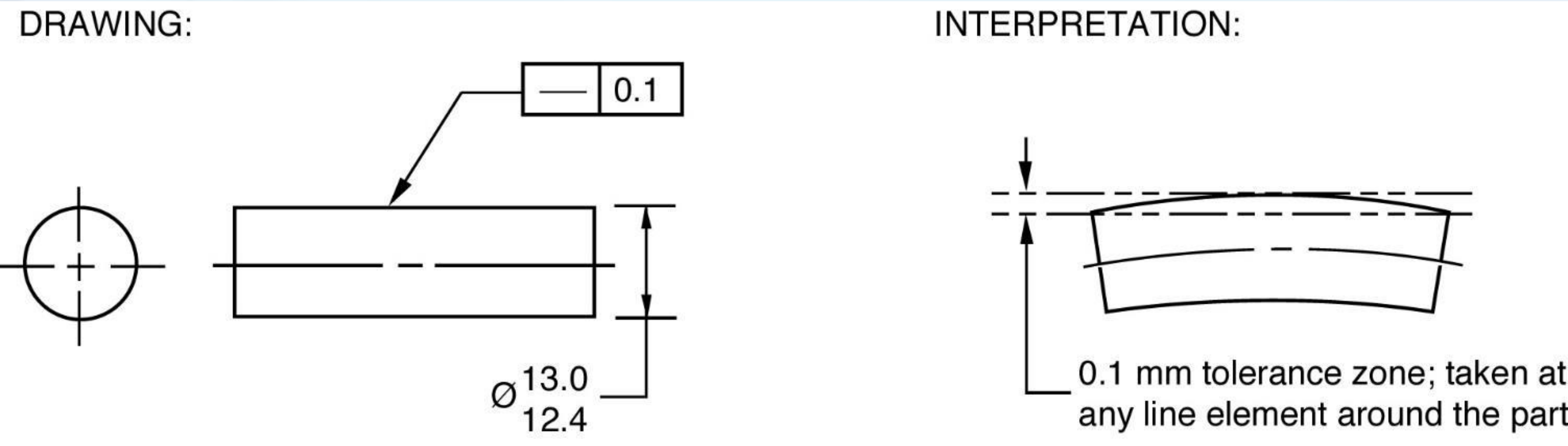
- By default, it is controlled by Rule #1, but it can be separated out to a feature control frame using the straightness symbol.

Straightness



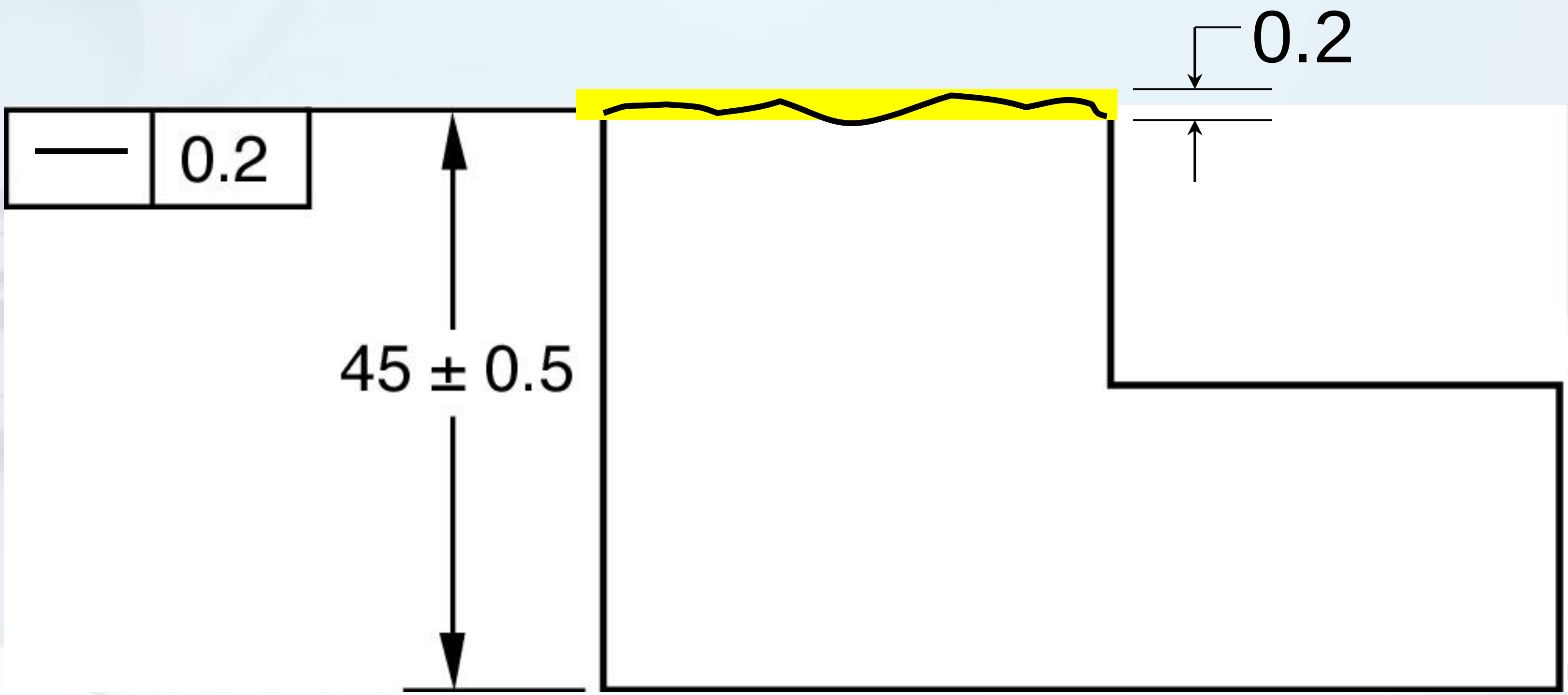
Straightness

- Two separate checks:
  - Size (Rule #1 go/no-go check still applies)
  - Form (additional requirement to apply to each surface element)



Straightness: A Subset of Flatness

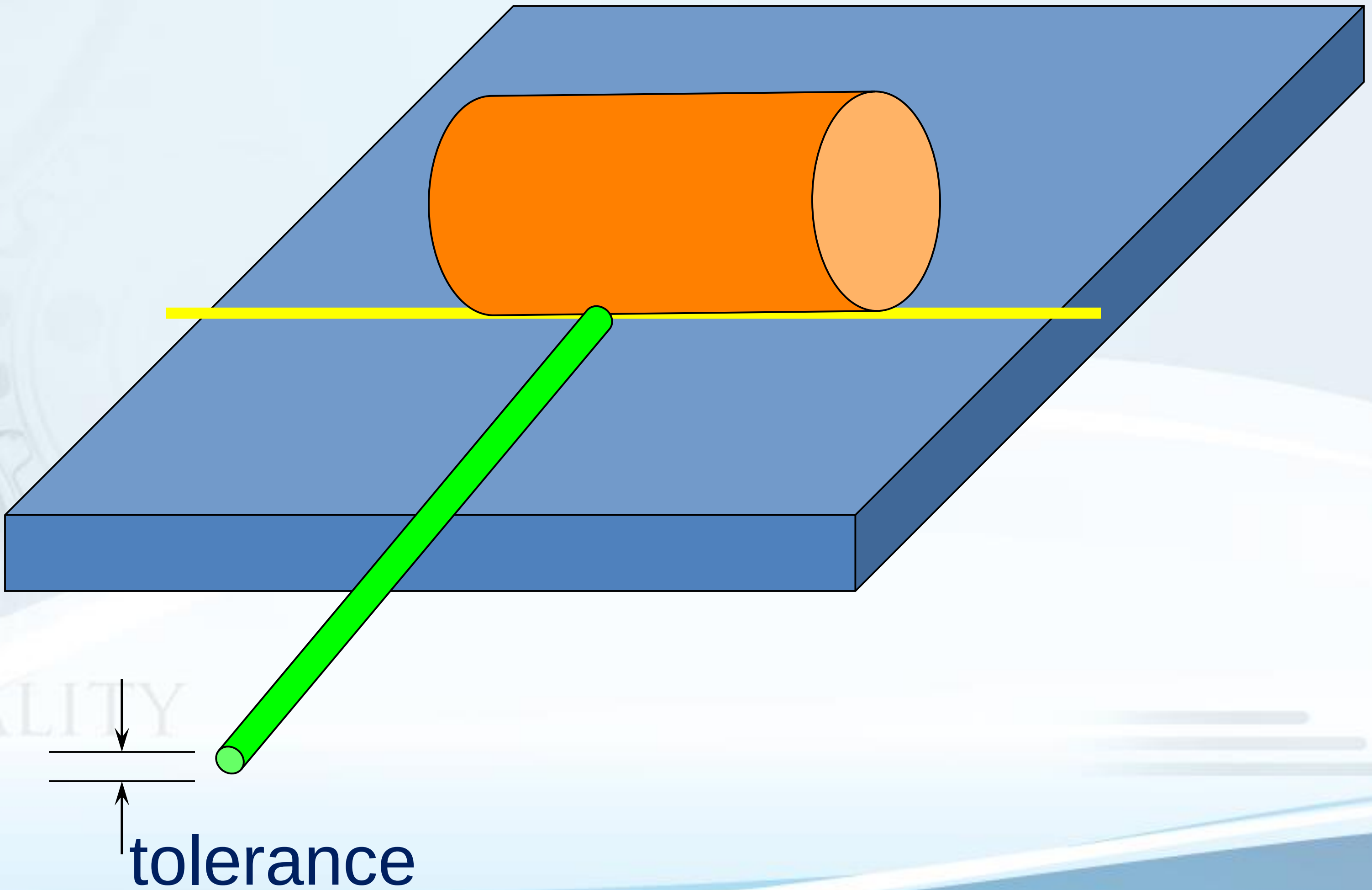
Two parallel lines



Things to Remember About Straightness

- It never references a datum
- The tolerance is not a plus/minus, but the total deviation
- There is no depth; it is just a line element (or axis)
- In the previous drawing, it is applied to the surface because it points to the surface
- When straightness is applied to surface elements, size-related modifiers (MMC, LMC, the diameter symbol) are not allowed
- It can be applied on a per-unit basis

Measuring Straightness



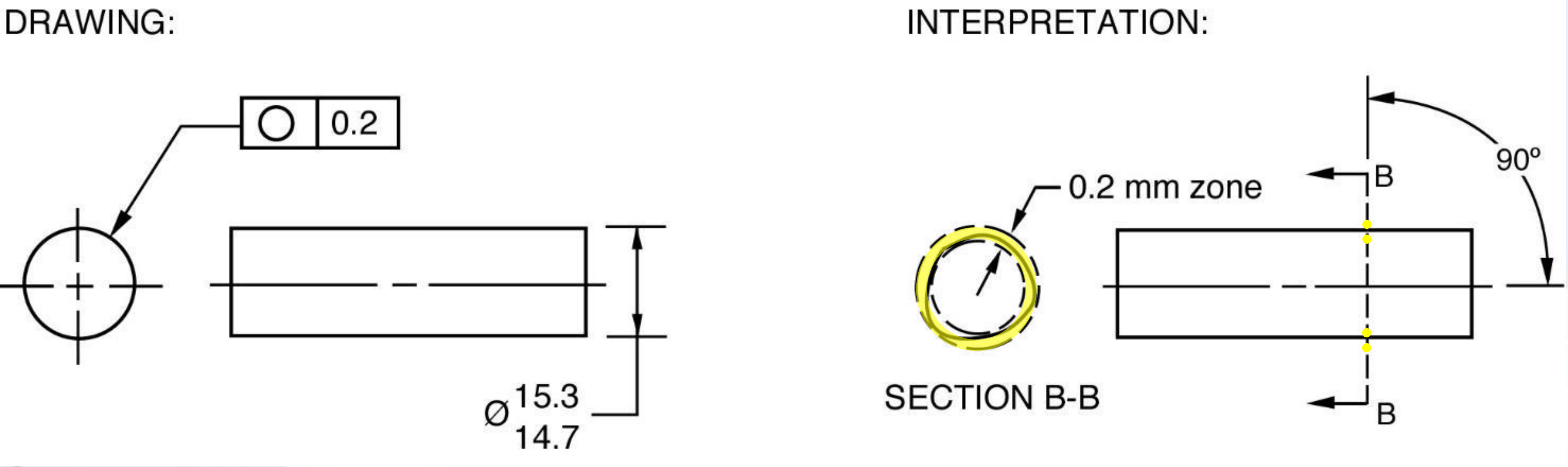
Circularity Definition

- Definition is divided into two:
  - For a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.
  - For a feature other than a sphere, all points of the surface intersected by any plane perpendicular to an axis are equidistant from that axis.

- By default, it's controlled by Rule #1, but it can be separated out to a feature control frame using the circularity symbol.

Circularity

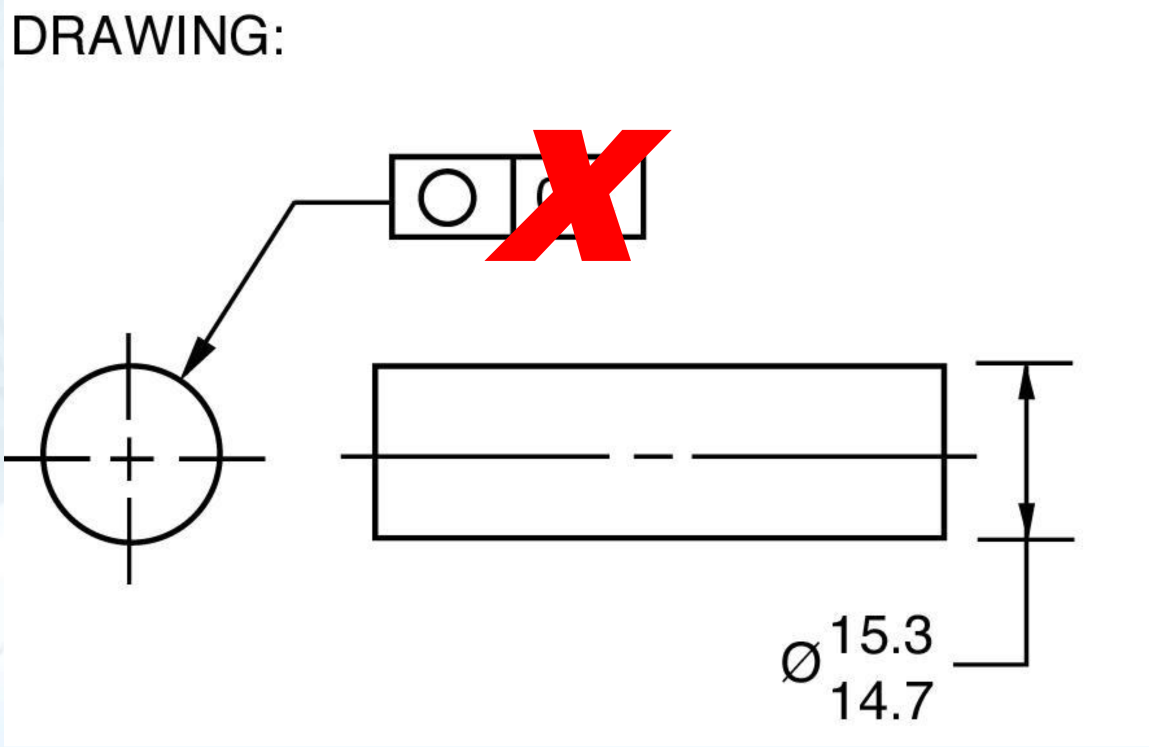
- Two dimensional cross-sectional check



Also goes by the name “roundness.”

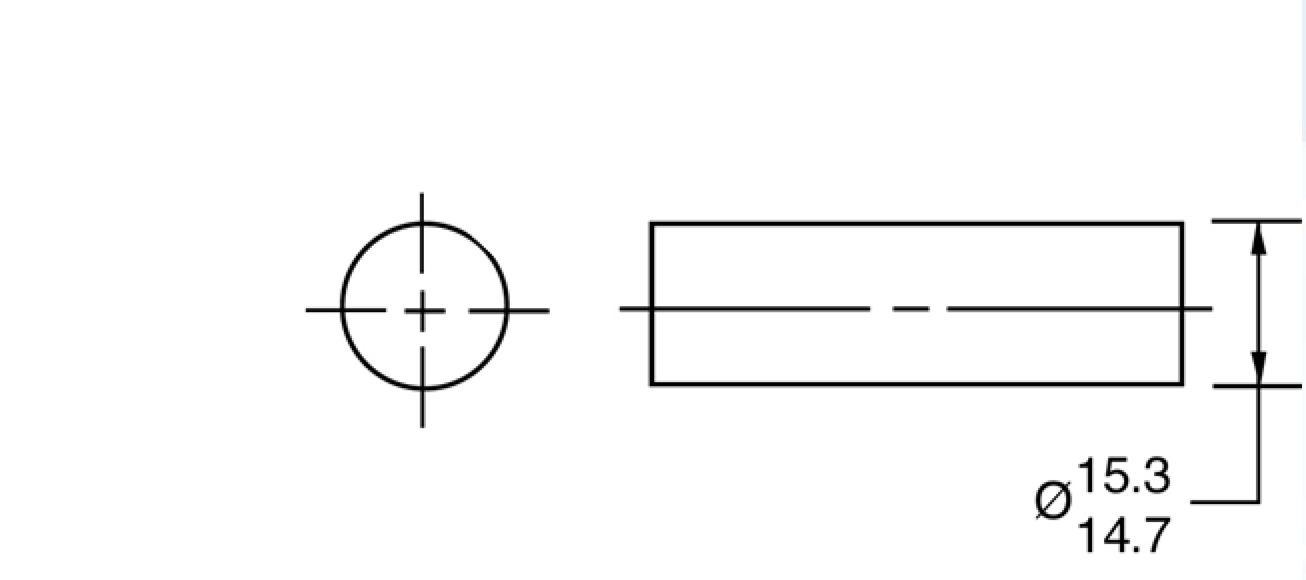
Circularity

- What would control the circularity if the FCF were not on the drawing?

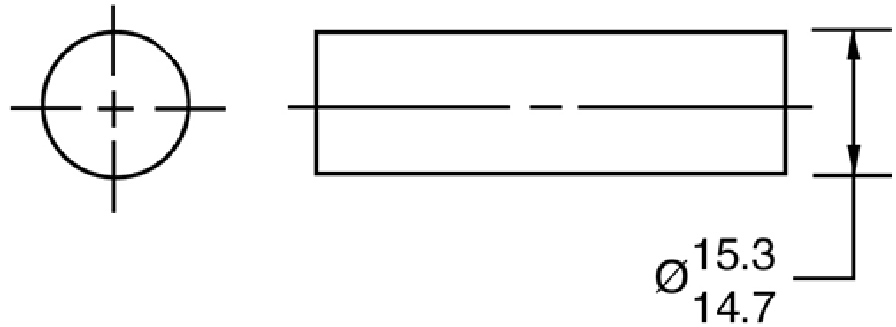


Circularity

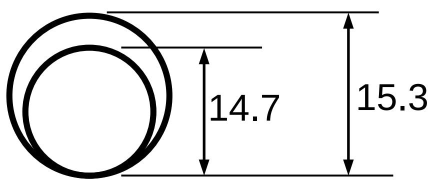
- **Rule #1** (the size tolerance) controls the circularity of this shaft
- The maximum circularity error is 15.3 - 14.7 = **0.6**



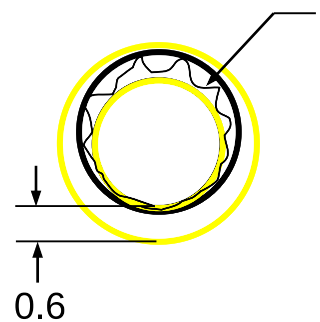
Size Tolerance Zone



Worst case actual mating envelope is 15.3 (Rule #1)



Minimum size is 14.7



Potential circularity error (in terms of the “two coaxial circles a specified distance apart”) as defined by the circularity control

Things to Remember About Circularity

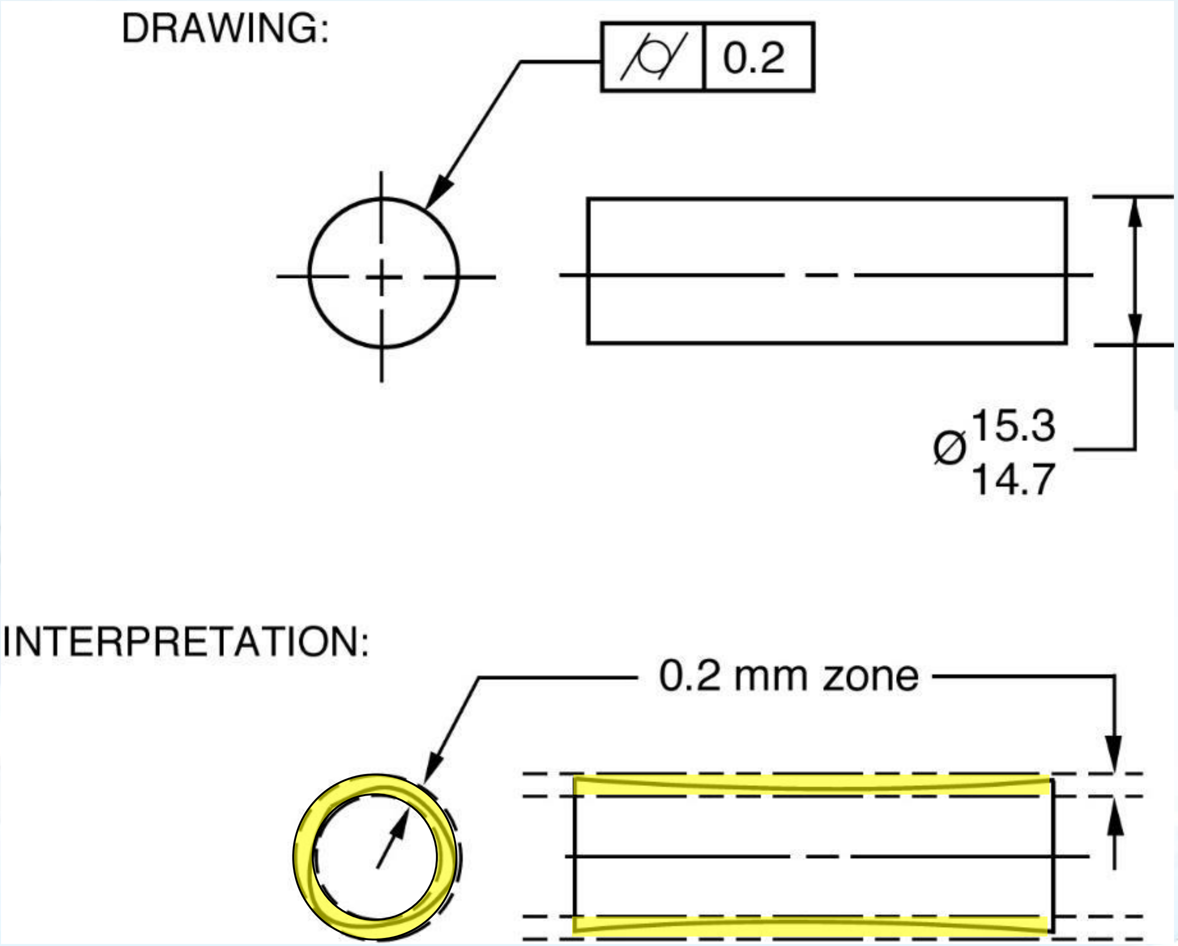
- Never references a datum; never uses MMC or LMC
- Tolerance is not a plus/minus, it’s the radial distance between two co-axial **circles**
- Measured at individual cross-sections (so it does not include straightness)
- Rule #1 still applies to control the axis straightness
- Tolerance value must be *less than* the size tolerance

Cylindricity Defintion

- Defined as the condition of a surface of revolution in which all points of the surface are equidistant from a common axis.

- By default, it’s controlled by Rule #1, but it can be separated out to a feature control frame using the cylindricity symbol.

Cylindricity



Things to Remember About Cylindricity

- Never references a datum
- Never uses MMC or LMC
- Tolerance is not a plus/minus, it's the radial distance between two co-axial **cylinders**
- Covers the entire length, so straightness is controlled (both surface straightness and axial straightness)
- Tolerance value must be *less than* the size tolerance

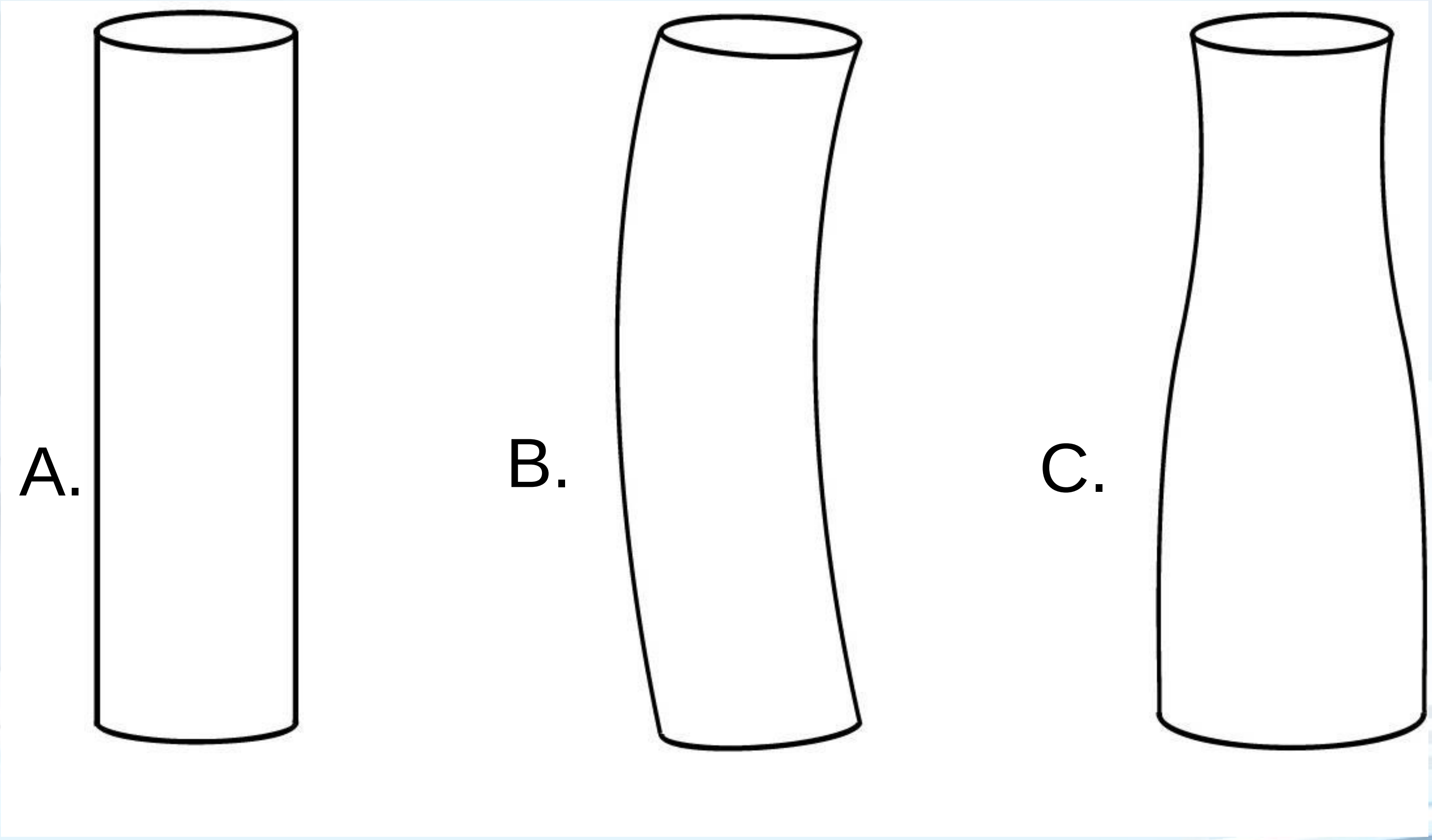
But Wait…

- All examples in this chapter up to this point have been showing surface form. But form can also be applied to an axis or center plane (for straightness and flatness).

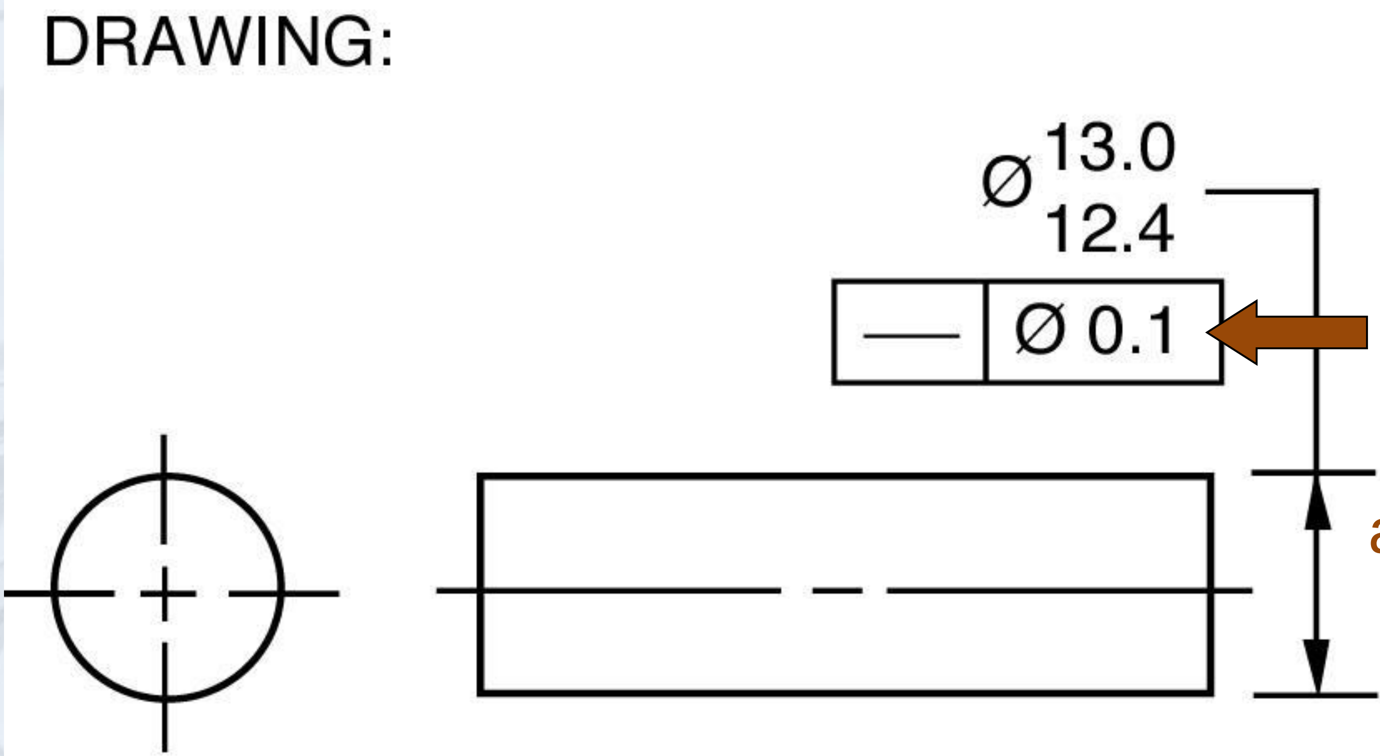
- The key is to know when a GD&T symbol applies to a surface or an axis.

Straightness

Which part is straight?

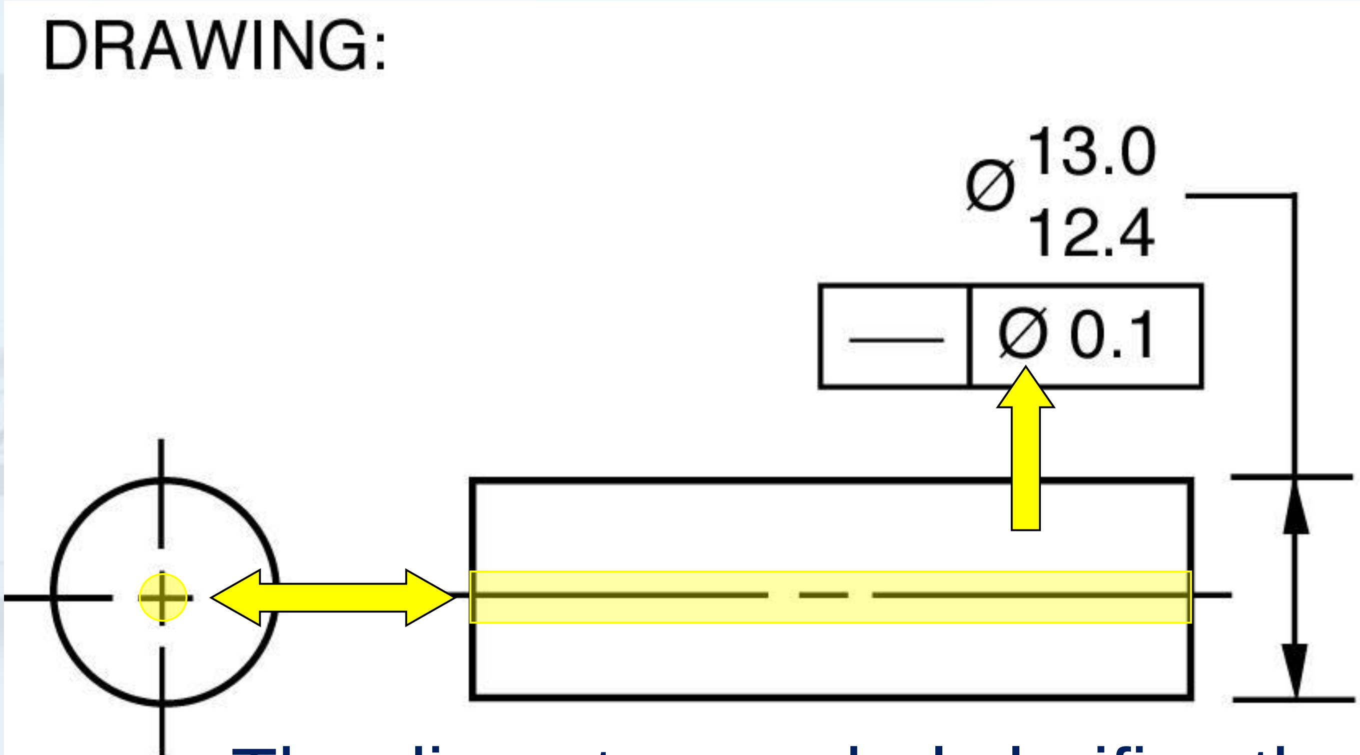


Straightness Applied to FOS



Frame's placement indicates it applies to a feature of size

Straightness Applied to FOS

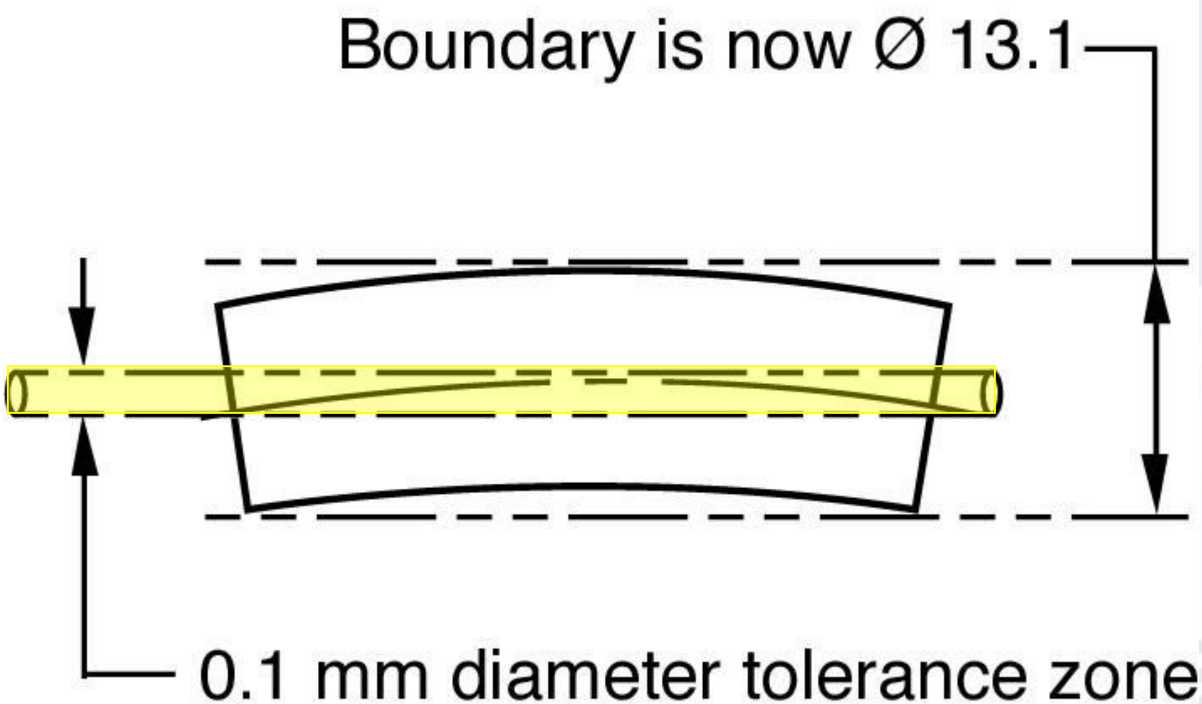


The diameter symbol clarifies that the tolerance zone is cylindrical

Straightness Applied to FOS

INTERPRETATION:

Worst-Case



Sizes and Envelopes

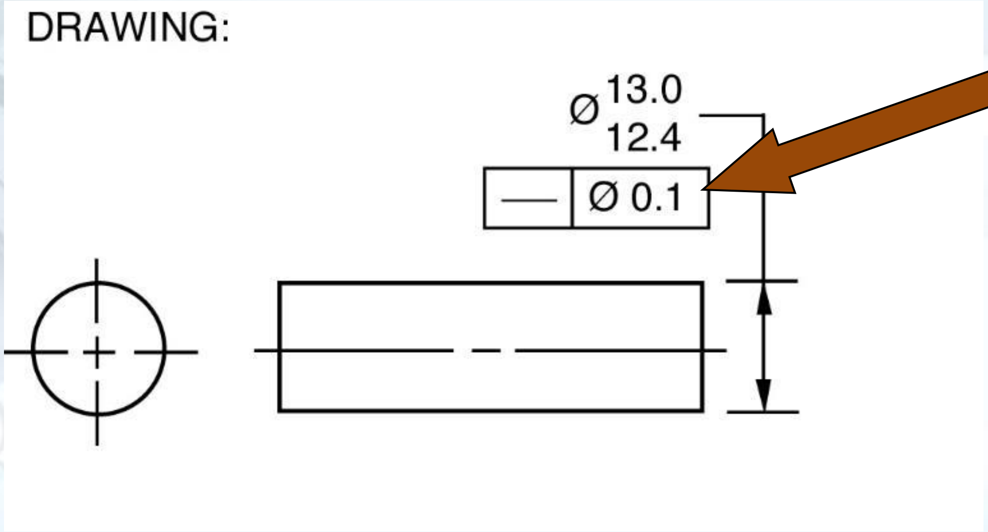
Actual Size	Straightness Tolerance	Actual Mating Envelope
12.4	0.1	12.5
12.5	0.1	12.6
12.6	0.1	12.7
12.7	0.1	12.8
12.8	0.1	12.9
12.9	0.1	13.0
13.0	0.1	**13.1**

Without Straightness

Actual Size	Implied Straightness	Actual Mating Envelope
12.4	0.6	13.0
12.5	0.5	13.0
12.6	0.4	13.0
12.7	0.3	13.0
12.8	0.2	13.0
12.9	0.1	13.0
13.0	0	13.0

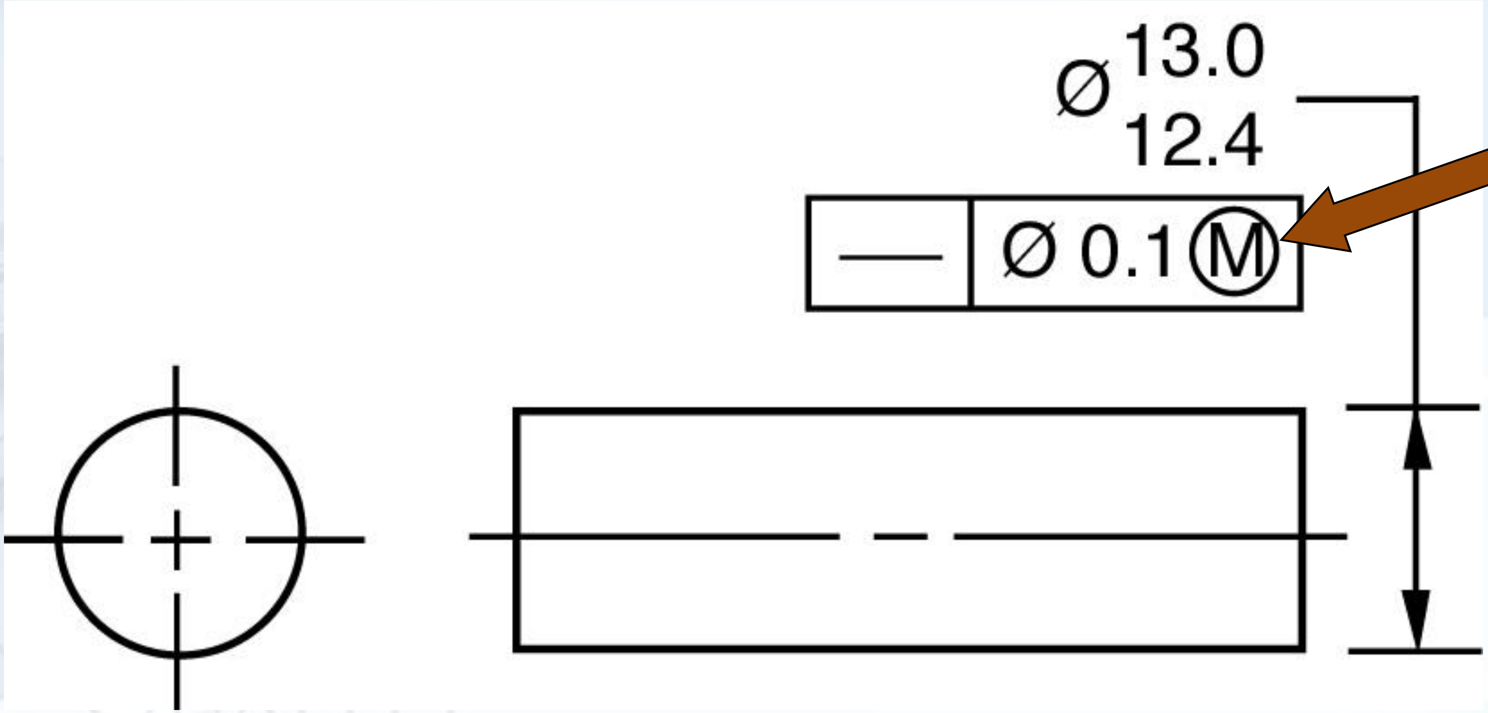
Recall Rule #2

- Unless a modifier is given, assume that a geometric tolerance (and any datum references) are RFS.
- RFS stands for regardless of feature size.



Constant tolerance of 0.1 -- RFS

Using MMC w/ Straightness

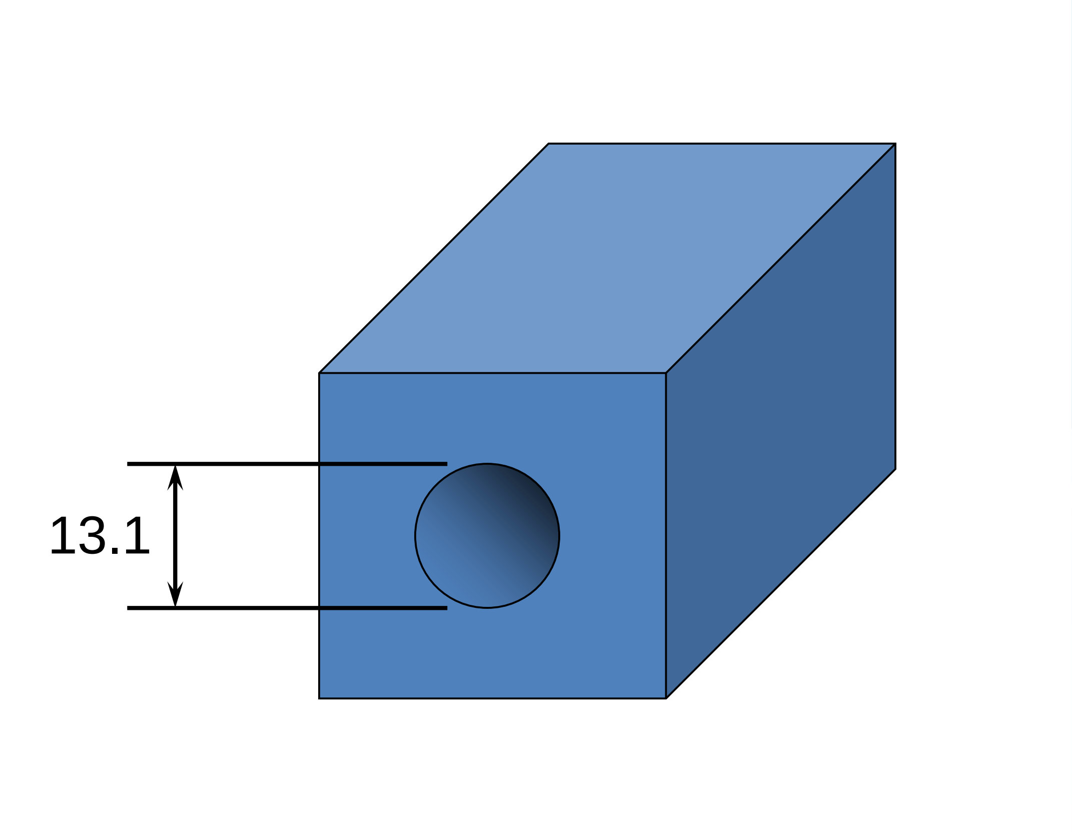


No longer RFS

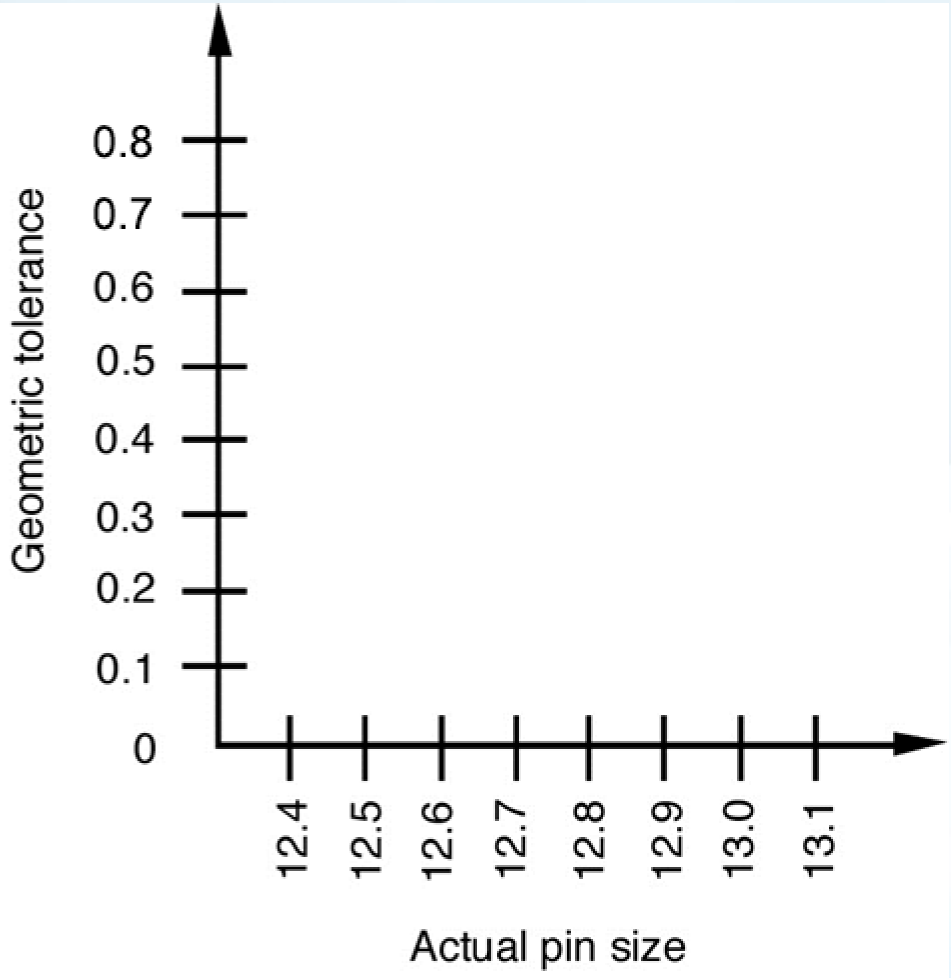
Sizes and Envelopes

Actual Size	Bonus Tolerance	Stated Tolerance	Act. Mating Envelope
12.4	0.6	0.1	
12.5	0.5	0.1	
12.6	0.4	0.1	
12.7	0.3	0.1	
12.8	0.2	0.1	
12.9	0.1	0.1	
13.0	0	0.1	

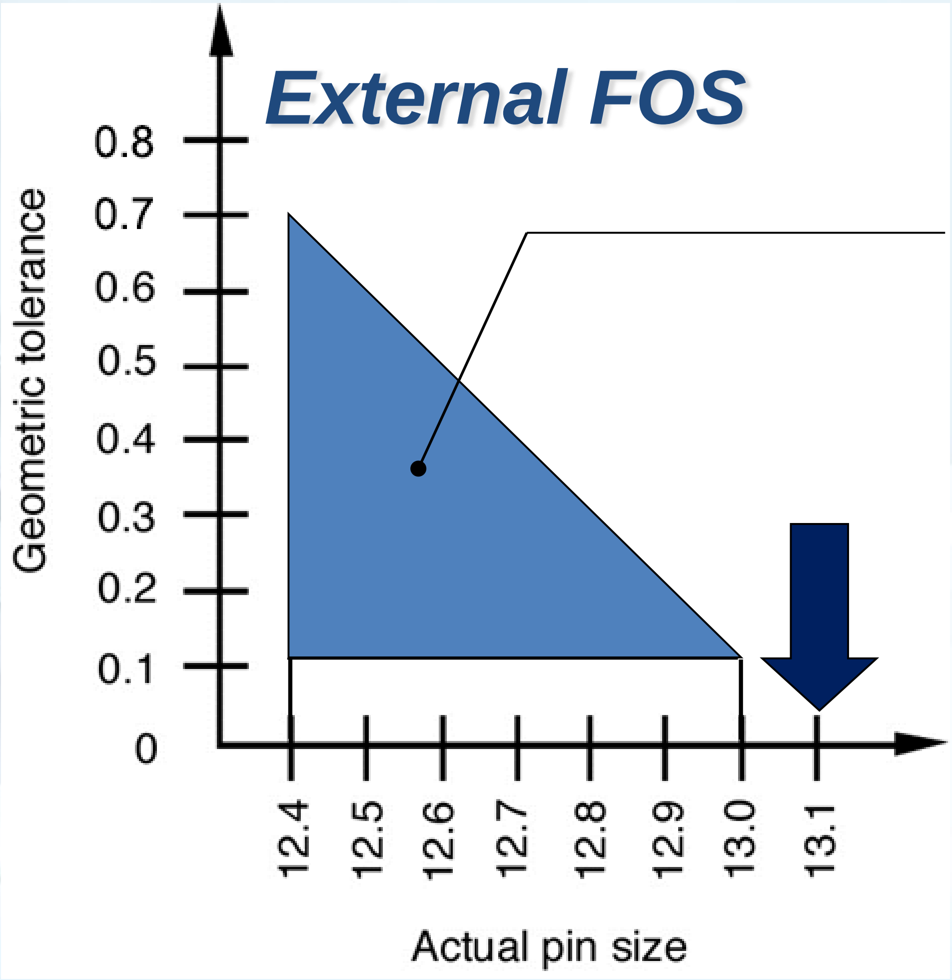
Constant Boundary Implies Fixed Gage



Graph of Tolerance vs. Size



Graph of Tolerance vs. Size



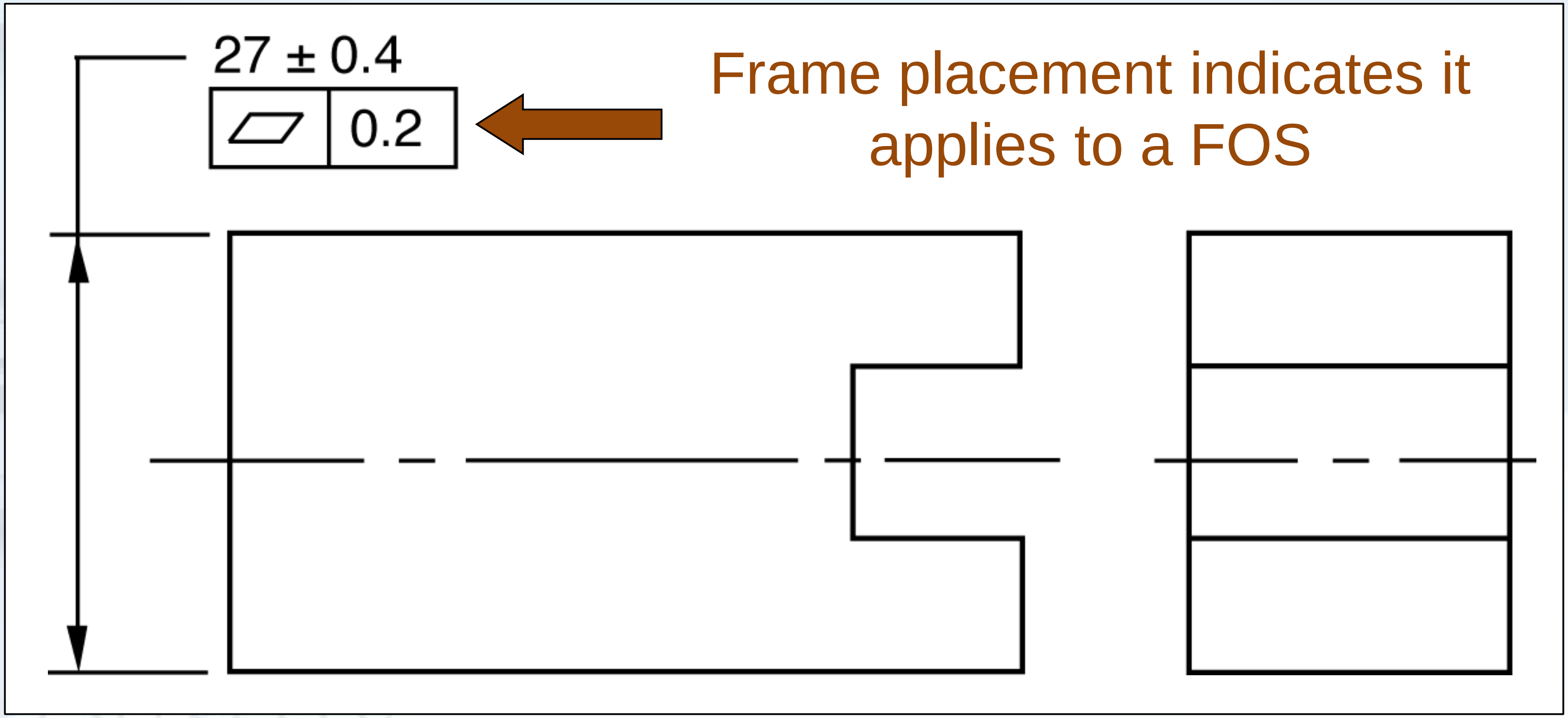
Bonus tolerance = MMC - actual size

Virtual condition = MMC + stated tol.

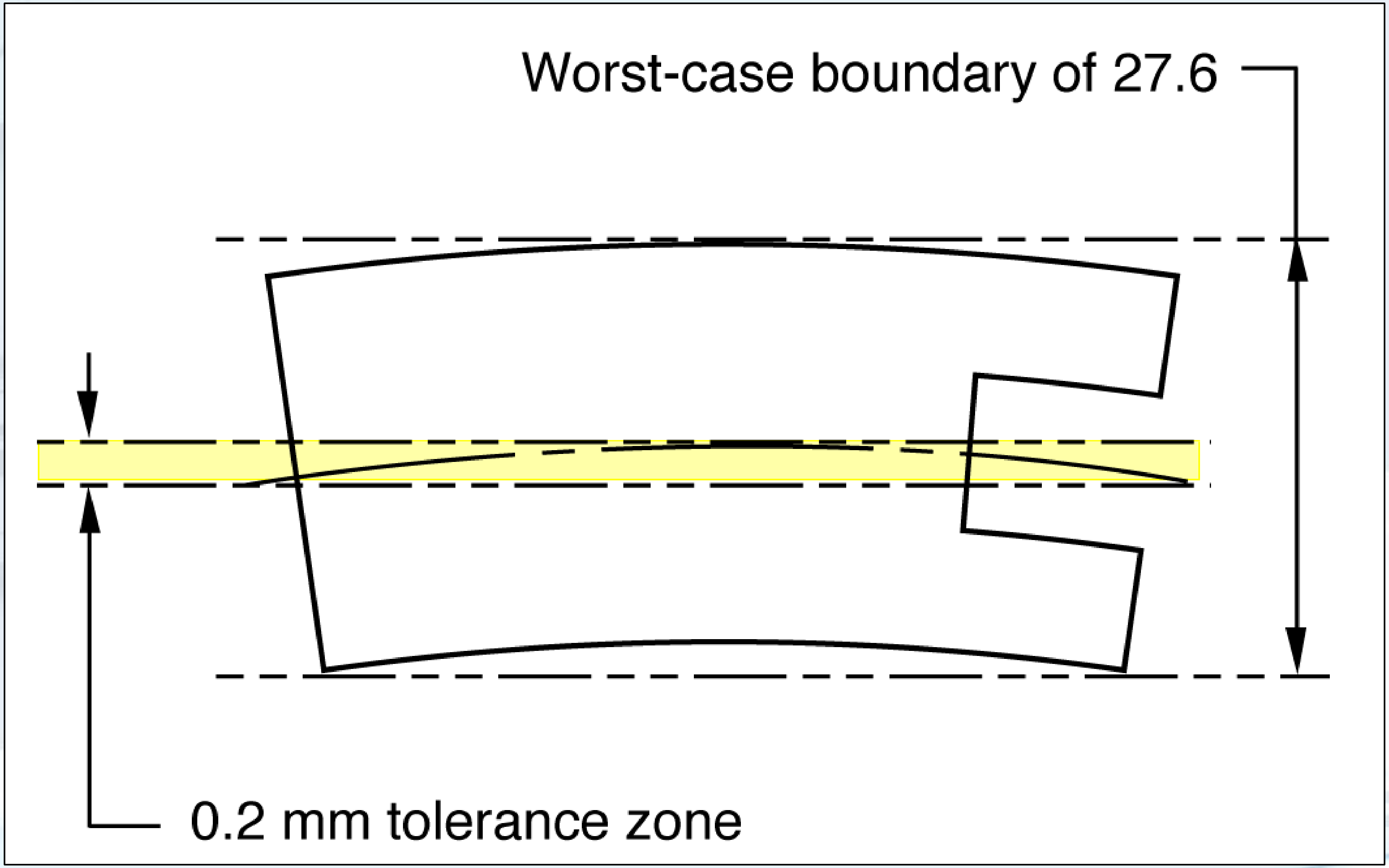
Things to Remember About FOS Straightness

- It never references a datum
- The tolerance is not a plus/minus, but the total deviation
- It is cylindrical (if dia. symbol) or two parallel planes
- It's on a FOS if applied to a size dimension
- When straightness is applied to a feature of size, size-related modifiers (MMC, LMC, the diameter symbol) are allowed
- It can be applied on a unit basis

Flatness Applied to FOS



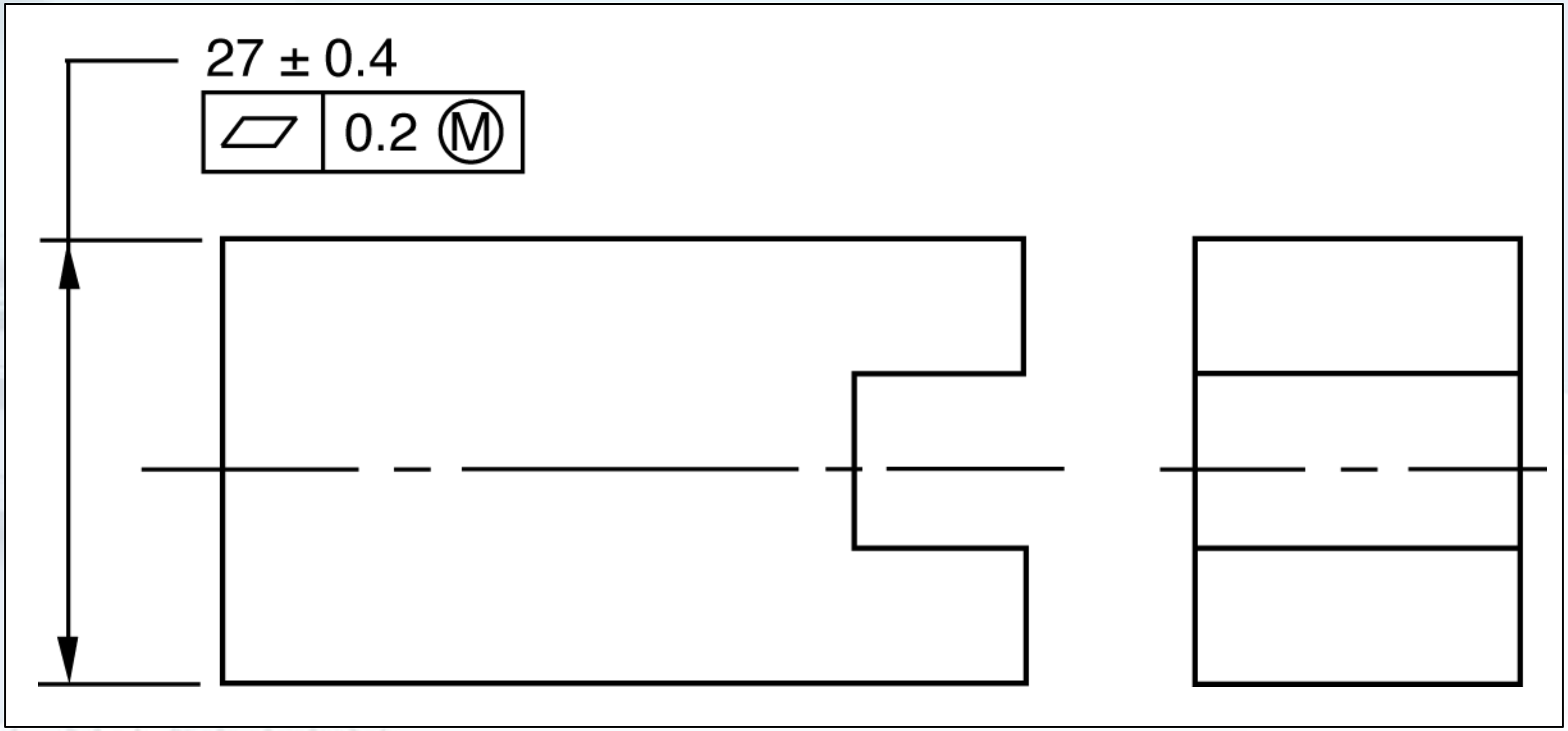
Flatness Applied to FOS



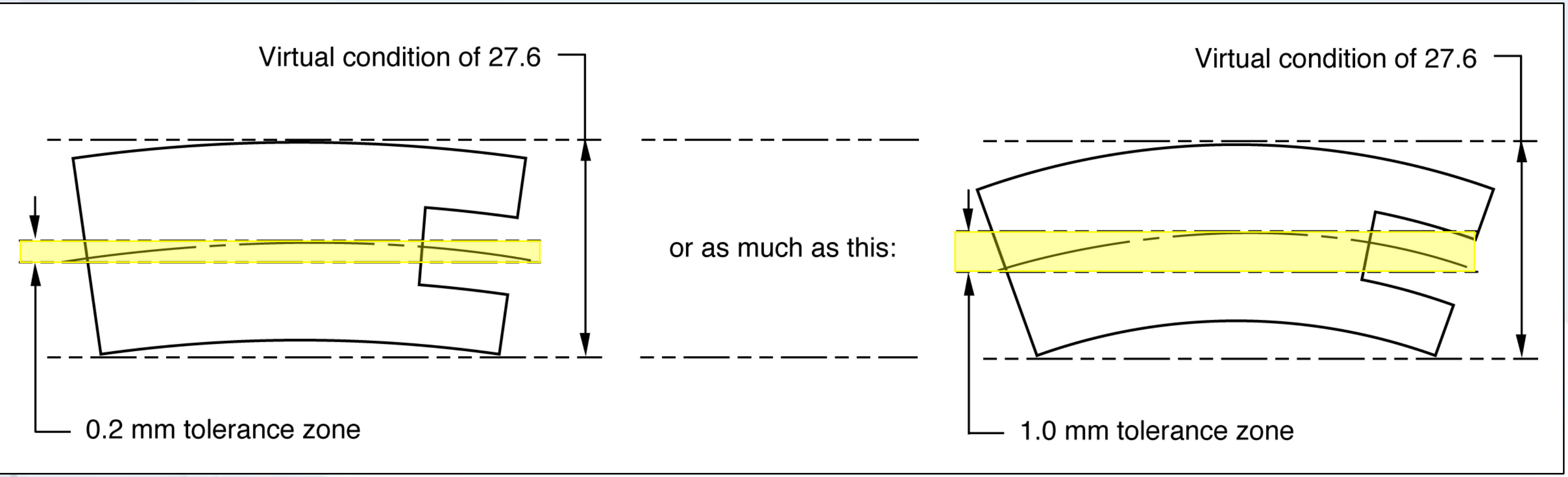
Sizes and Envelopes

Actual Local Size	Stated Tolerance	Actual Mating Envelope
26.6	0.2	26.8
26.7	0.2	26.9
26.8	0.2	27.0
26.9	0.2	27.1
27.0	0.2	27.2
27.1	0.2	27.3
27.2	0.2	27.4
27.3	0.2	27.5
27.4	0.2	**27.6**

Flatness Applied at MMC



Flatness Applied at MMC



Chapter 3: Form – What We Covered

Learning Objectives

You should now be able to:

- Describe the tolerance zone for flatness, and how it is inspected
- Describe the tolerance zone for straightness, and how it is inspected
- Describe the tolerance zone for circularity, and how it is inspected
- Describe the tolerance zone for cylindricity, and how it is inspected
- Name the two form tolerances that may be used on features of size
- Explain how bonus tolerance is calculated

Chapter Agenda

- Flatness
- Straightness
- Circularity
- Cylindricity

Chapter 4

Datums

Selecting, Identifying and Referencing Datums

Chapter 4: Datums – What We Will Cover

Learning Objectives

At the end of this chapter, you will be able to:

- Define datum, and datum feature
- Identify primary, secondary, and tertiary datums for a given feature control frame.
- Select appropriate datums based on function
- Explain how to determine if a datum is derived from a surface or a feature of size
- Explain the effect of “M” after a datum reference
- Identify and explain datum targets

Chapter Agenda

- Identifying Datums
- Selecting Datums
- Simulating a Surface Datum
- 6 Degrees of Freedom
- Datum Targets

Datums

- Datums are the key to consistency:

Product Design

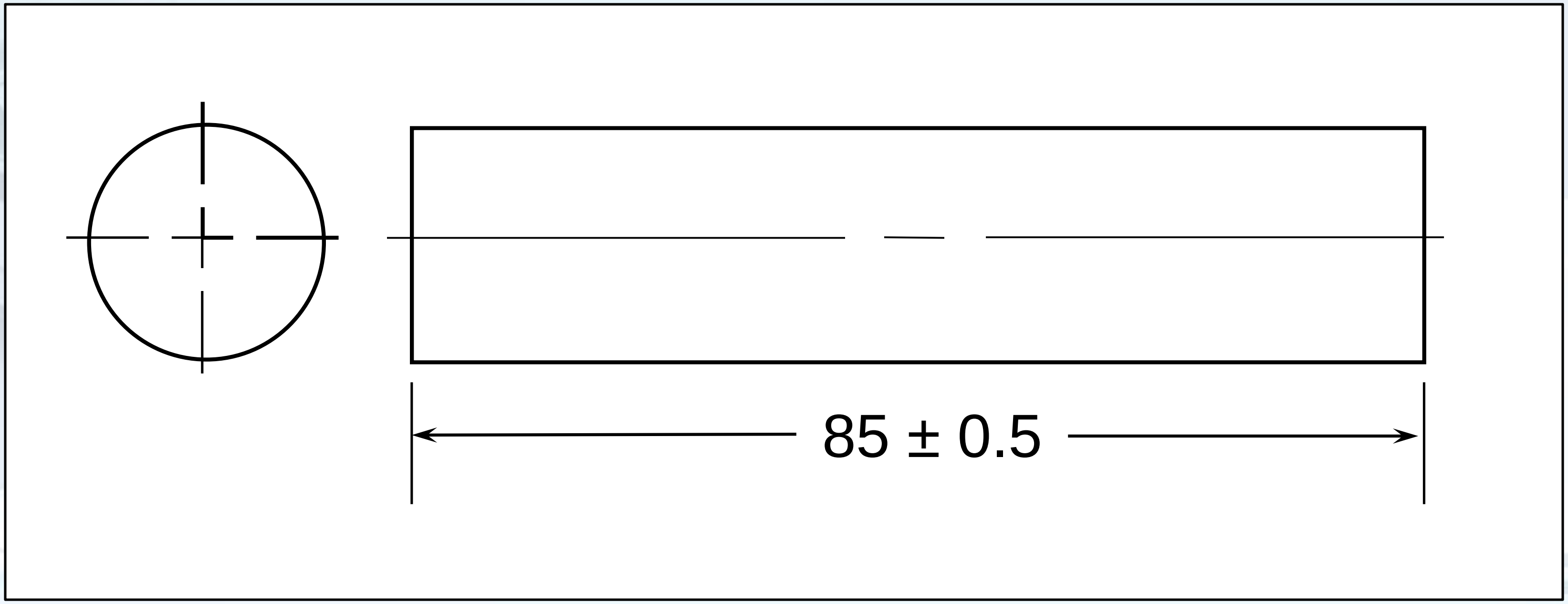
→ Process Design

→ Manufacturing

→ Inspection

Datums

- Origin for a measurement
  - *From where* does a location, orientation, runout, or profile dimension originate?

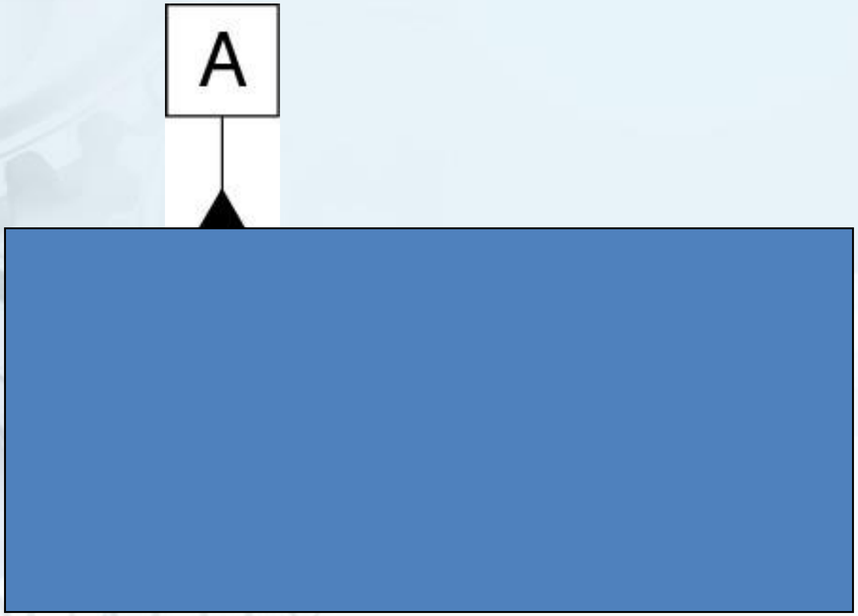


Datums

- **Datum:** A theoretically exact point, axis, or plane that serves as the origin for a measurement.
- **Datum Feature:** The actual surface that the datum is derived from.

Datum Feature Identification

How it looks on print:

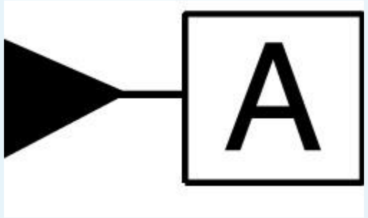


Definitions:

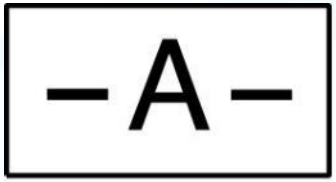
Datum: Theoretical plane

Datum Feature: Top surface of part

Symbol



(Triangle may be filled or unfilled)



(Old datum feature symbol)

Using Datums Effectively

1. Selecting the appropriate datum features

2. Simulating the datums correctly

Selecting Datums

The most important consideration is…

… part / product **FUNCTION**

- What ***features*** on the detail orient and locate it in the assembly?

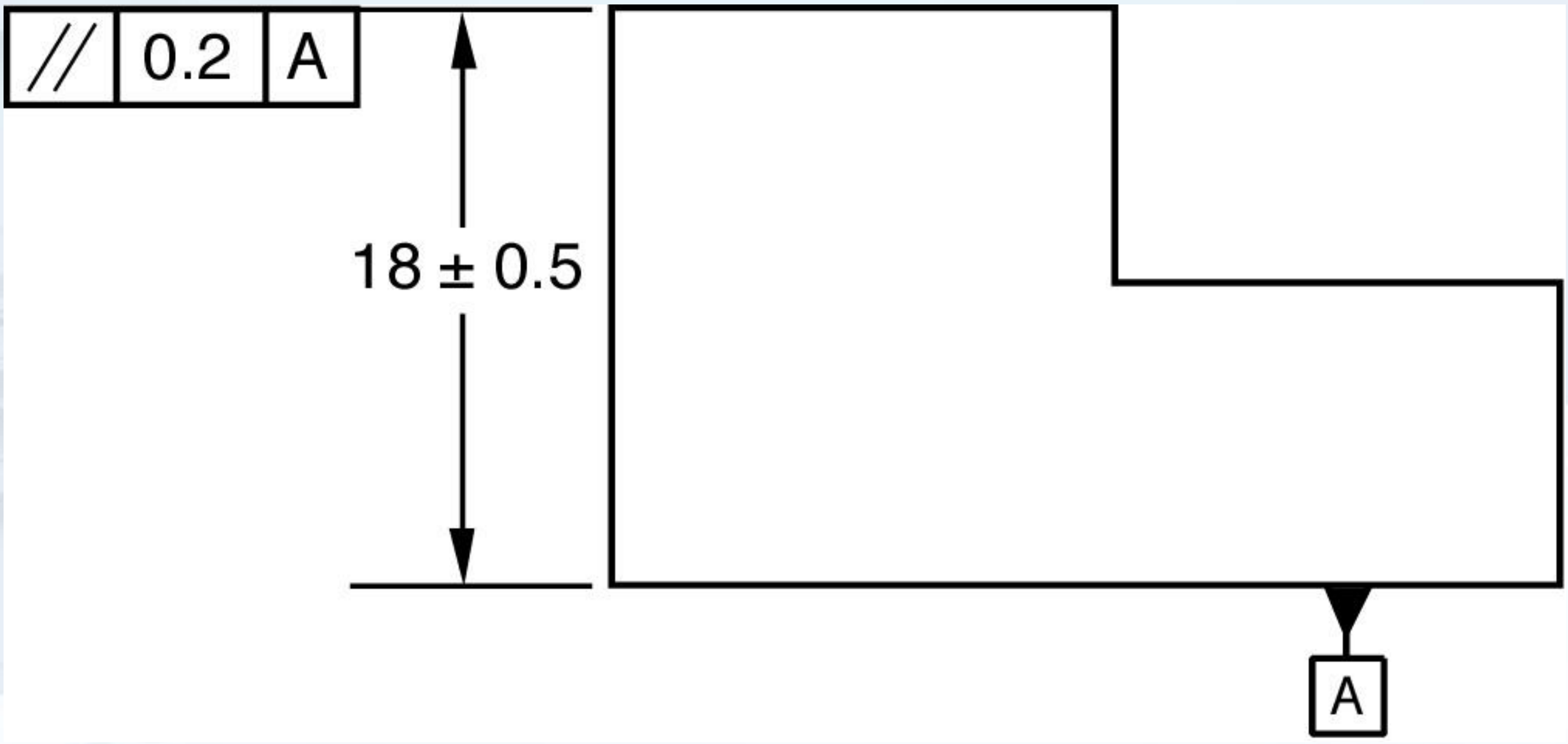
Selecting Datums

- Other concerns:
  - Accessibility
  - Repeatability
  - Stability
  - Cost
  - Time

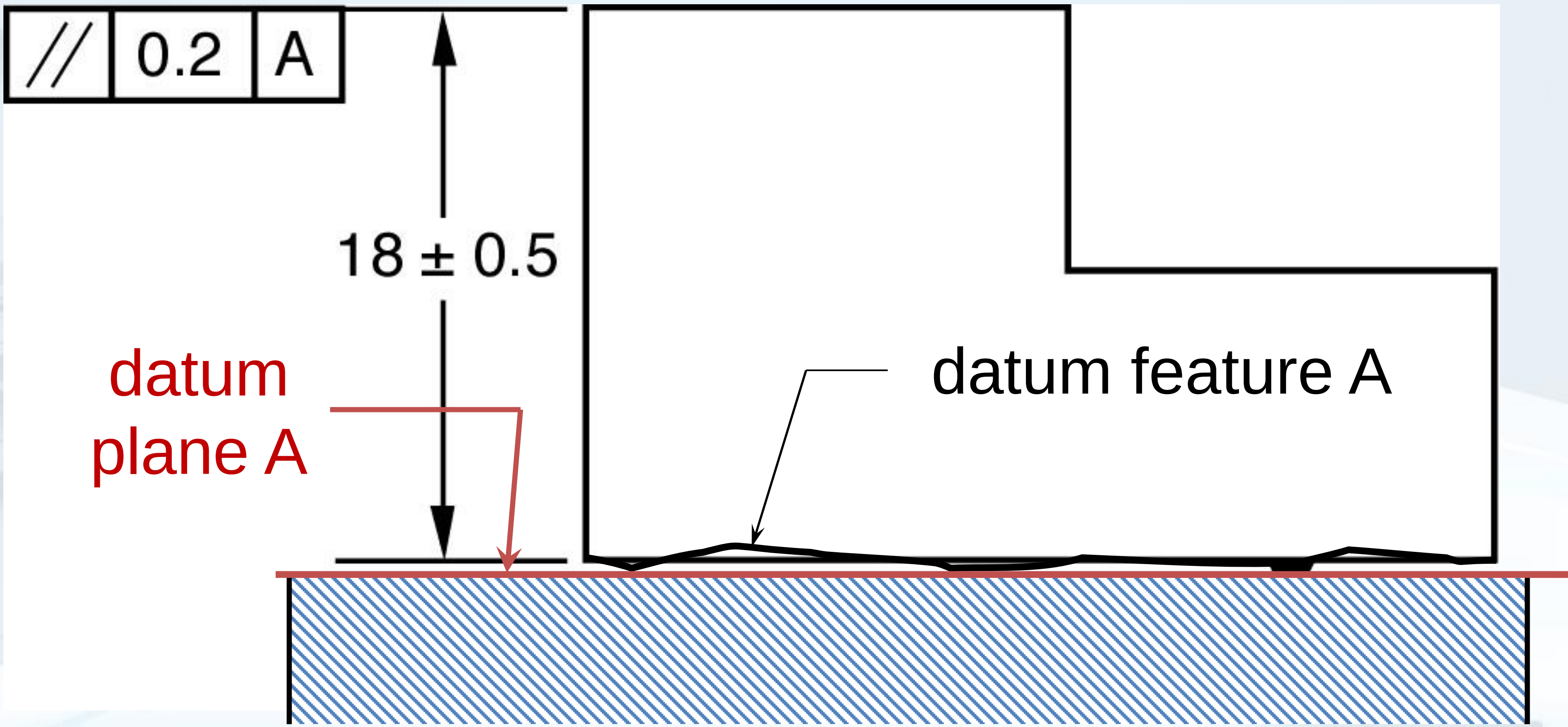
Selecting Datums

- Product functionality: *What **features** on the detail orient and locate it in the assembly?*
- Mounting surfaces
- Interfaces
- Locating holes
- Guide pins
- Bearing surfaces

Simulating a Surface Datum

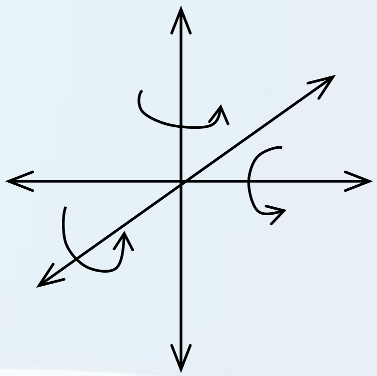


Simulating a Surface Datum



6 Degrees of Freedom

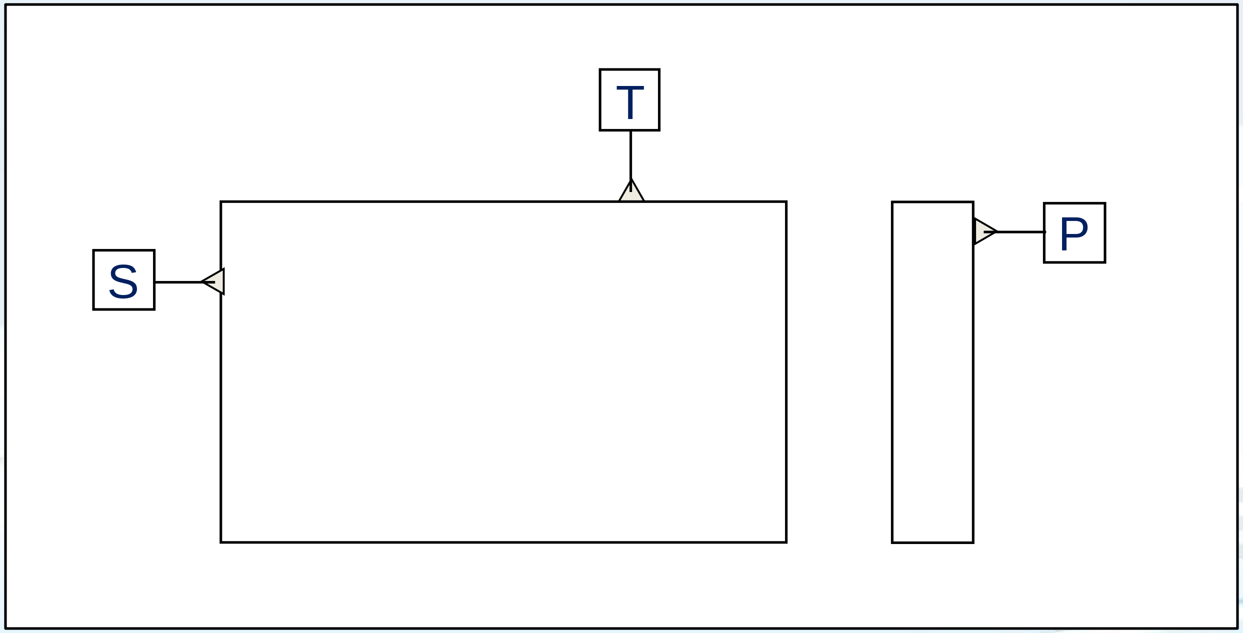
1. Up/down
2. Forward/back
3. Left/right
4. Rotation around up/down
5. Rotation around forward/back
6. Rotation around left/right



Restricting 6 Degrees

Three mutually perpendicular planes:

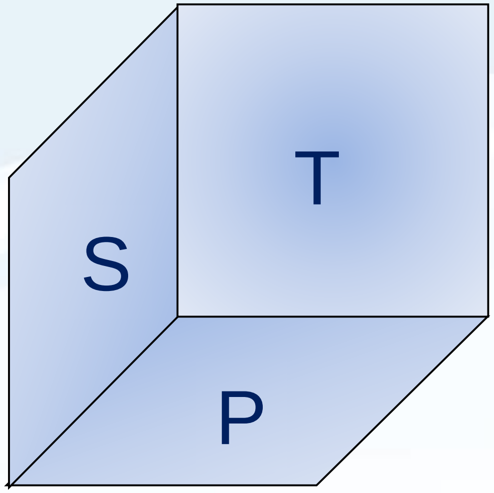
1. Primary
2. Secondary
3. Tertiary



Restricting 6 Degrees

Three mutually perpendicular planes:

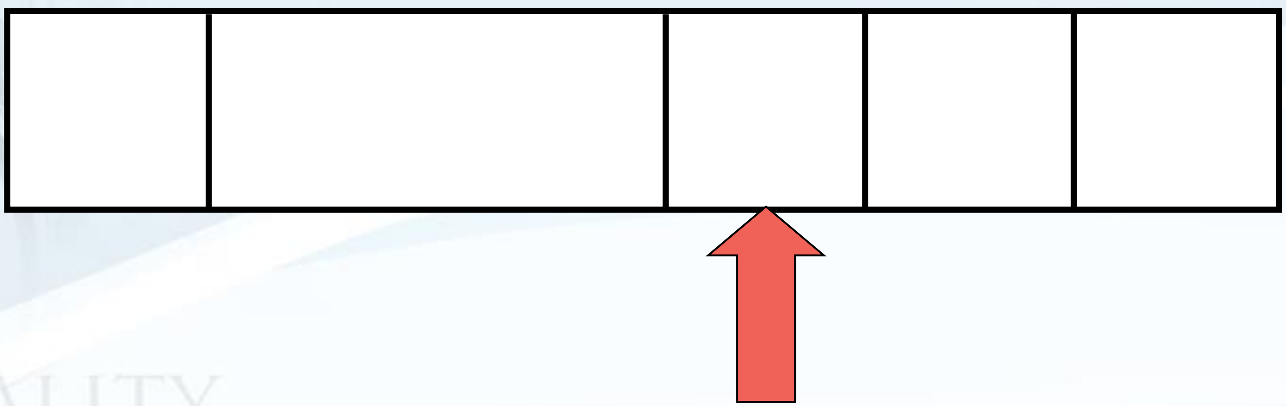
1. Primary
2. Secondary
3. Tertiary



First Plane

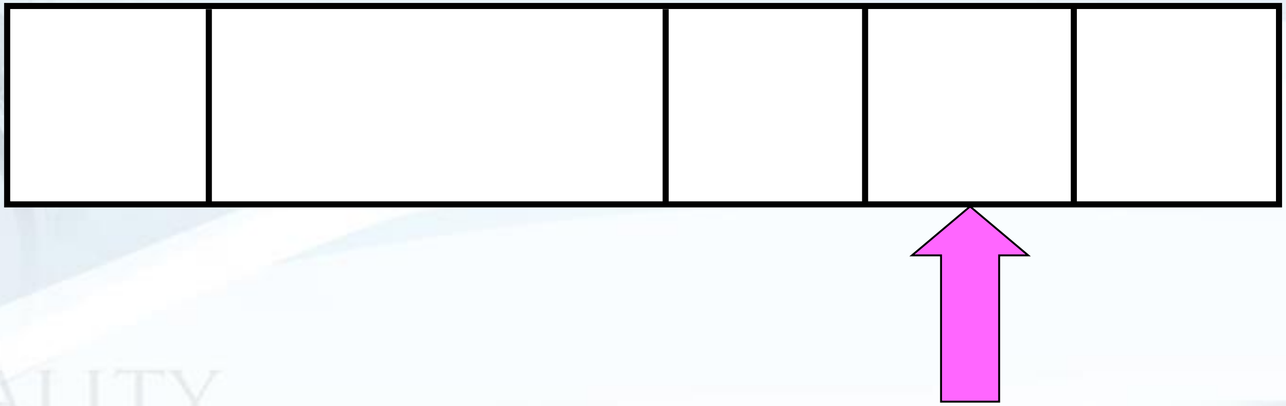
- Primary datum plane:

  Requires ____ points of contact (min)



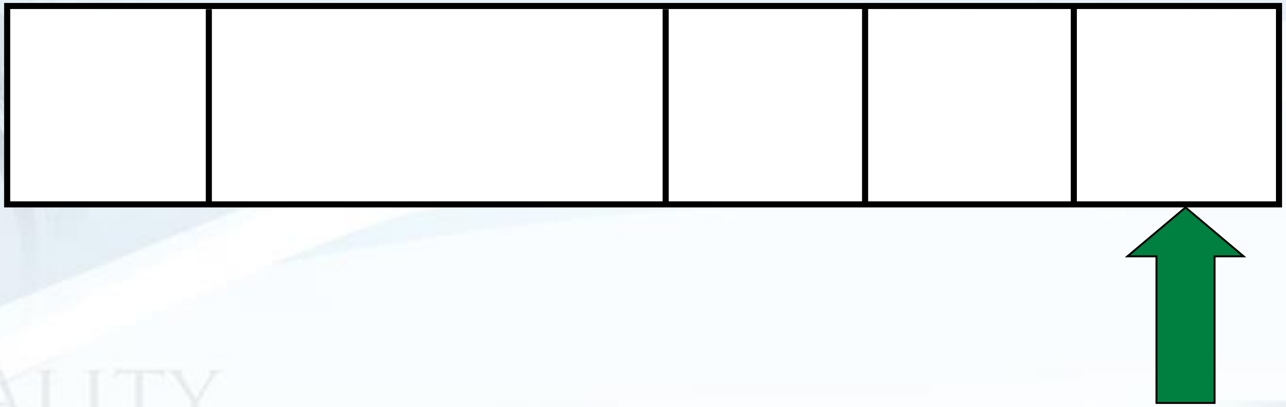
Second Plane

- Secondary datum plane:
  Requires ____ points of contact (min)



Third Plane

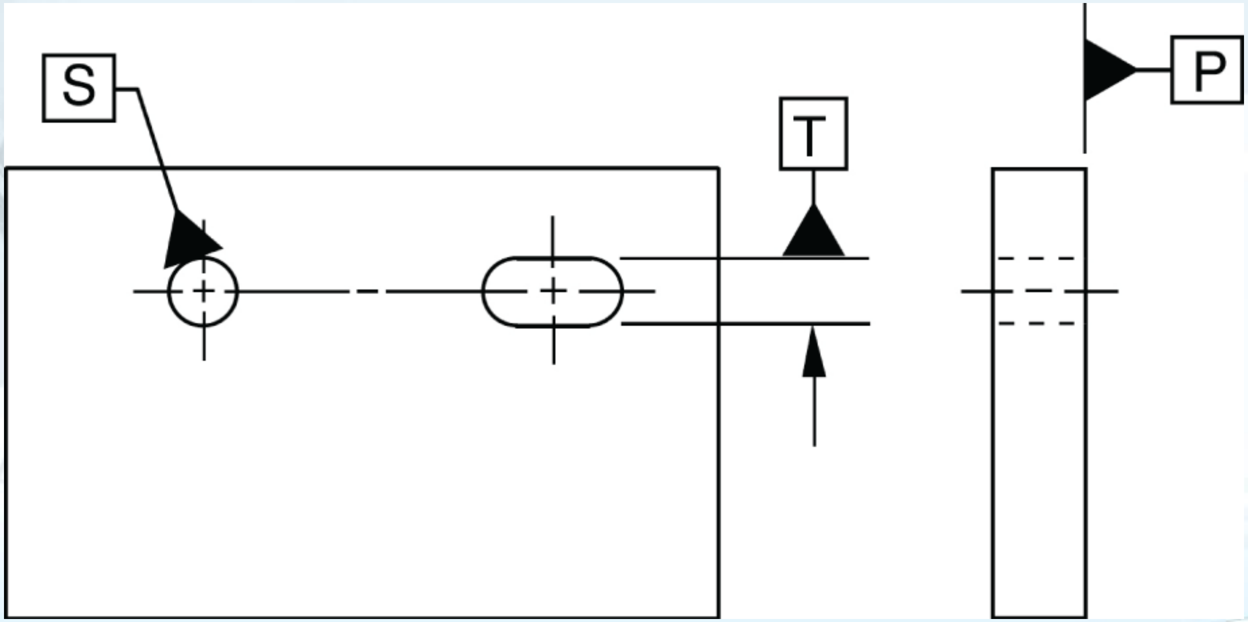
- Tertiary datum plane:
  Requires ____ point of contact (min)



Restricting 6 Degrees

One plane, one hole, one slot:

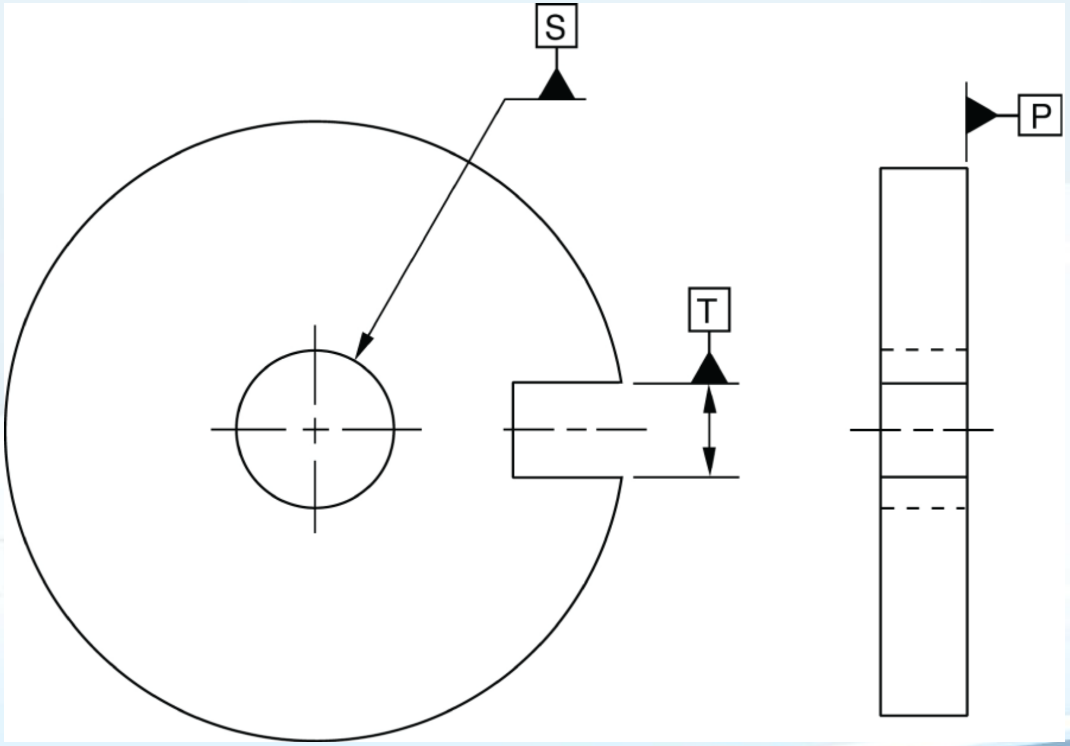
1. Primary
2. Secondary
3. Tertiary



Restricting 6 Degrees

One plane, one hole, one slot:

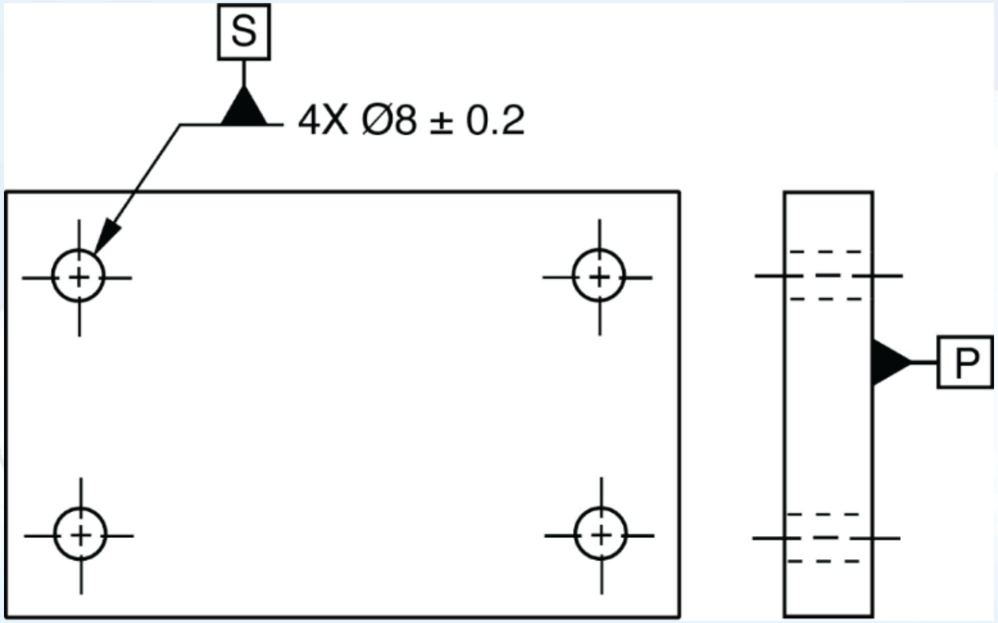
1. Primary
2. Secondary
3. Tertiary



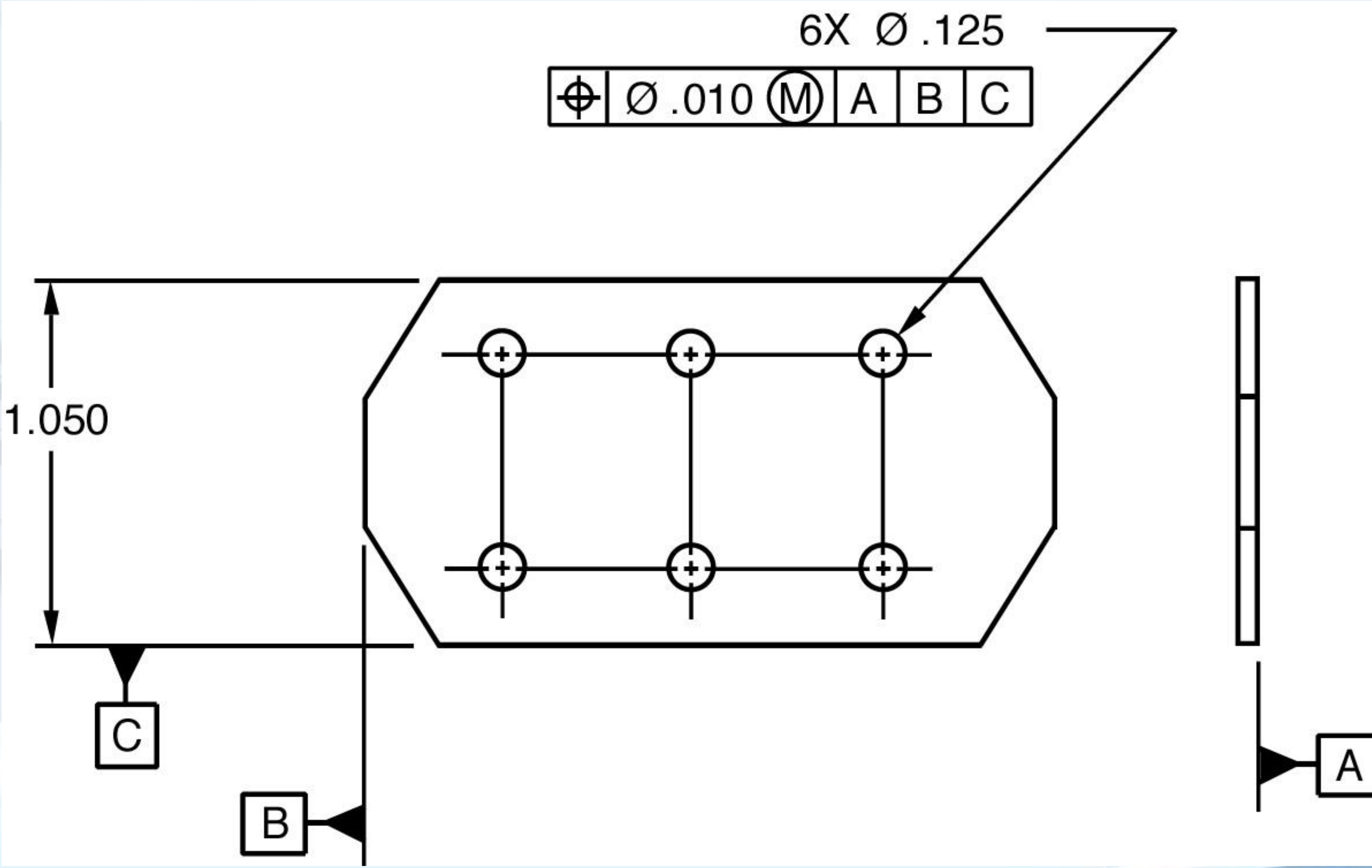
Restricting 6 Degrees

One plane, one hole pattern:

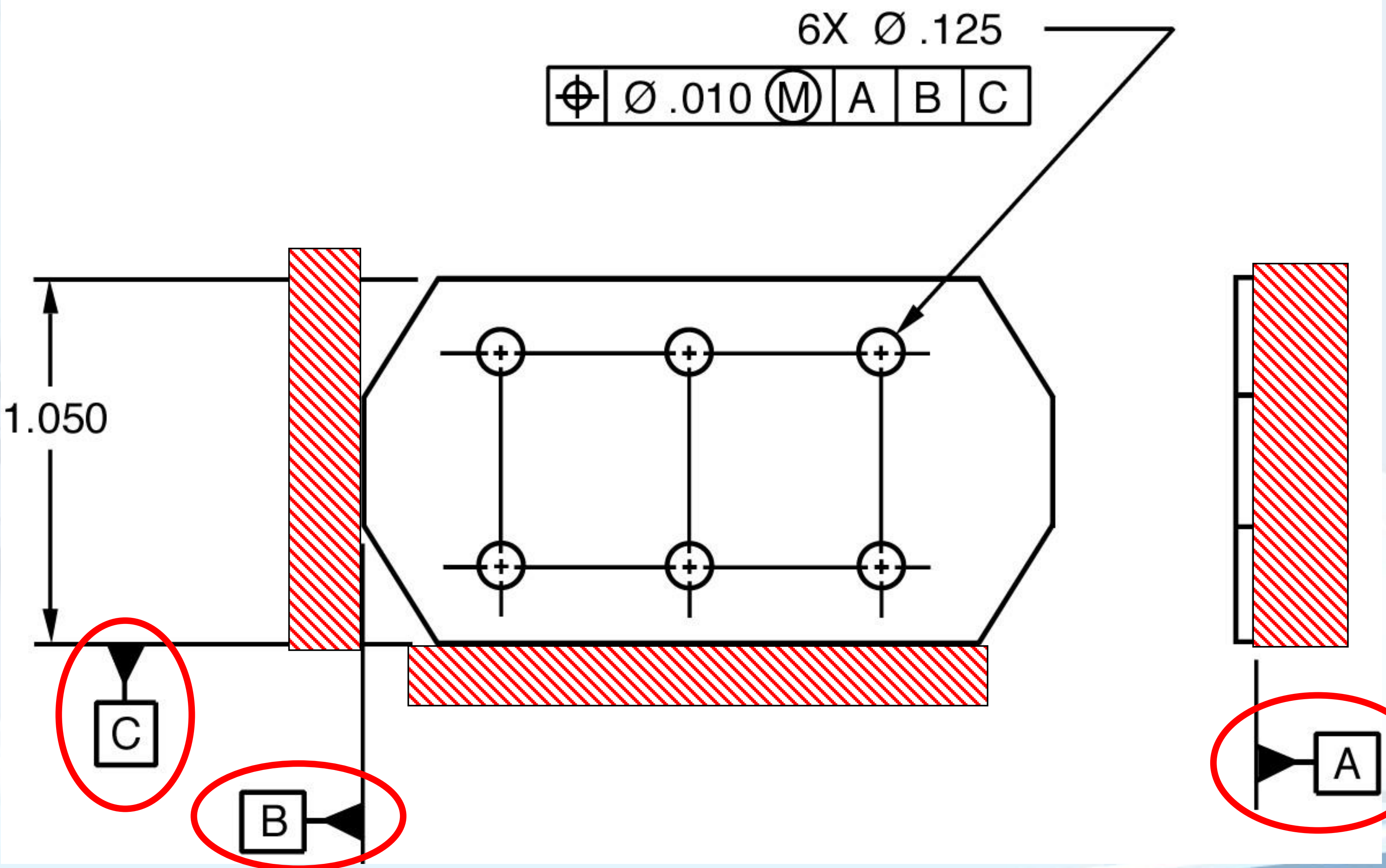
1. Primary – 3 degrees of freedom
2. Secondary – 3 degrees of freedom



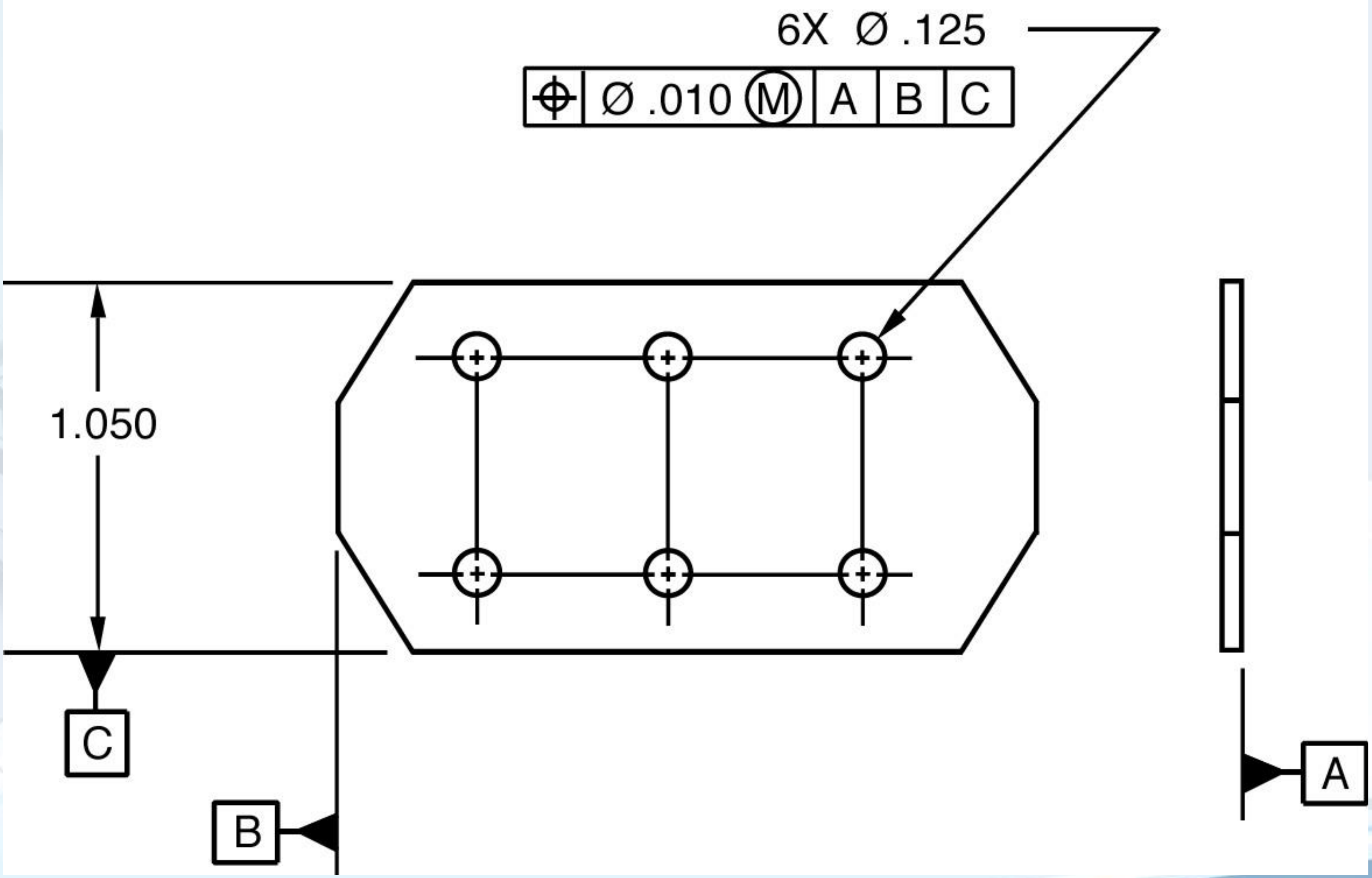
Surface Datum



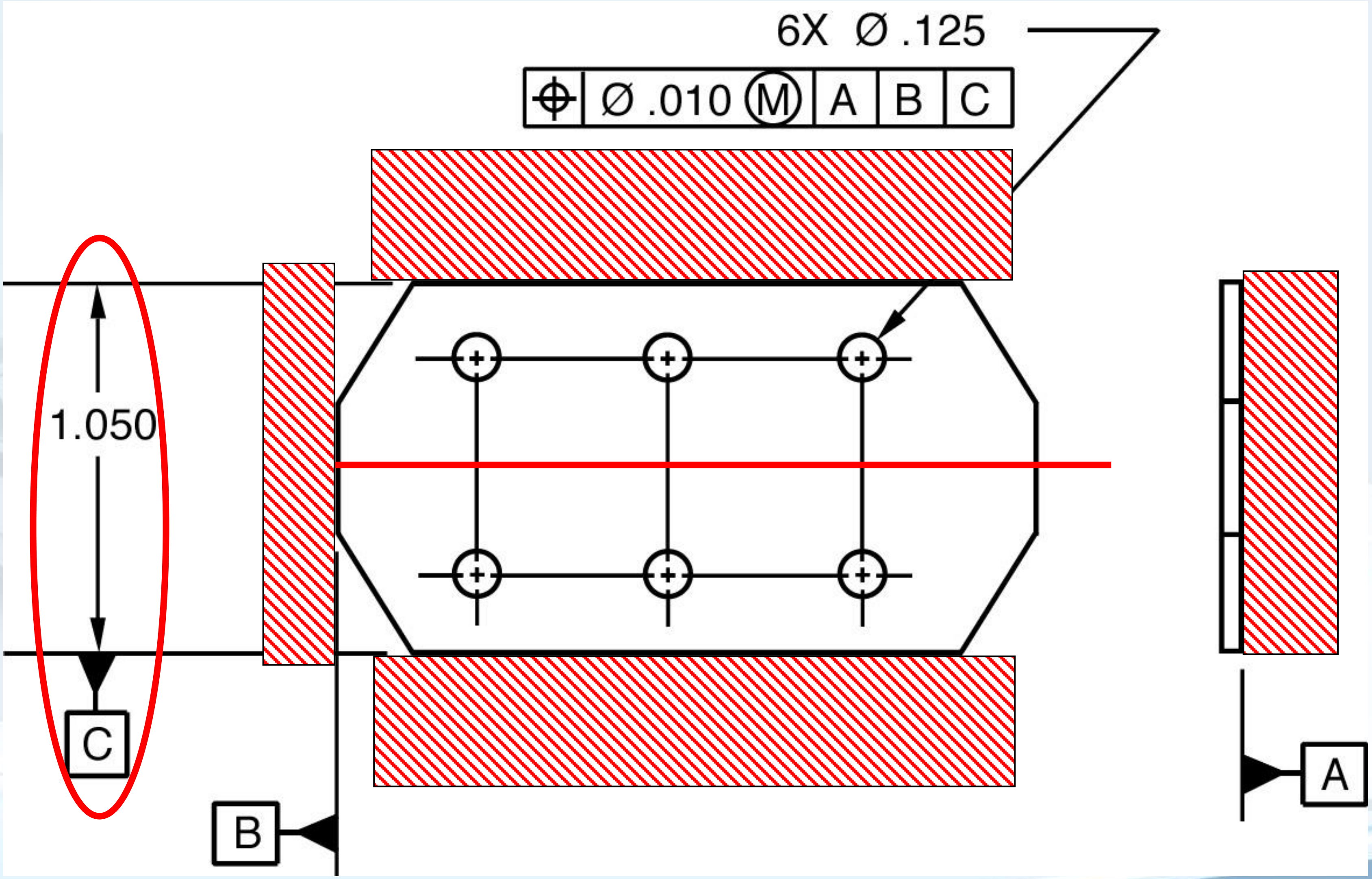
Surface Datum



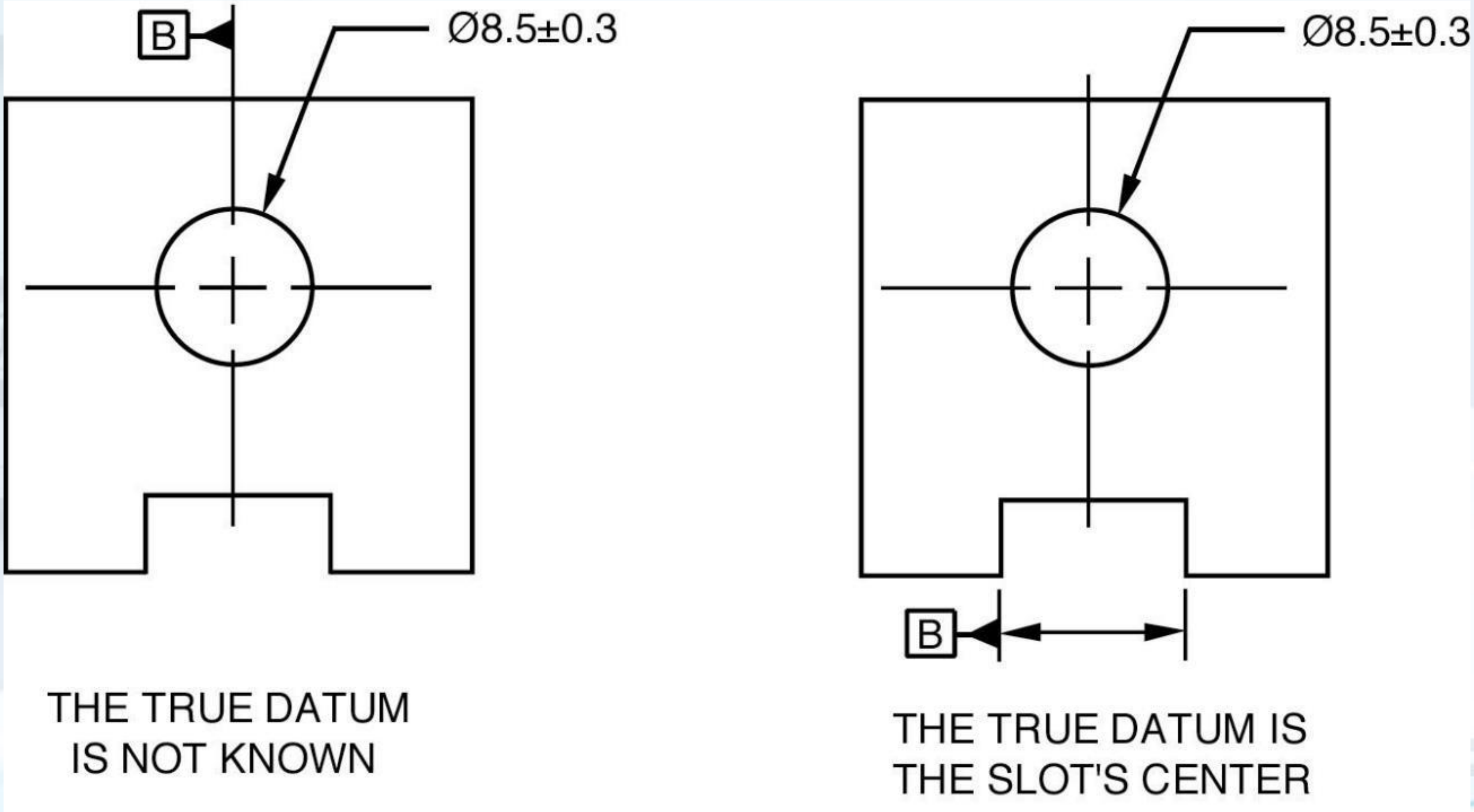
Feature-of-Size Datum



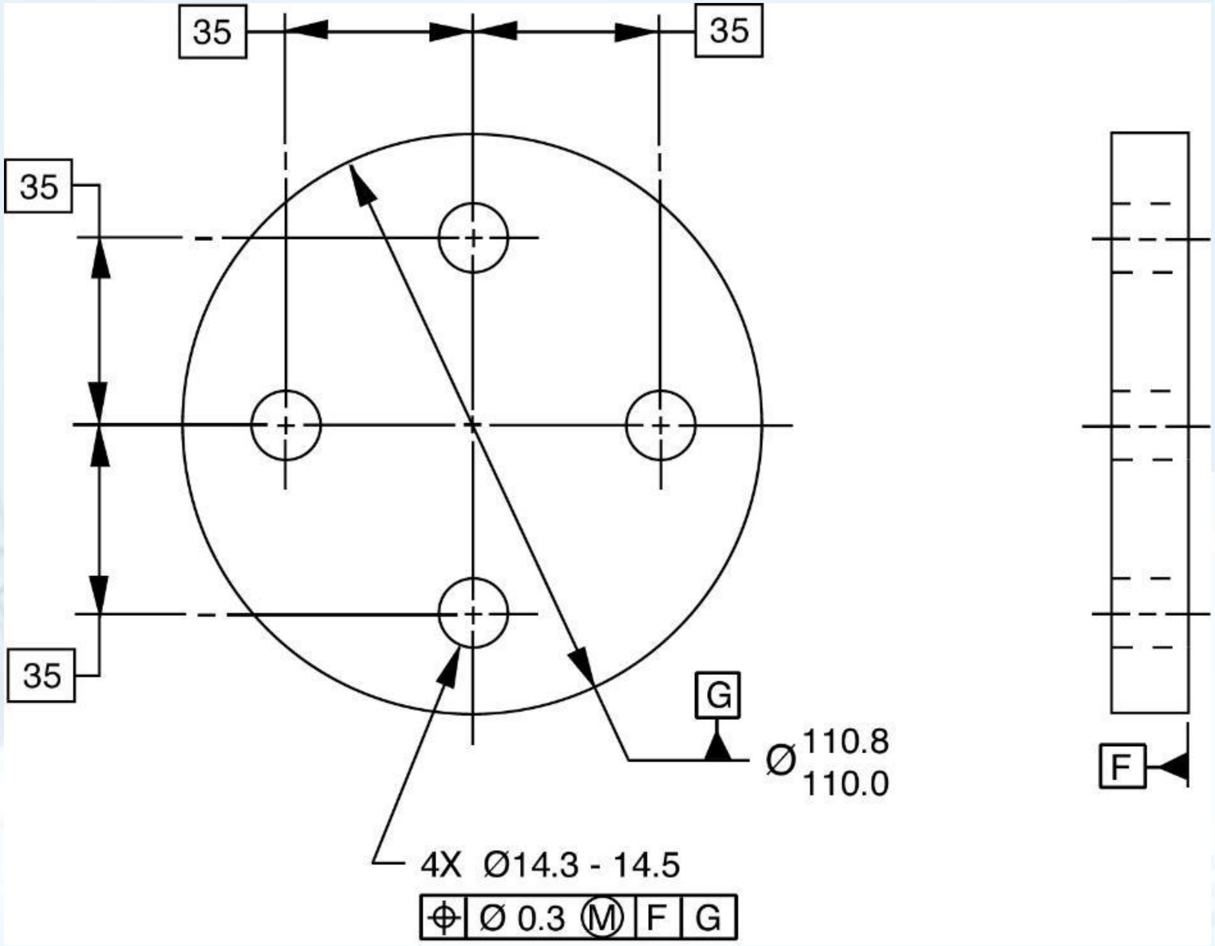
Feature-of-Size Datum



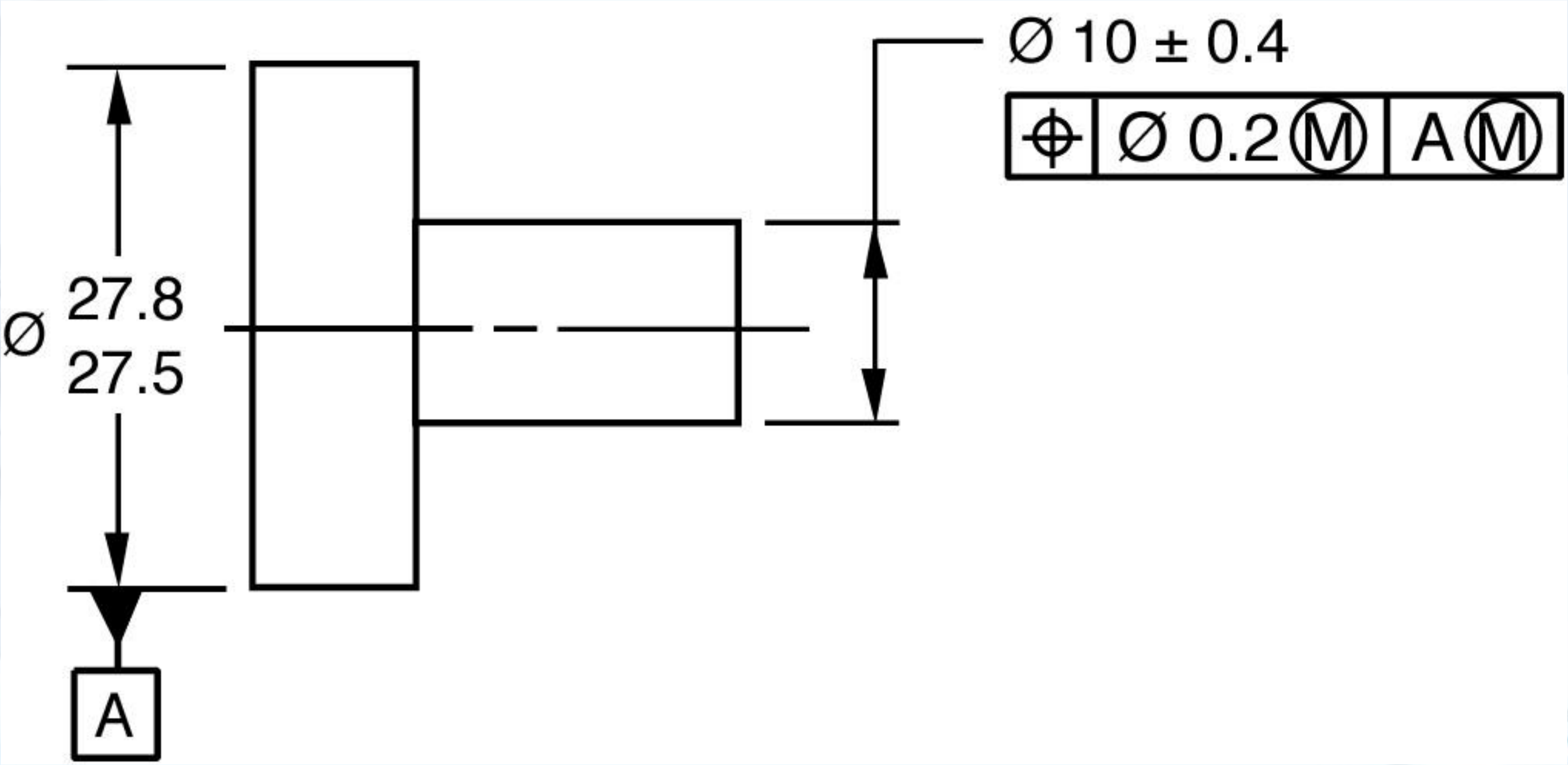
Label the Datum *Feature*



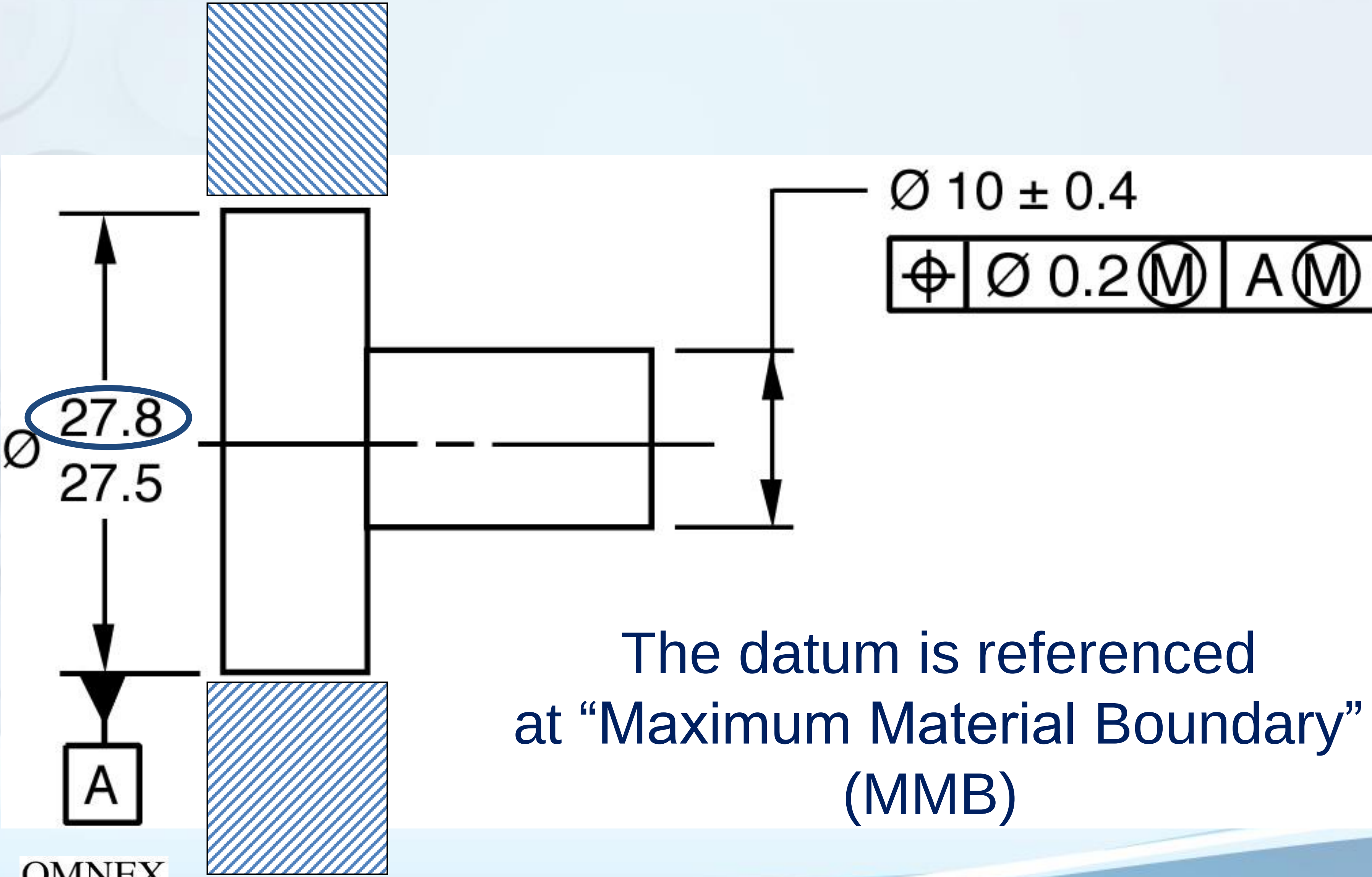
Datum from a Circular Feature



Another "M" Modifier

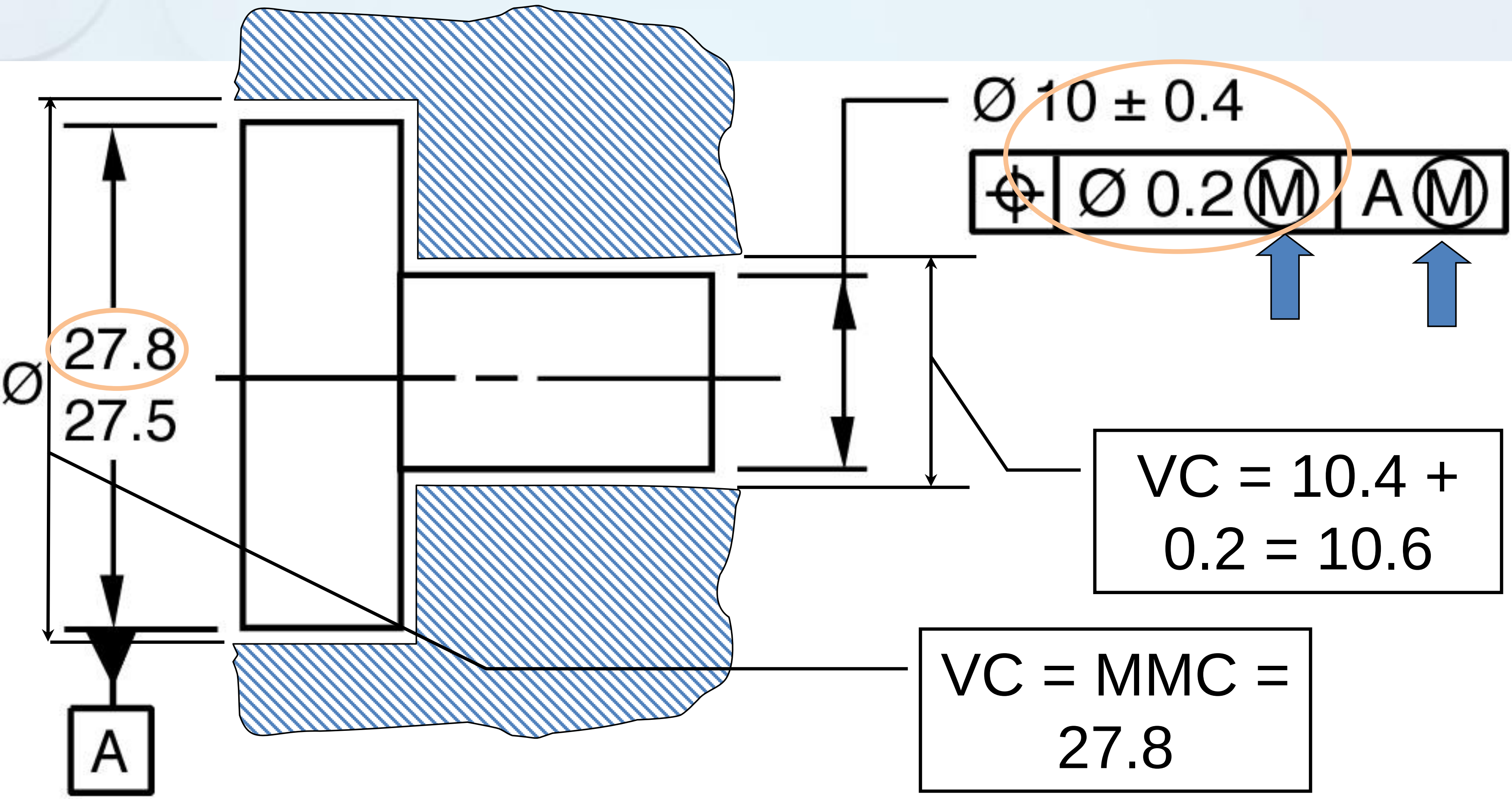


Another “M” Modifier

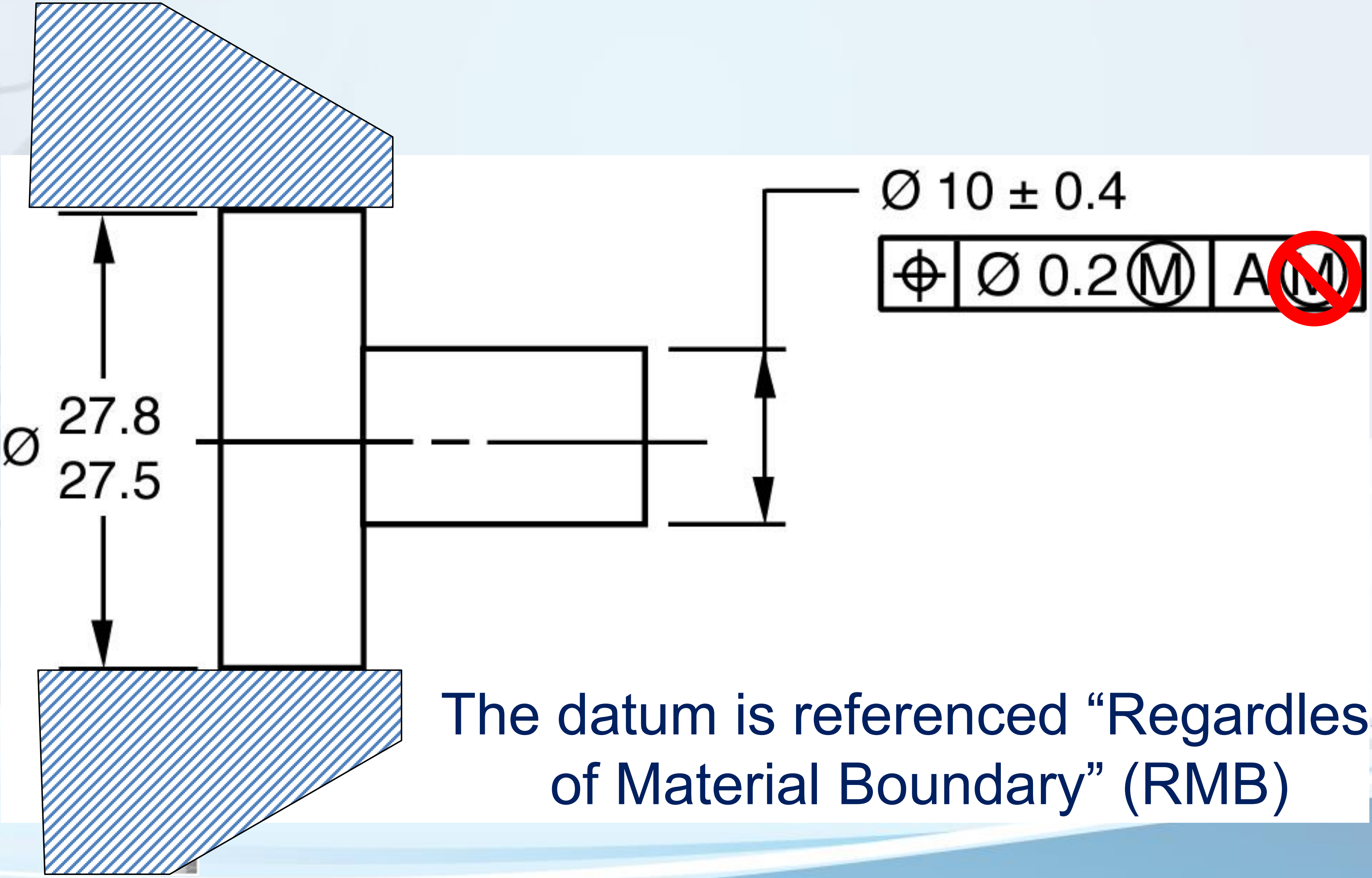


The datum is referenced at “Maximum Material Boundary” (MMB)

Functional Gage

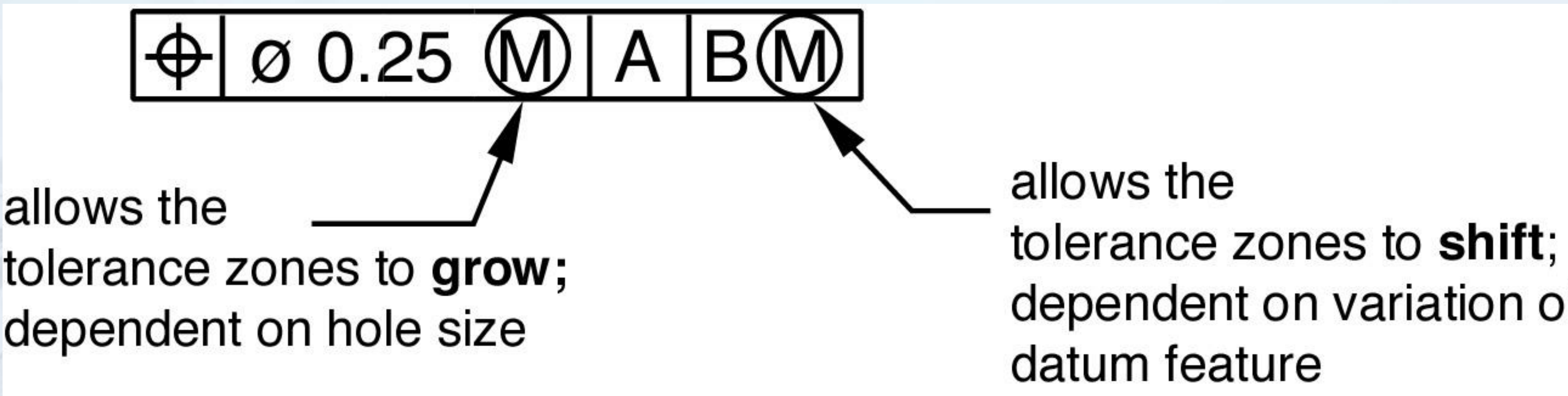


Without the "M"

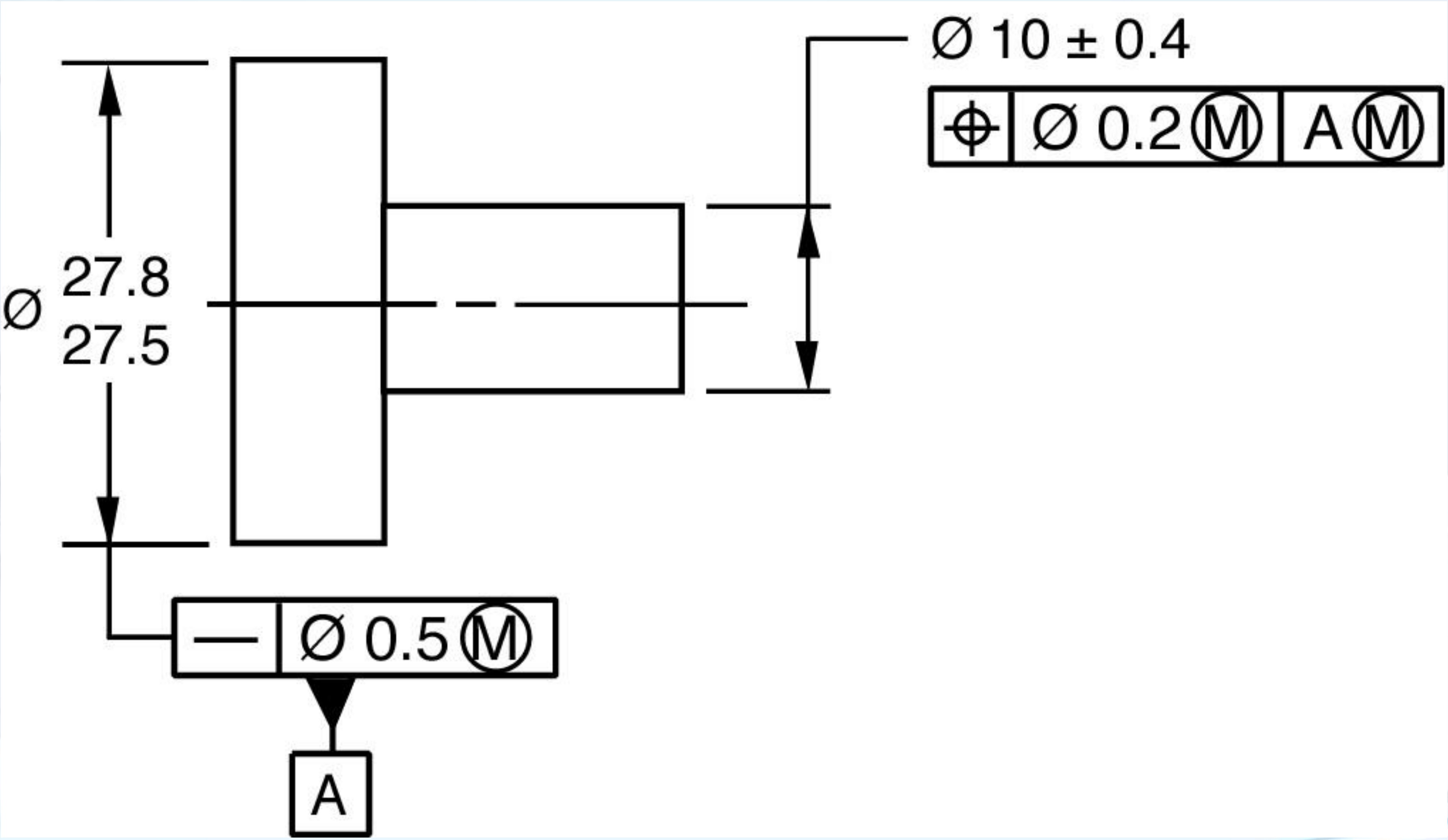


The datum is referenced "Regardless of Material Boundary" (RMB)

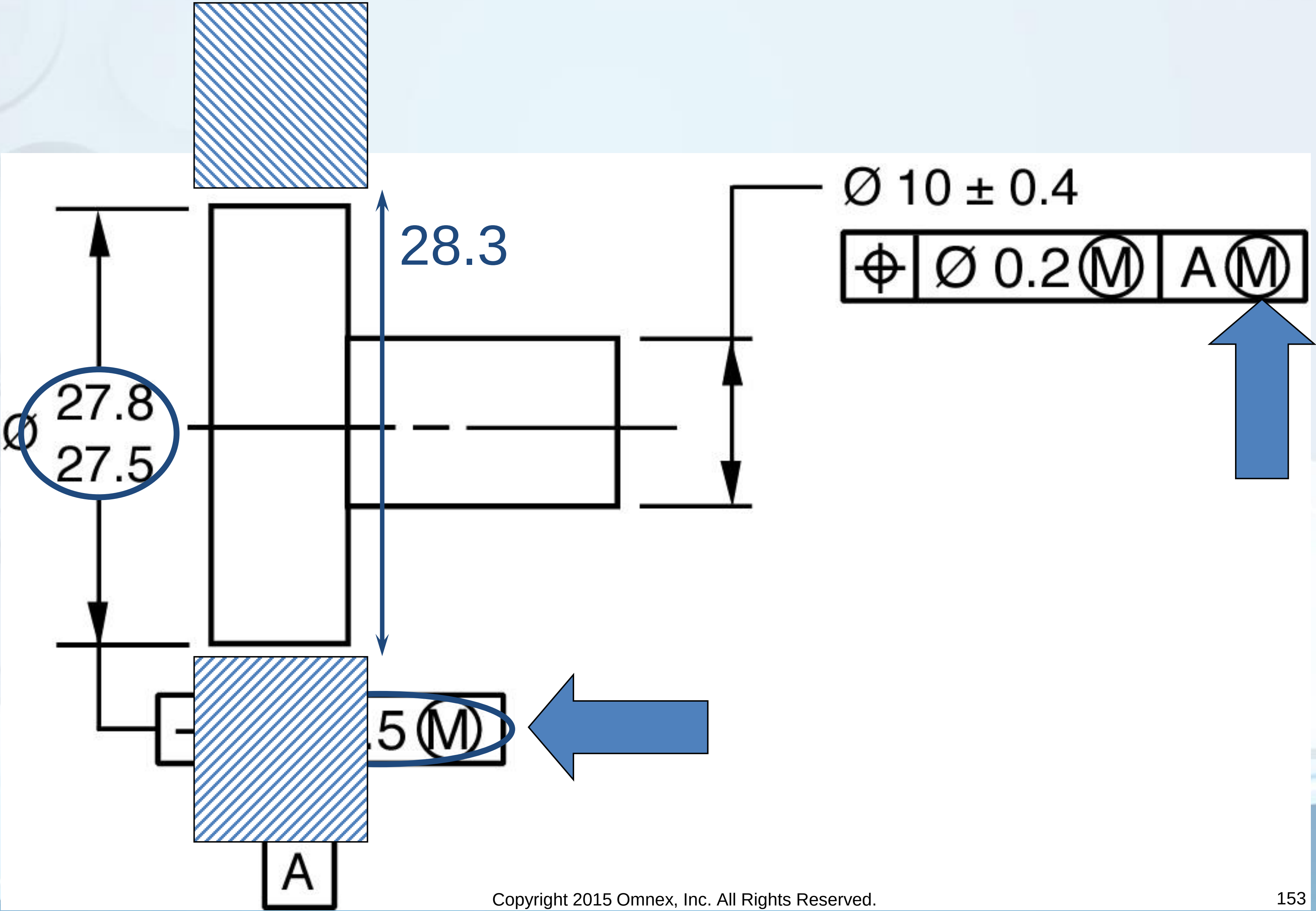
Shift vs. Bonus



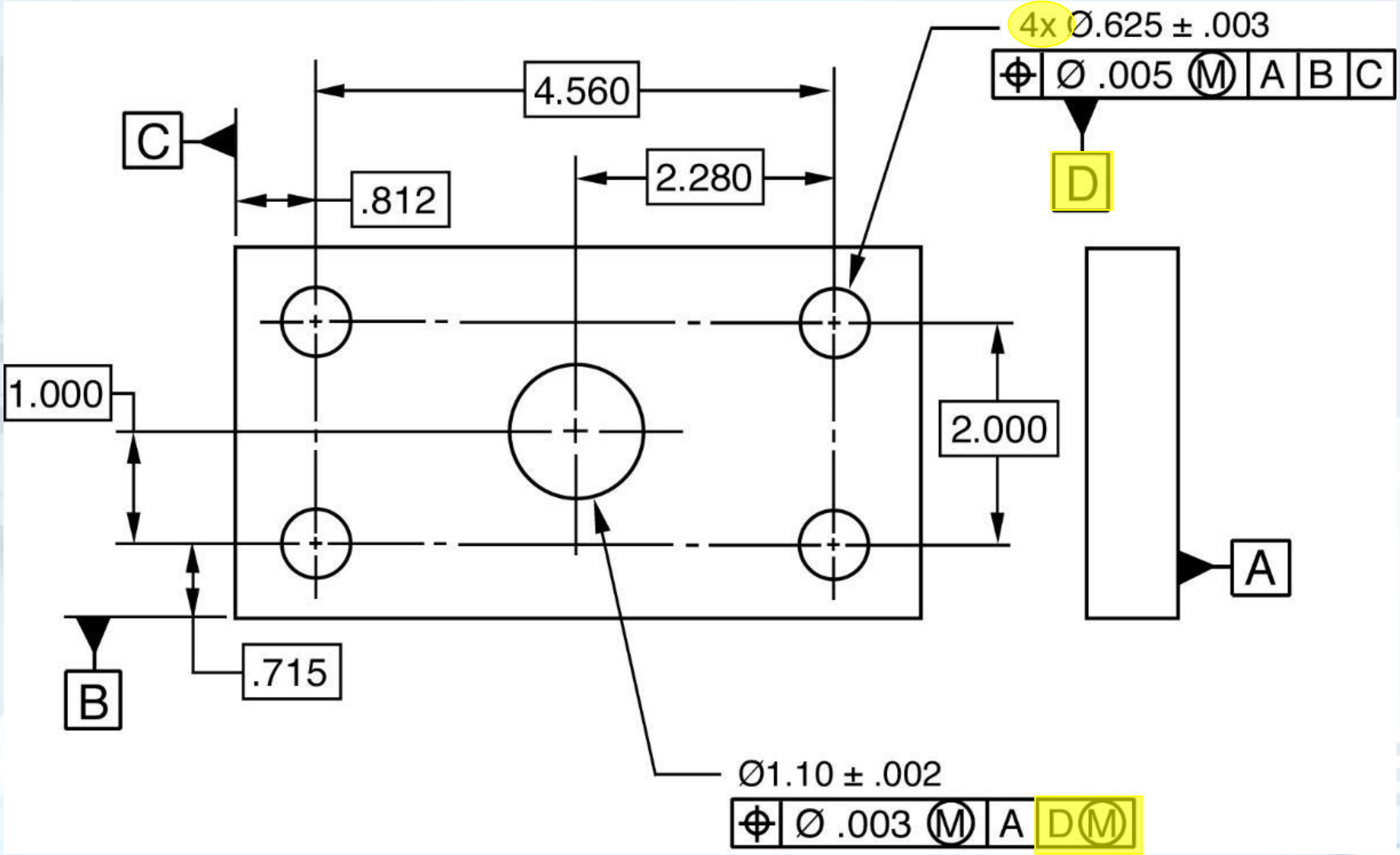
More Datum Shift



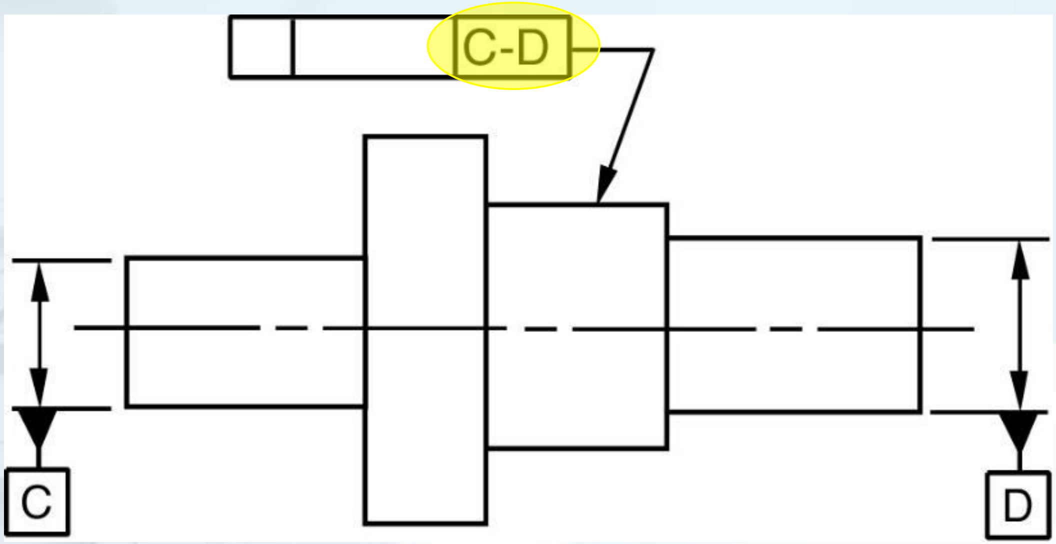
More Datum Shift

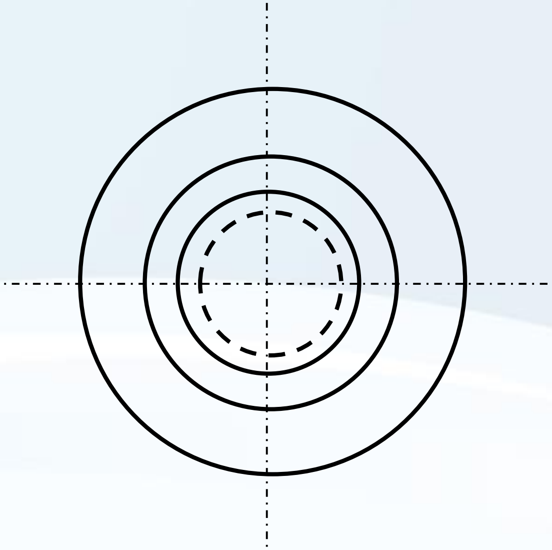


Pattern as a Datum

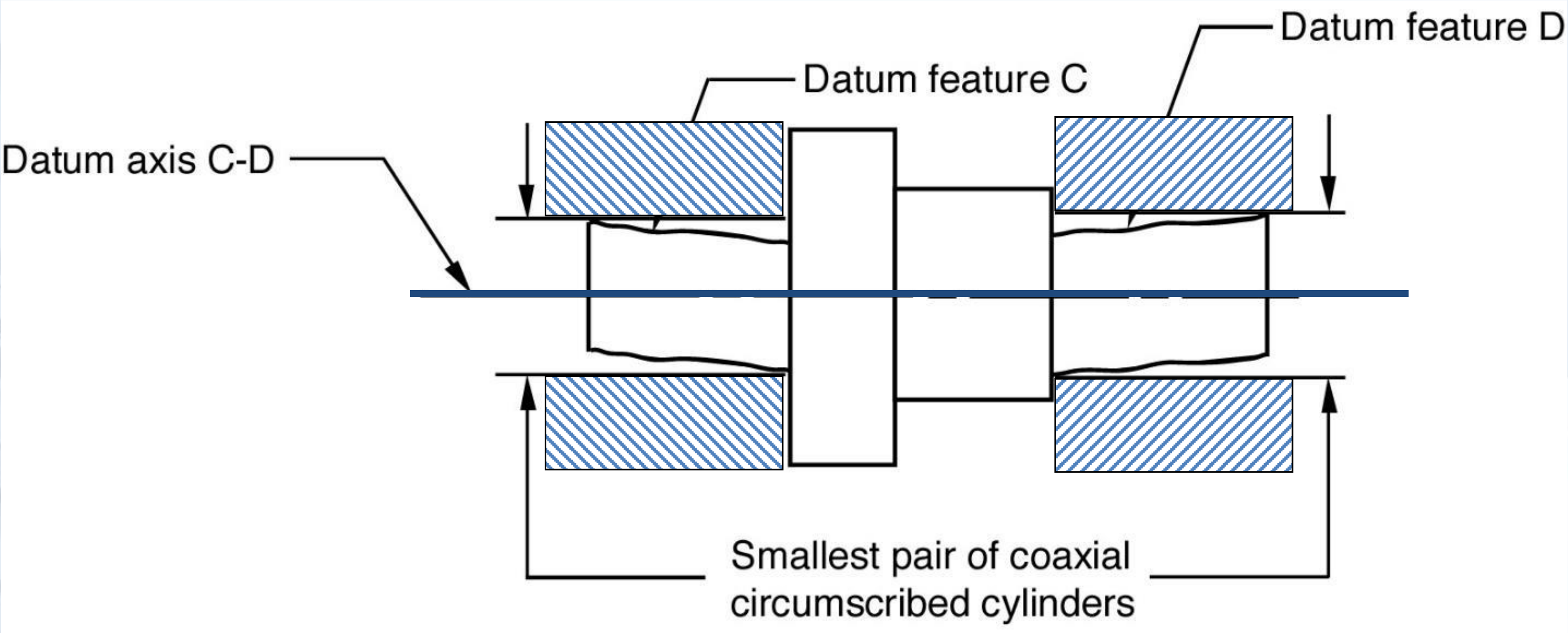


Multiple Datum Features





Multiple Datum Features

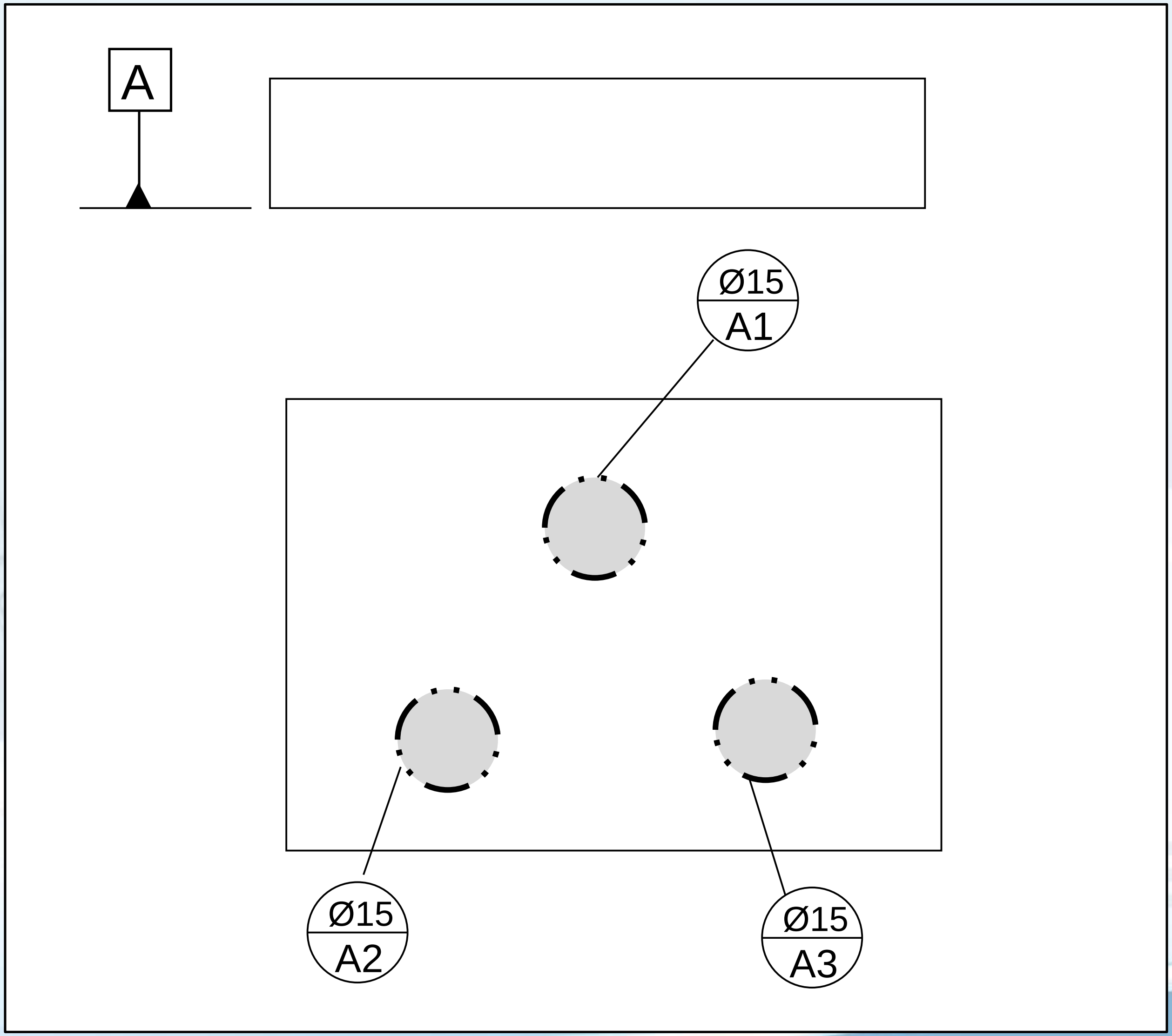


Datum Targets

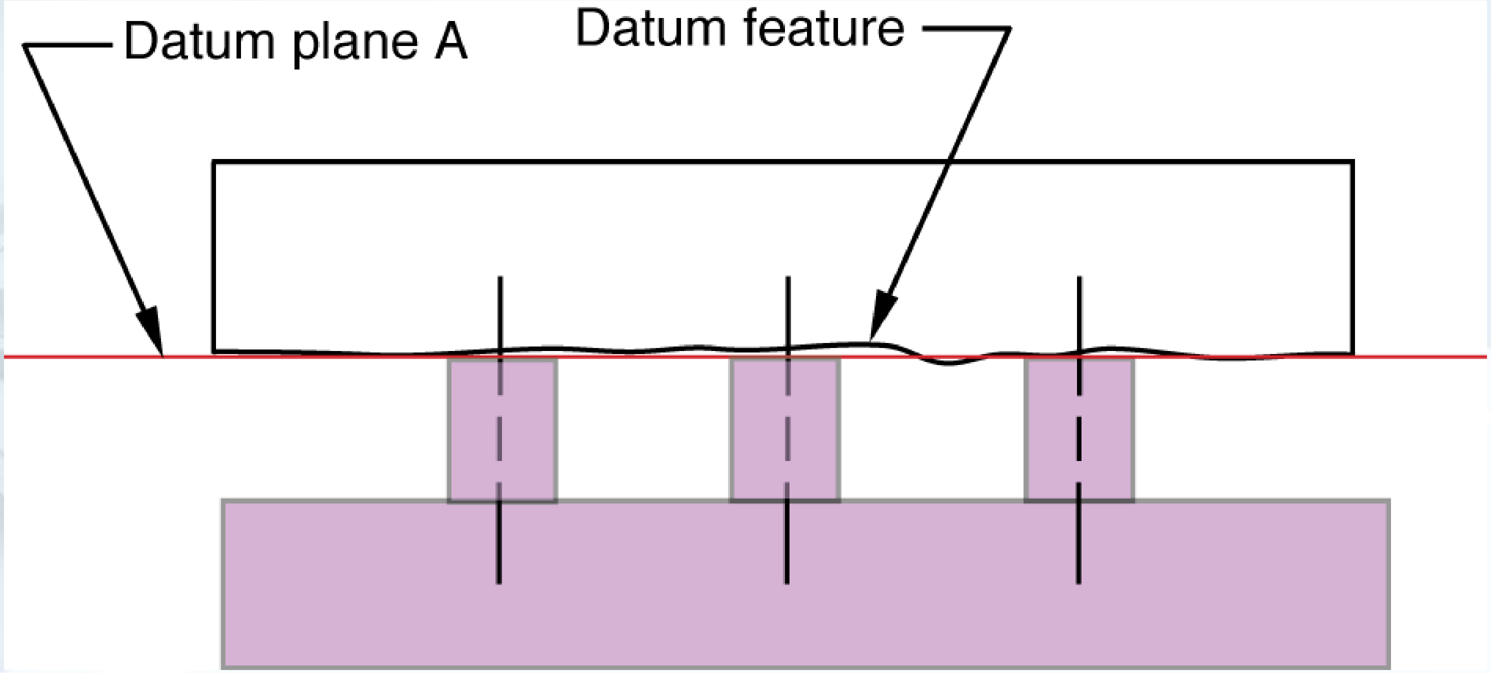
- A "datum target" is a way to identify specific
  - points
  - lines
  - areas

which are then used to establish the true datum.

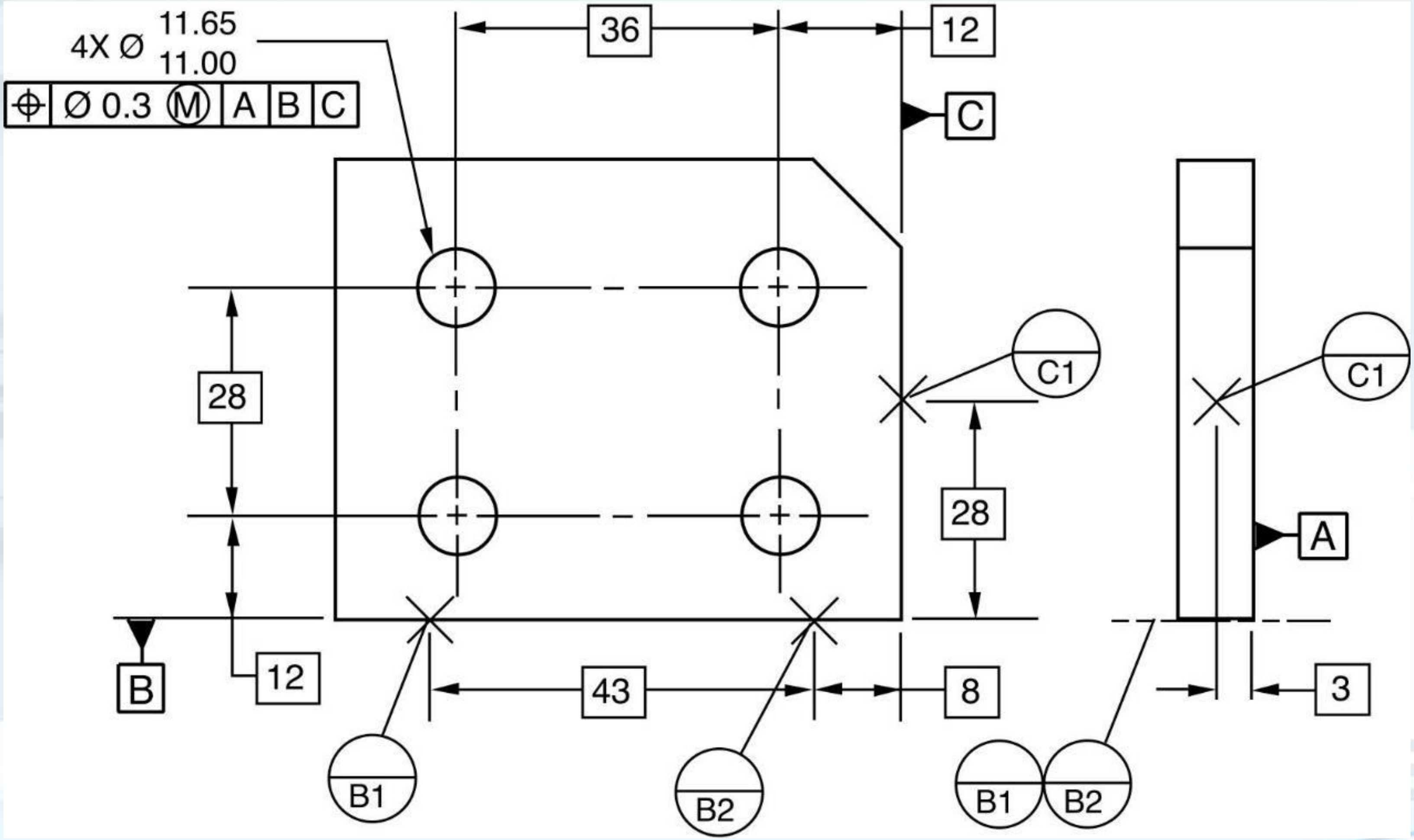
Datum Targets



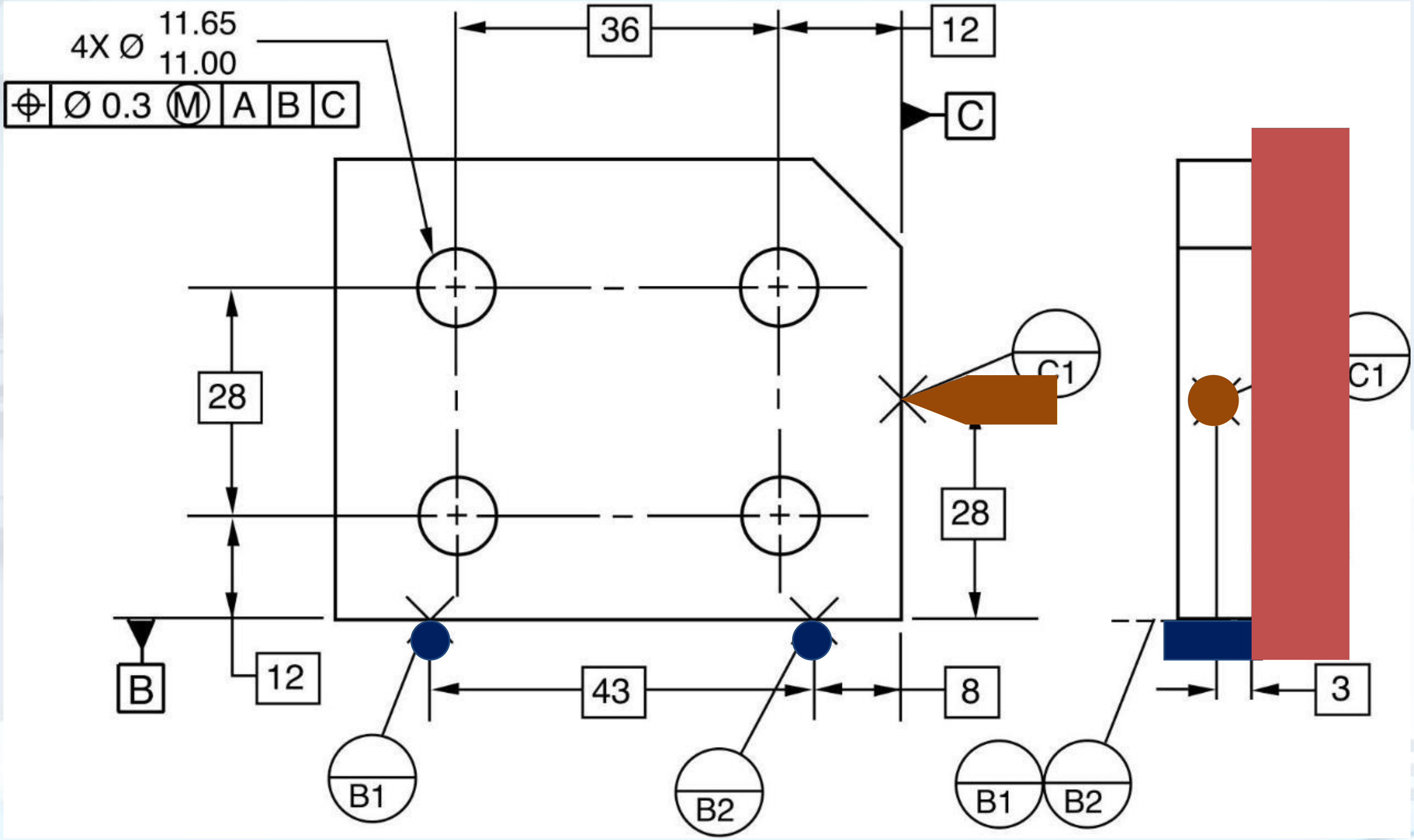
Datum Target Simulator



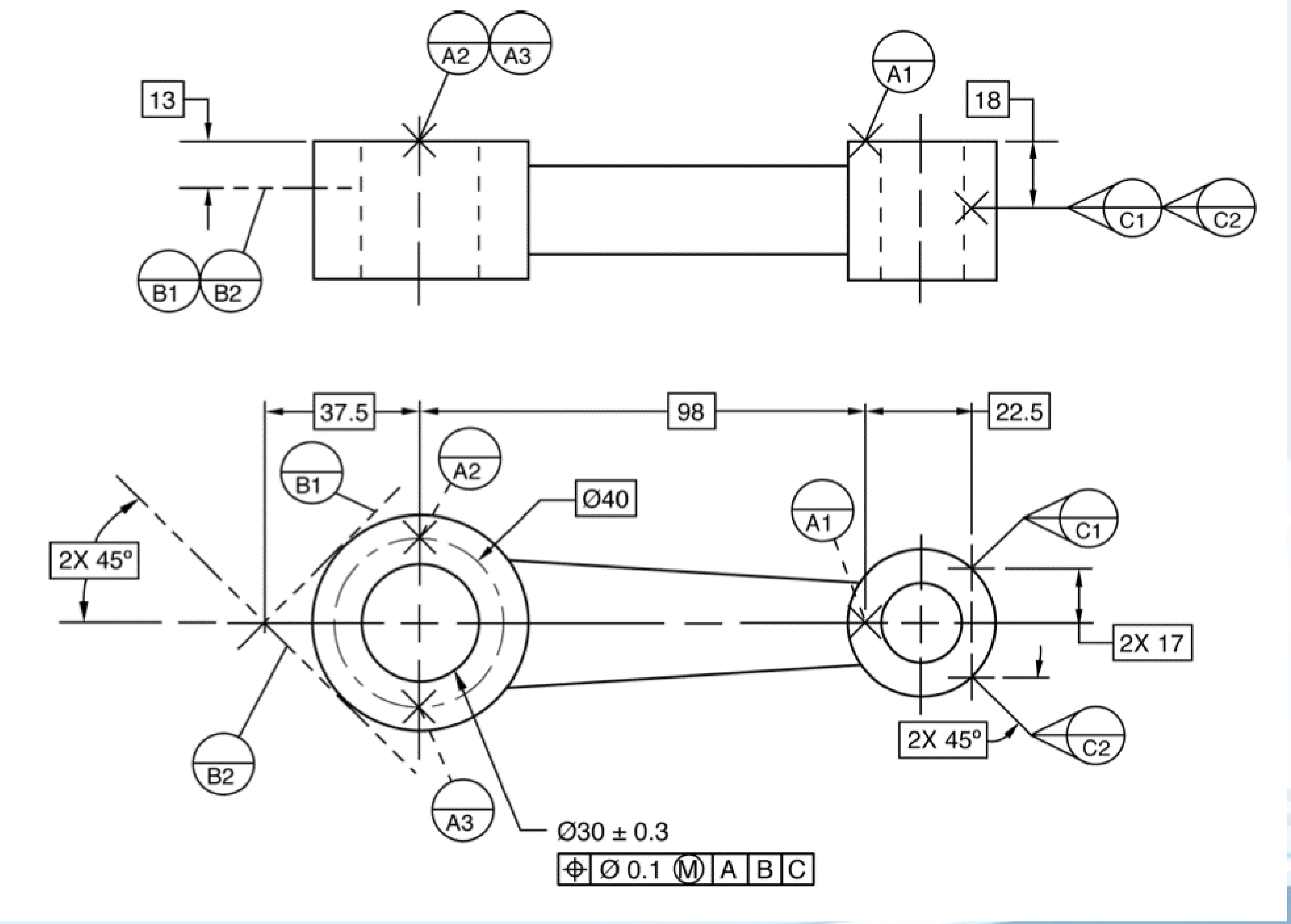
Datum Targets



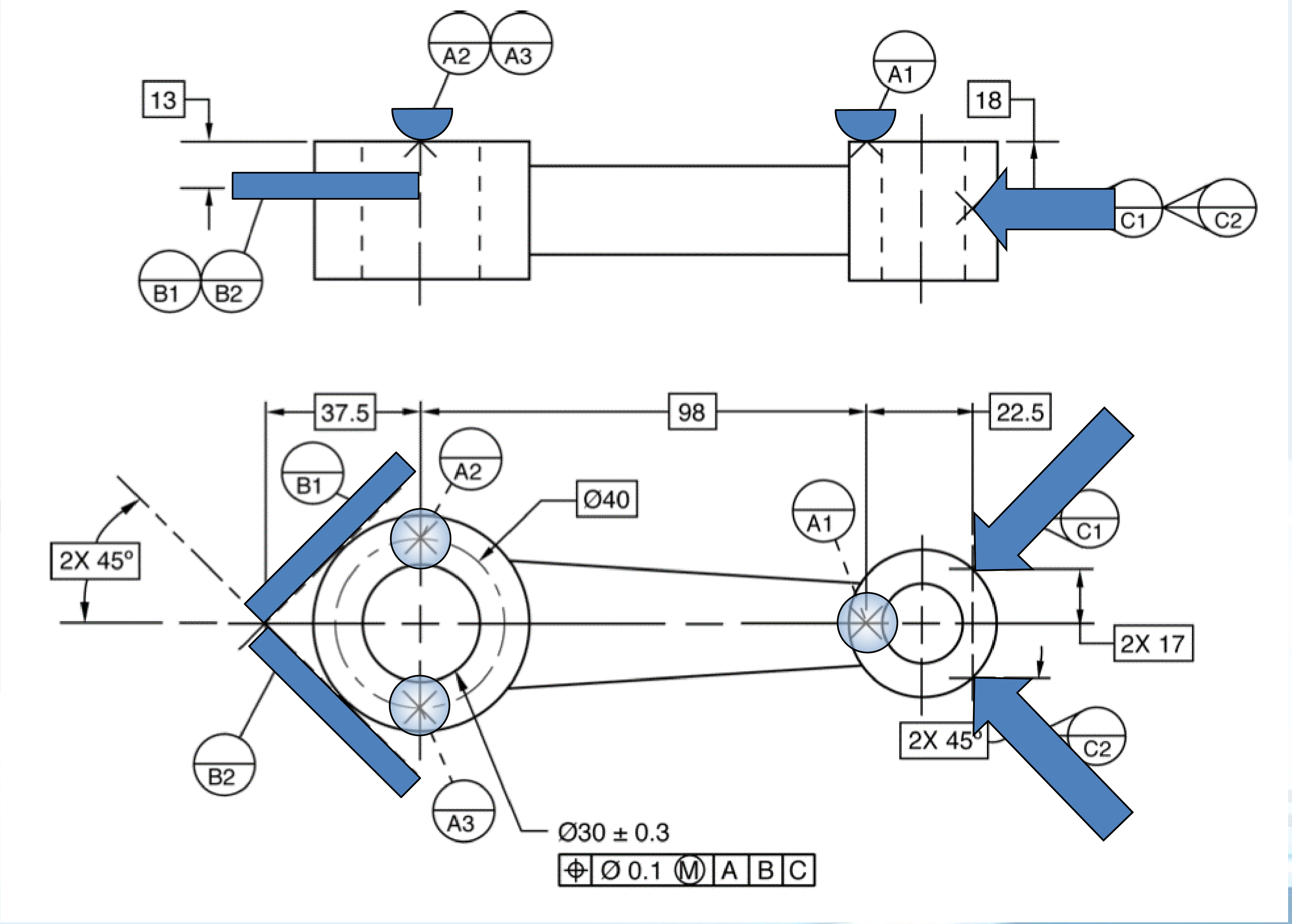
Datum Targets



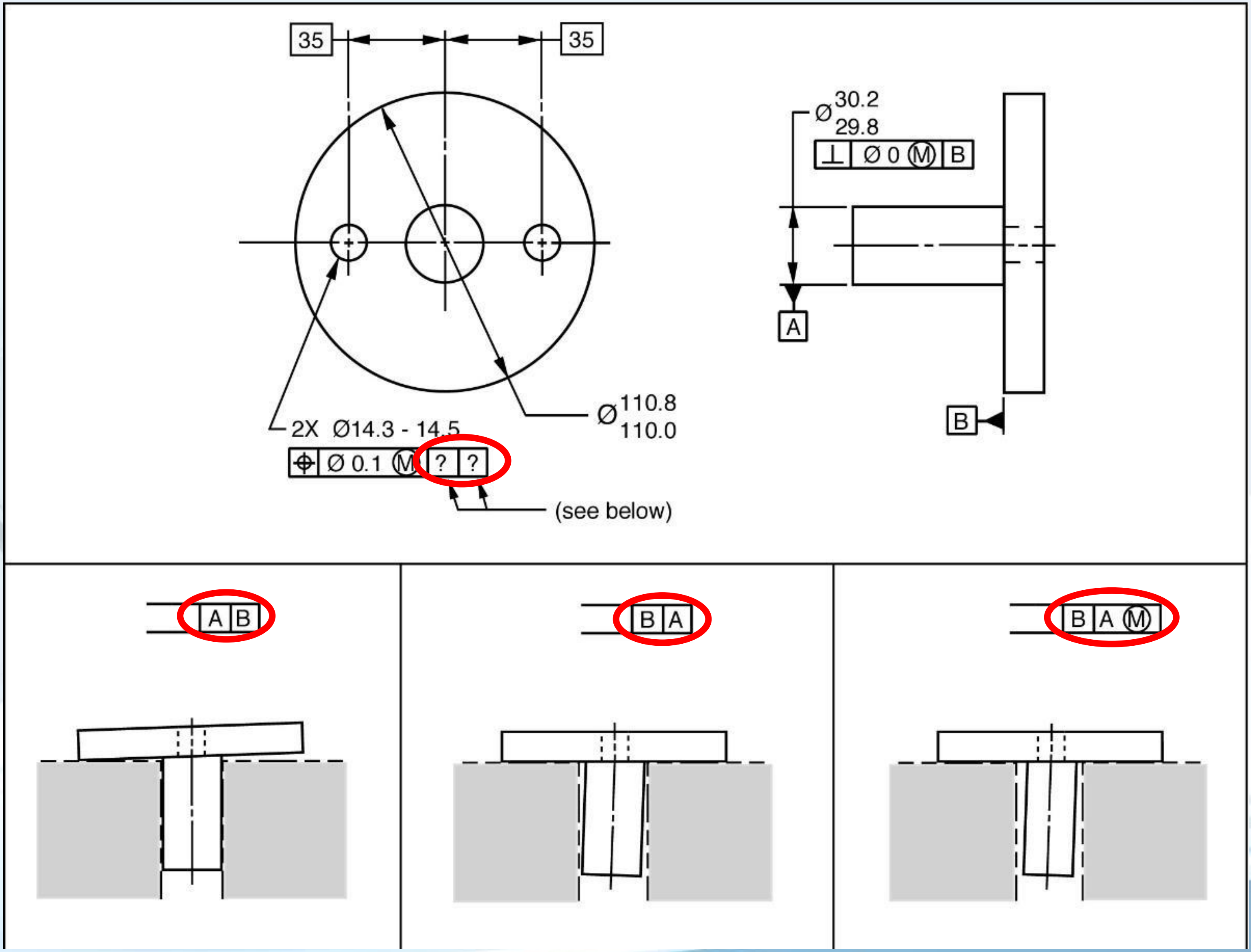
Movable Target



Movable Target



Datum Summary



Chapter 4: Datums – What We Covered

Learning Objectives

You should now be able to:

- Define datum, and datum feature
- Identify primary, secondary, and tertiary datums for a given feature control frame
- Select appropriate datums based on function
- Explain how to determine if a datum is derived from a surface or a feature of size
- Explain the effect of “M” after a datum reference
- Identify and explain datum targets

Chapter Agenda

- Identifying Datums
- Selecting Datums
- Simulating a Surface Datum
- 6 Degrees of Freedom
- Datum Targets

Chapter 5

Profile Tolerances

… the most versatile of the GD&T Symbols

Chapter 5: Profile Tolerances – What We Will Cover

Learning Objectives

At the end of this chapter, you will be able to:

- Explain the difference between profile of a line and profile of a surface
- Determine if a profile tolerance references a datum, and how that impacts the tolerance zone
- Identify the special callouts for all around, all over, non-uniform, and unequal
- Interpret a composite profile tolerance, and what qualities each tolerance number controls

Chapter Agenda

- Profile Tolerances
- Profile with Datums
- Composite Profile

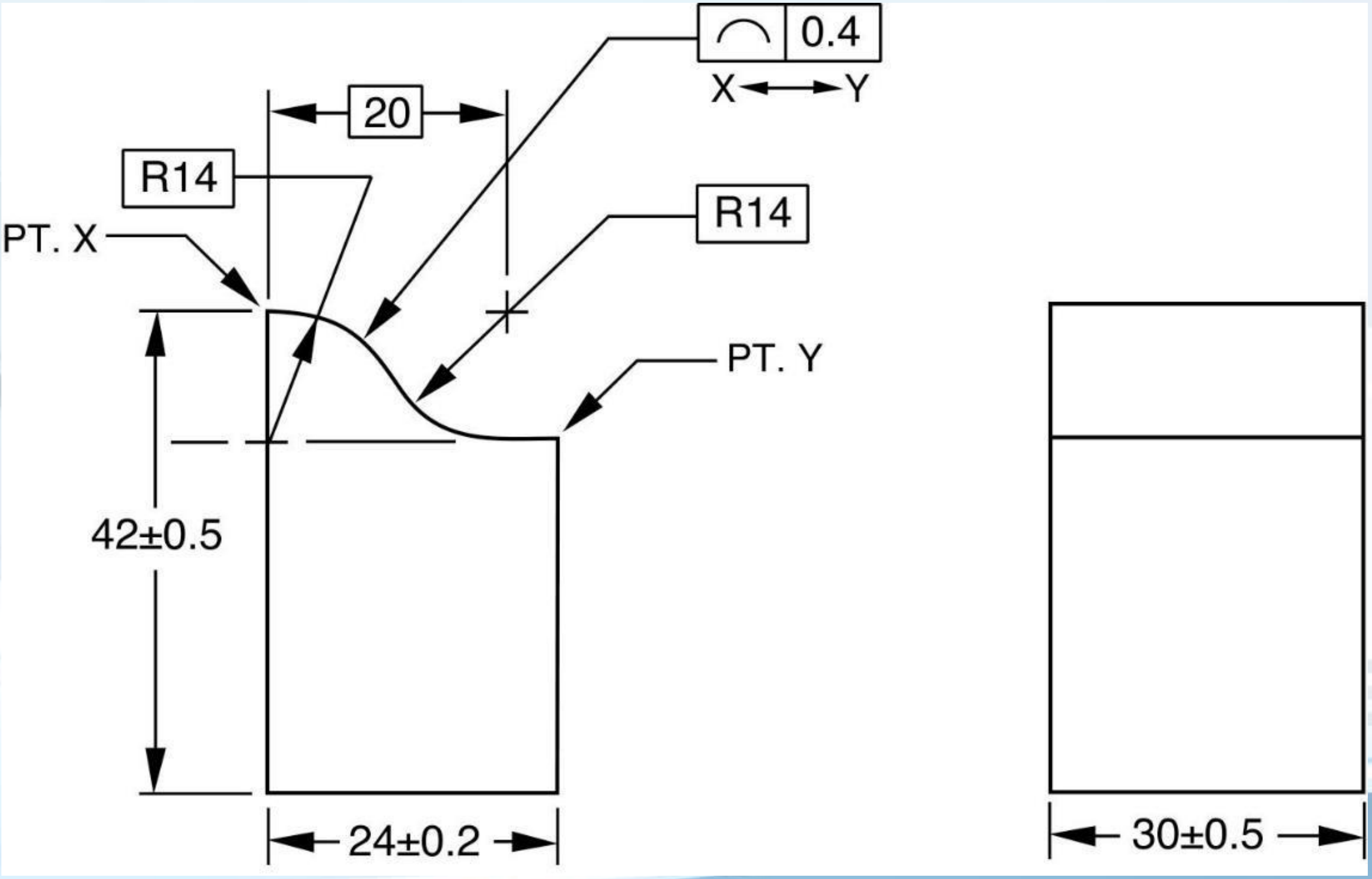
Profile Tolerances

⌒ Profile of a Line (a 2-D control)

⌓ Profile of a Surface (a 3-D control)

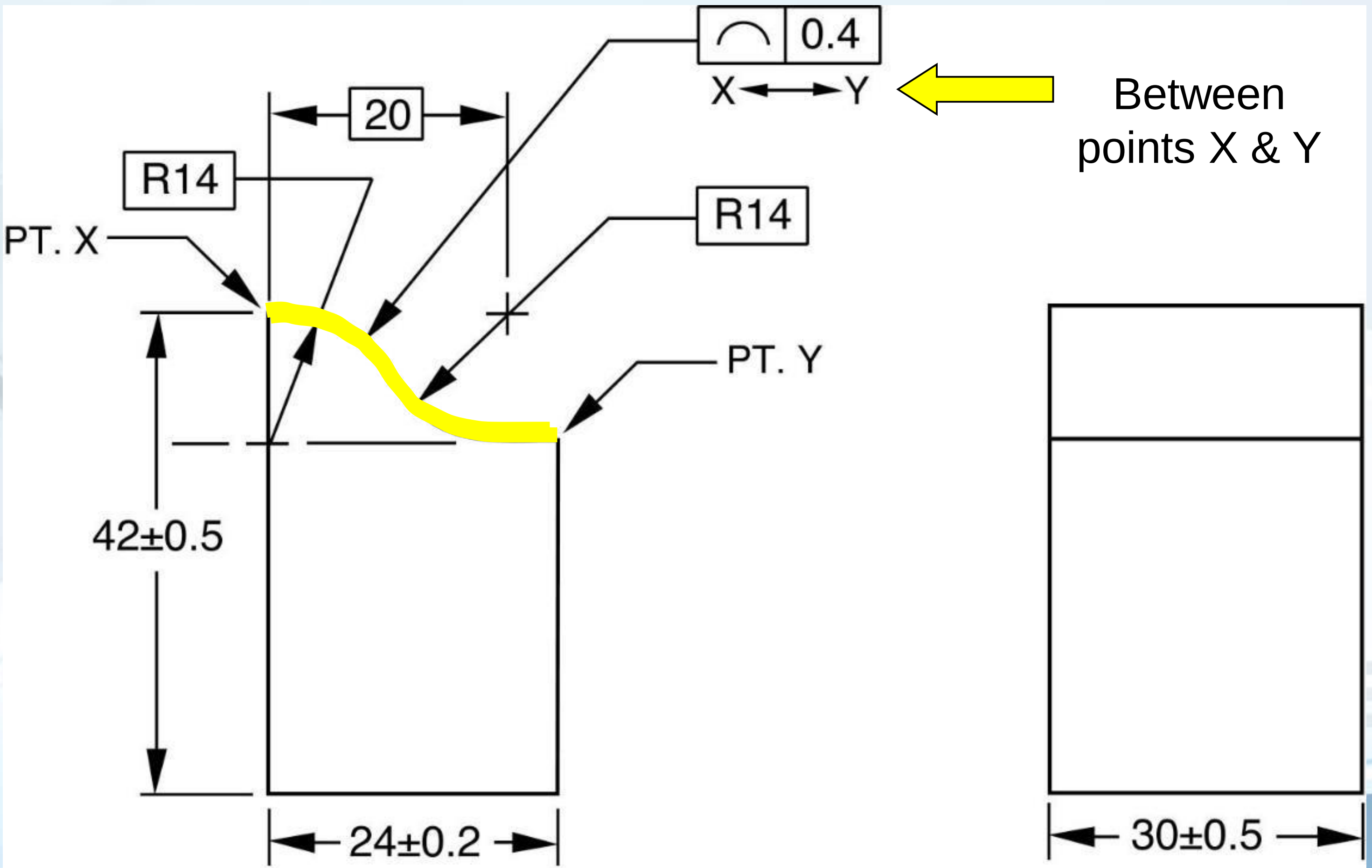
Profile of a Line

- **Basic dimensions** are required to define the "true profile."



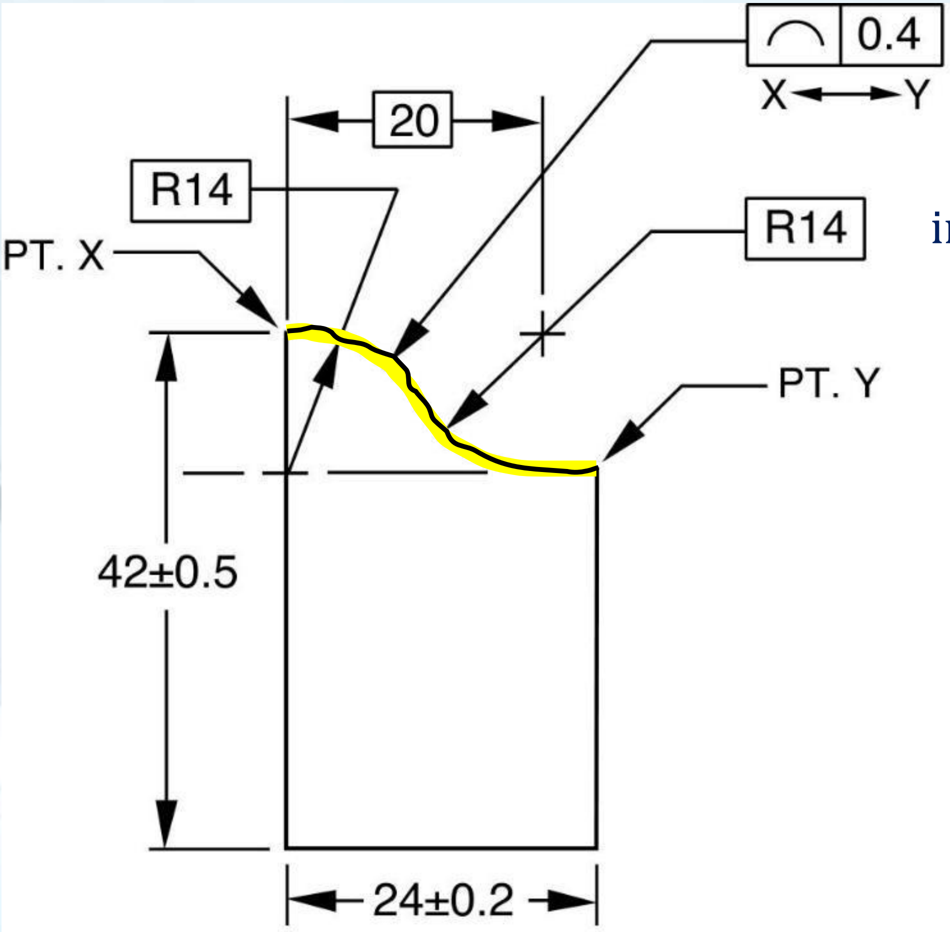
Profile of a Line

- The **tolerance zone** is equally spaced on both sides of the true profile.

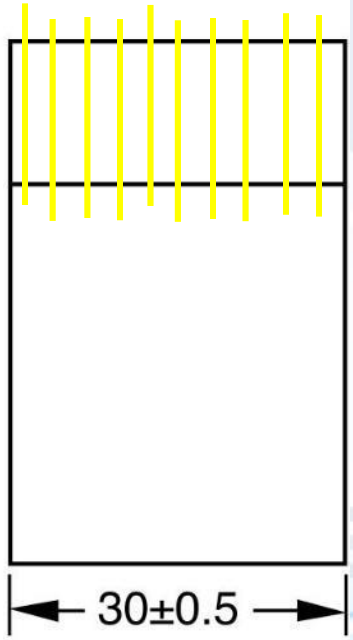


Between points X & Y

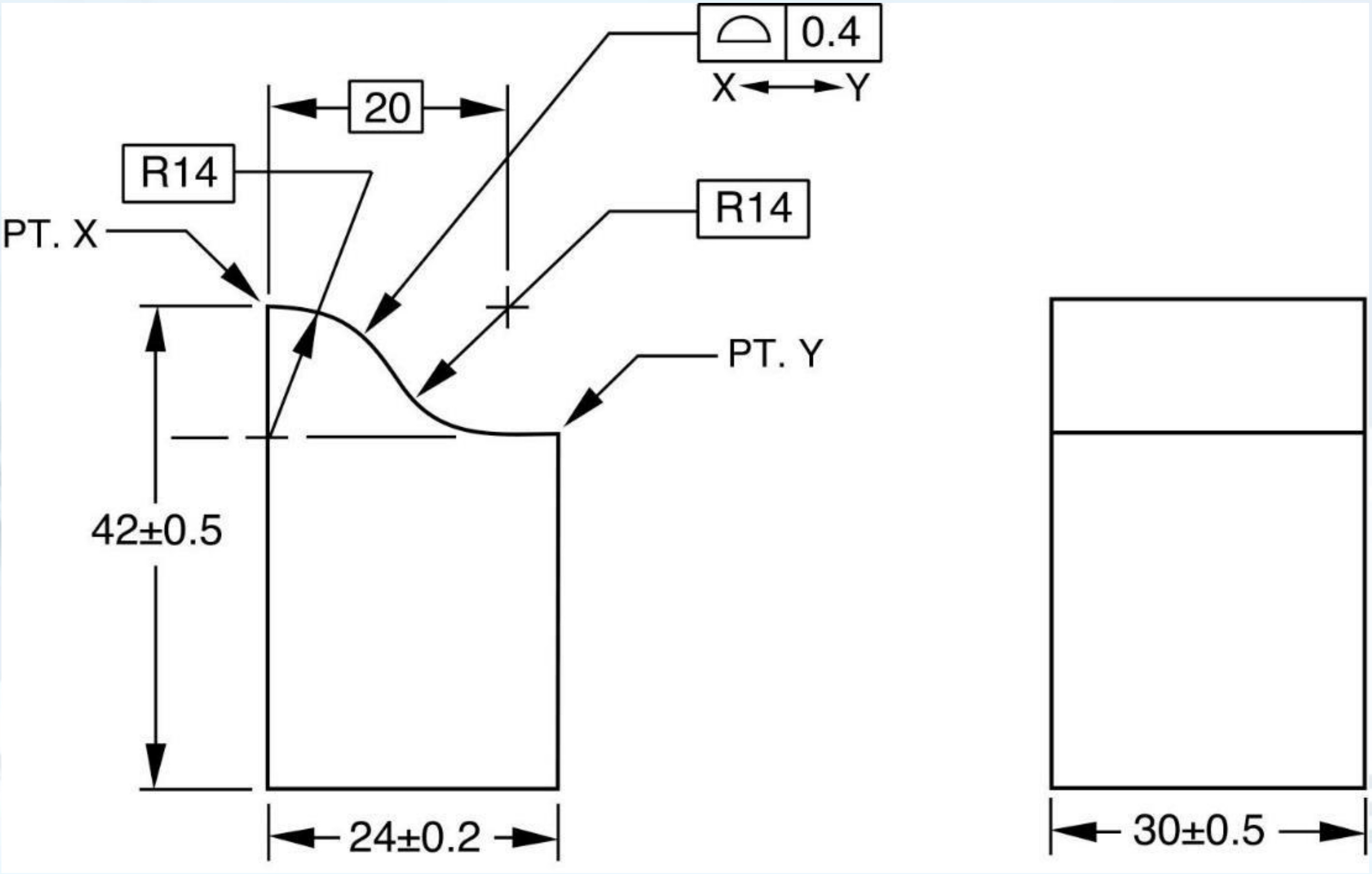
Profile of a Line



Notice that it looks at individual cross-sections

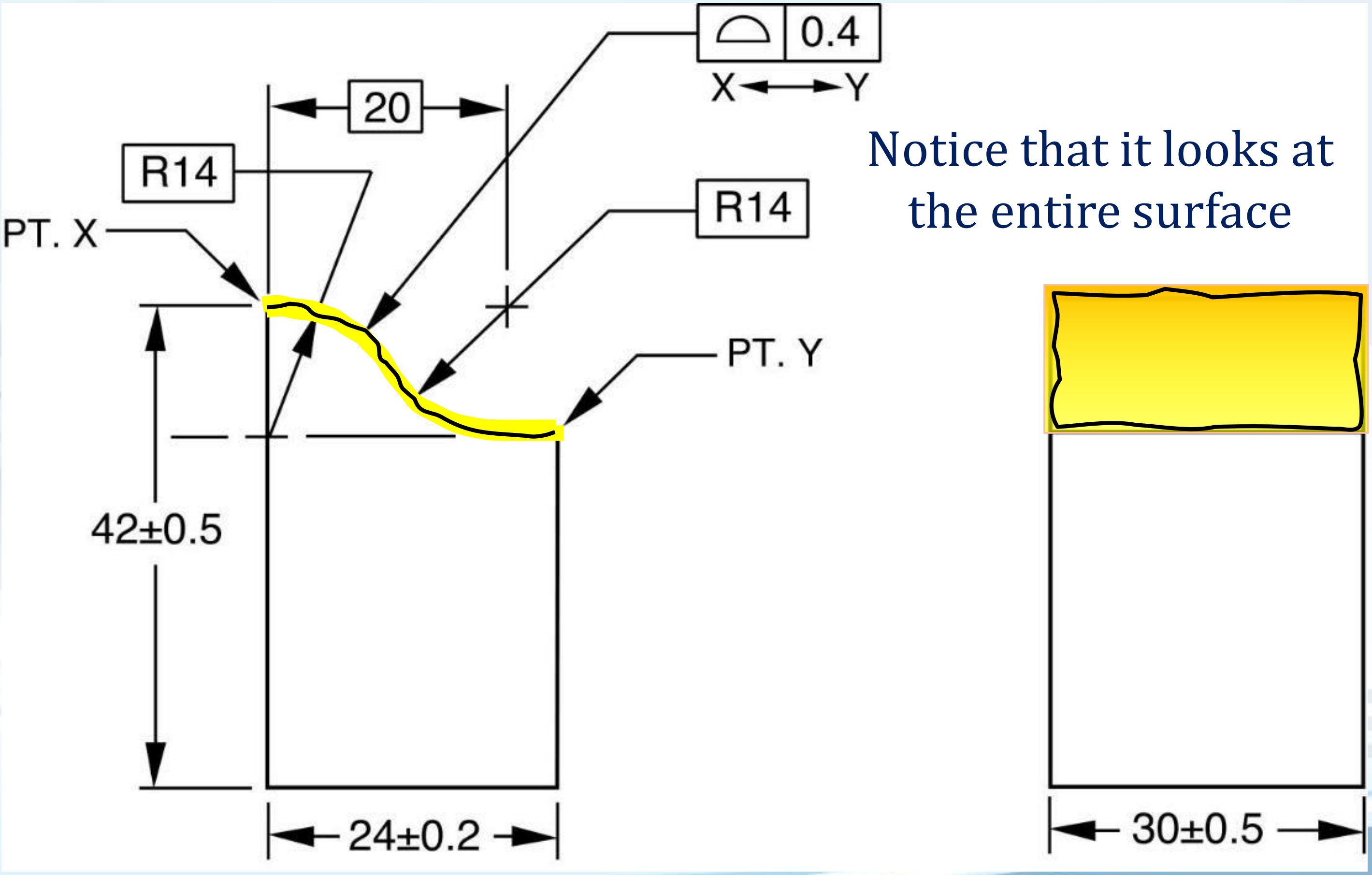


Profile of a Surface



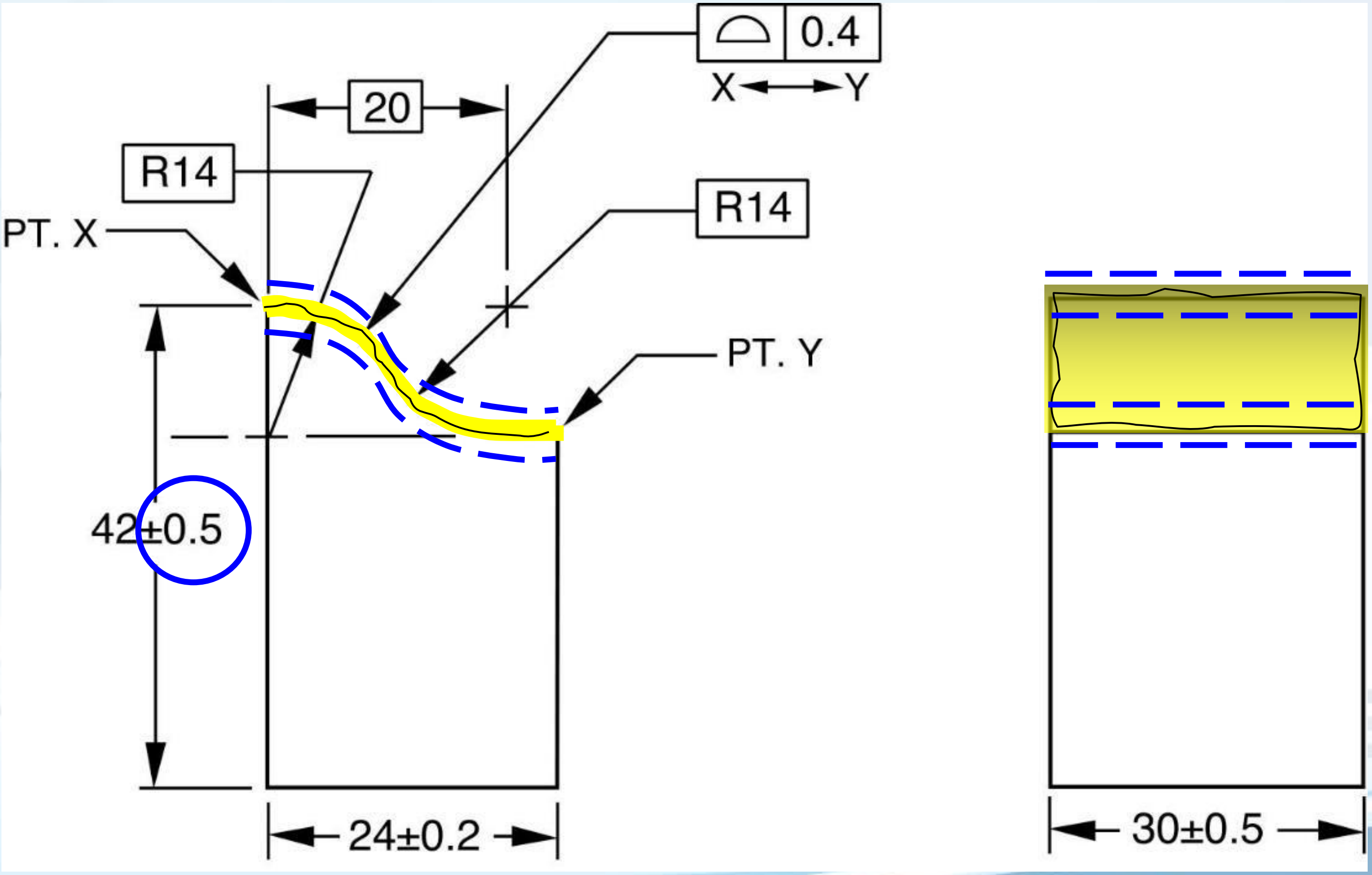
Profile of a Surface

- All points of the toleranced surface must lie within the **tolerance zone**.

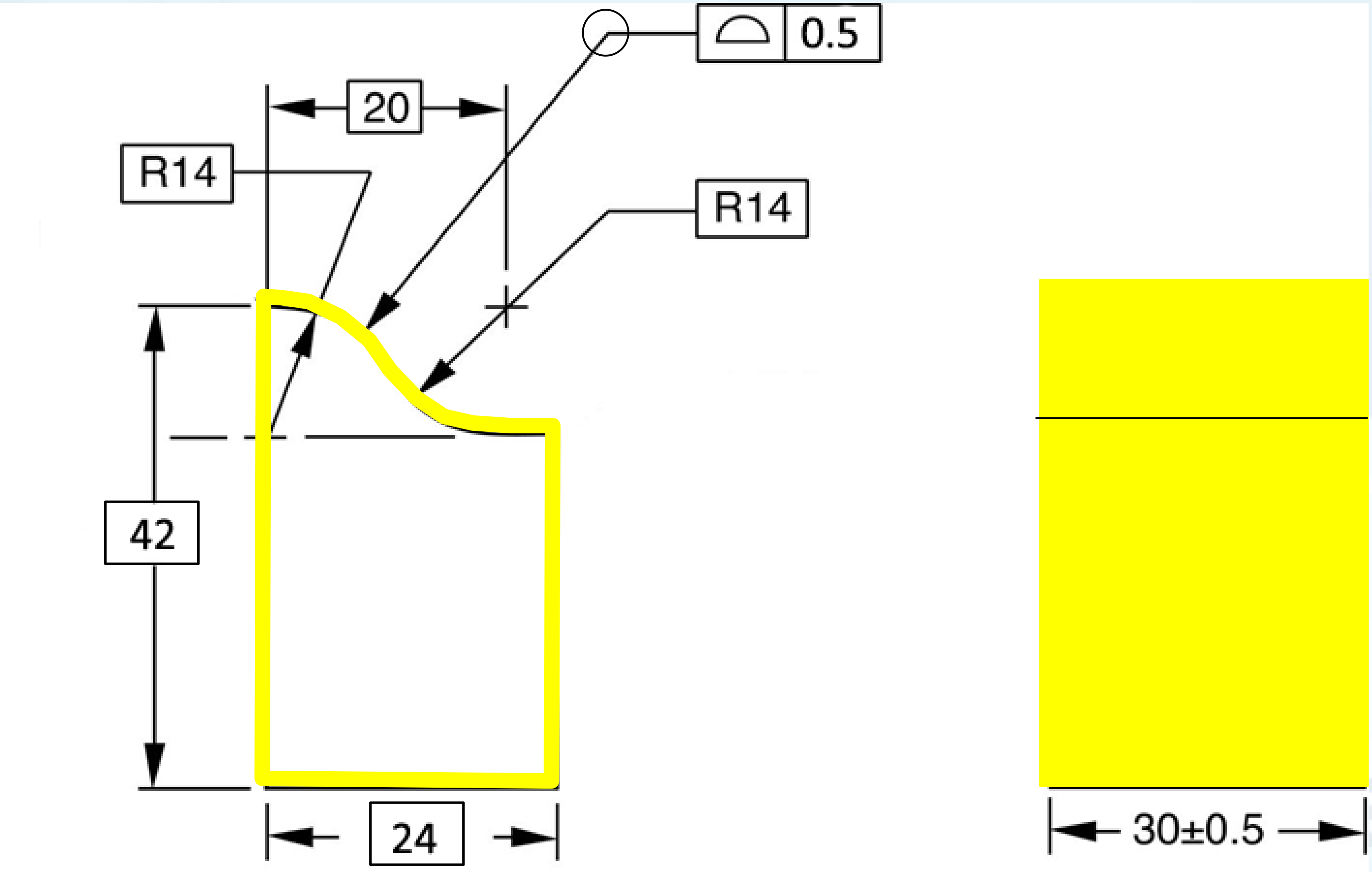


Profile of a Surface

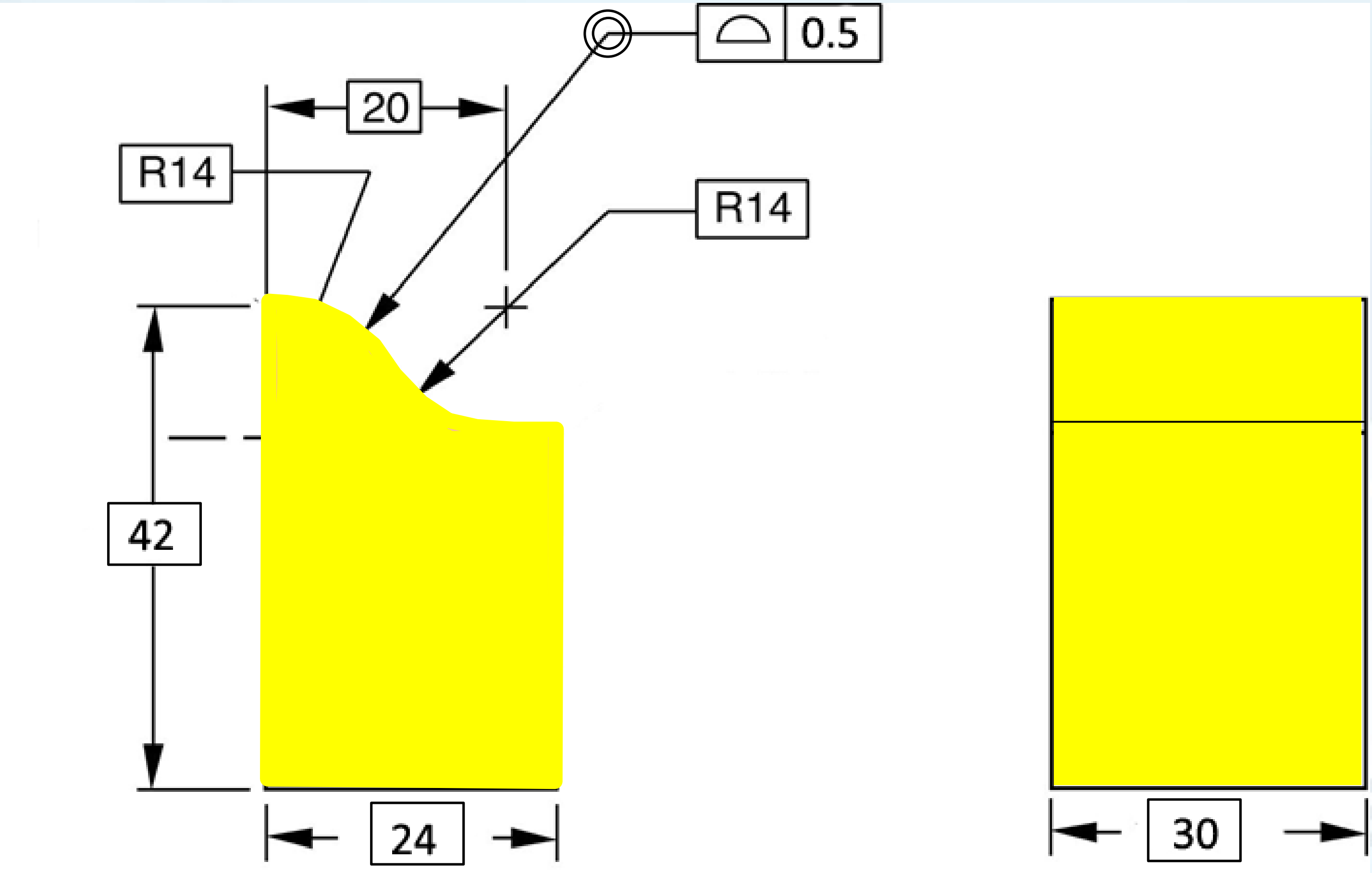
- In these examples, the controlled surface is not related to datums, so can be located anywhere within the limits of size.



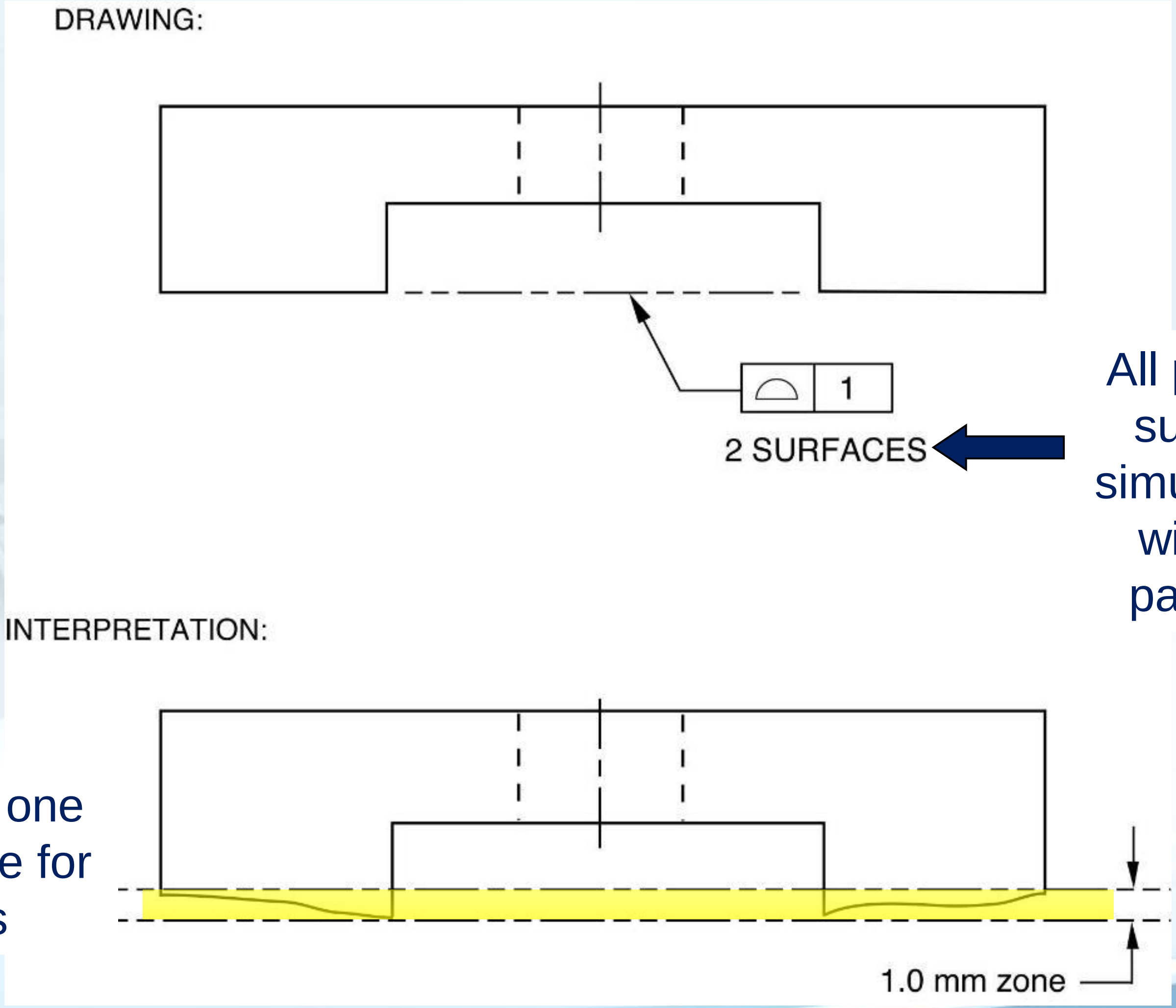
All Around



All Over



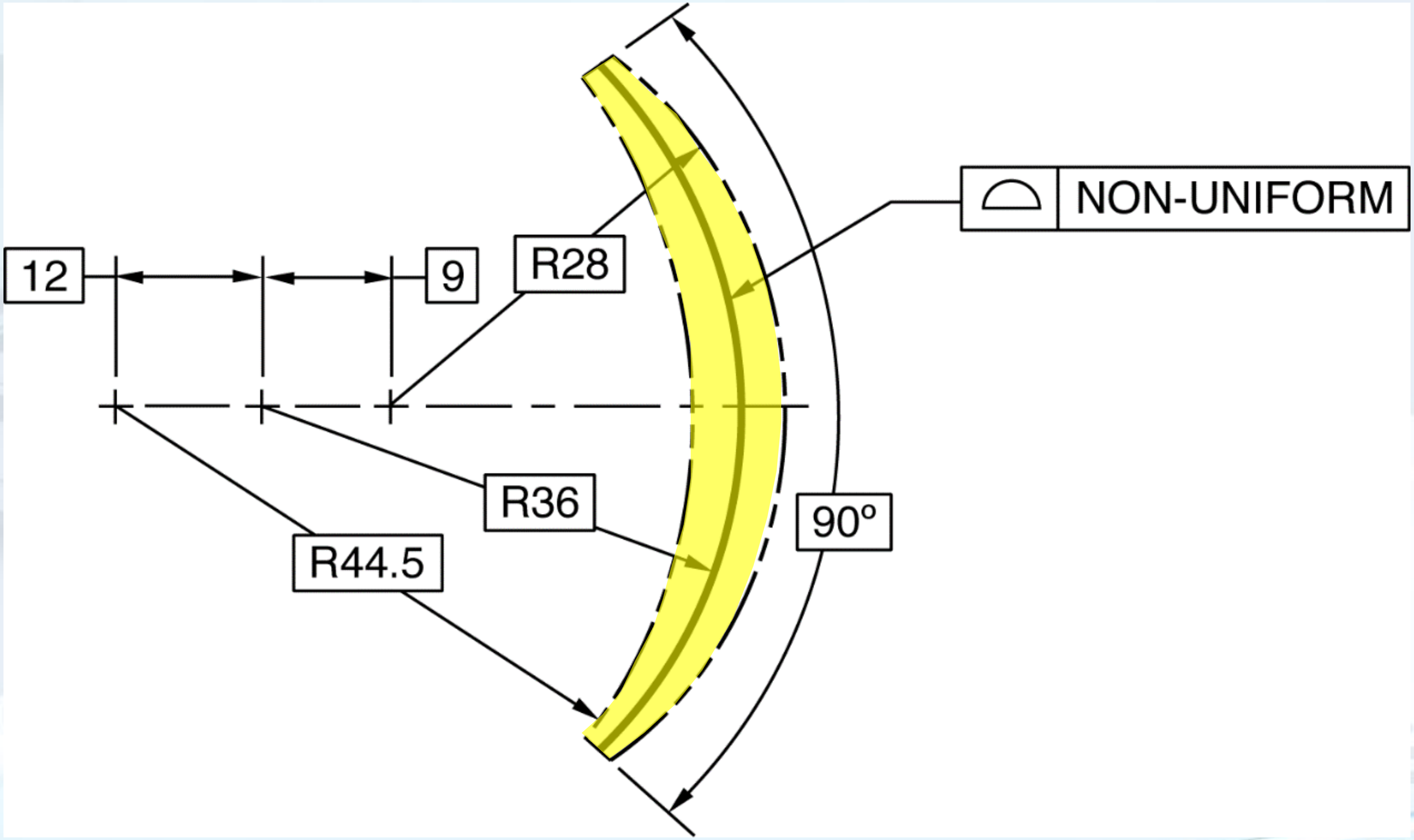
Coplanar Surfaces



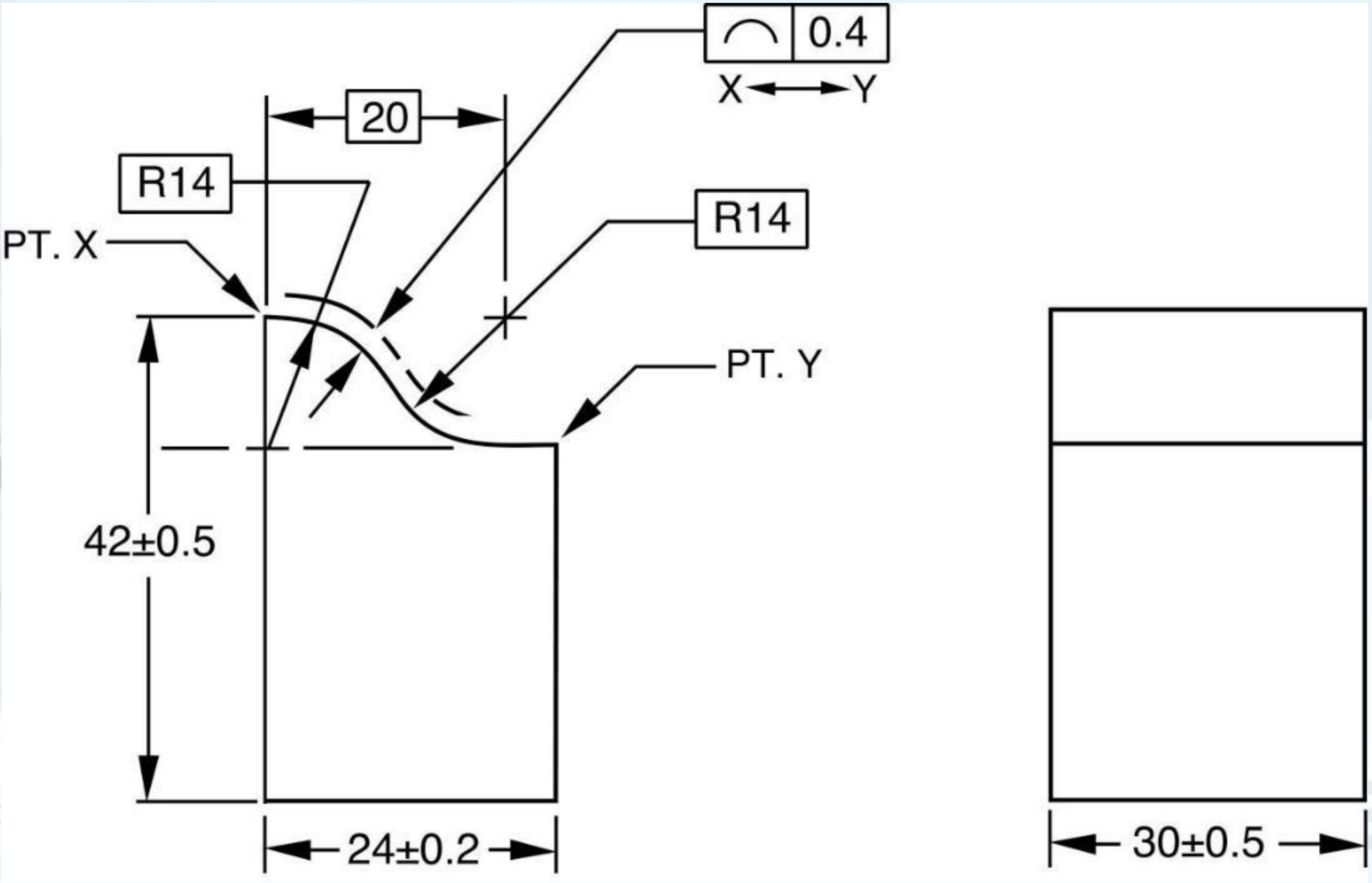
All points of both surfaces must simultaneously lie within the two parallel planes

There is only one tolerance zone for 2 features

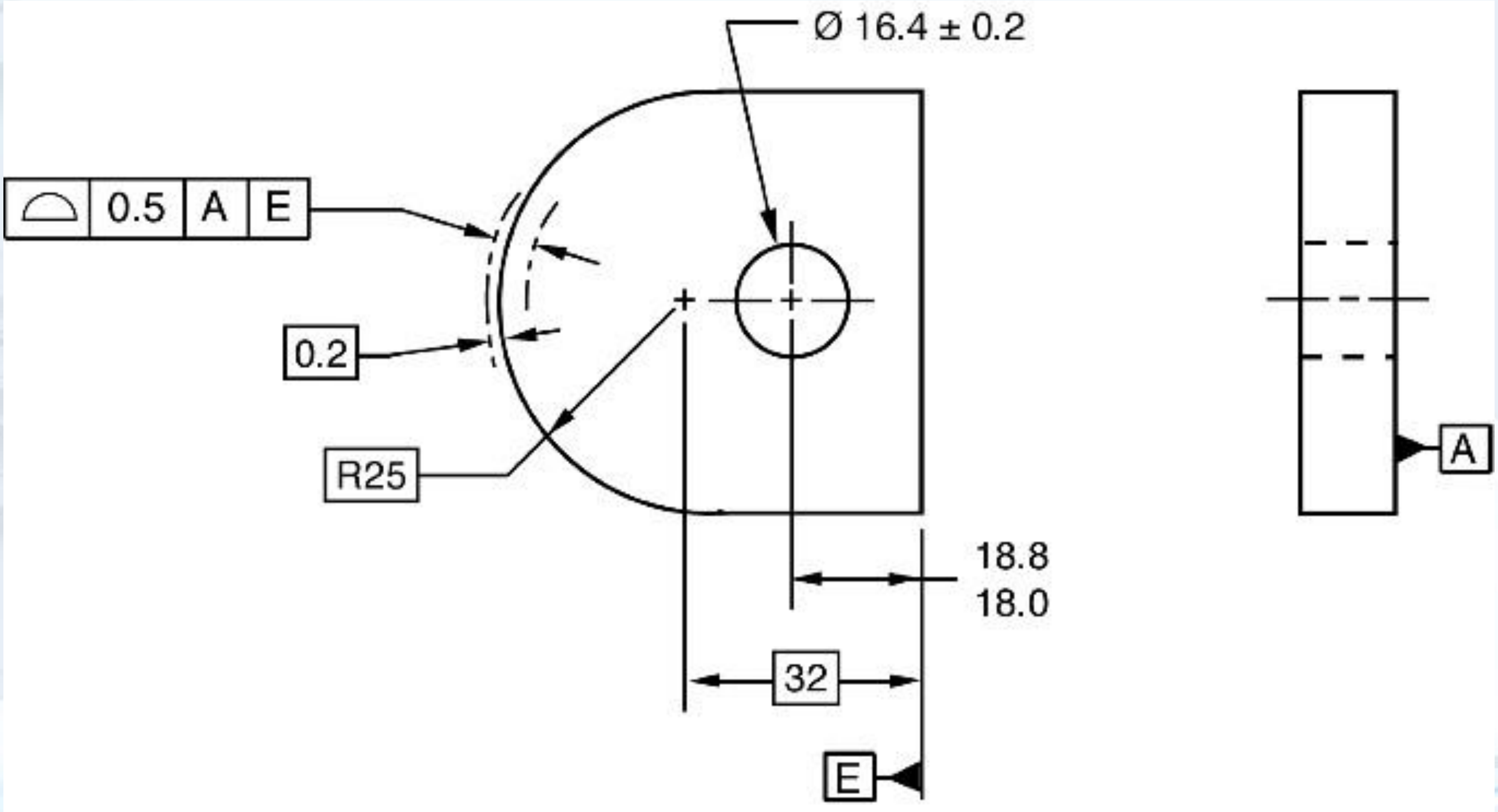
Non-Uniform



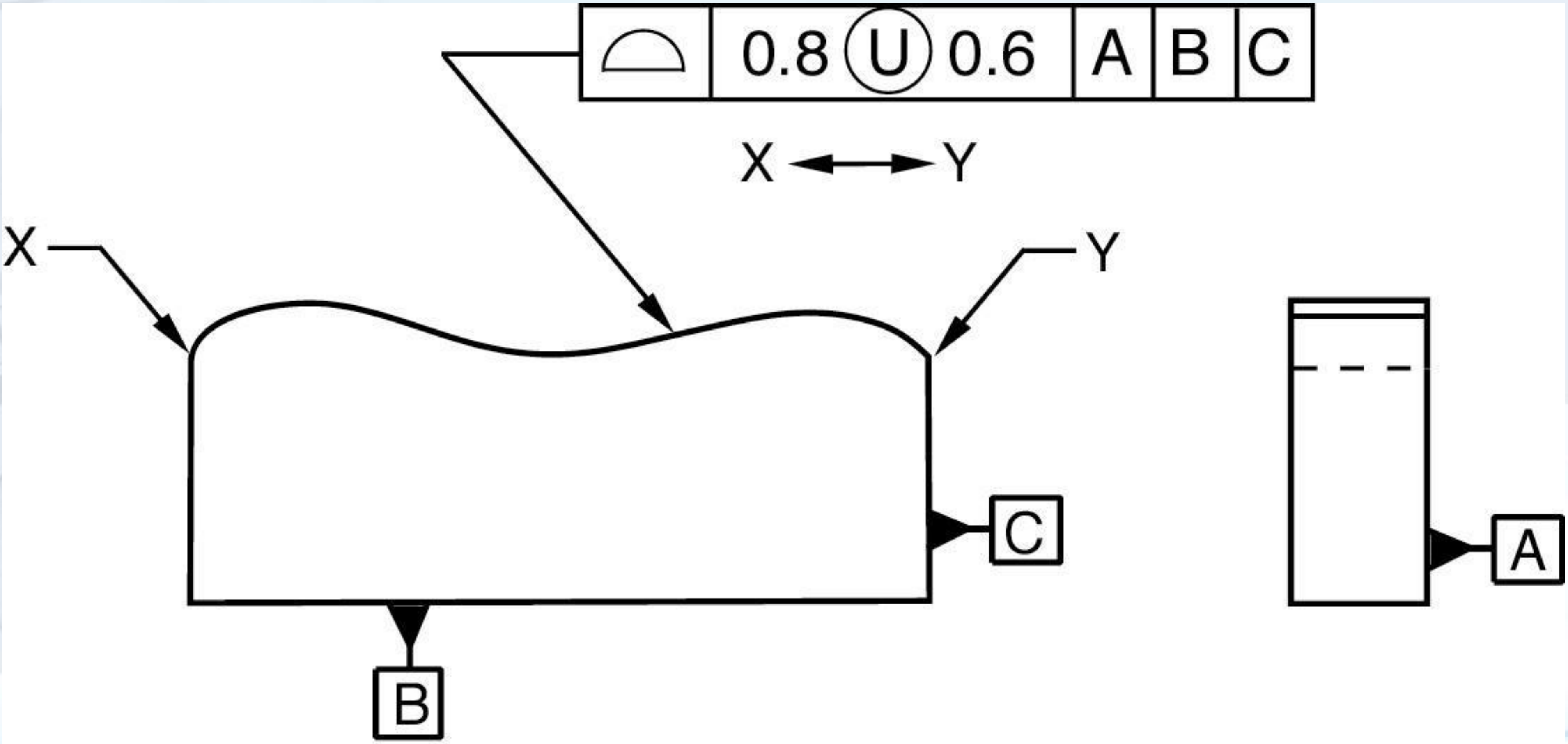
Unilateral Profile



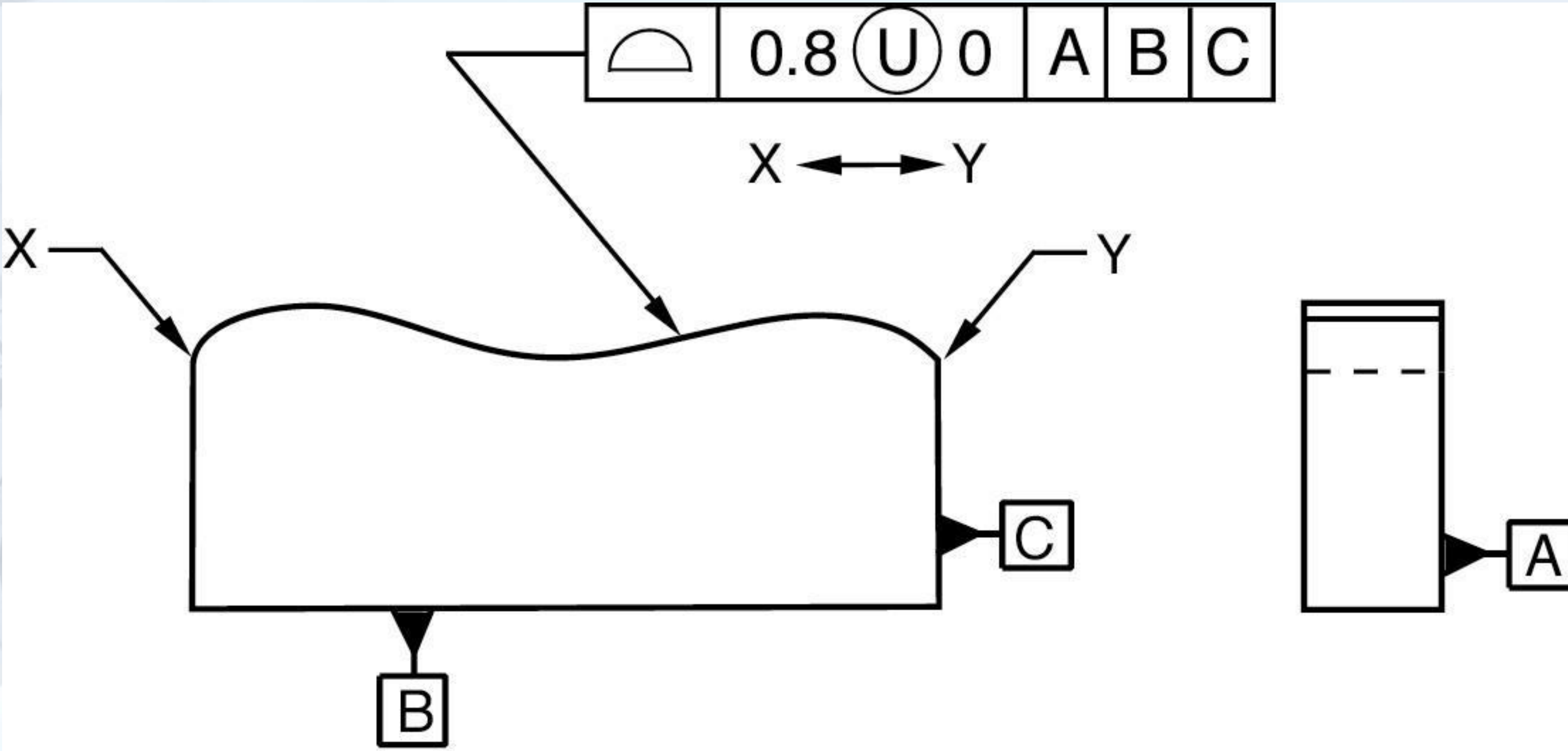
Unequal Profile



Unequal Bilateral – New Way



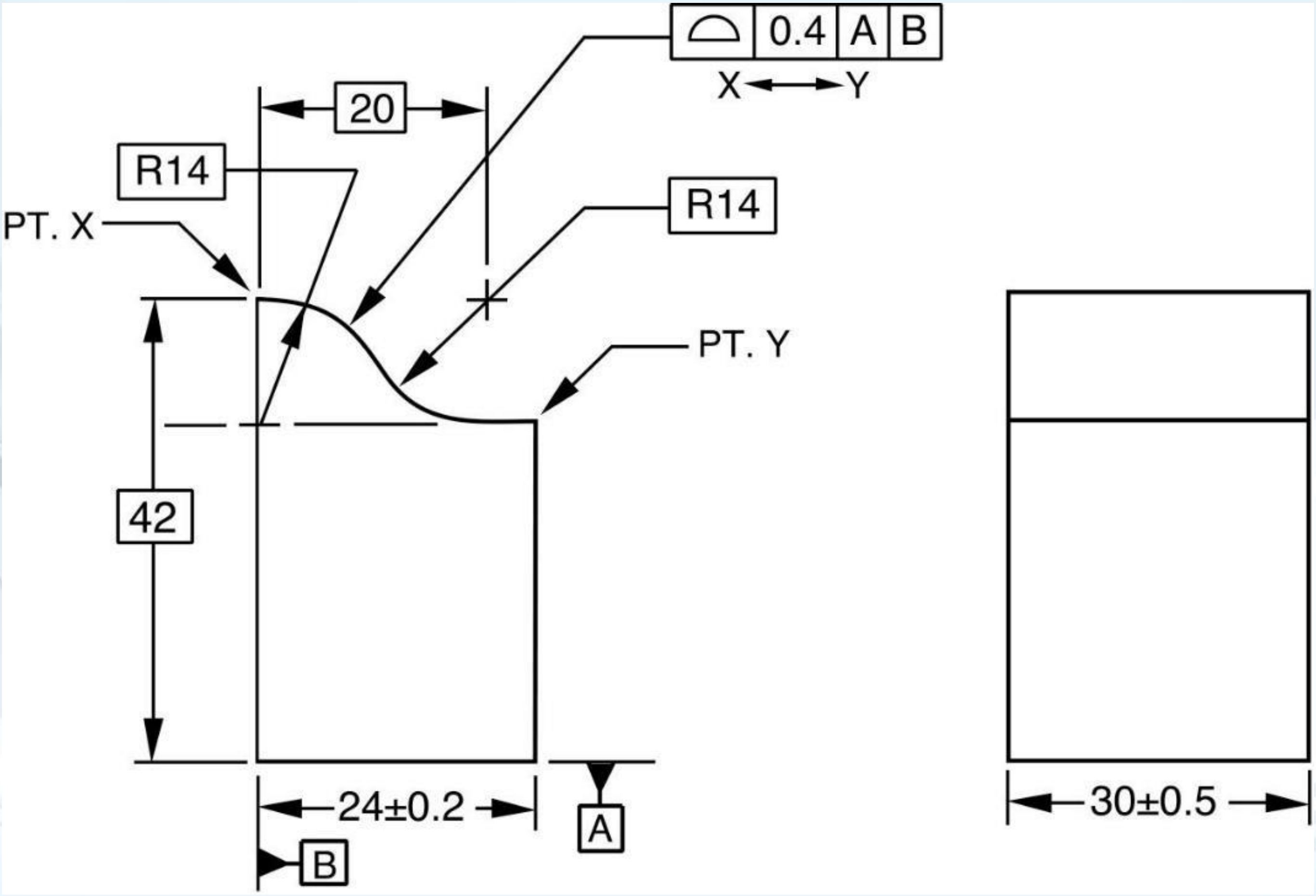
Unilateral – New Way



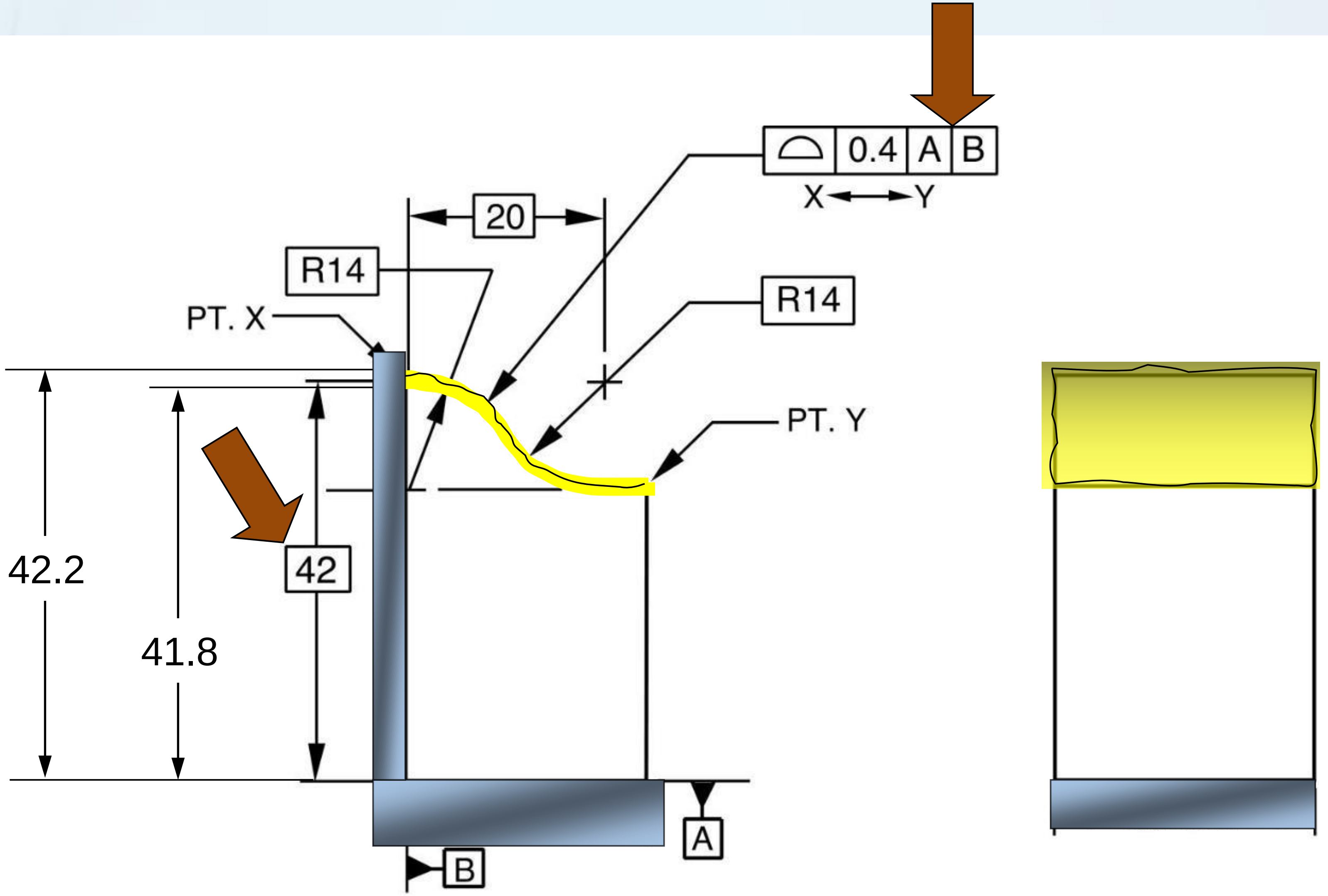
Profile Tolerances

- There are two types of profile – be careful!
- The desired profile must be basic dimensions
- The tolerance is assumed to be equal bilateral
- It may or may not reference datums
- No MMC or LMC modifiers used on the tolerance value

Profile with Datums



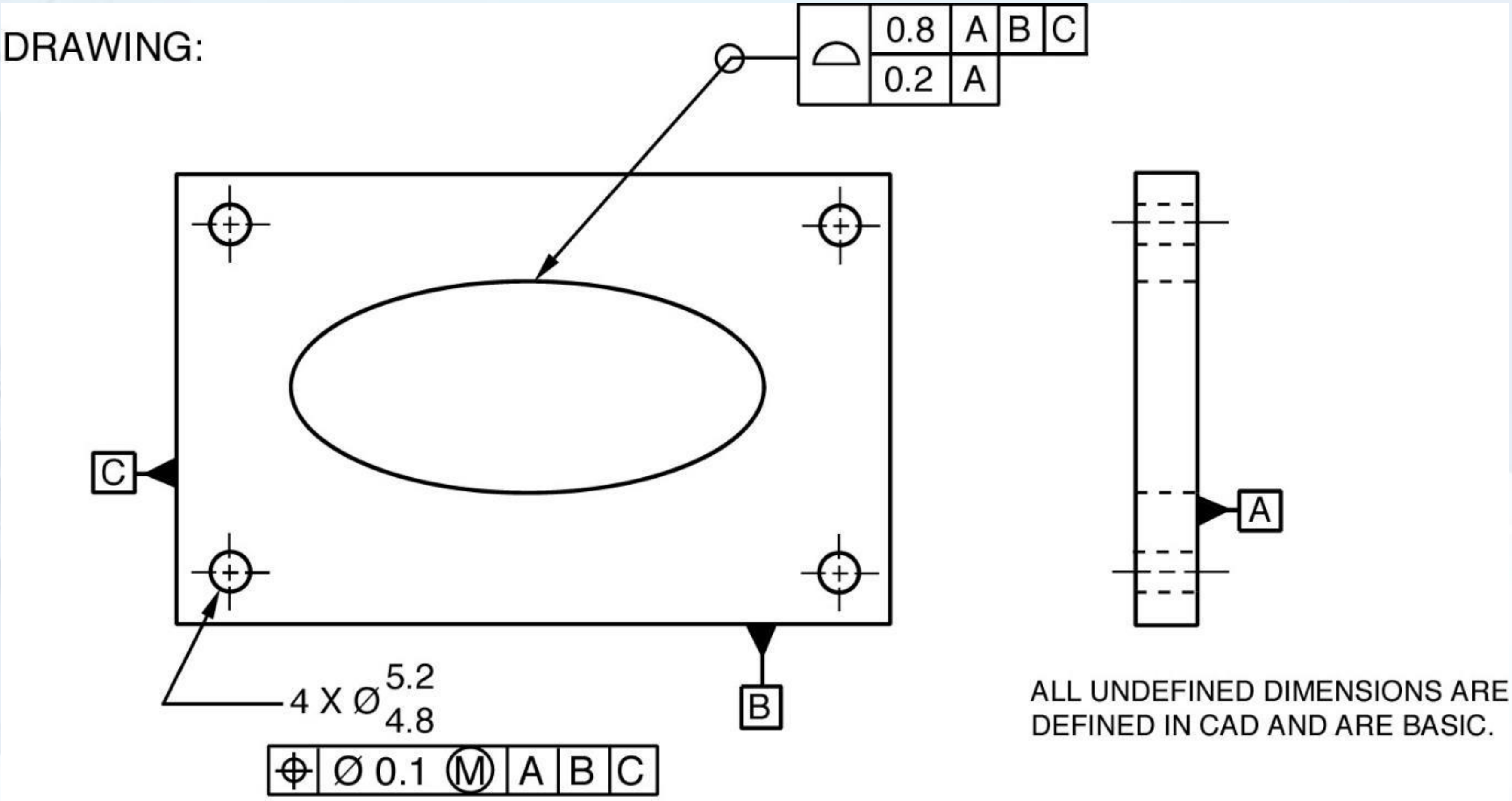
Datum Ref & Basic Dim



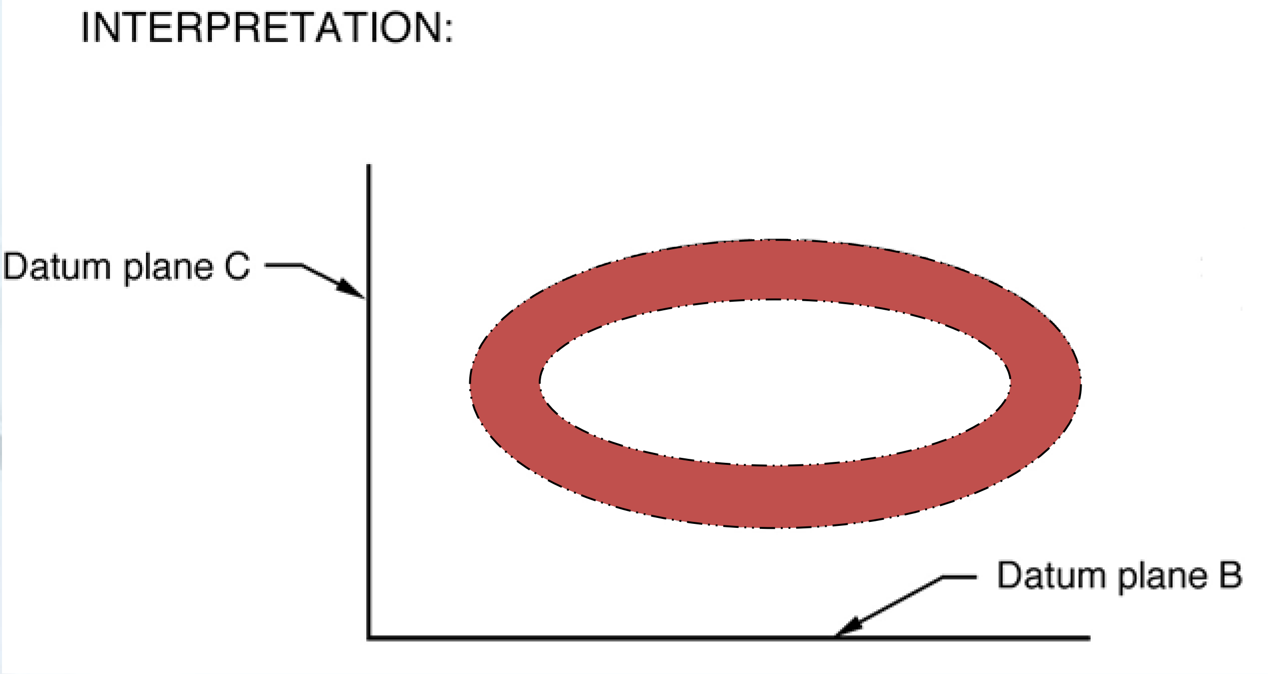
Profile with Datums

- In this drawing, profile controls:
  - Form
  - Orientation
  - Location
  - Size

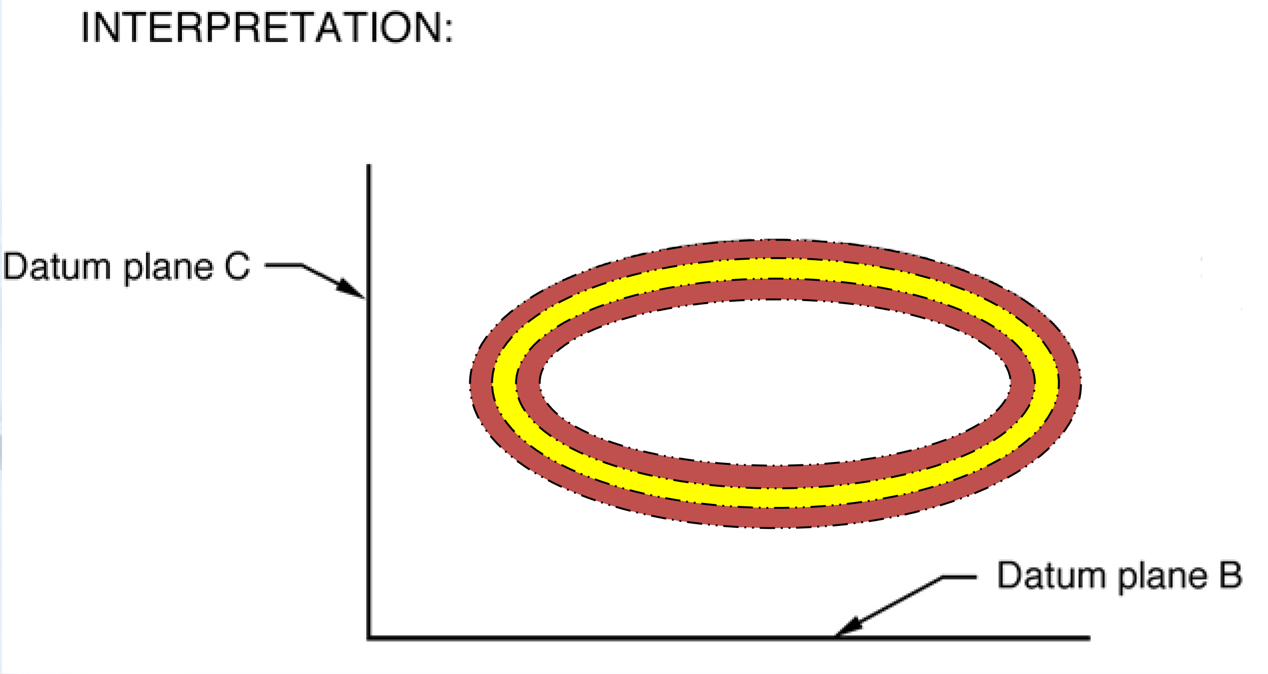
Composite Profile



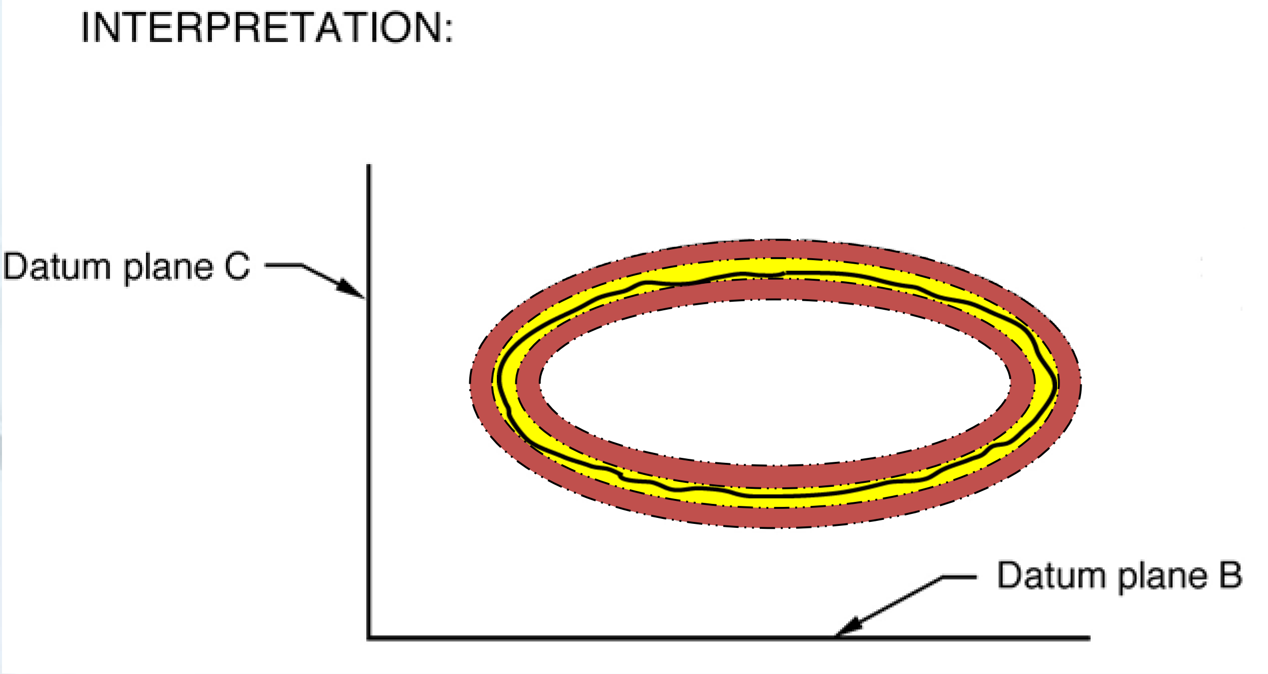
Composite Profile – The 0.8 Zone



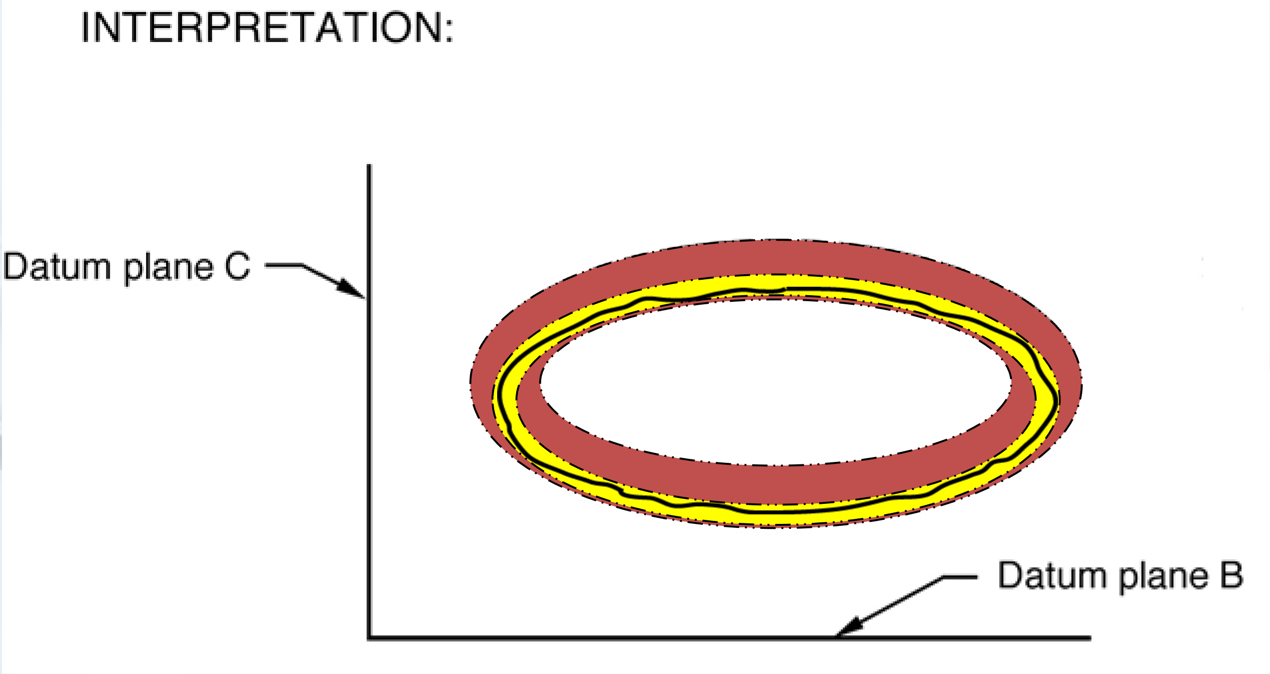
Composite Profile – The 0.2 Zone Inside the 0.8



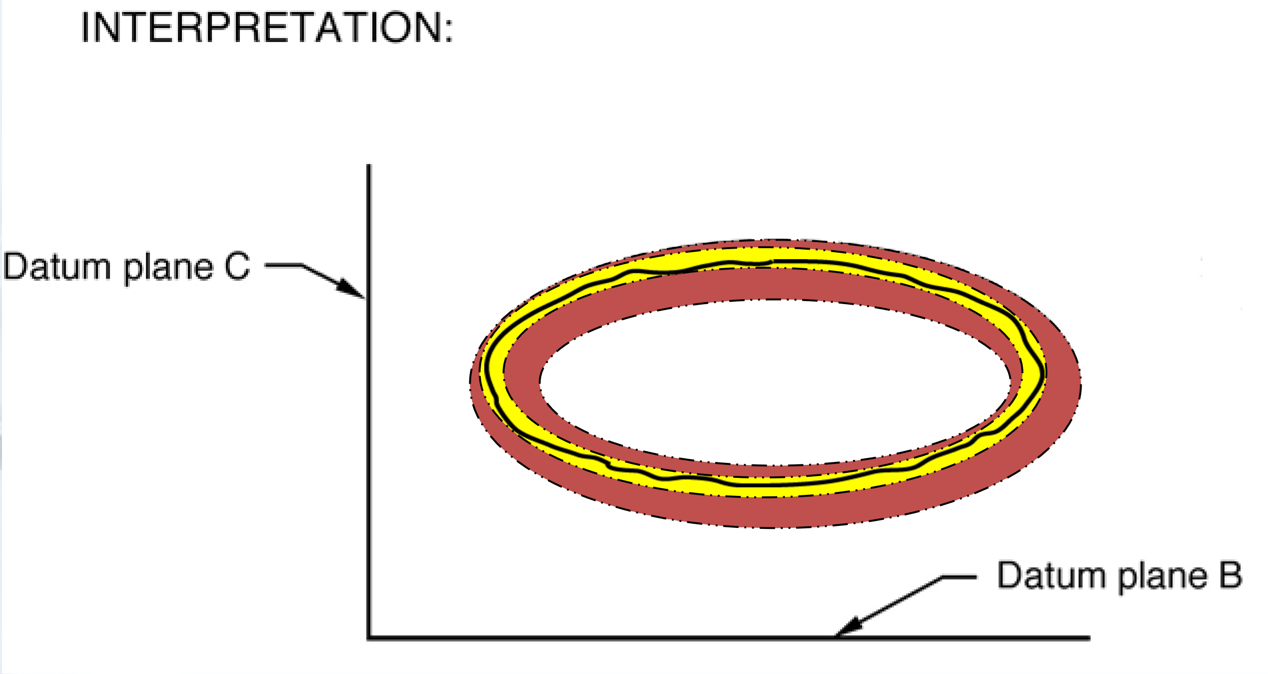
Composite Profile – The Actual Part in Zones



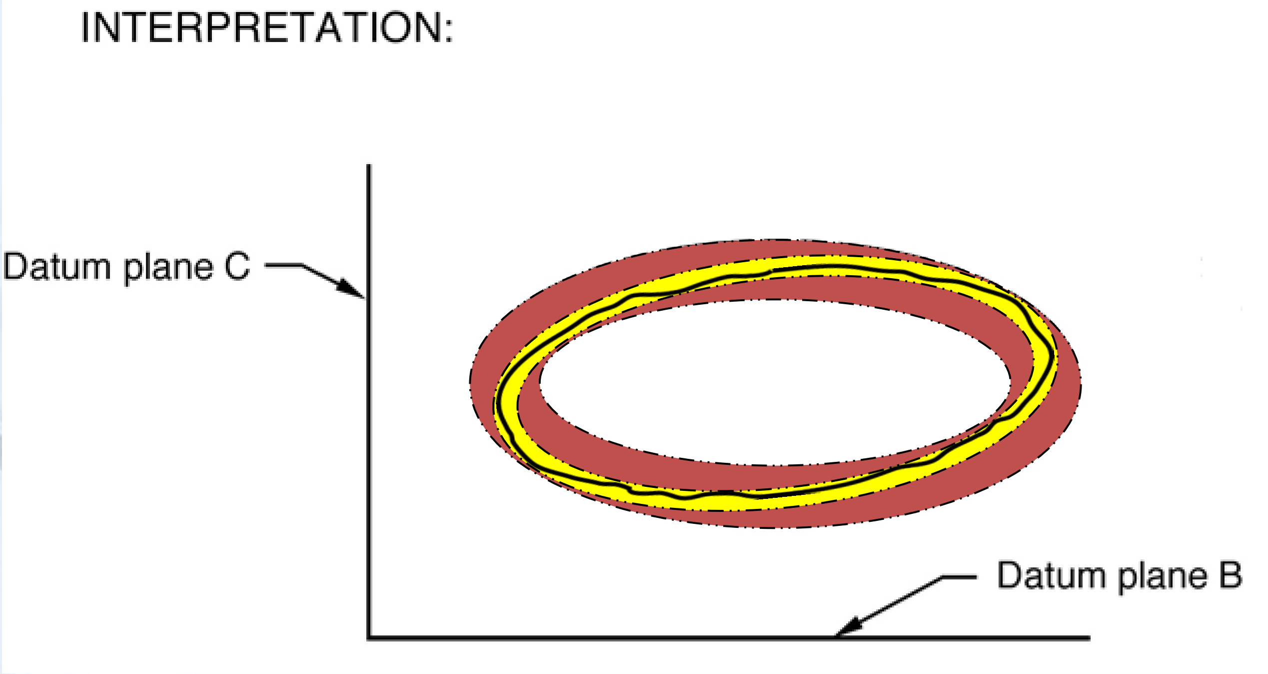
Composite Profile – The Actual Part in Zones



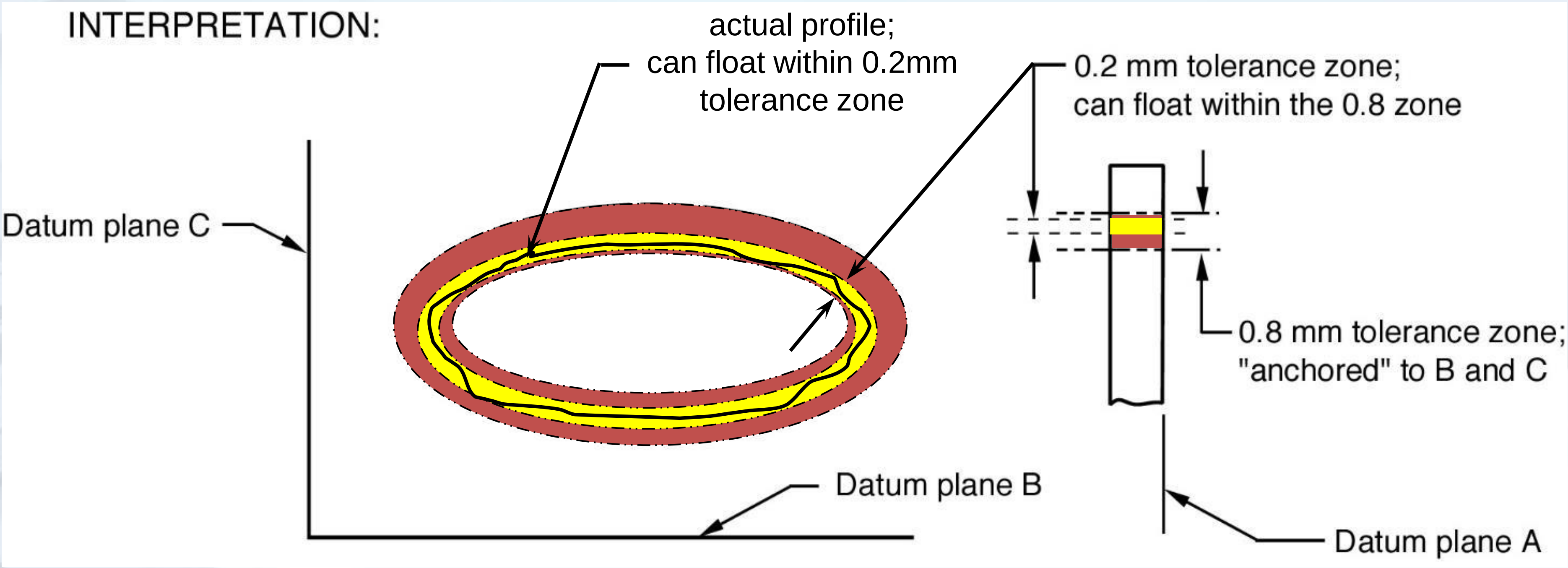
Composite Profile – The Actual Part in Zones



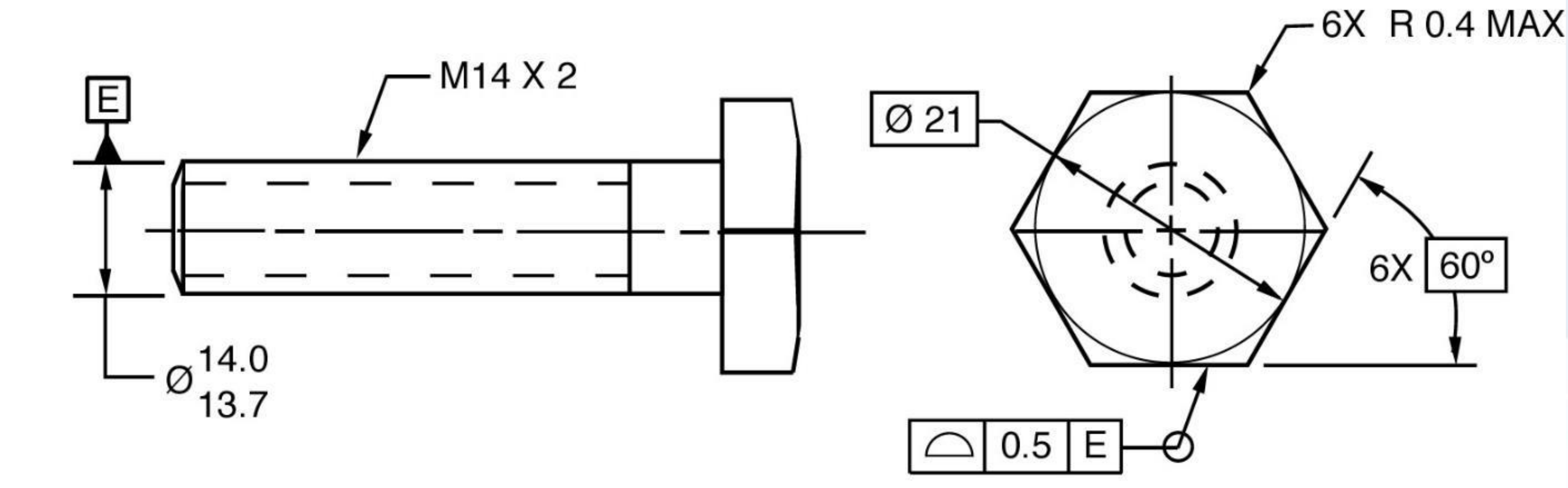
Composite Profile – The Actual Part in Zones



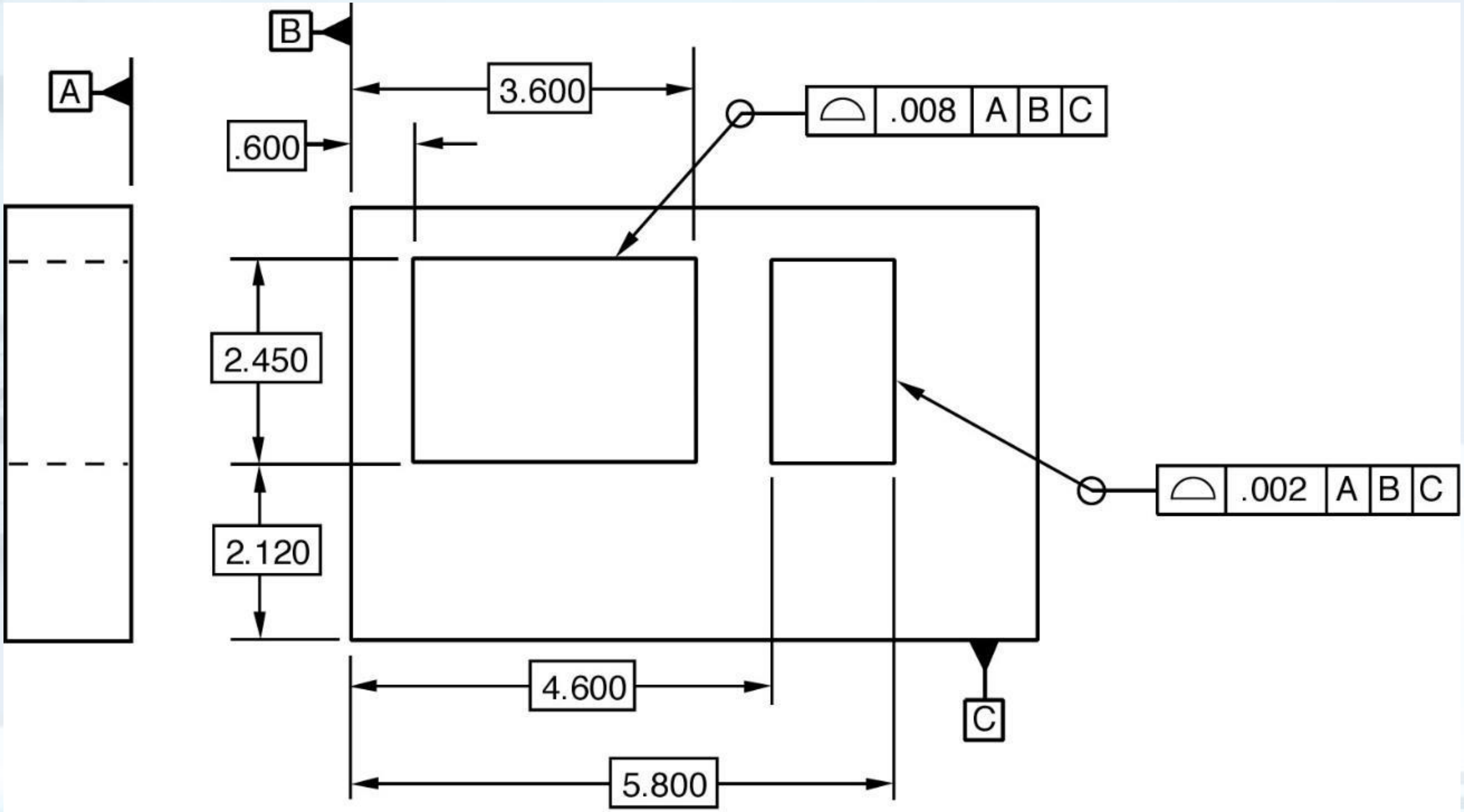
Composite Profile – The Actual Part in Zones



Other Profile Examples



Other Profile Examples



Chapter 5: Profile Tolerances – What We Covered

Learning Objectives

You should now be able to:

- Explain the difference between profile of a line and profile of a surface
- Determine if a profile tolerance references a datum, and how that impacts the tolerance zone
- Identify the special callouts for all around, all over, non-uniform, and unequal
- Interpret a composite profile tolerance, and what qualities each tolerance number controls

Chapter Agenda

- Profile Tolerances
- Profile with Datums
- Composite Profile

Chapter 6

Orientation

Perpendicularity	⊥
Angularity	∠
Parallelism	//

Chapter 6: Orientation – What We Will Cover

Learning Objectives

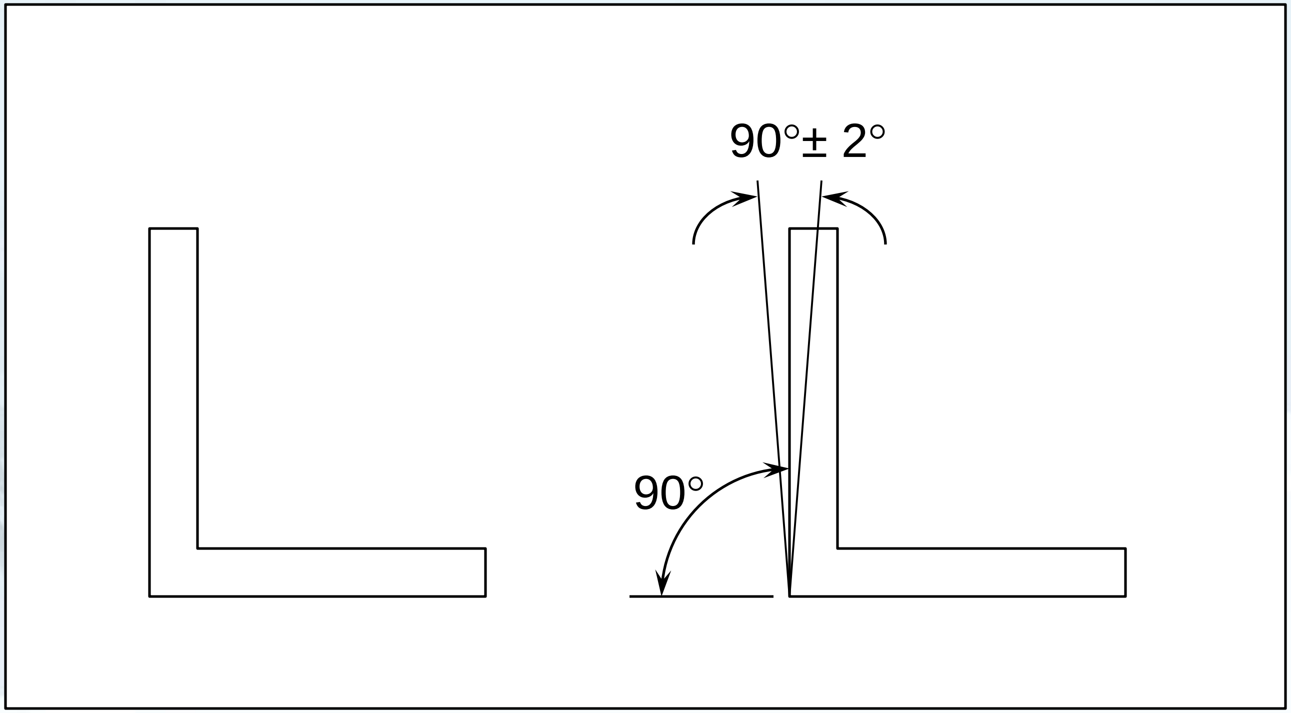
At the end of this chapter, you will be able to:

- Identify perpendicularity, angularity, and parallelism callouts on a print, and determine if they apply to surfaces or features of size
- Explain the effect of the tangent plane modifier on an orientation control

Chapter Agenda

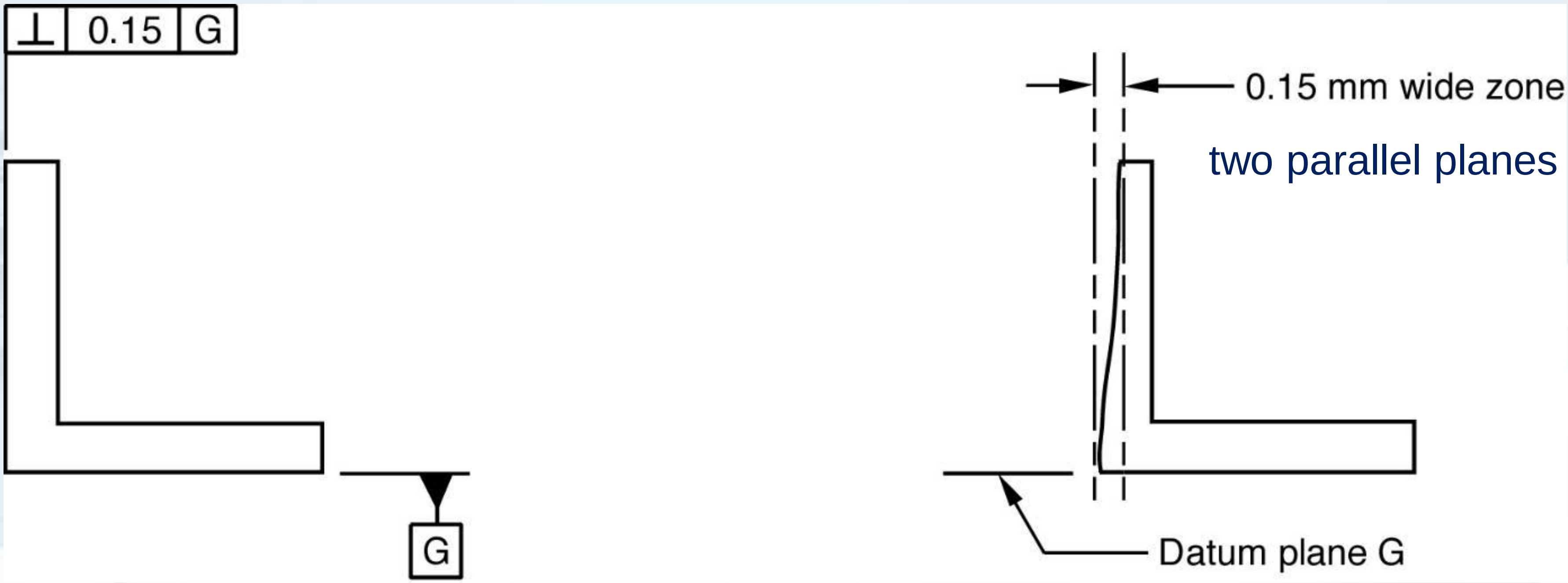
- Perpendicularity
- Angularity
- Parallelism

Consider This:



Perpendicularity

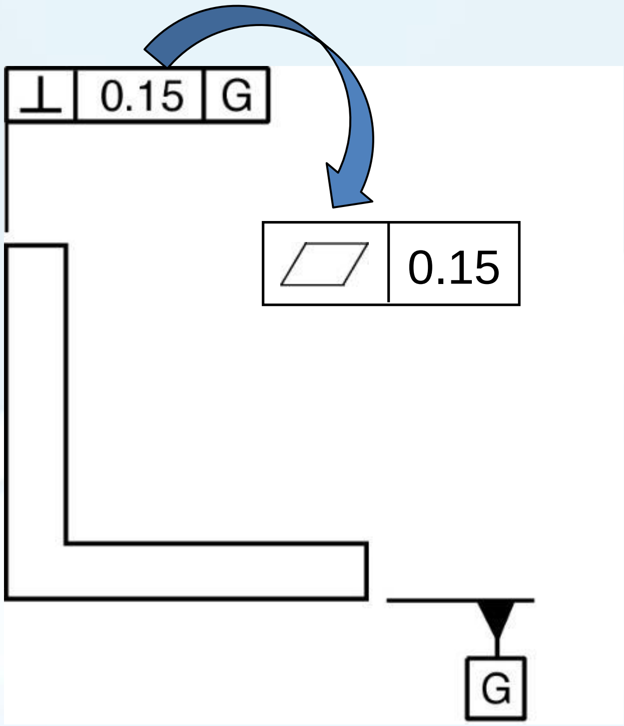
- Defined as the condition of a surface, center plane, or axis at a right angle to a datum plane or axis.



Perpendicularity

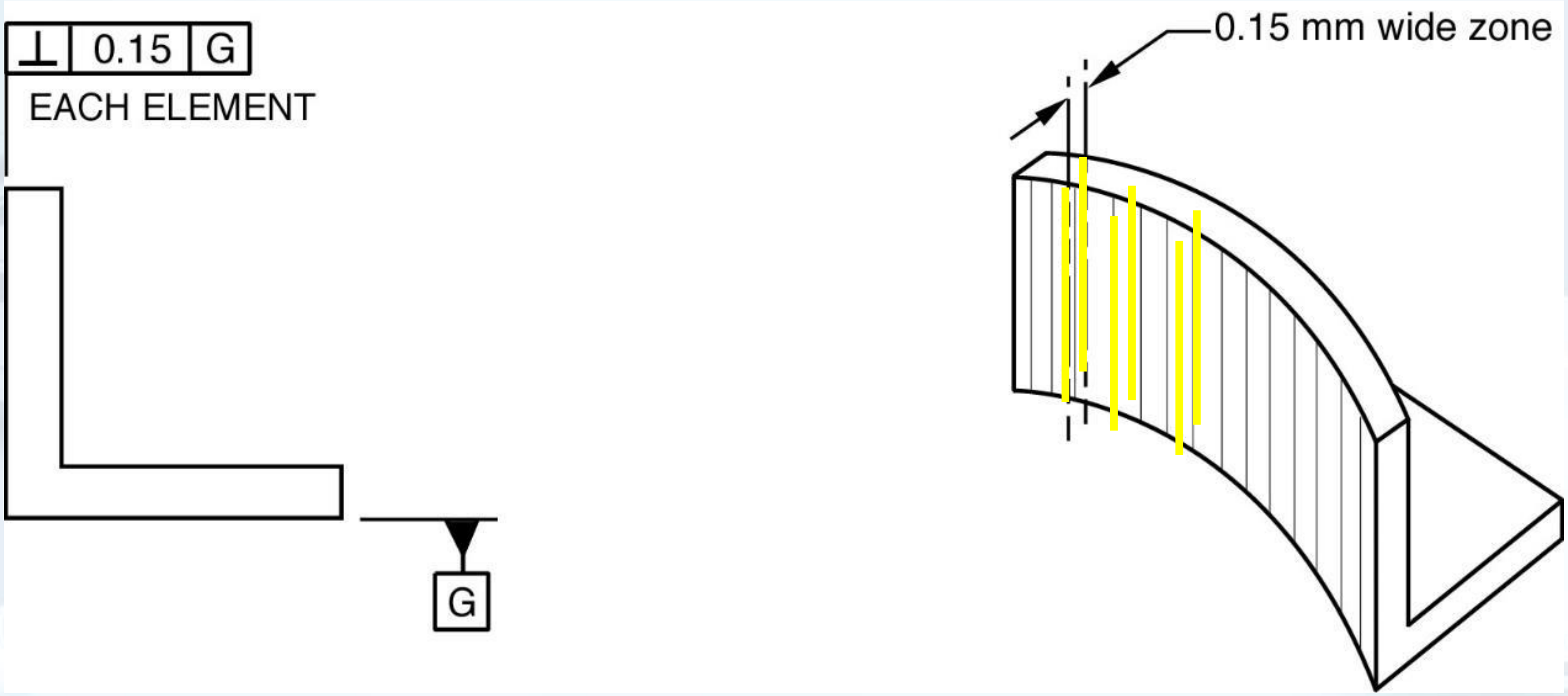
Surface

- The perpendicularity control also includes flatness.



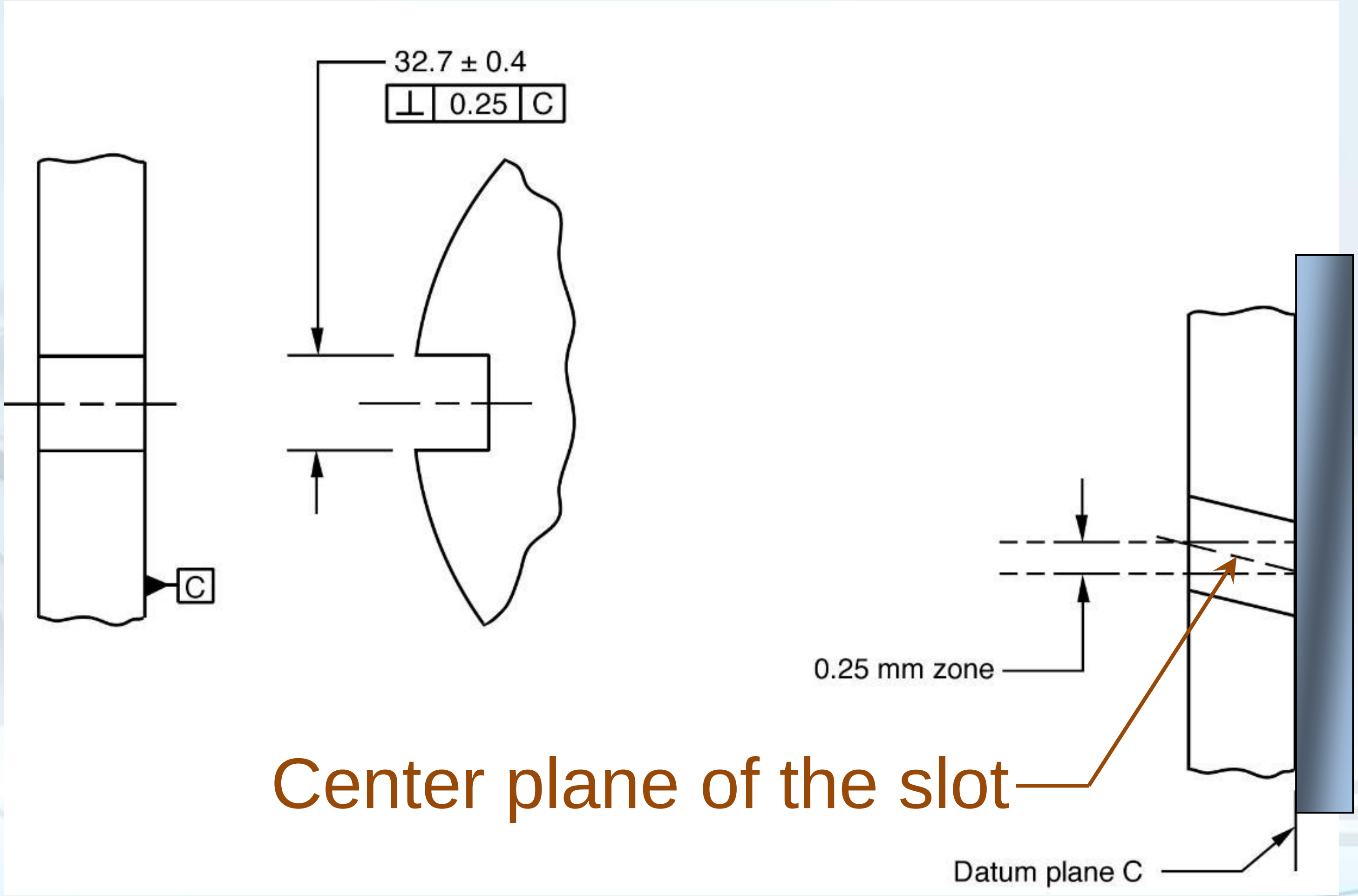
Perpendicularity

Each Element (of a Surface)

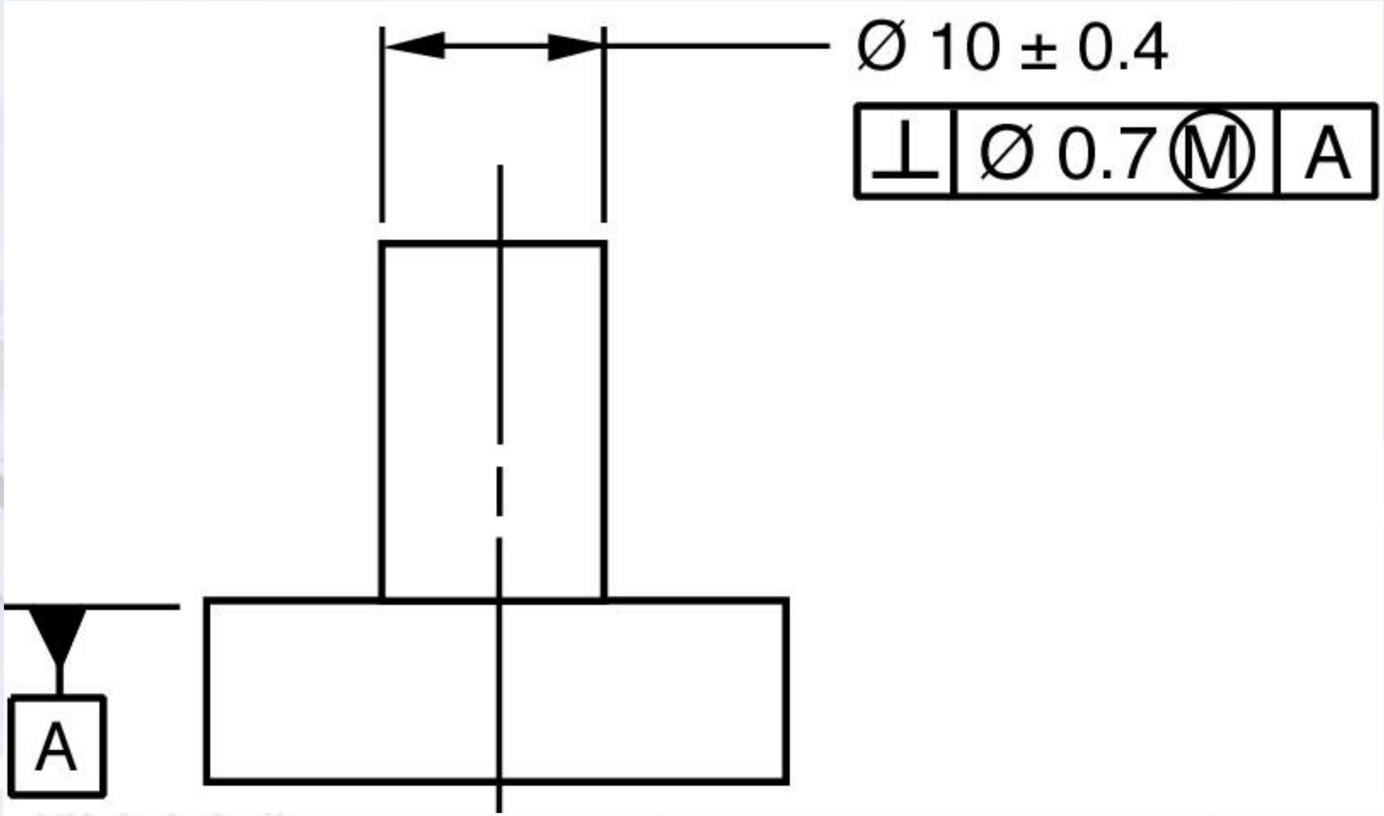


Perpendicularity

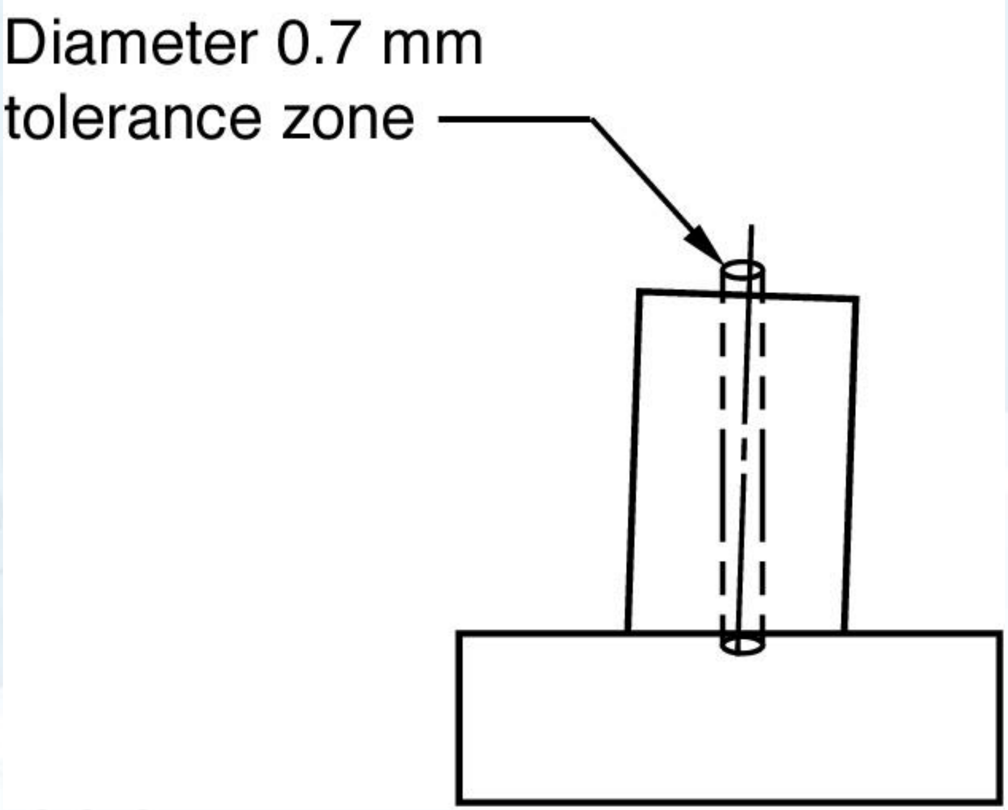
Feature of Size



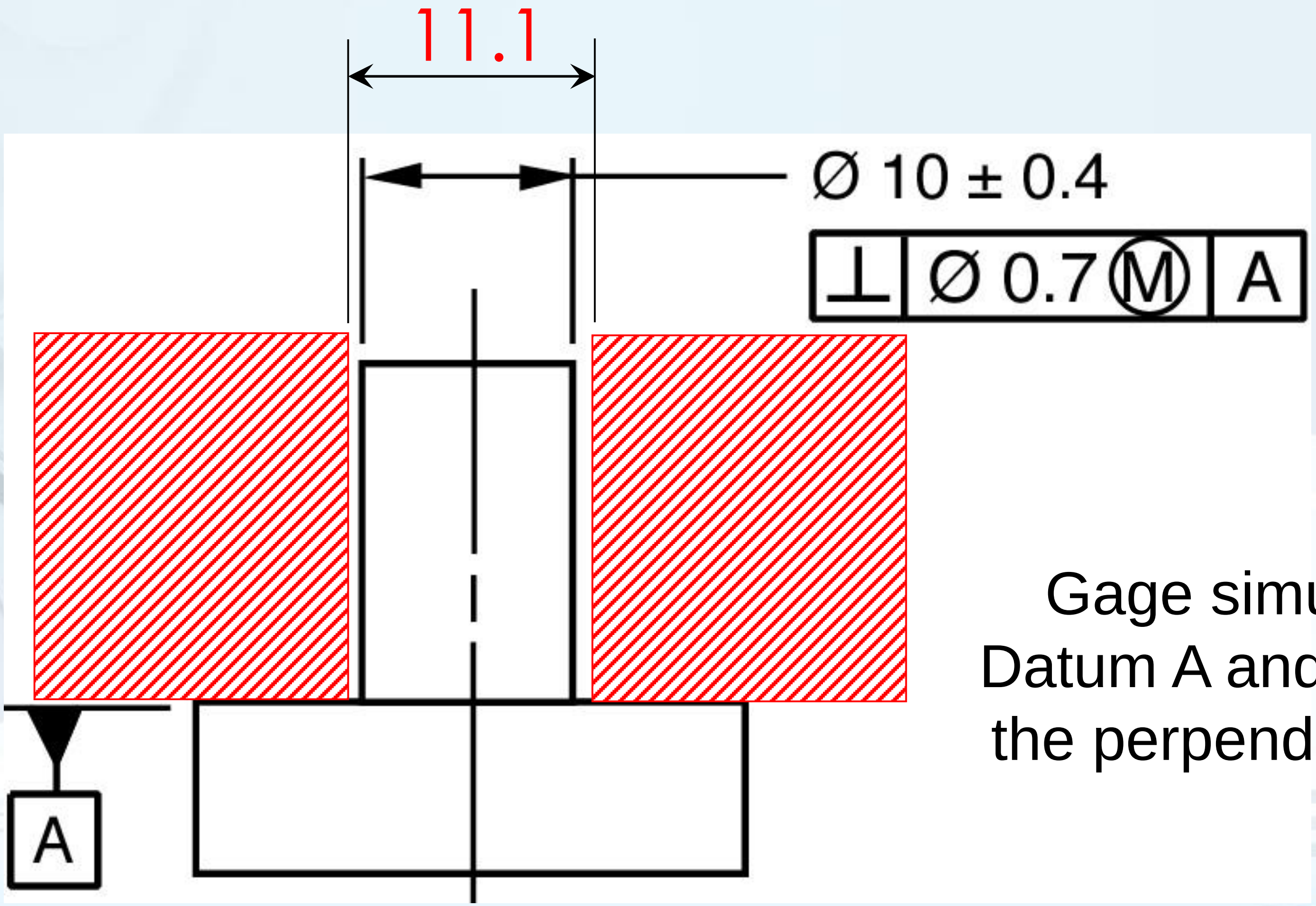
Perpendicularity -- MMC



Perpendicularity -- MMC



Perpendicularity -- MMC



Gage simulates Datum A and checks the perpendicularity

Perpendicularity

- It must reference a datum
- Tolerance is in millimeters or inches
- 90º angle is understood to be basic; general tolerance for angles do not apply
- Perpendicularity does not affect the size dimensions
- When applied to an axis, the diameter symbol is required to describe a cylindrical tolerance zone
- When applied to a surface, it also controls flatness
- Material condition modifiers (MMC & LMC) are allowed if it’s applied to a feature of size

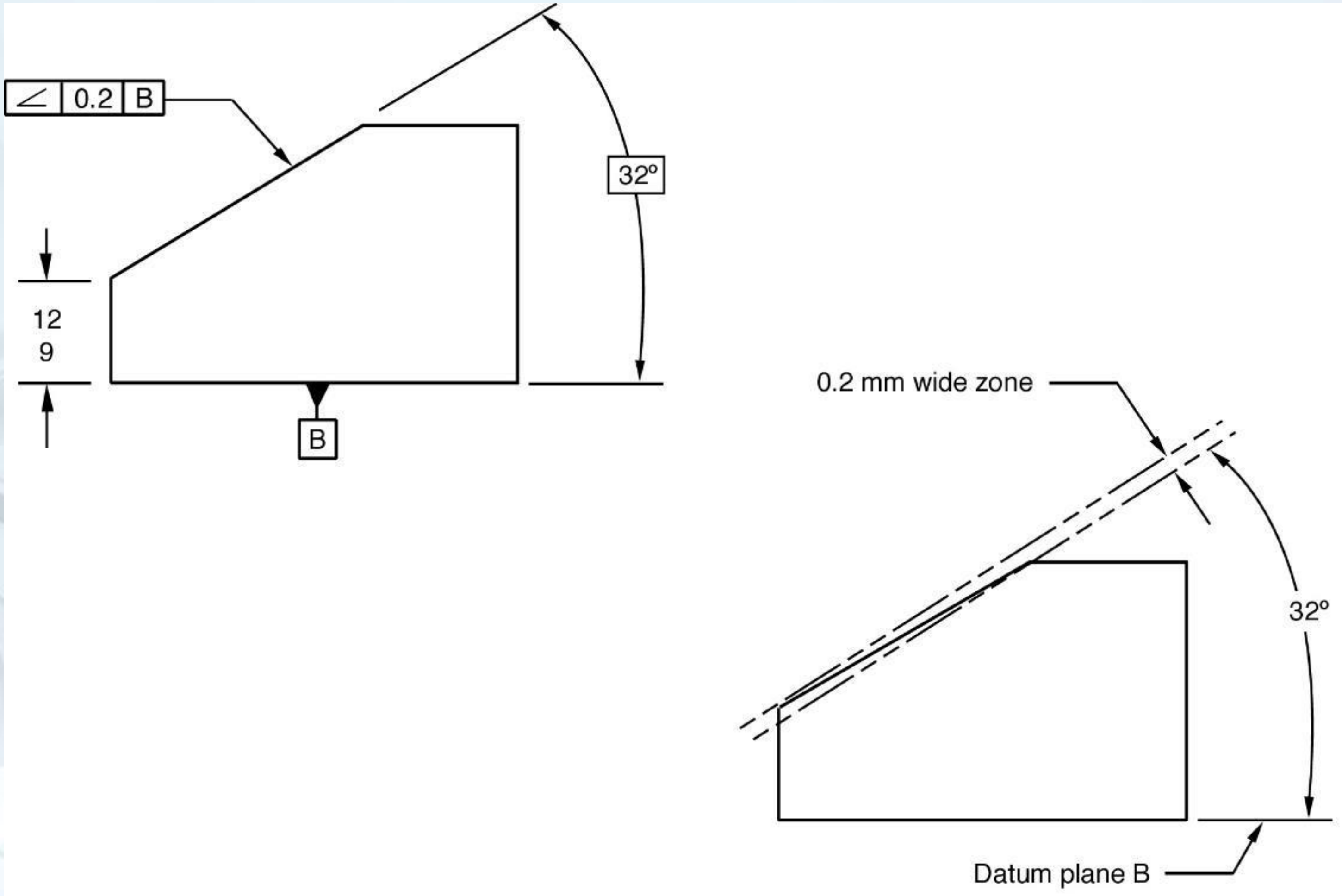
Angularity

- Angularity is the condition of a surface, center plane, or axis at any specified angle from a datum plane or axis.

- The most important thing to notice is that the tolerance value is given in linear units, NOT degrees.

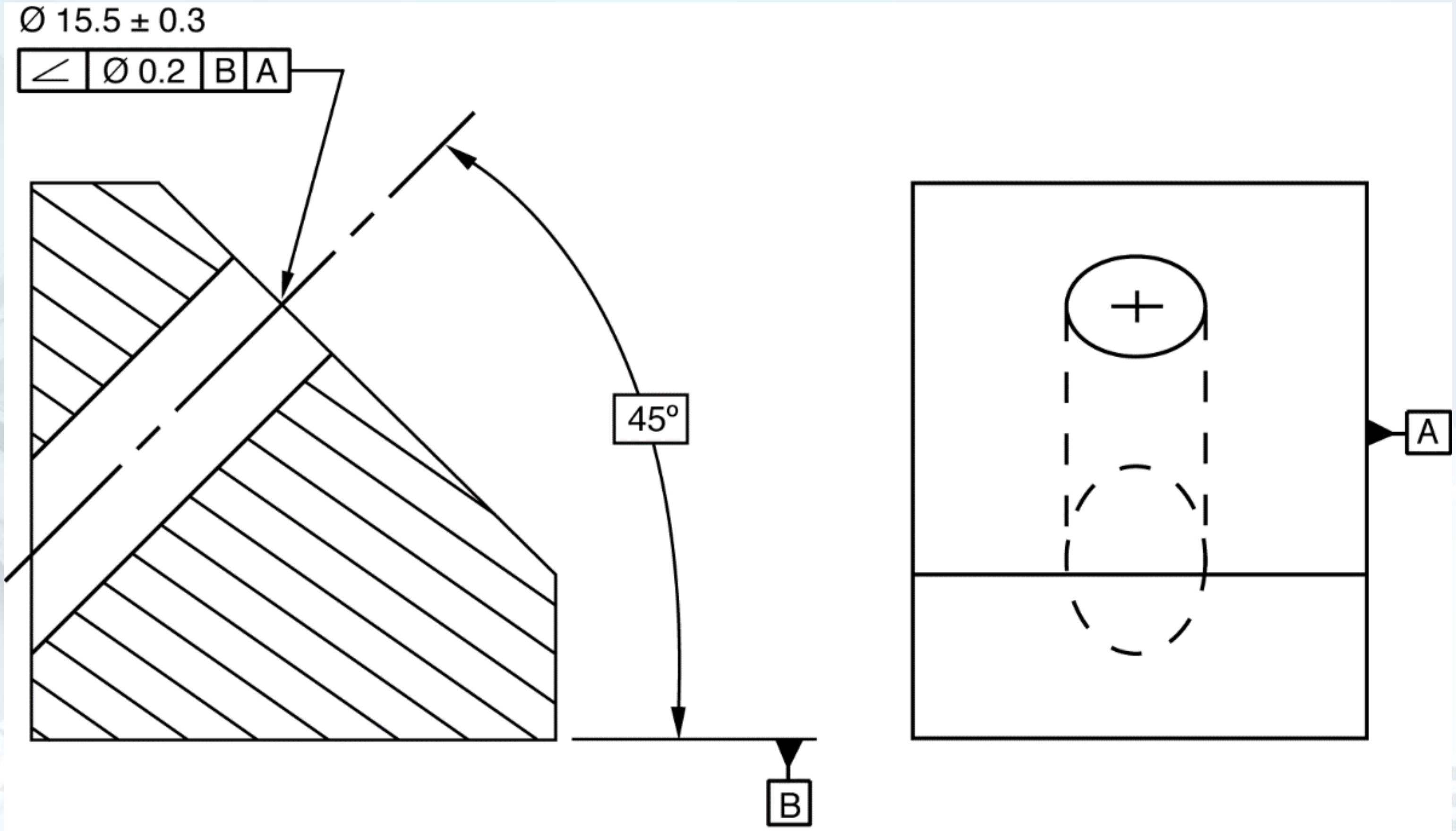
Angularity

Surface

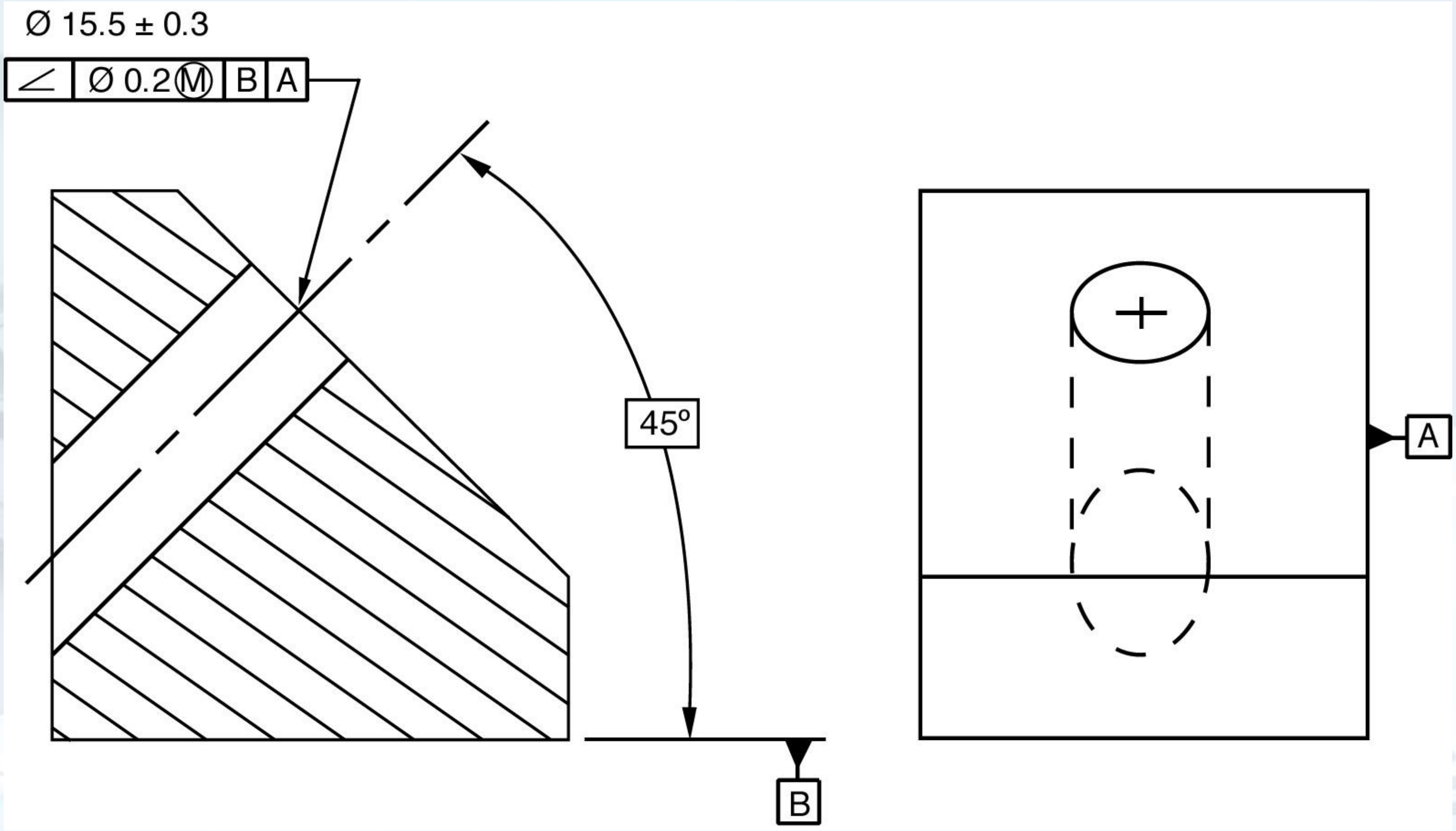


Angularity

Feature of Size



Angularity -- MMC

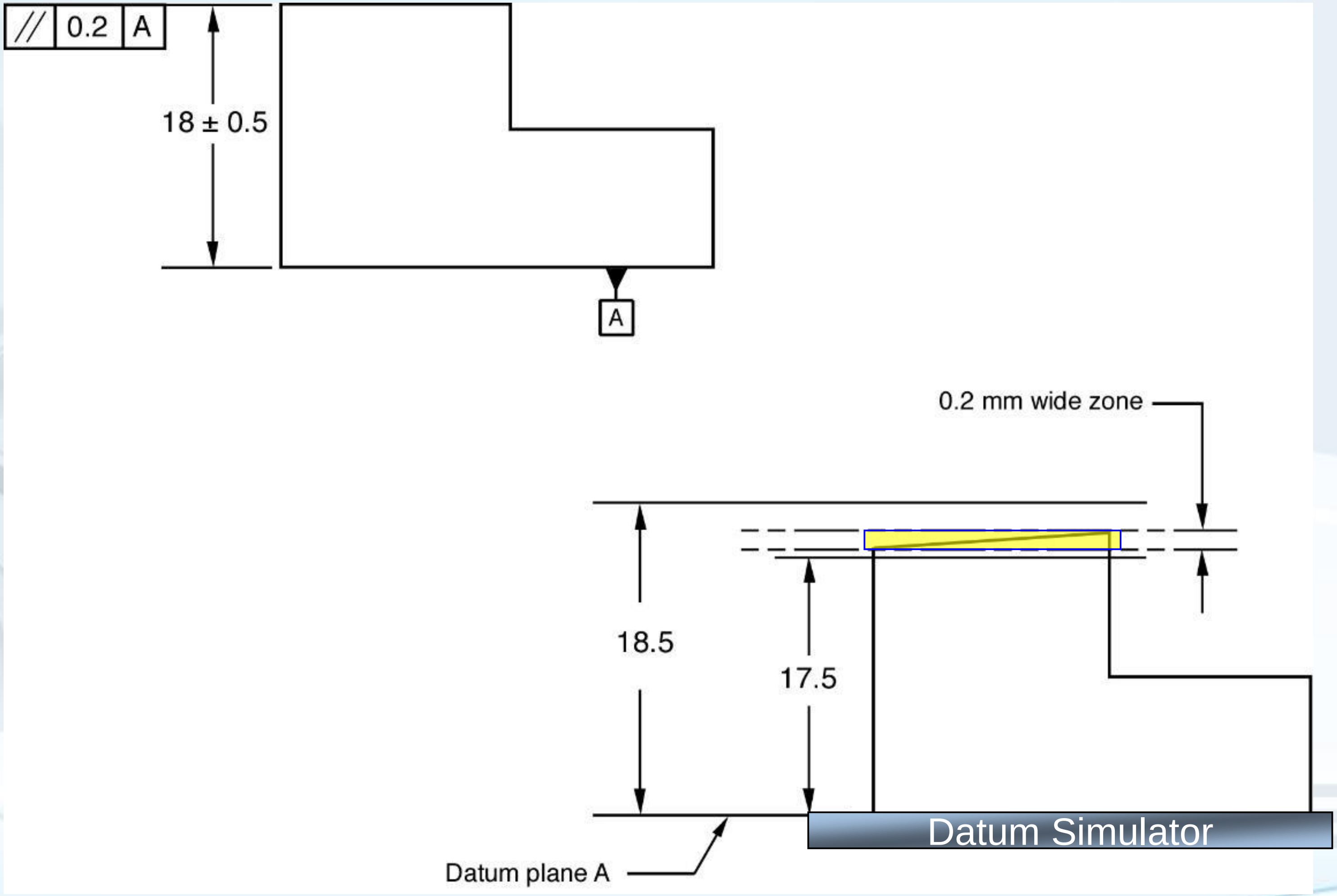


Angularity

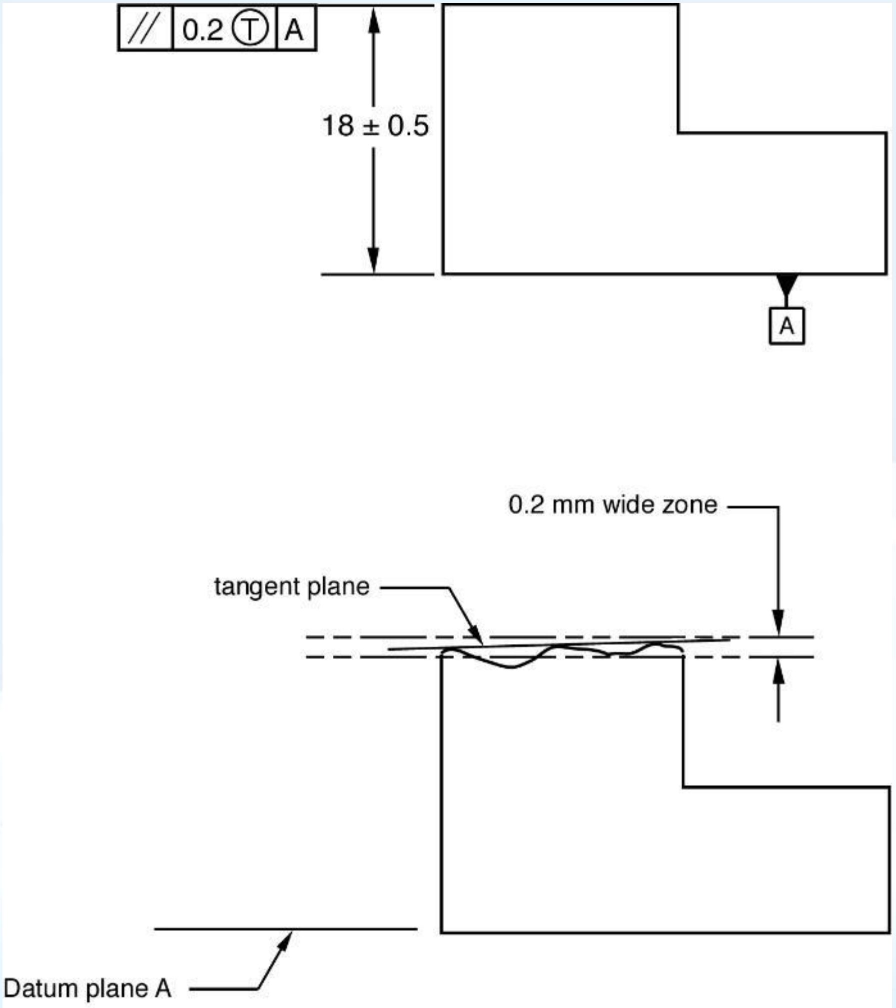
- Must reference a datum
- Tolerance: millimeters or inches (not degrees)
- When applied to an axis, the diameter symbol is required to describe a cylindrical tolerance zone
- Material condition modifiers (MMC & LMC) are allowed if applied to a feature of size
- The specified angle must be a basic dimension
- Does not affect the size dimensions

Parallelism

Surface

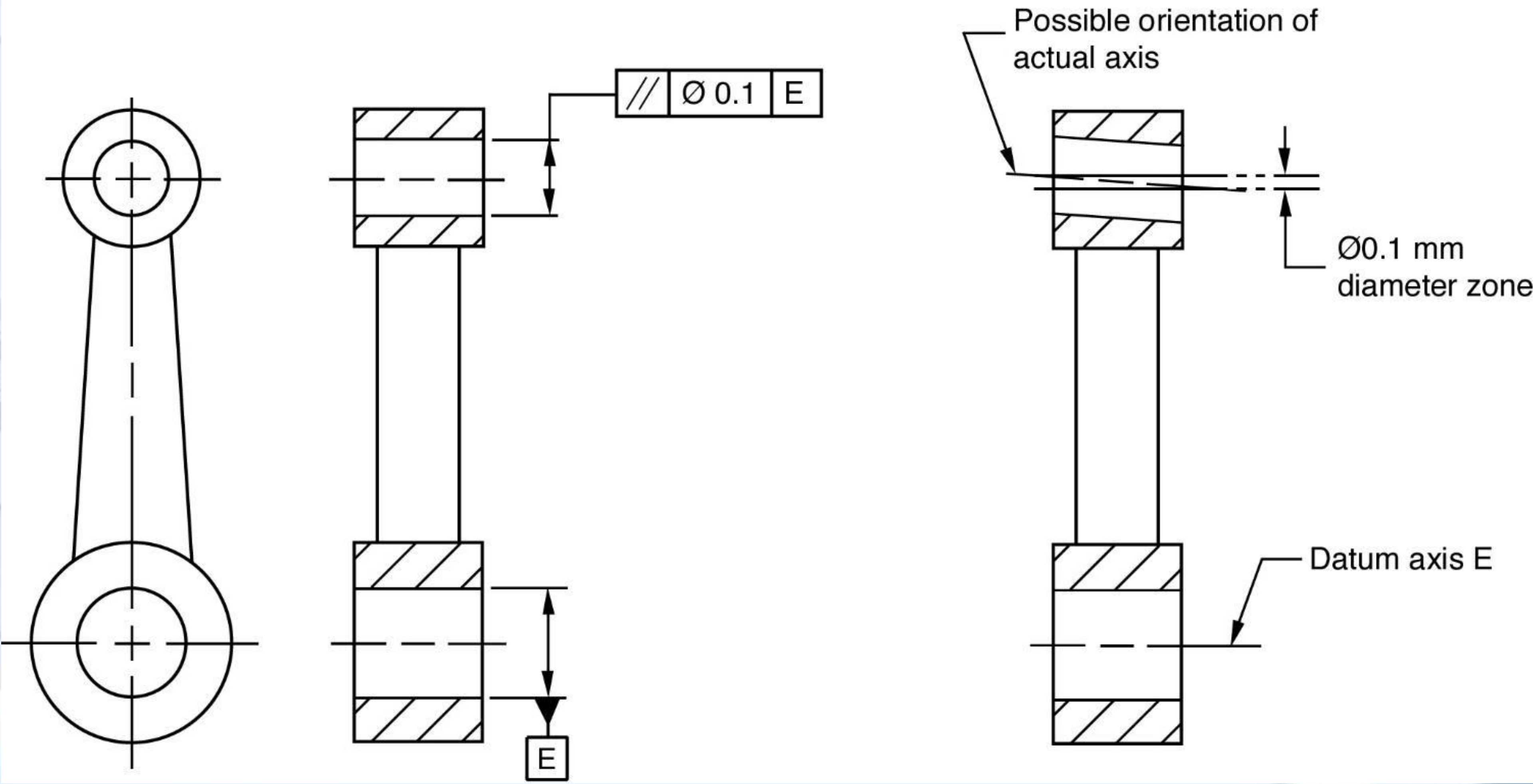


Tangent Plane Modifier



Parallelism

Feature of Size



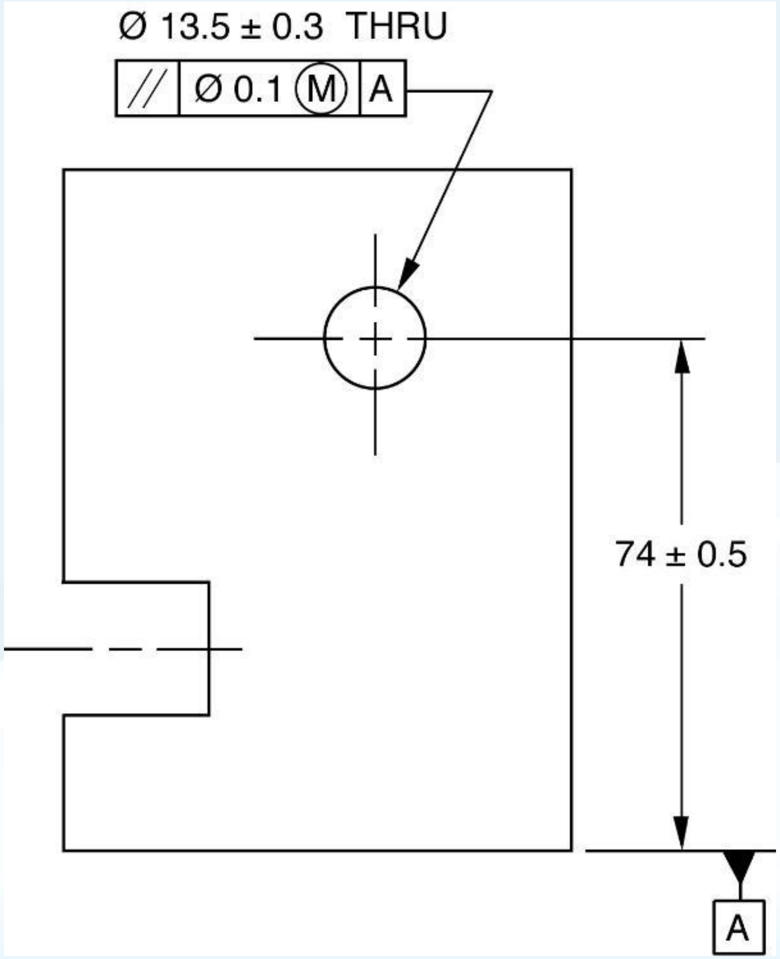
OMNEX

For This Example

- The hole is not being **located** by the geometric tolerance, but only **oriented**.

- A position tolerance is more common, and would control orientation and location.

Parallelism -- MMC



Parallelism

- Must reference a datum
- Tolerance is in mm (or inches)
- When applied to an axis, the diameter symbol is required to describe a cylindrical tolerance zone
- Material condition modifiers (MMC & LMC) are allowed if applied to a feature of size
- Controlled feature must be parallel to the referenced datum
- Does not affect the size dimensions

Chapter 6: Orientation – What We Covered

Learning Objectives

You should now be able to:

- Identify perpendicularity, angularity, and parallelism callouts on a print, and determine if they apply to surfaces or features of size
- Explain the effect of the tangent plane modifier on an orientation control

Chapter Agenda

- Perpendicularity
- Angularity
- Parallelism

Chapter 7

Position

The Most Common GD&T Symbol

Chapter 7: Position – What We Will Cover

Learning Objectives

At the end of this chapter, you will be able to:

- Define true position
- Explain the effect of the MMC and LMC modifiers on a position tolerance
- Explain the pitch diameter rule
- Interpret a zero position tolerance and calculate the maximum position tolerance available
- For a composite position tolerance, explain what qualities are controlled by each number

Chapter Agenda

- Position
- Variation vs. Functional
- Boundary
- Tolerance Zones

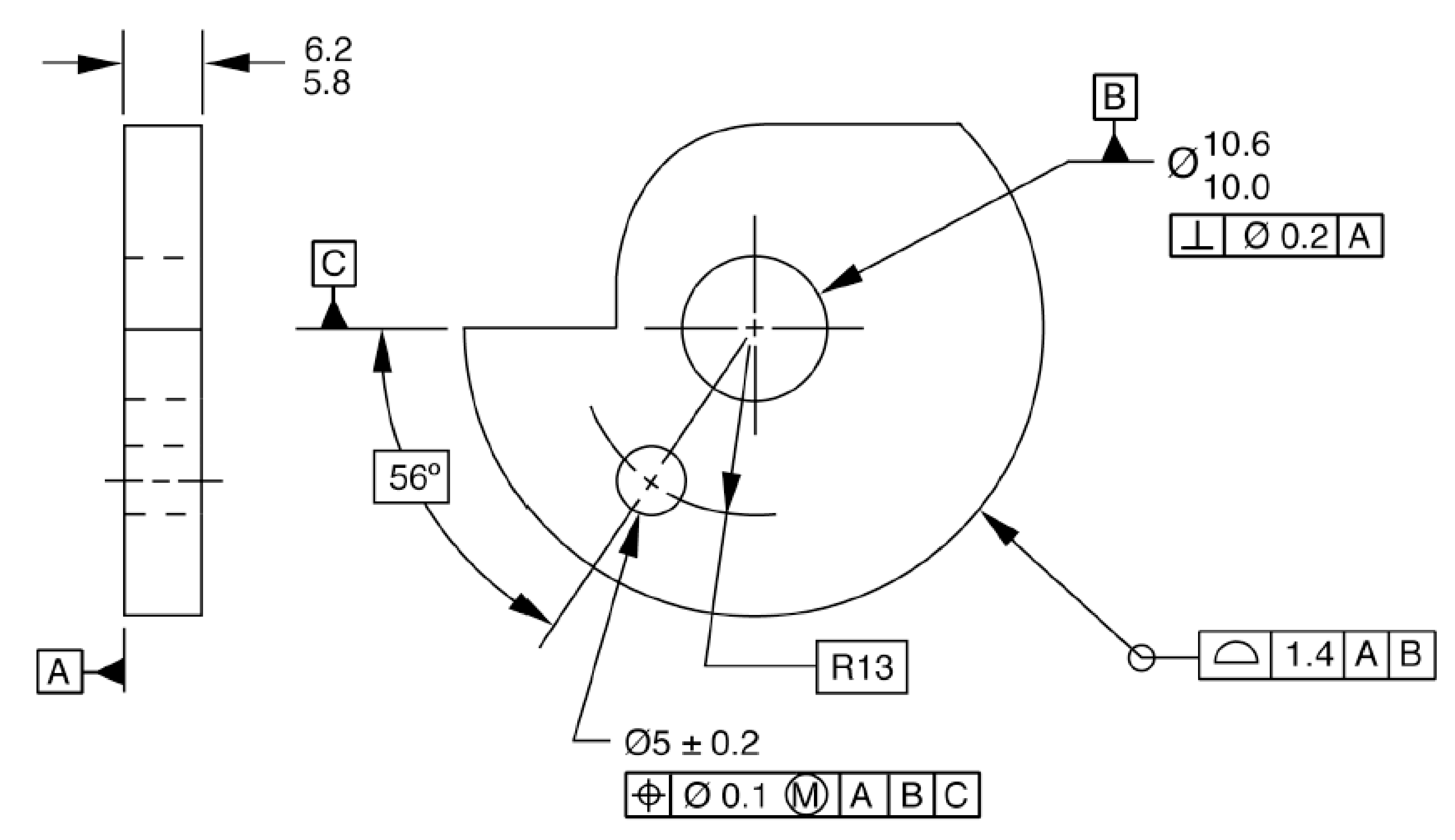
Position

- Allowable deviation for a feature of size from the "true position"

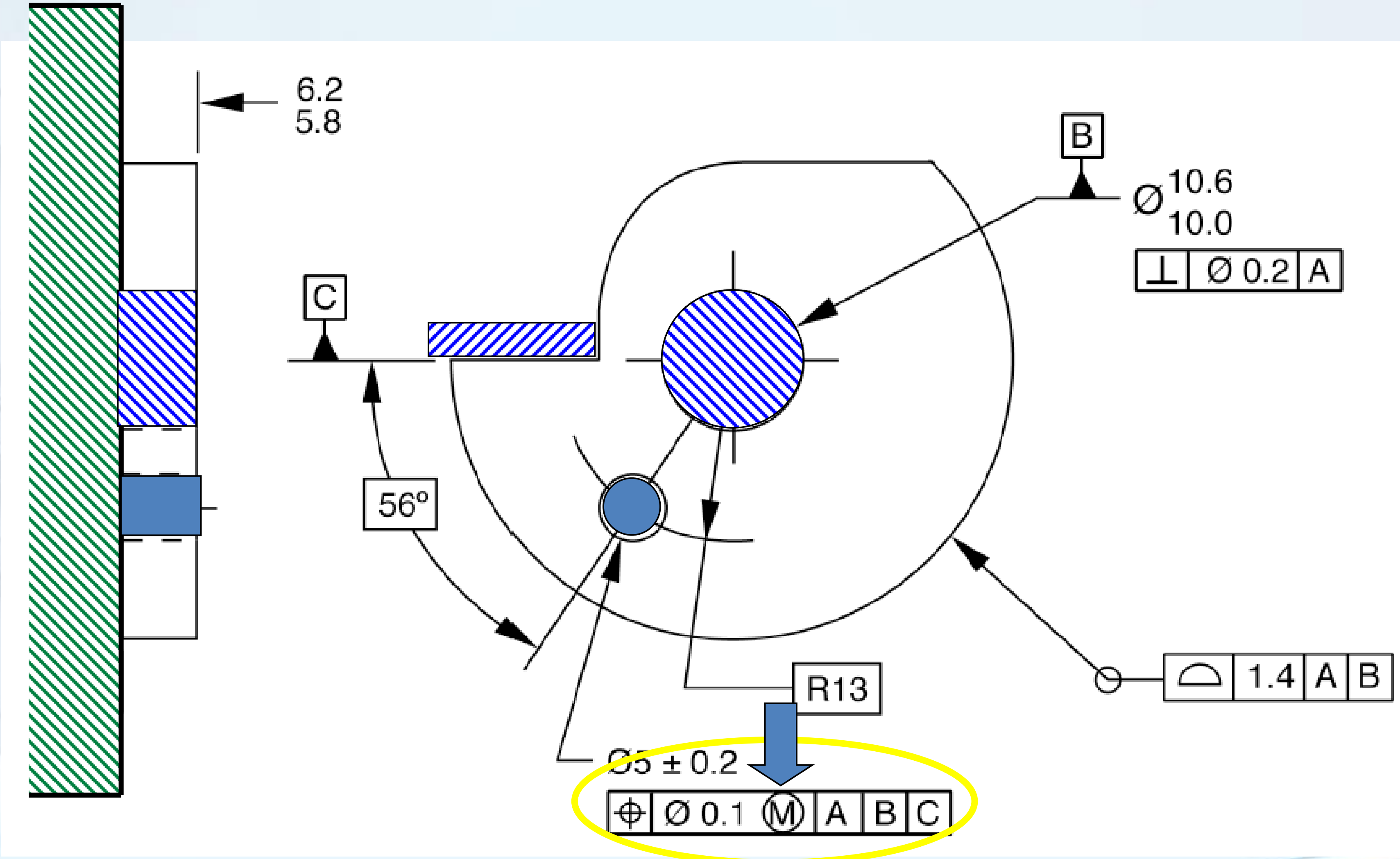
True Position:

The ________________________ location that is desired for a feature of size.

Position



Position



Position

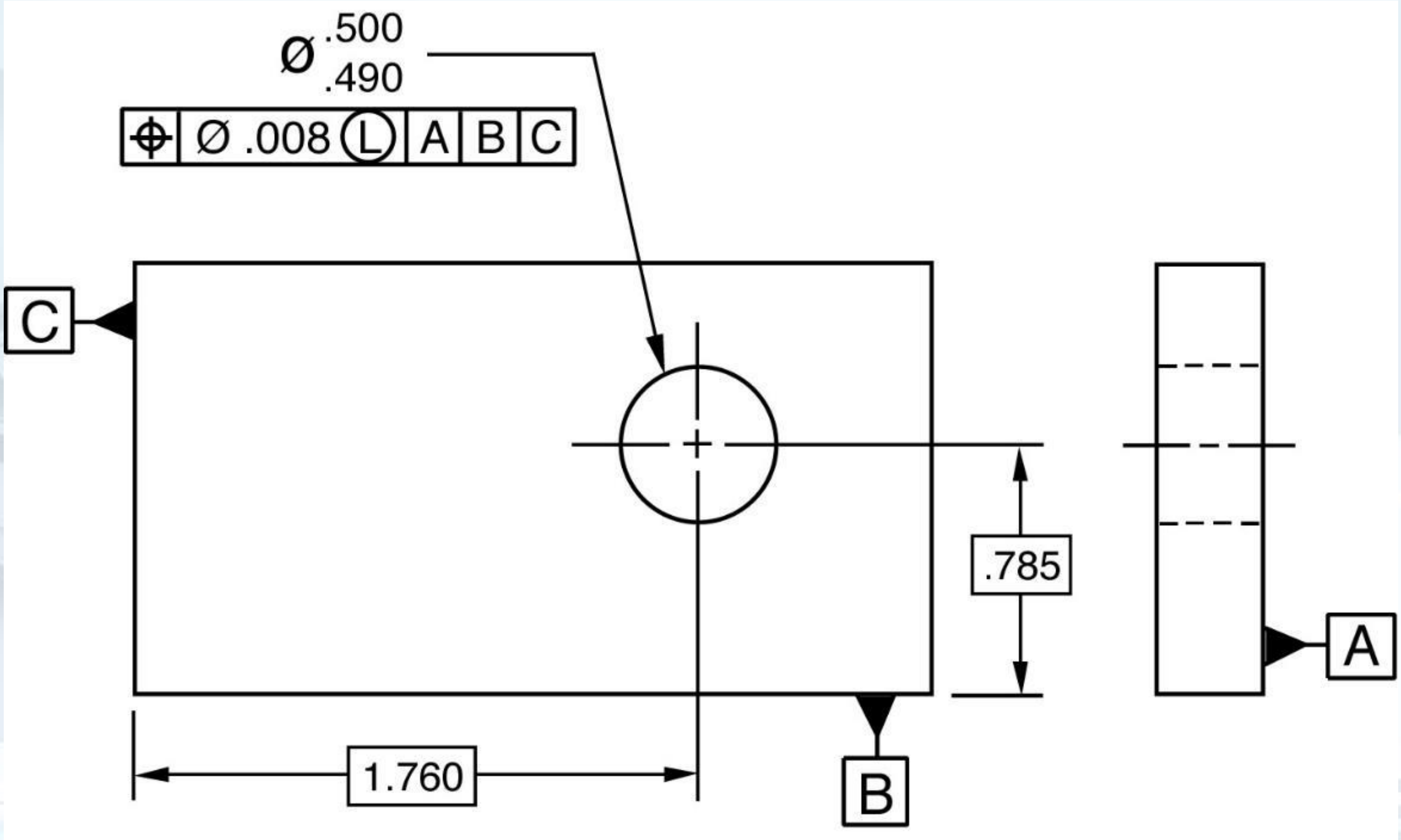
- It must reference a datum (one exception: for coaxial features positioned to each other)
- The tolerance is the total value, centered at the true position
- It may only be applied to a feature of size
- May use BOUNDARY concept
- It is often modified with the MMC symbol
- The relationship between the controlled feature and the datums must be clearly understood and defined as basic

Tolerance of Position

Common Position Applications (4 types)

1. Location of FOS
2. Control the distance between FOS
3. Coaxial relationship between FOS
4. Symmetrical relationship between FOS

Position @ LMC



OMNEX

Position @ LMC

- LMC not useful for mating parts
- LMC can be used for protecting wall thickness

QUALITY

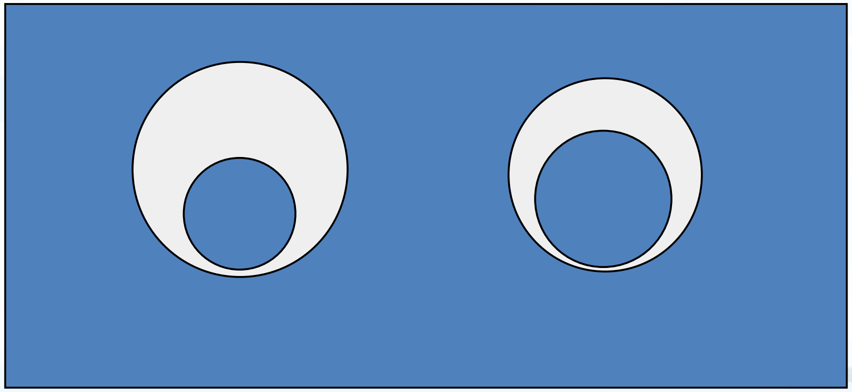
OMNEX

One of Three Choices…

MATERAL CONDITION	COMMON USAGE	COMMENTS
Ⓜ	Assembly (clearance fit)	• Very common modifier • Allows bonus tolerance • Always ensures clearance • Allows functional gaging
Ⓛ	To maintain a minimum wall thickness or machine stock	• Least common modifier • Allows bonus tolerance • Opposite effect of MMC • Requires variable gaging
RFS (No modifier)	Centering/alignment; a symmetrical relationship	• Most expensive condition • No bonus tolerance • Is implied per Rule #2 • Requires variable gaging

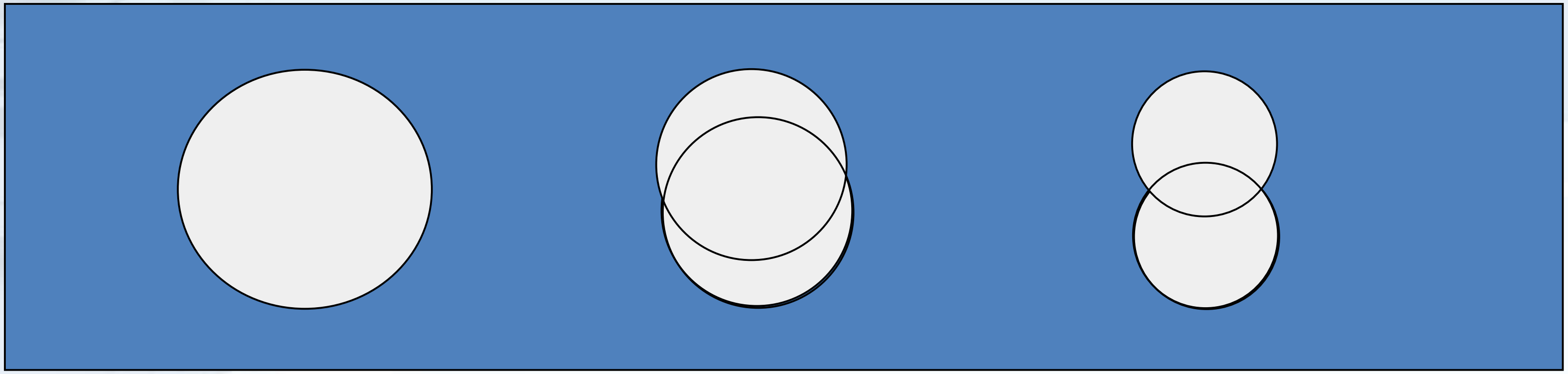
MMC

- Used for clearance fits
- Very common modifier
- Allows for bonus tolerance
- Ensures clearance
- Allows functional gaging



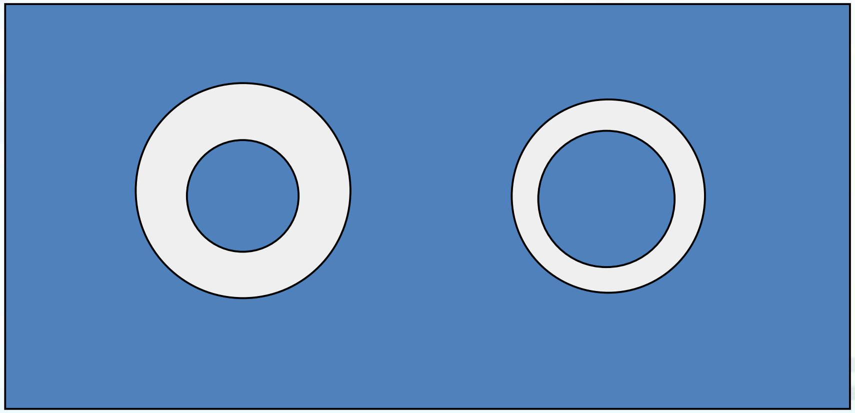
LMC

- Bonus (works opposite of MMC)
- Good for controlling minimum wall thickness:

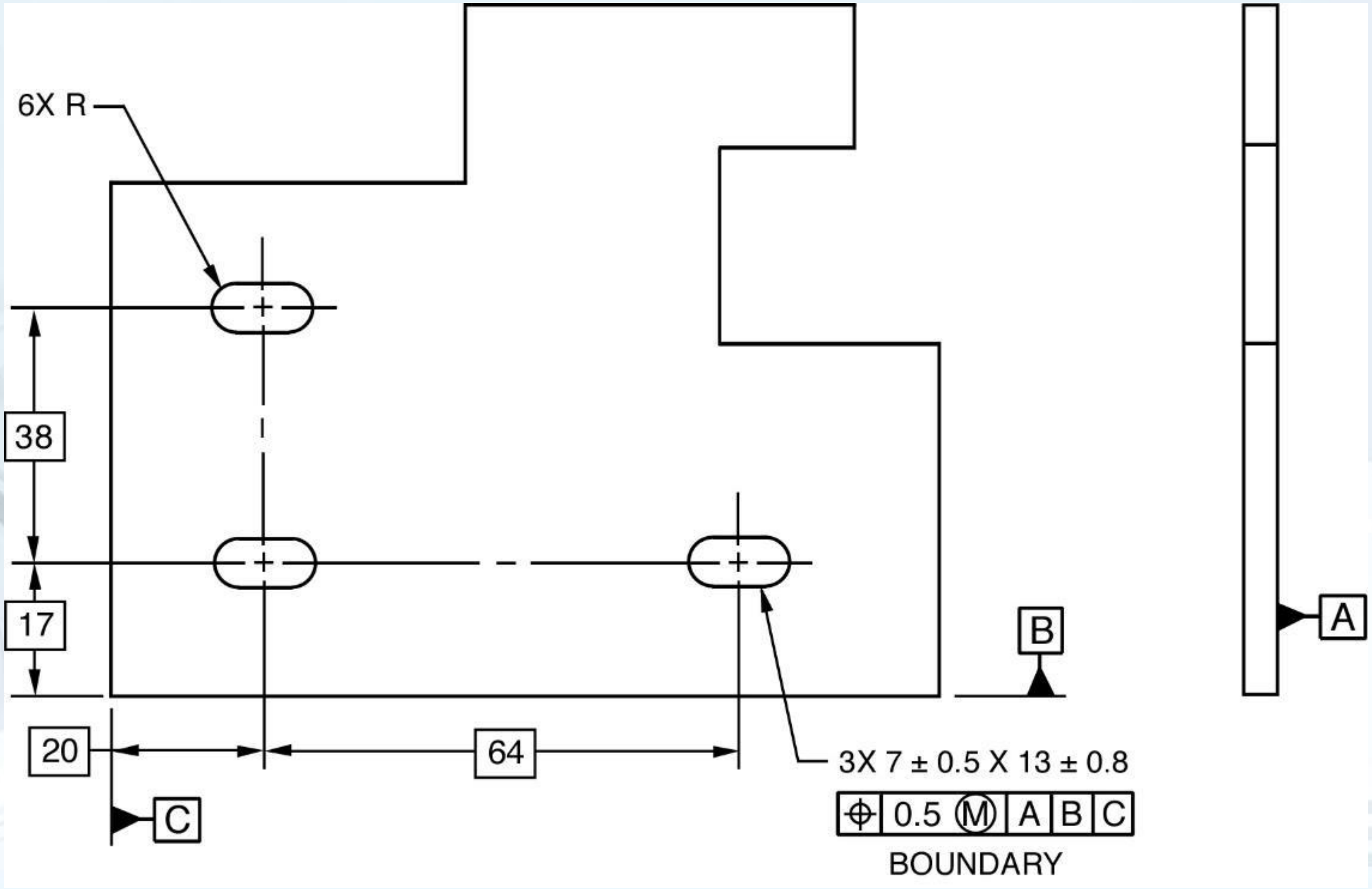


RFS

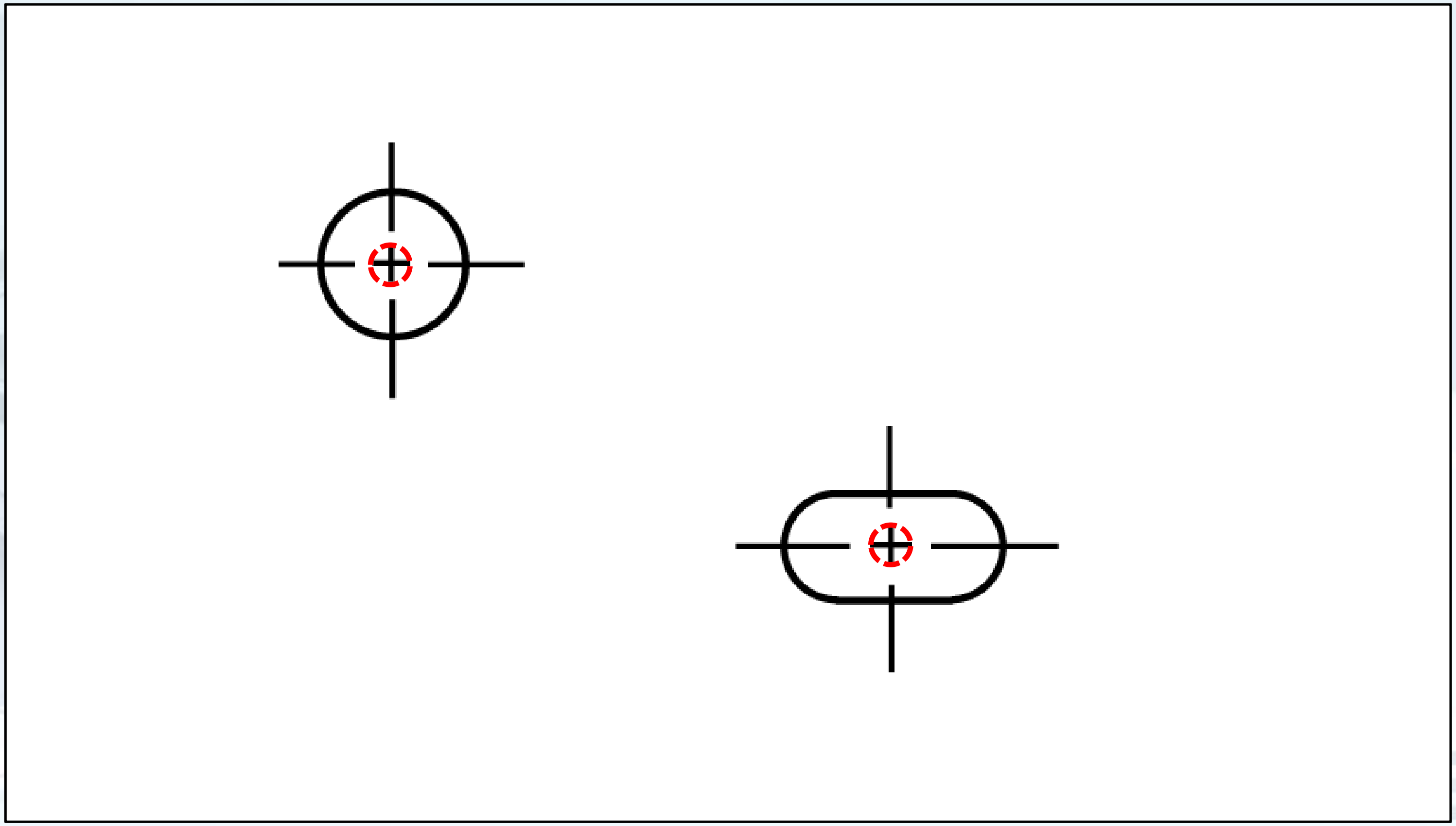
- Used for centering
- More expensive
- No bonus
- Is implied if no modifier
- Variable gaging



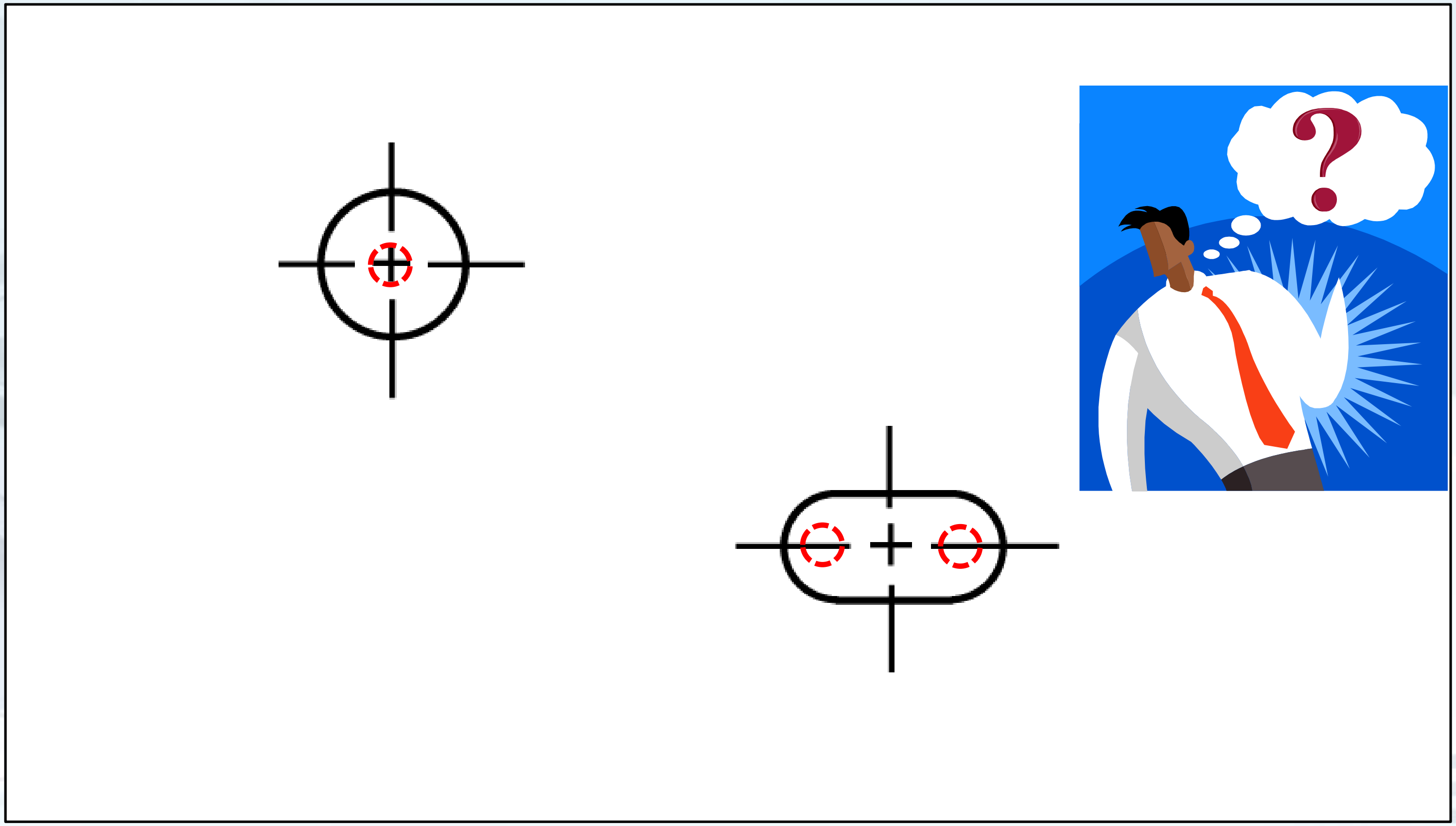
Boundary



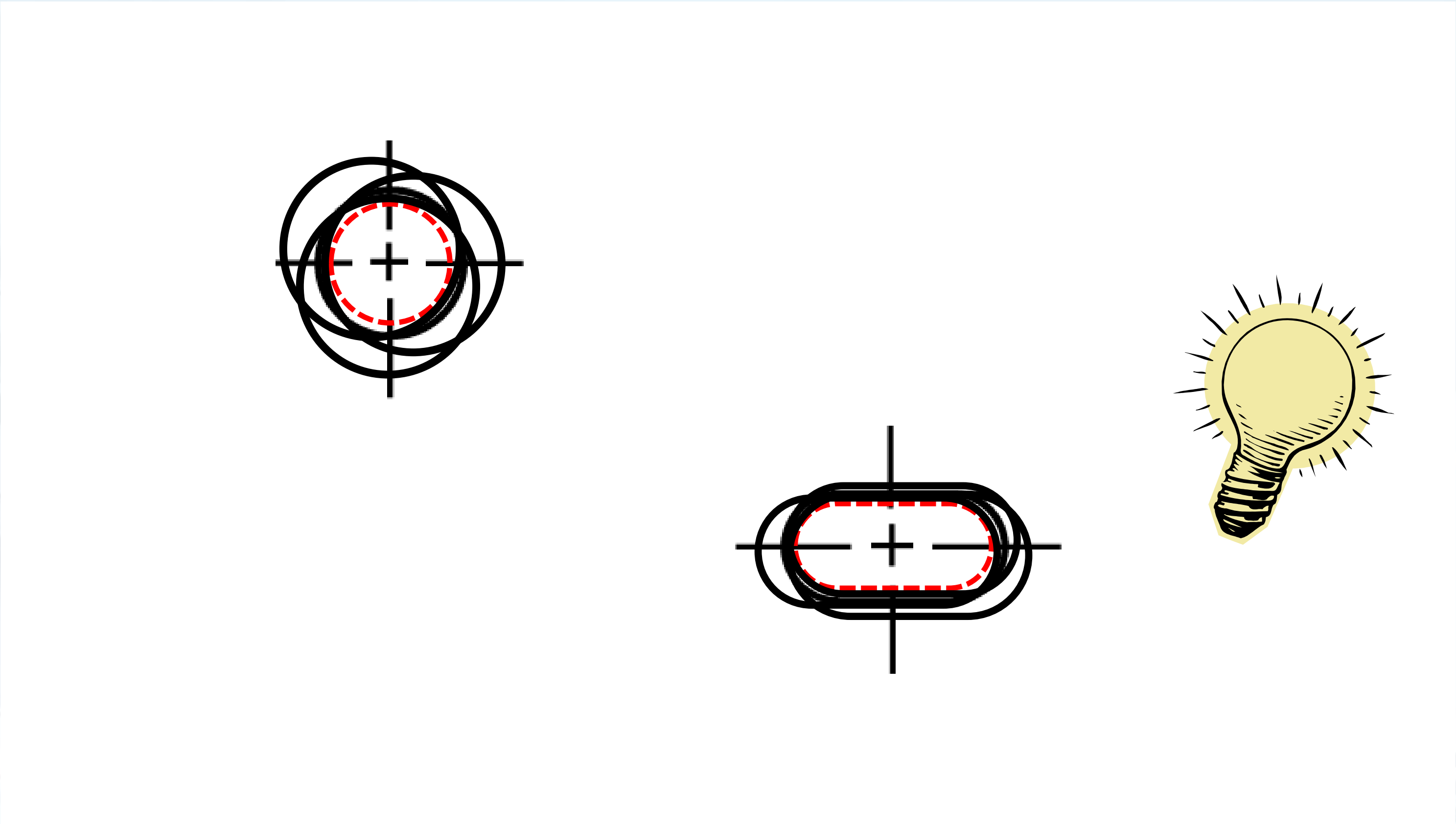
Where Is the Tolerance Zone?



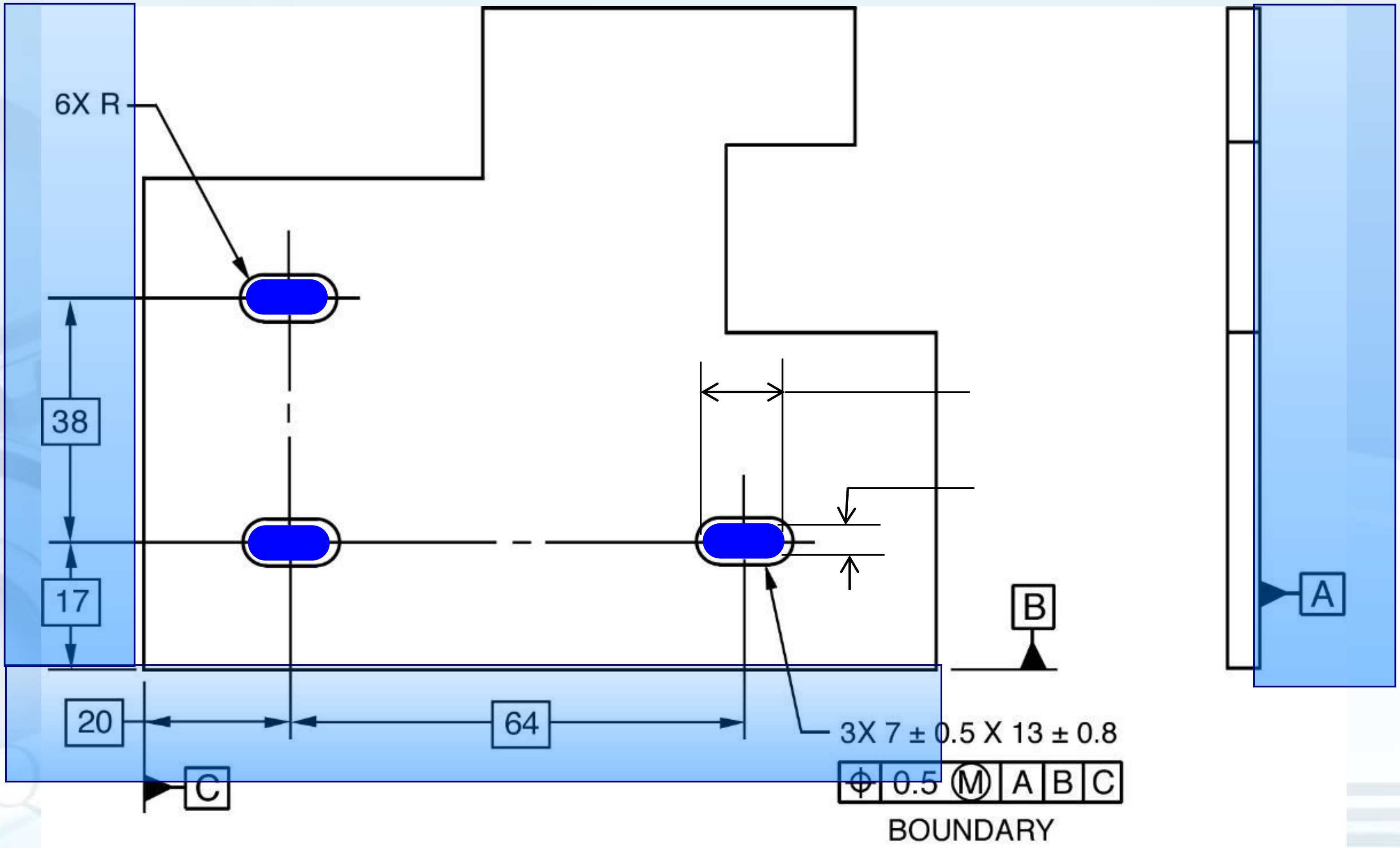
Where Is the Tolerance Zone?



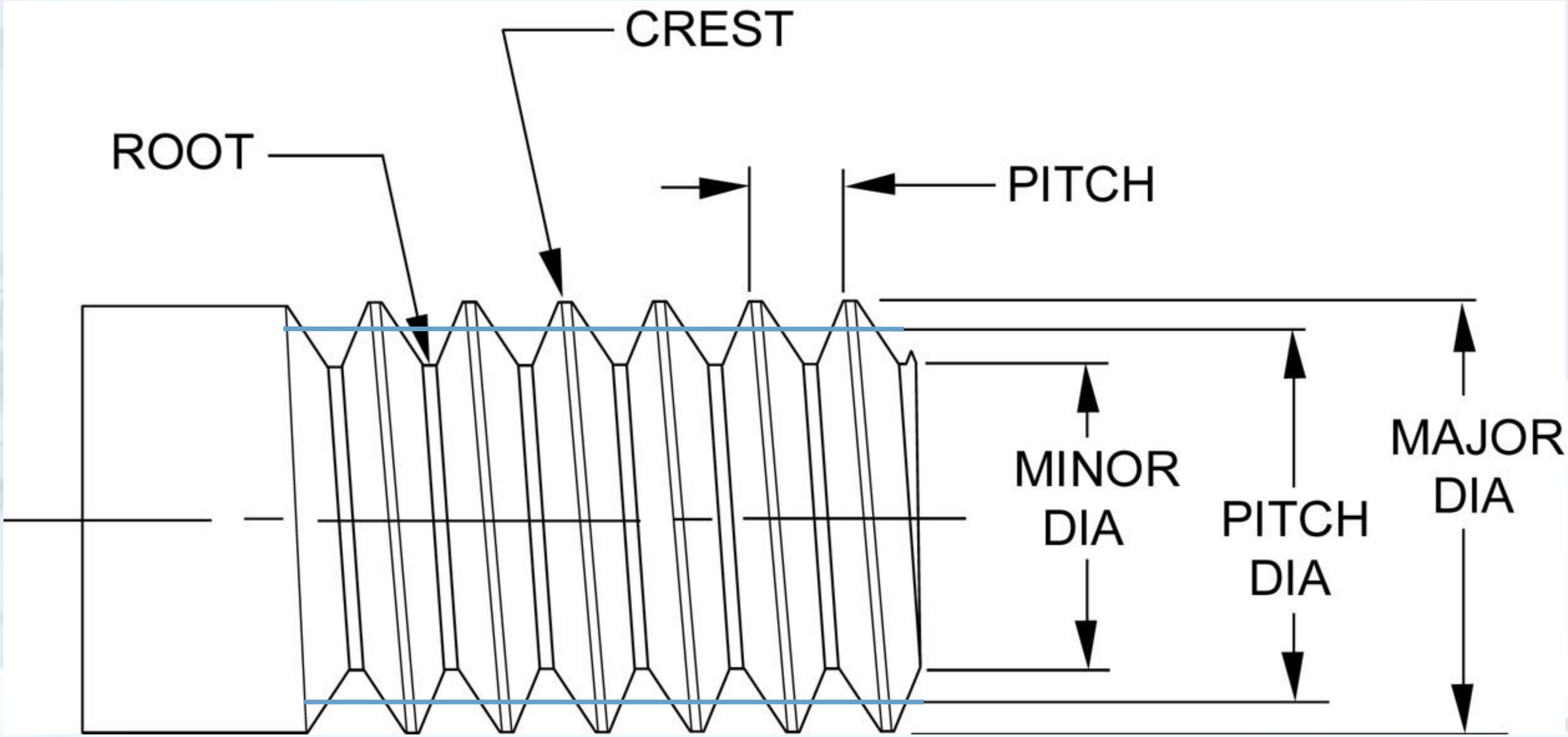
Remember Virtual Condition?



Boundary – A Functional Gage



Thread Terminology



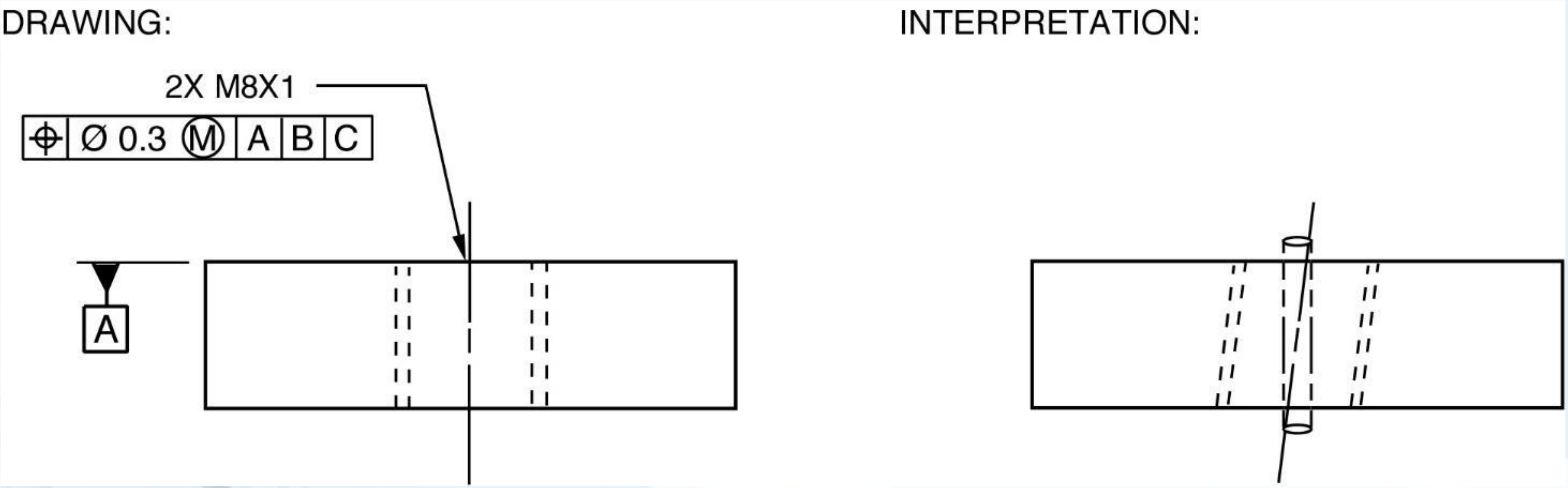
OMNEX

Screw Threads

- How to measure GD&T on threads?
  a) Major diameter
  b) Minor diameter
  c) Pitch diameter ← default

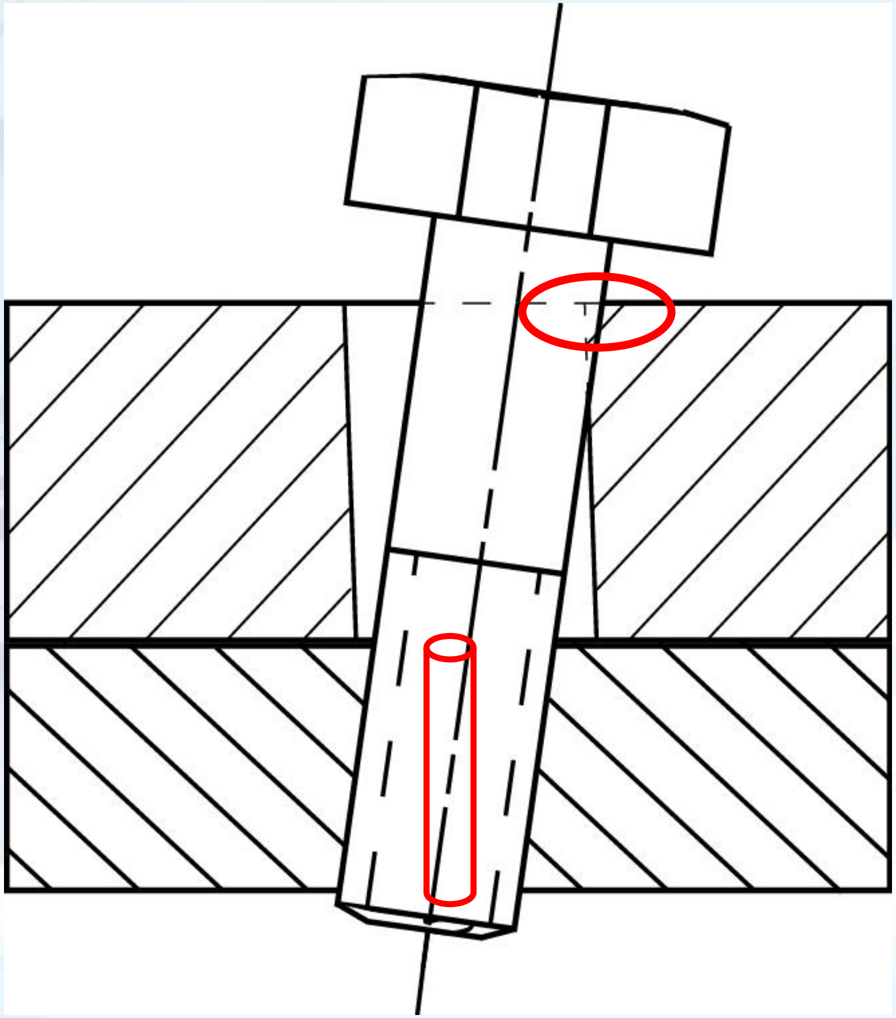
The ASME rule is to assume pitch diameter as the basis for geometric tolerances and datums on threads, unless otherwise specified.

Threaded Holes



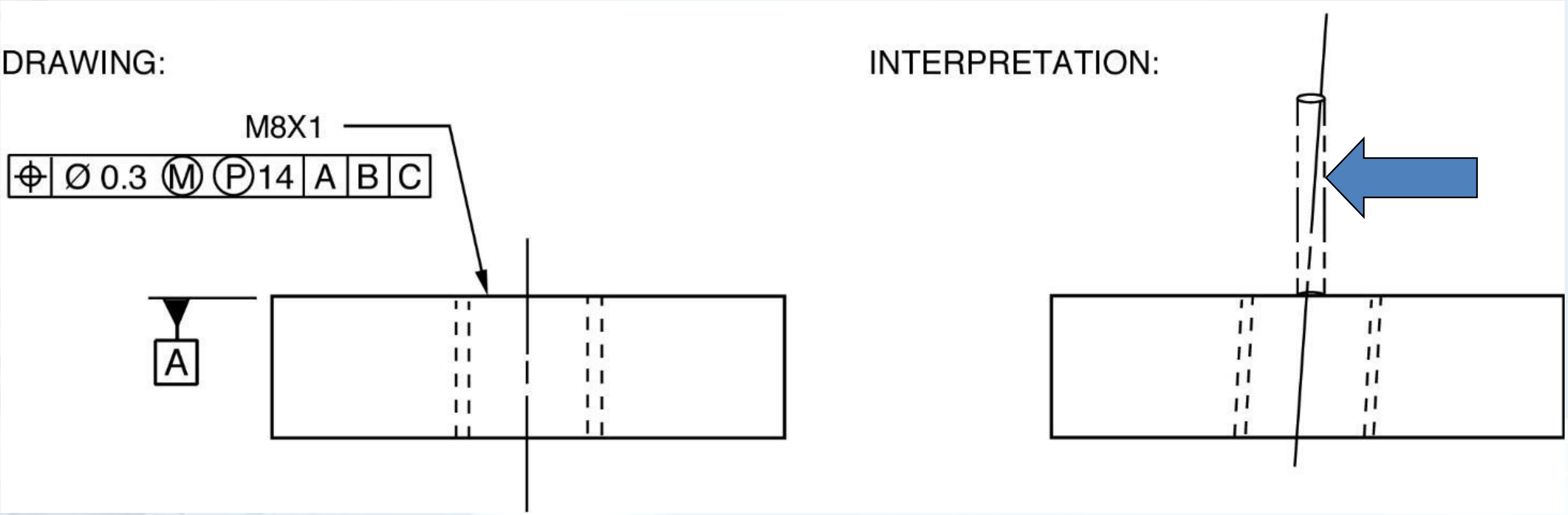
But wait: This tolerance extends only through this part. Yet there will be a fastener that carries this potential tilt into an adjacent part…

Interference



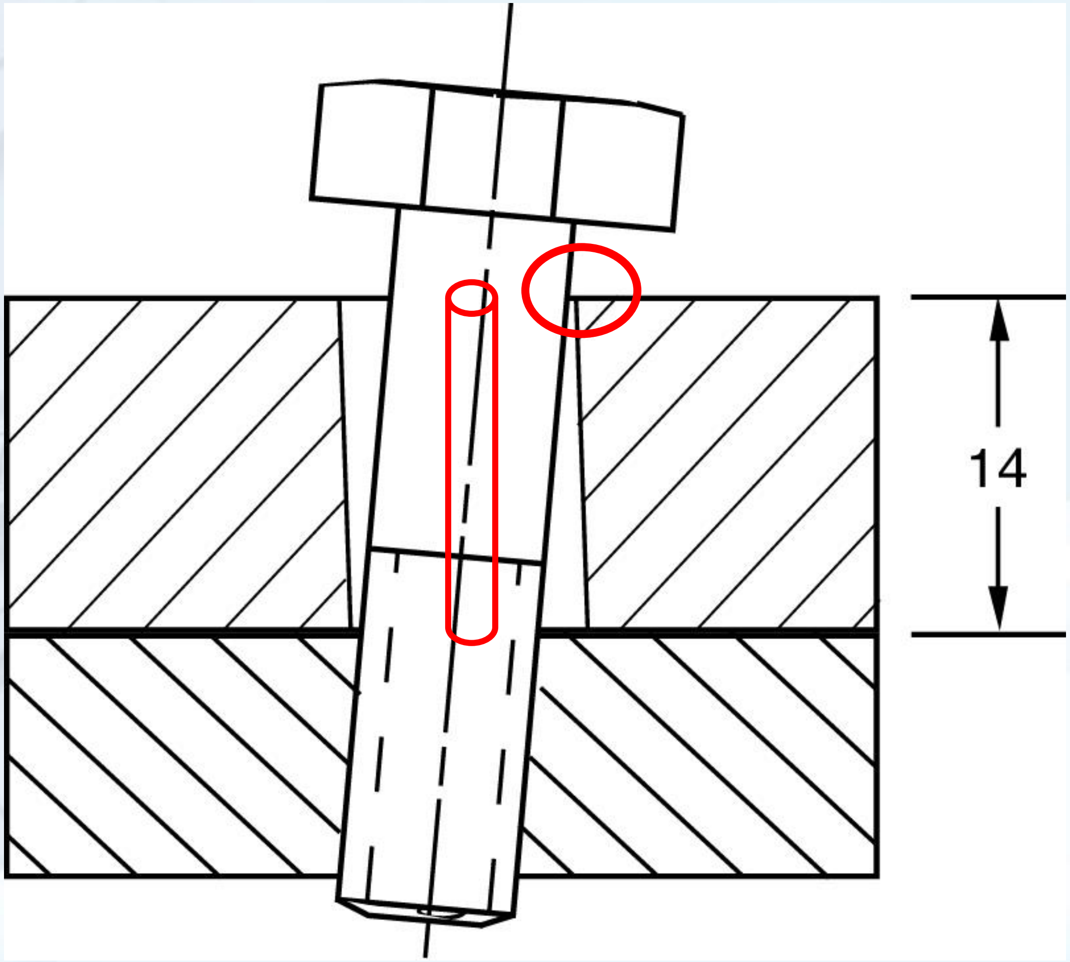
Possible interference between parts that fit the normal tolerance zones

Projected Tolerance Zone



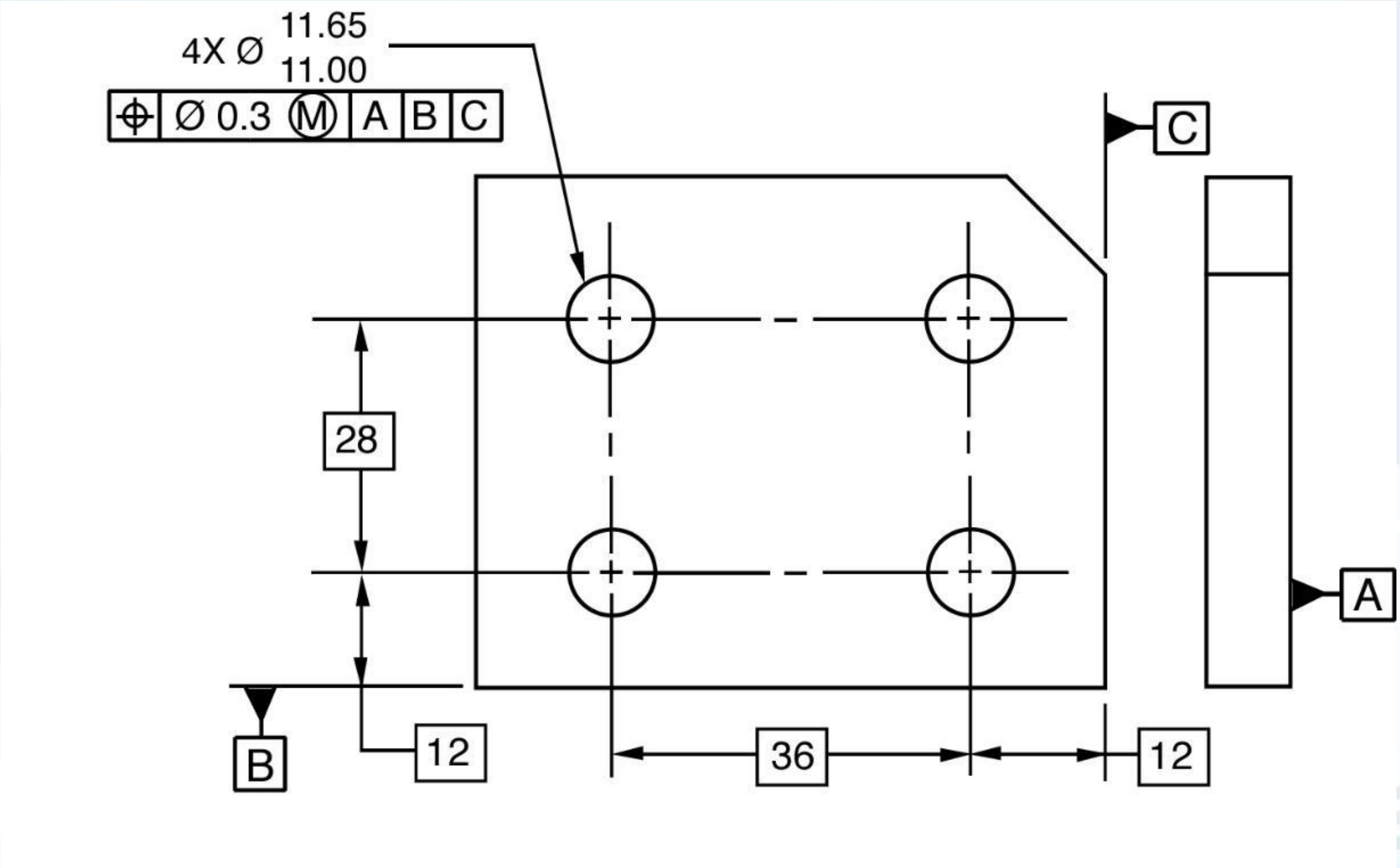
By designating a projected tolerance zone, the same diameter tolerance can be assigned to the functional area in the mating part...

Projected Tolerance Zone

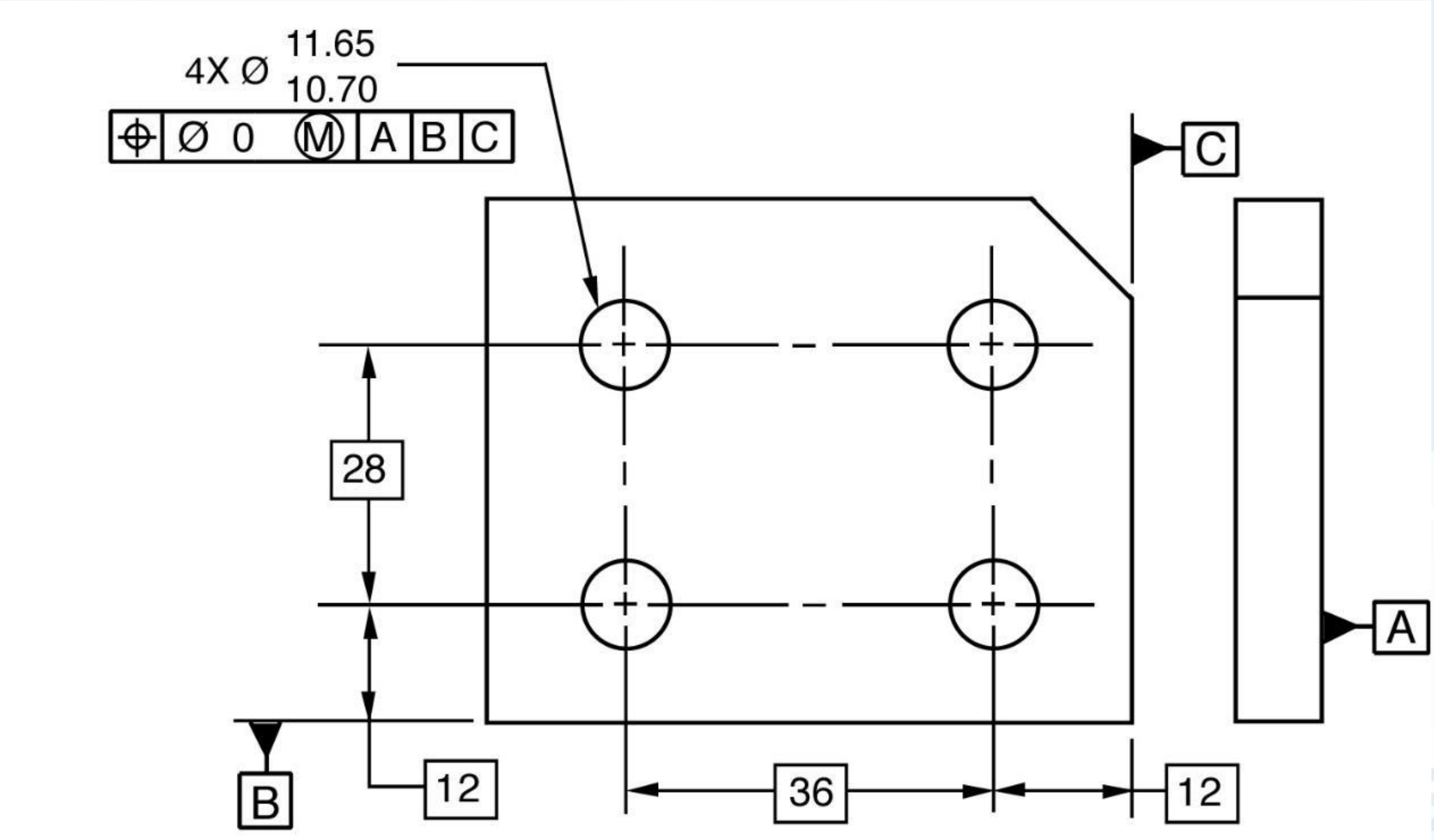


Amount zone needs to be projected is the height of the mating part

Compare This Tolerance…



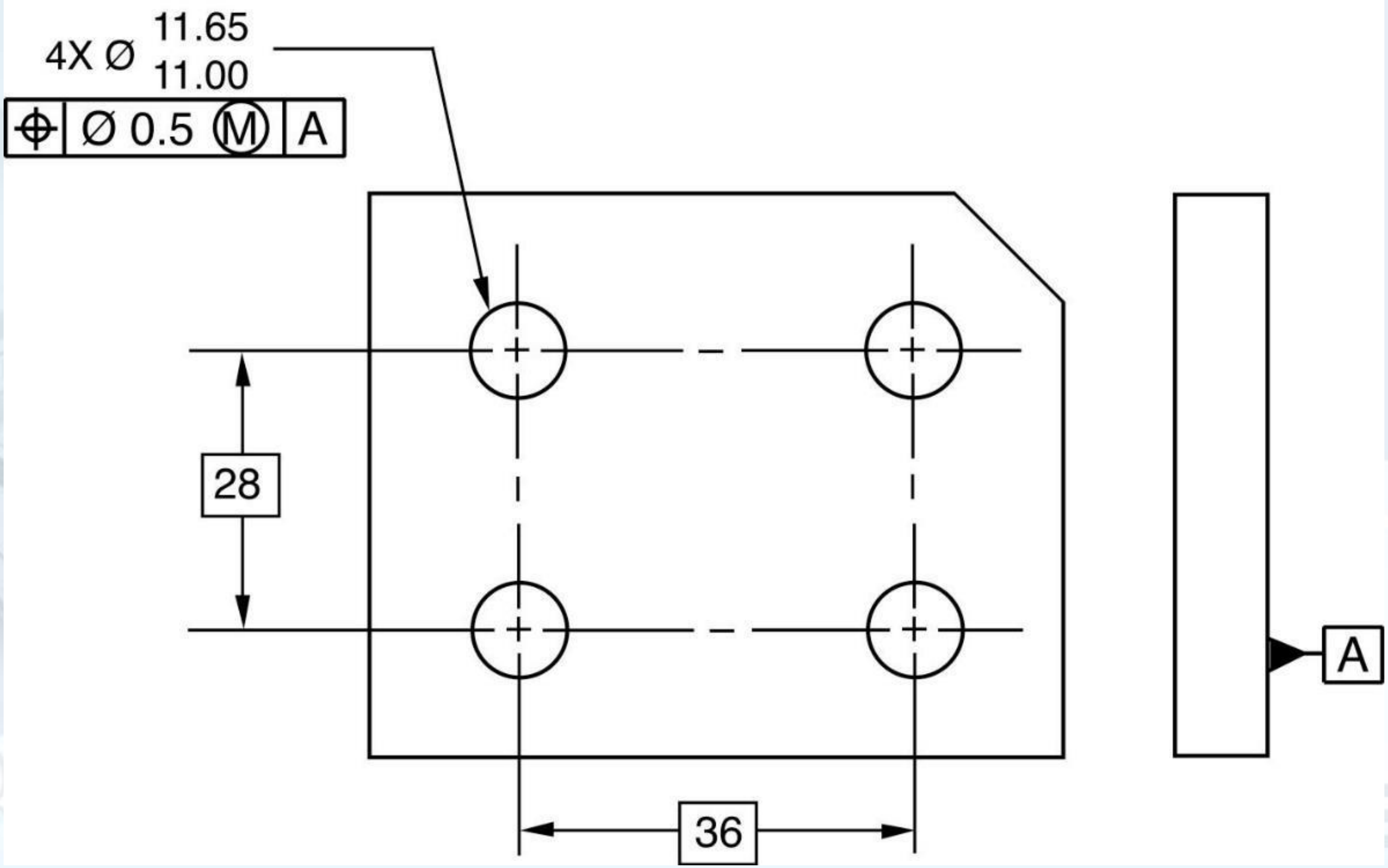
To This Tolerance



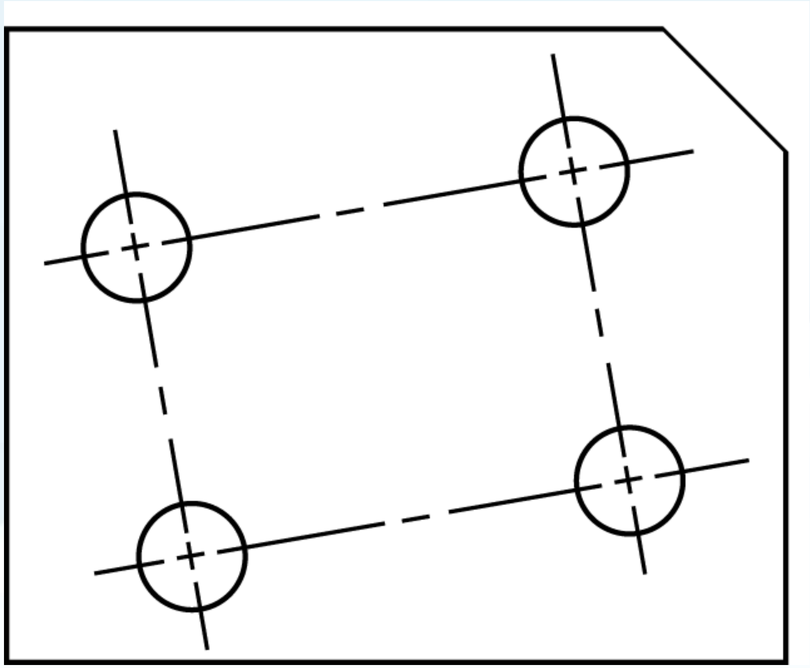
Zero Tolerance @ MMC

- Provides the same fit/assembly requirements
- Accepts **more** possible size/positional combinations
- Mass or threads may be a reason to avoid

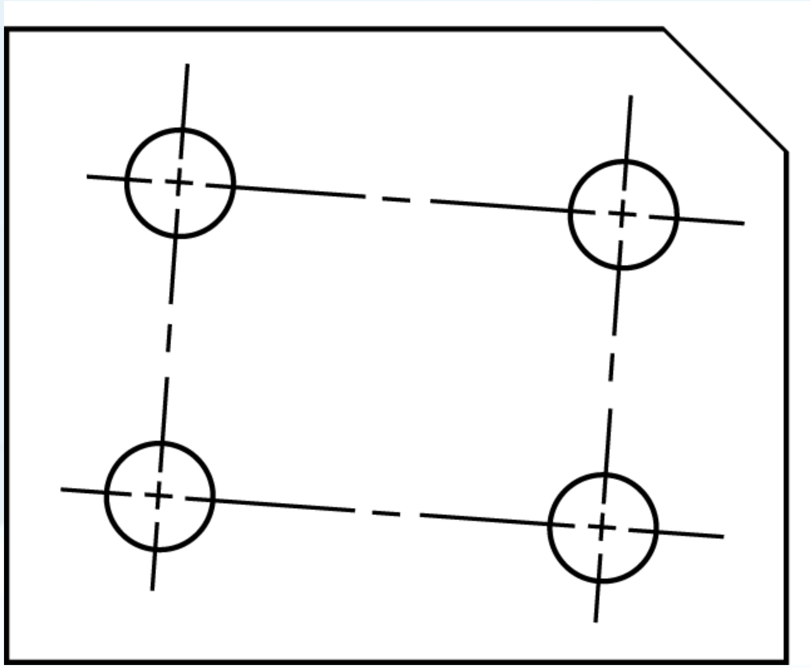
Hole-to-Hole Location



Locates Holes to Each Other

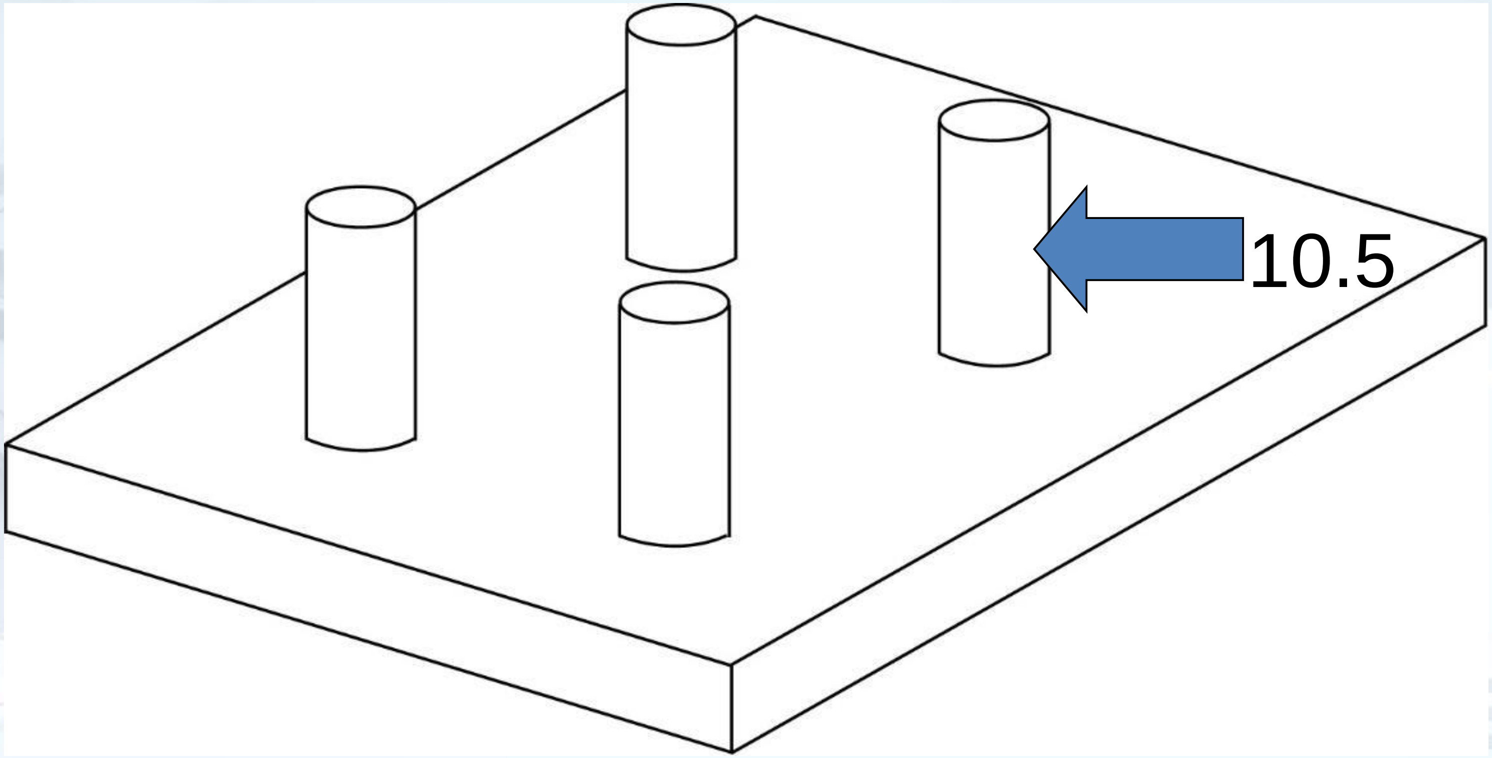


Locates Holes to Each Other

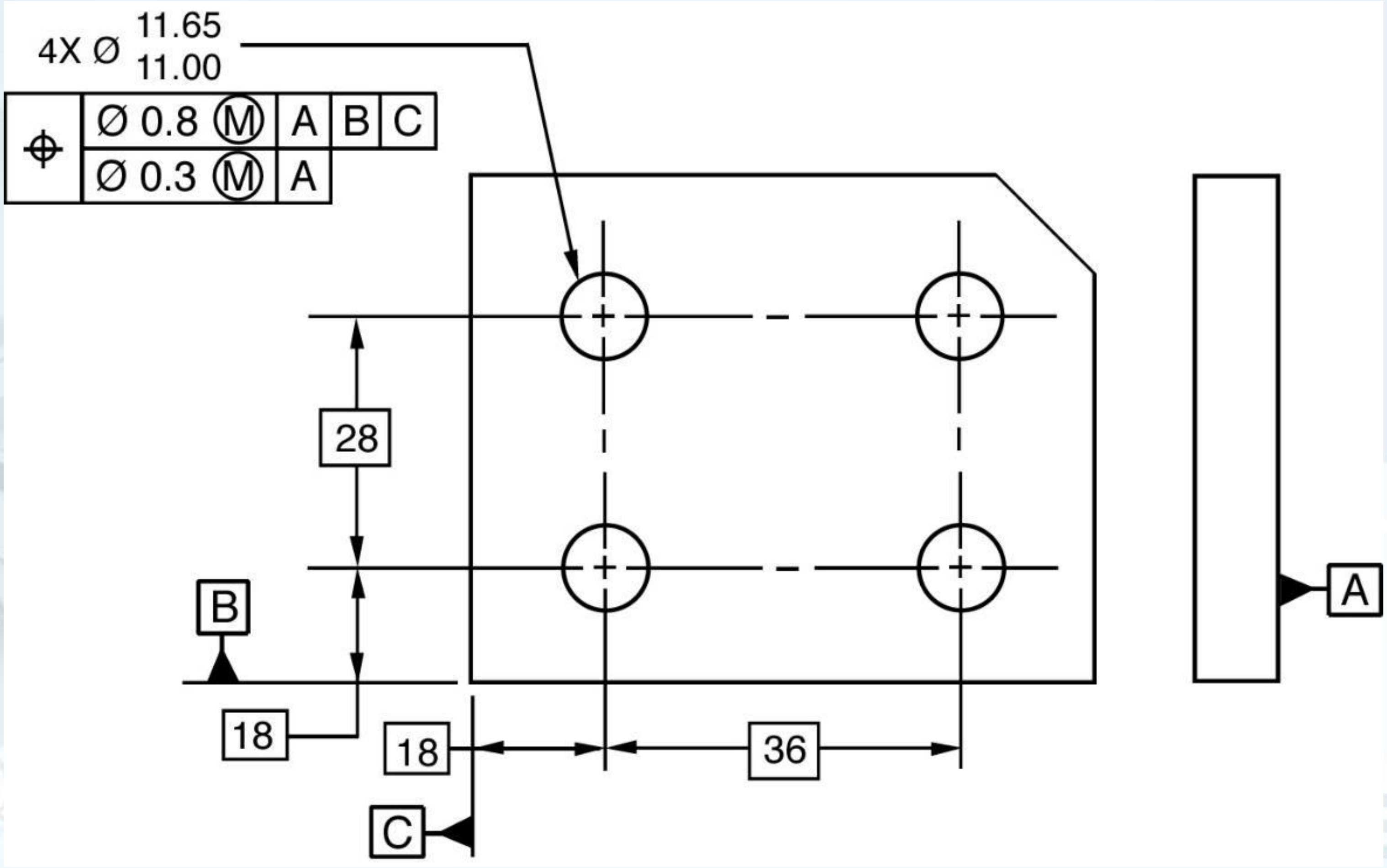


Functional Gage

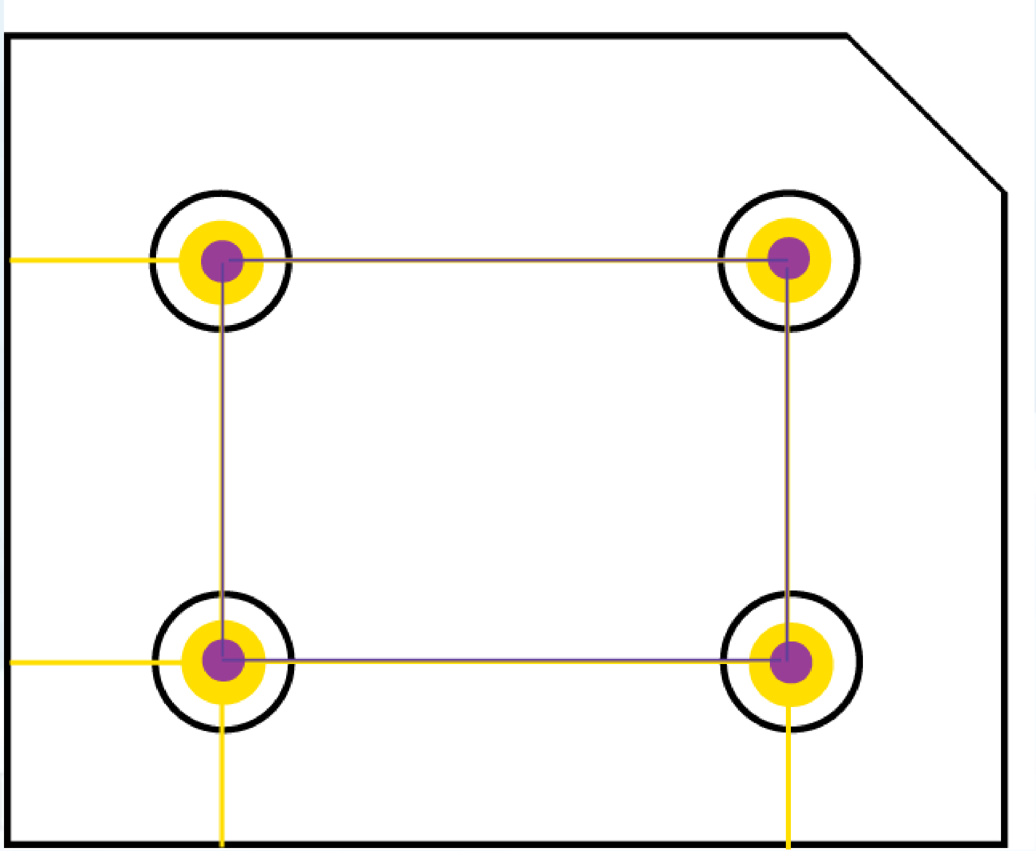
(Checks hole-to-hole location, and perpendicularity to A)



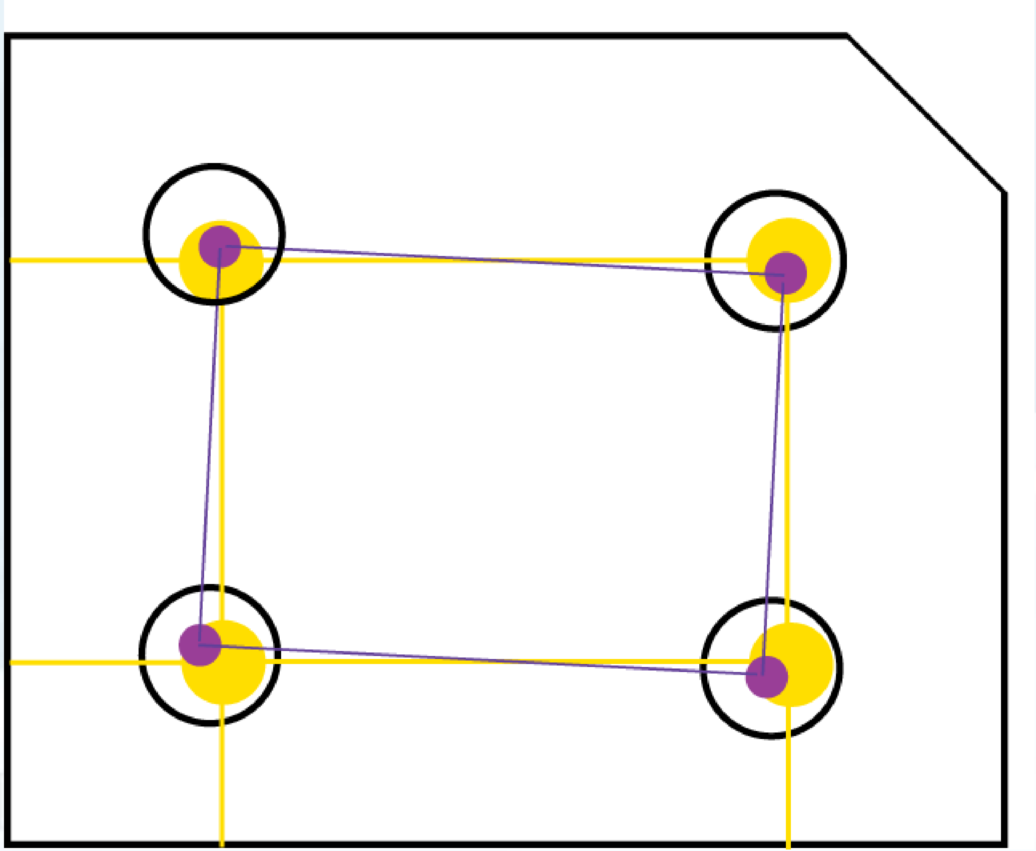
Composite Position



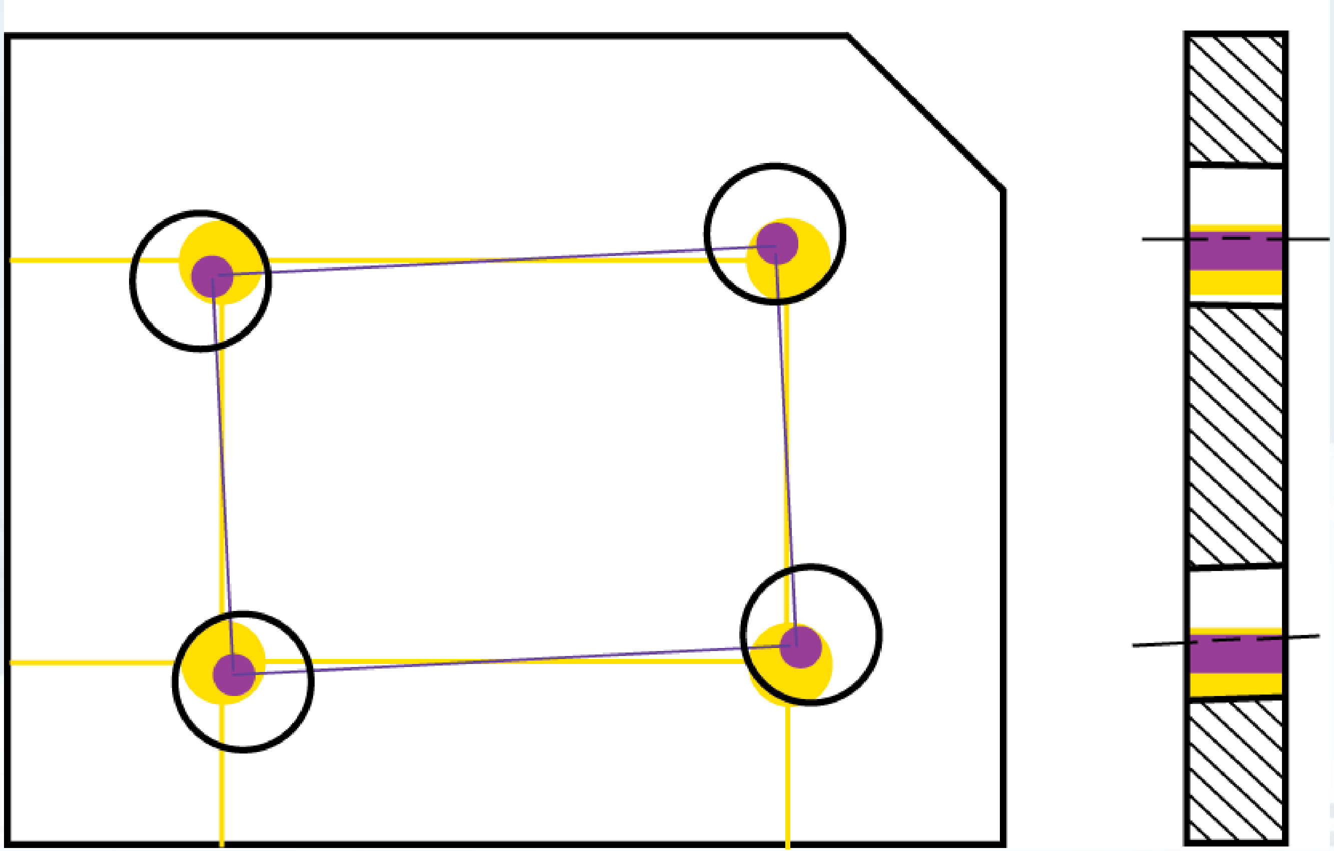
Two Tolerance Zones



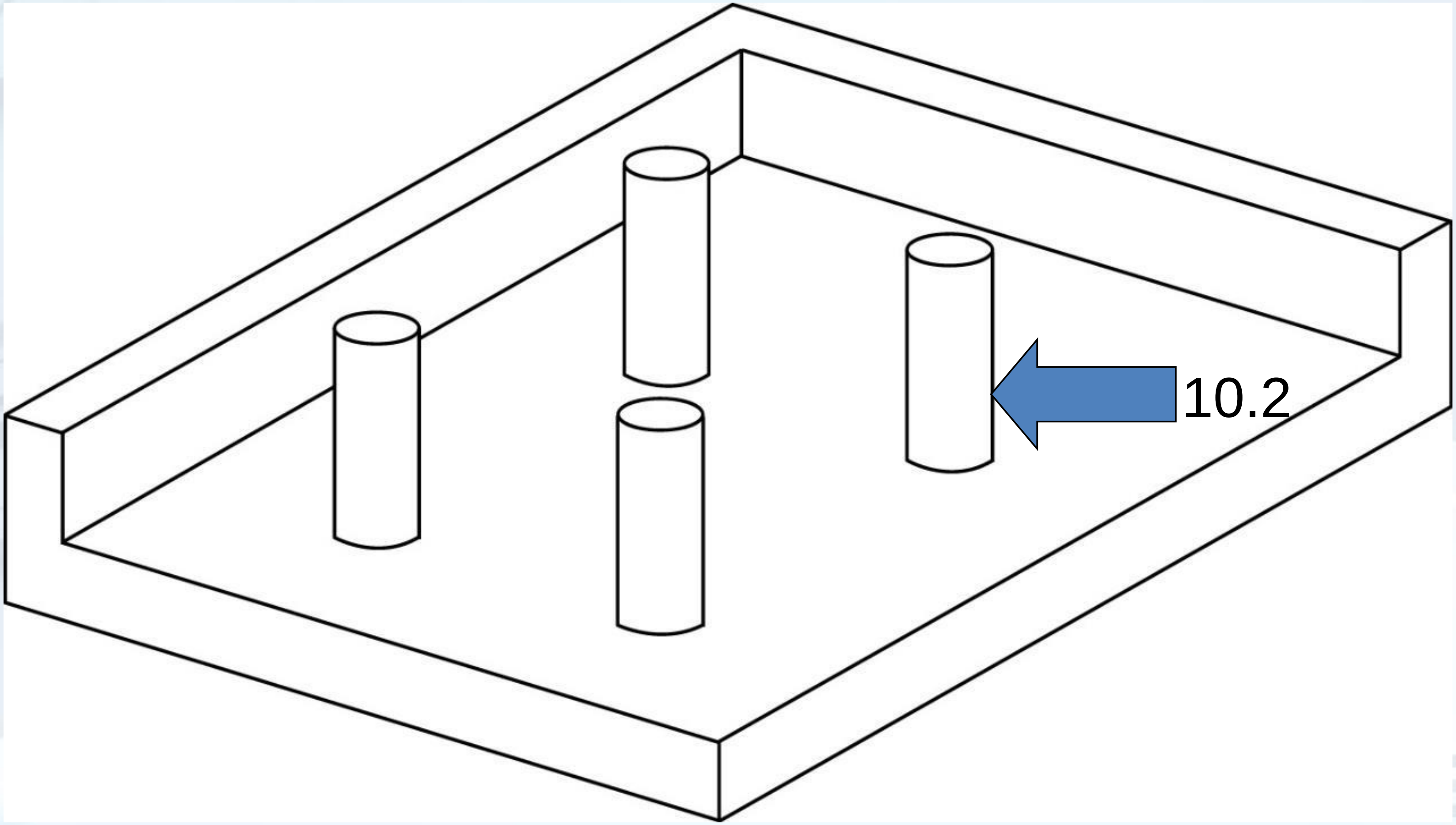
Two Tolerance Zones



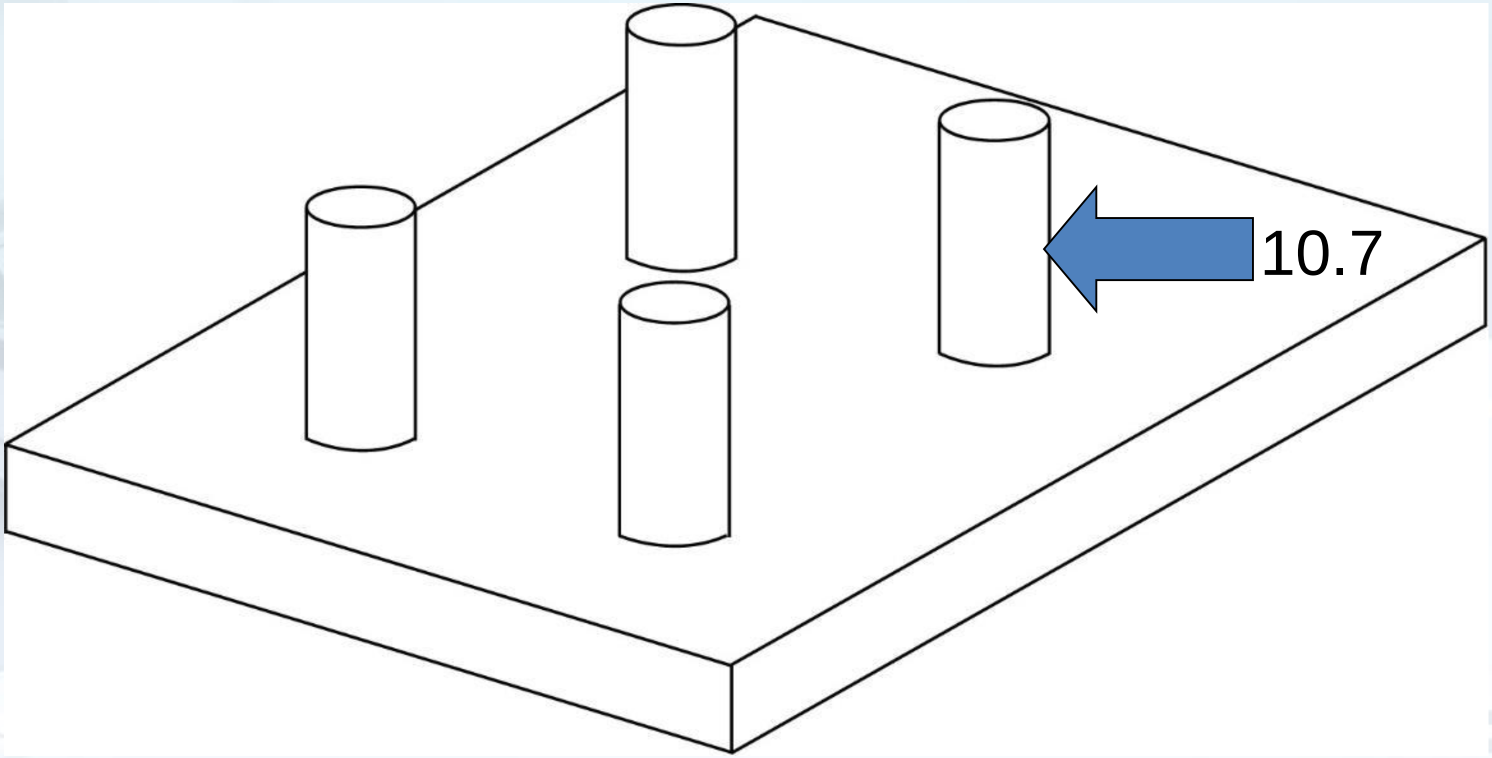
Two Tolerance Zones



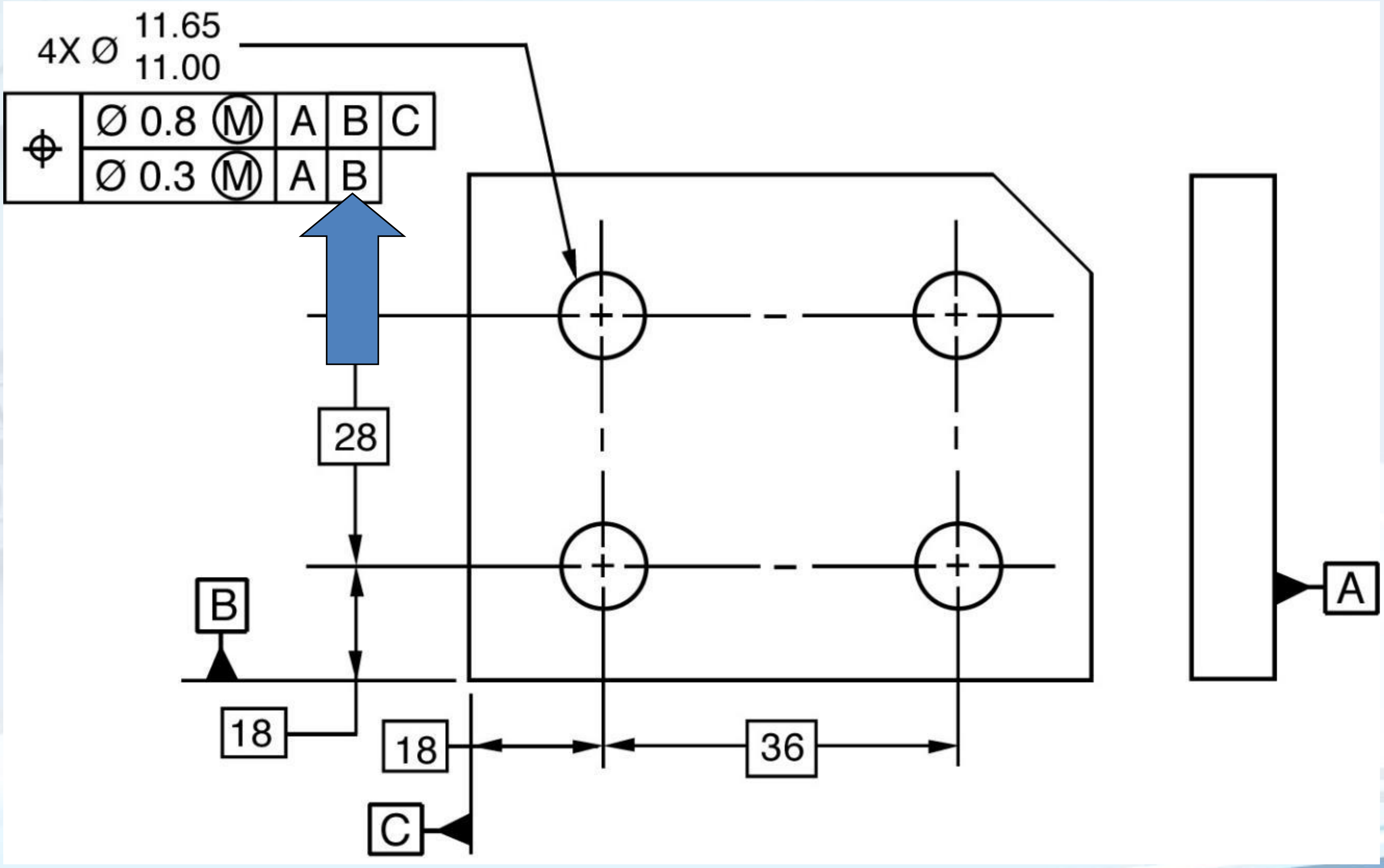
Gage for Upper Frame



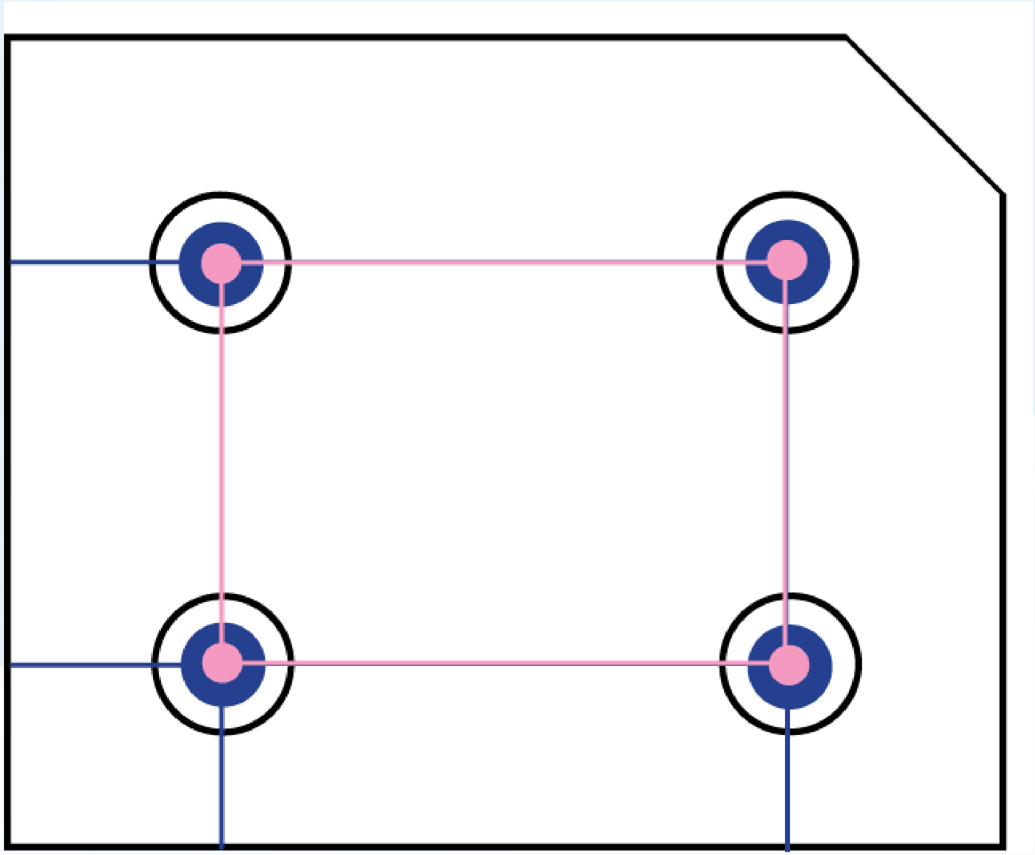
Gage for Lower Frame



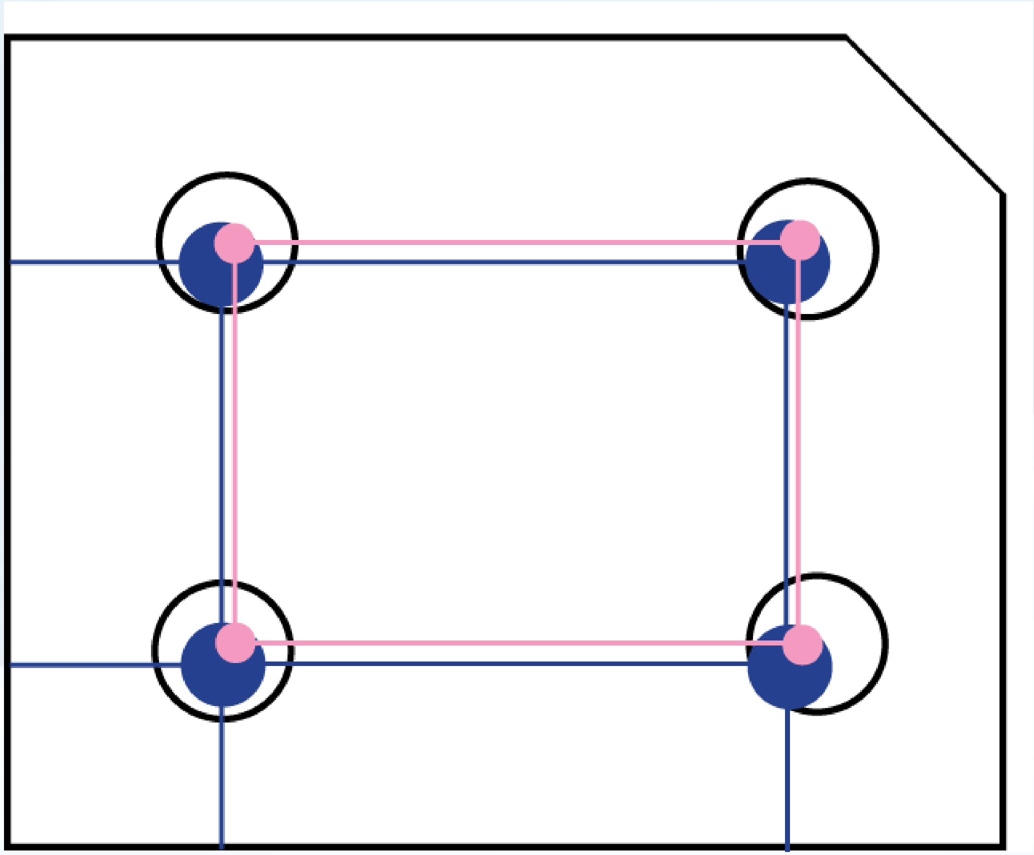
Composite Position



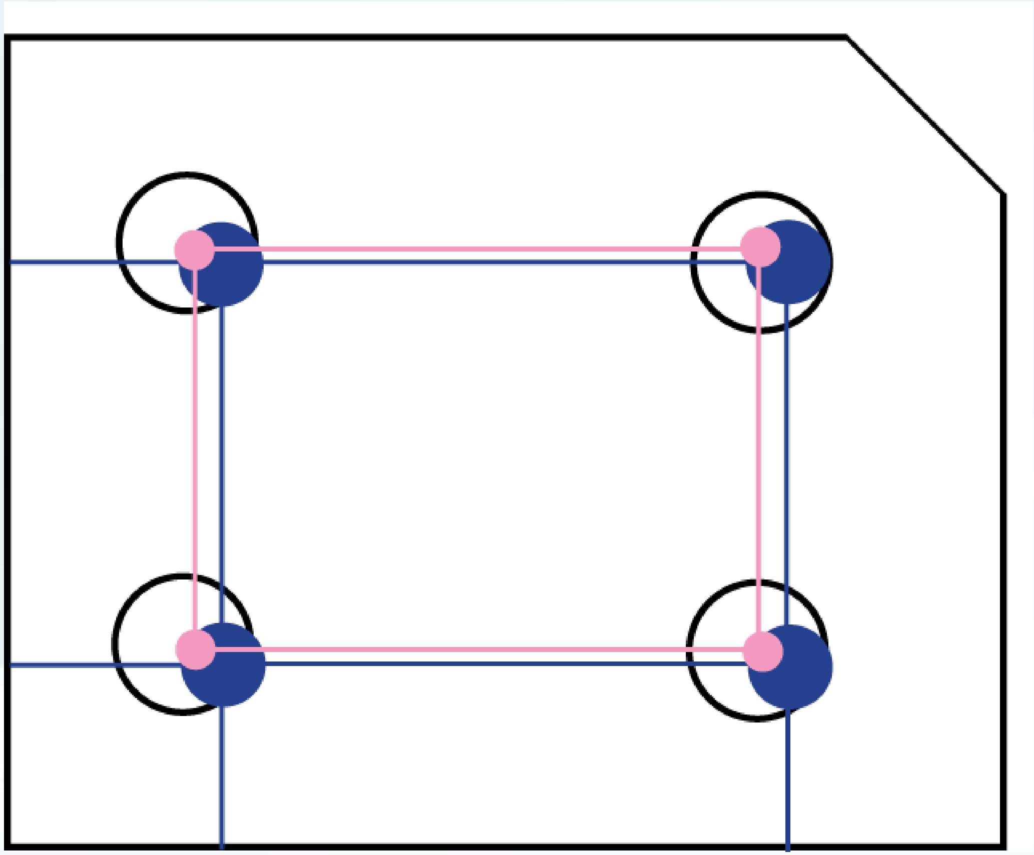
Bottom Frame Controls Orientation to the Datums



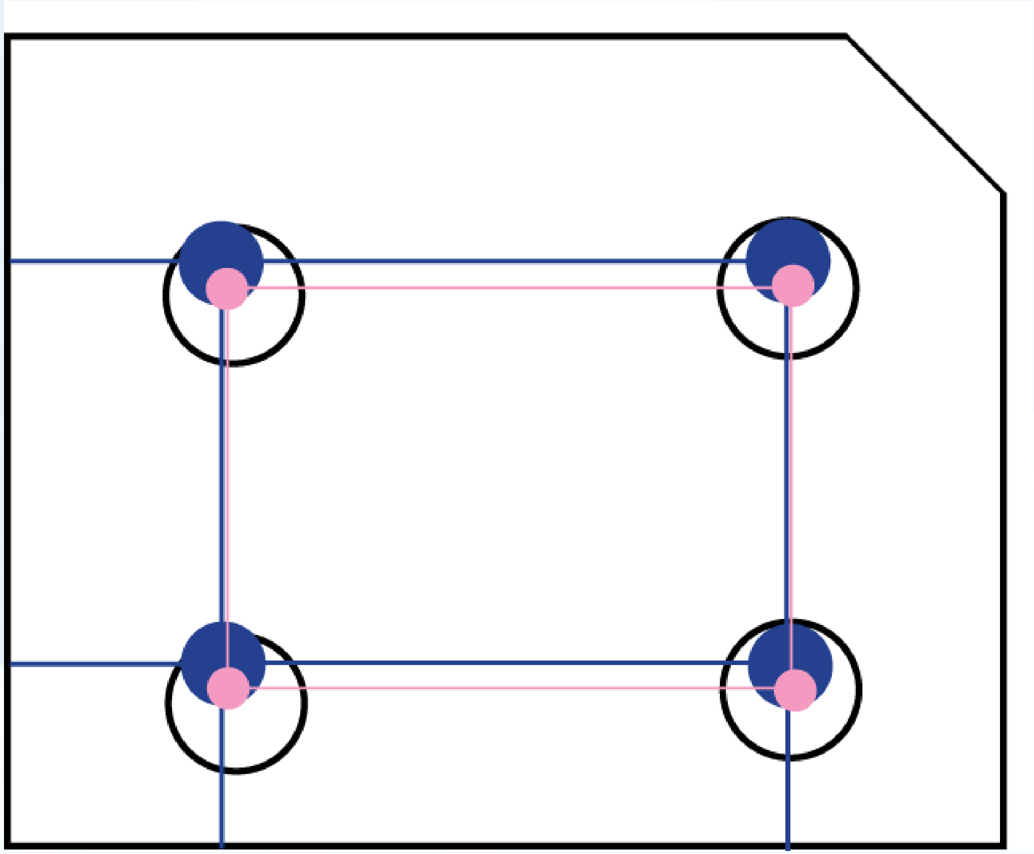
Bottom Frame Controls Orientation to the Datums



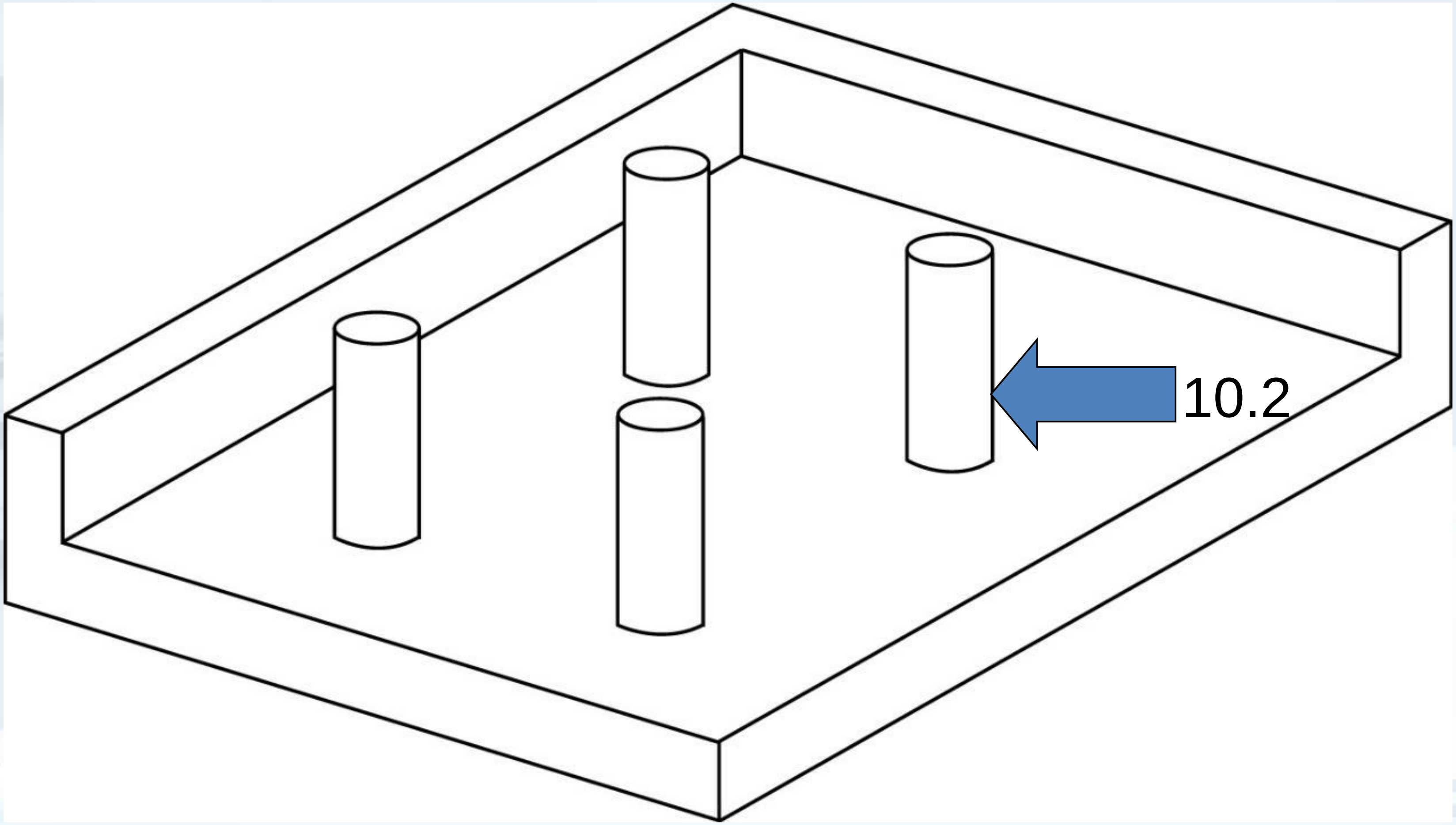
Bottom Frame Controls Orientation to the Datums



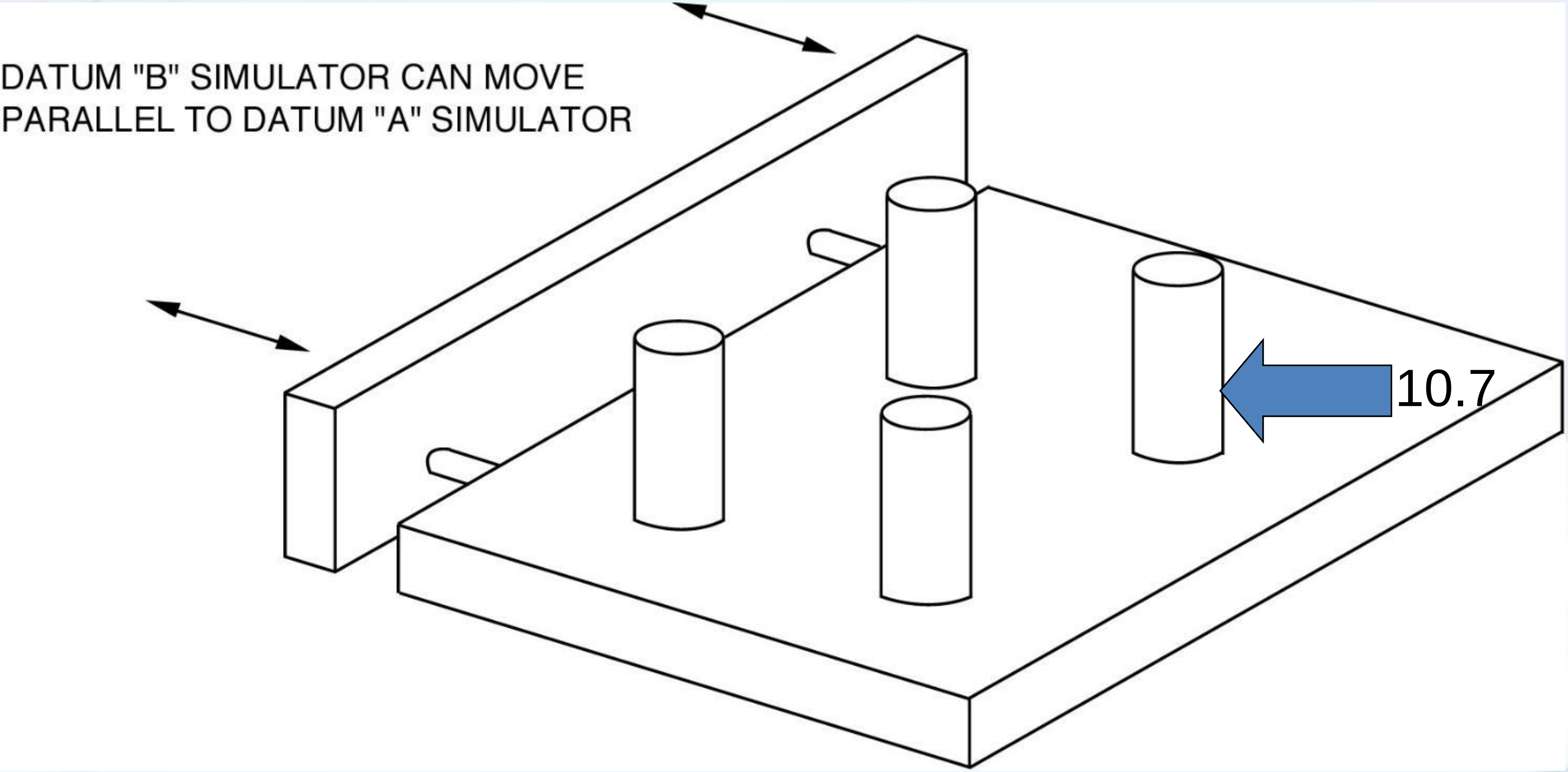
Bottom Frame Controls Orientation to the Datums



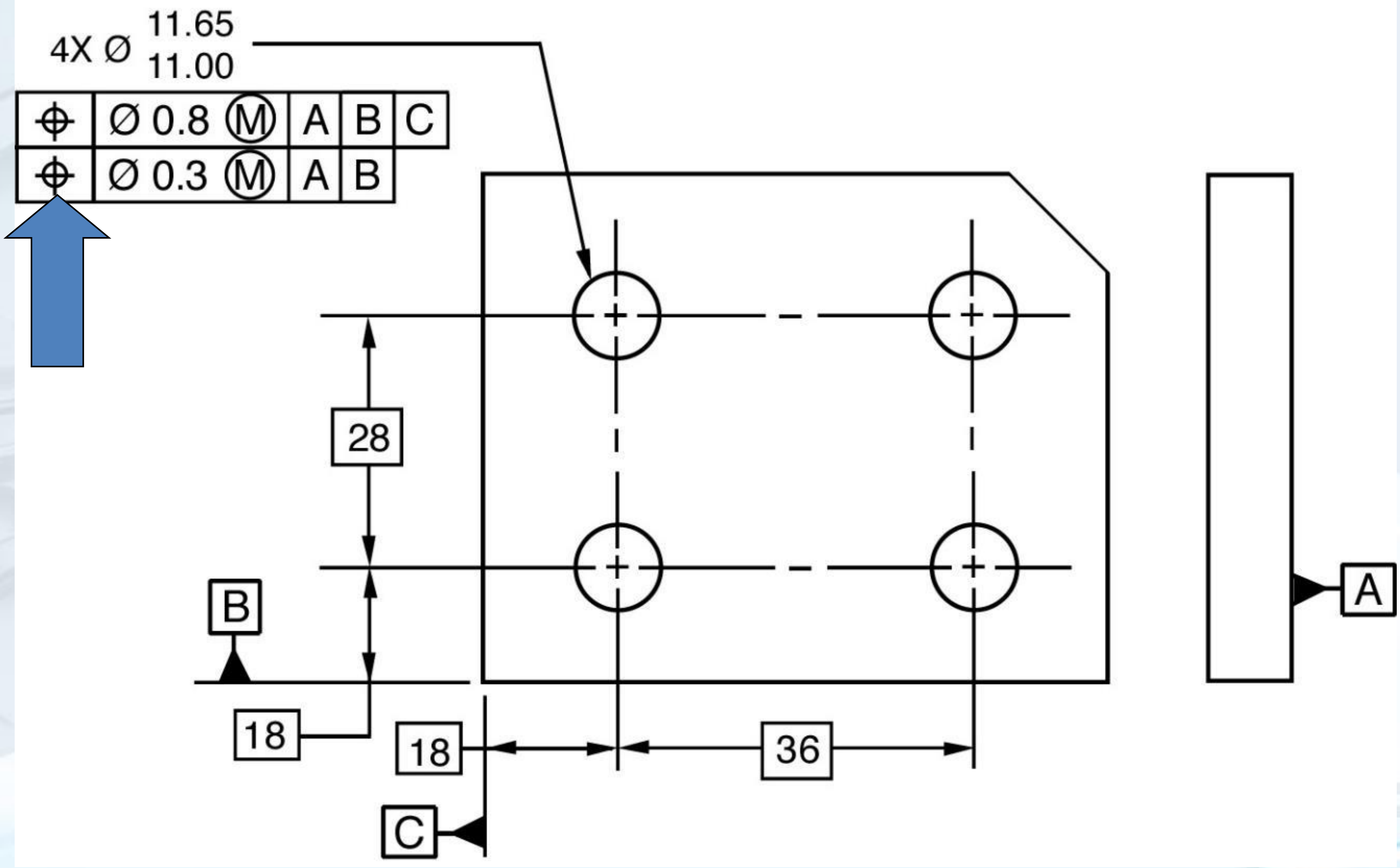
Gage for Upper Frame



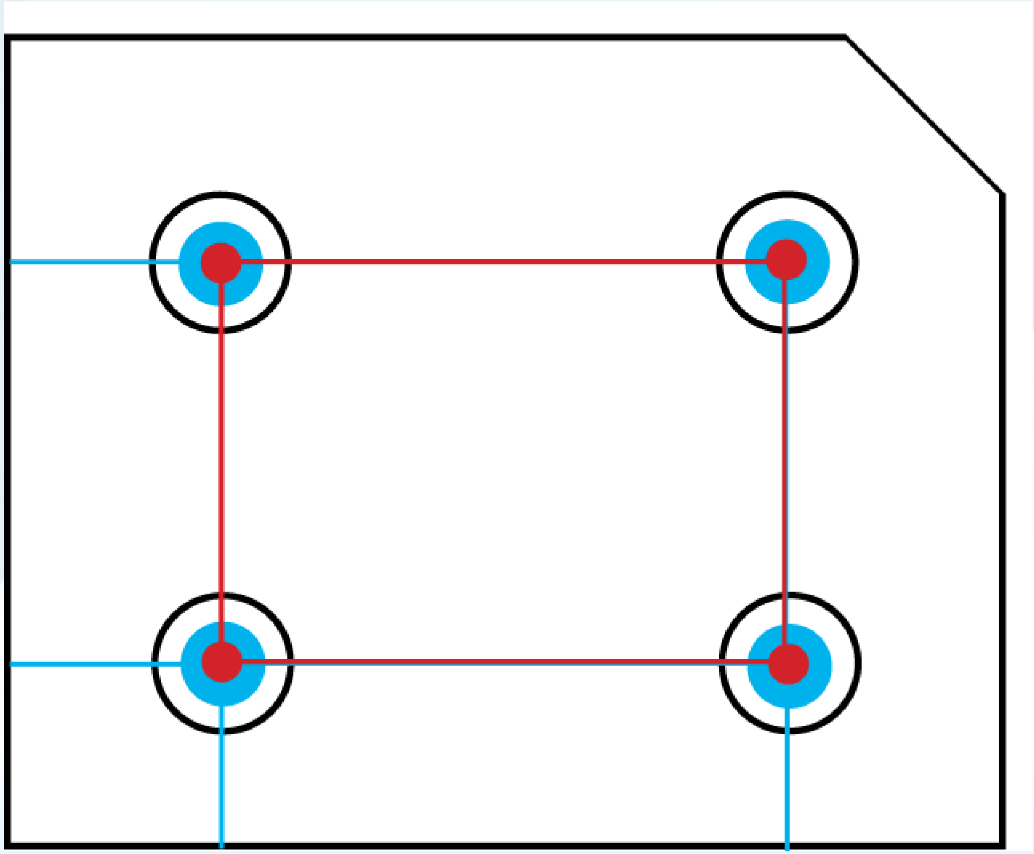
Gage for Lower Frame



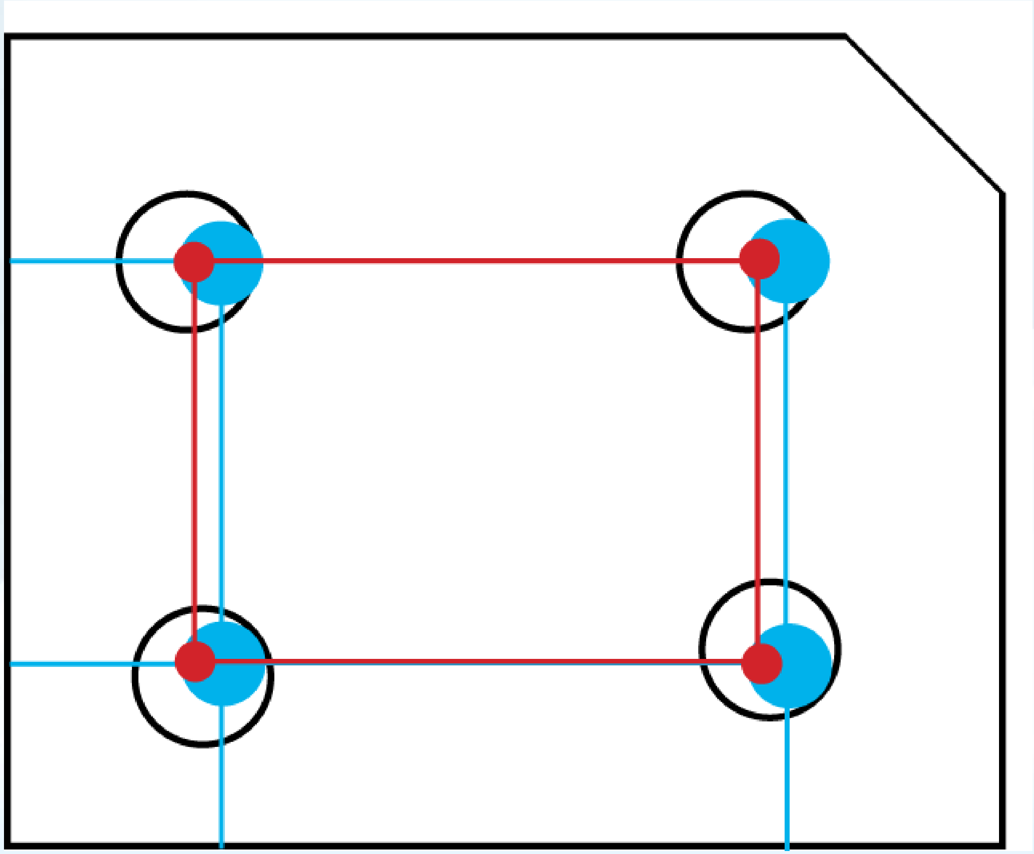
Two Single-Segment



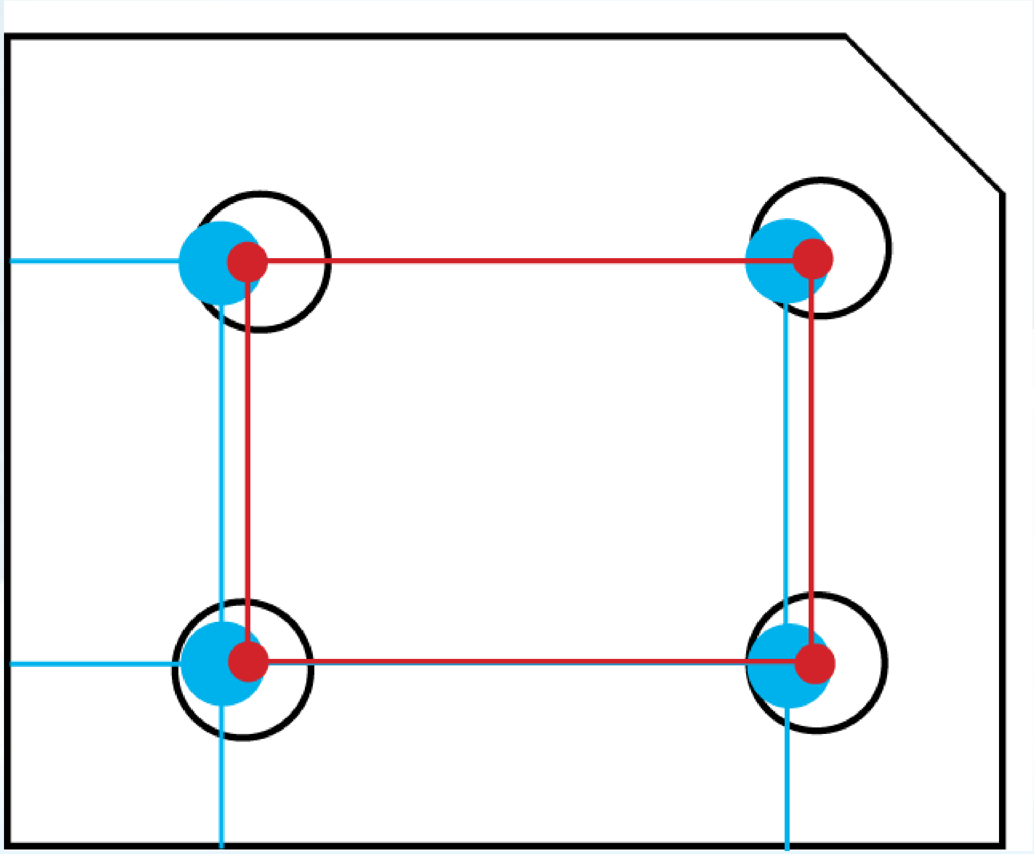
Bottom Frame Controls Orientation and Location to the Datums



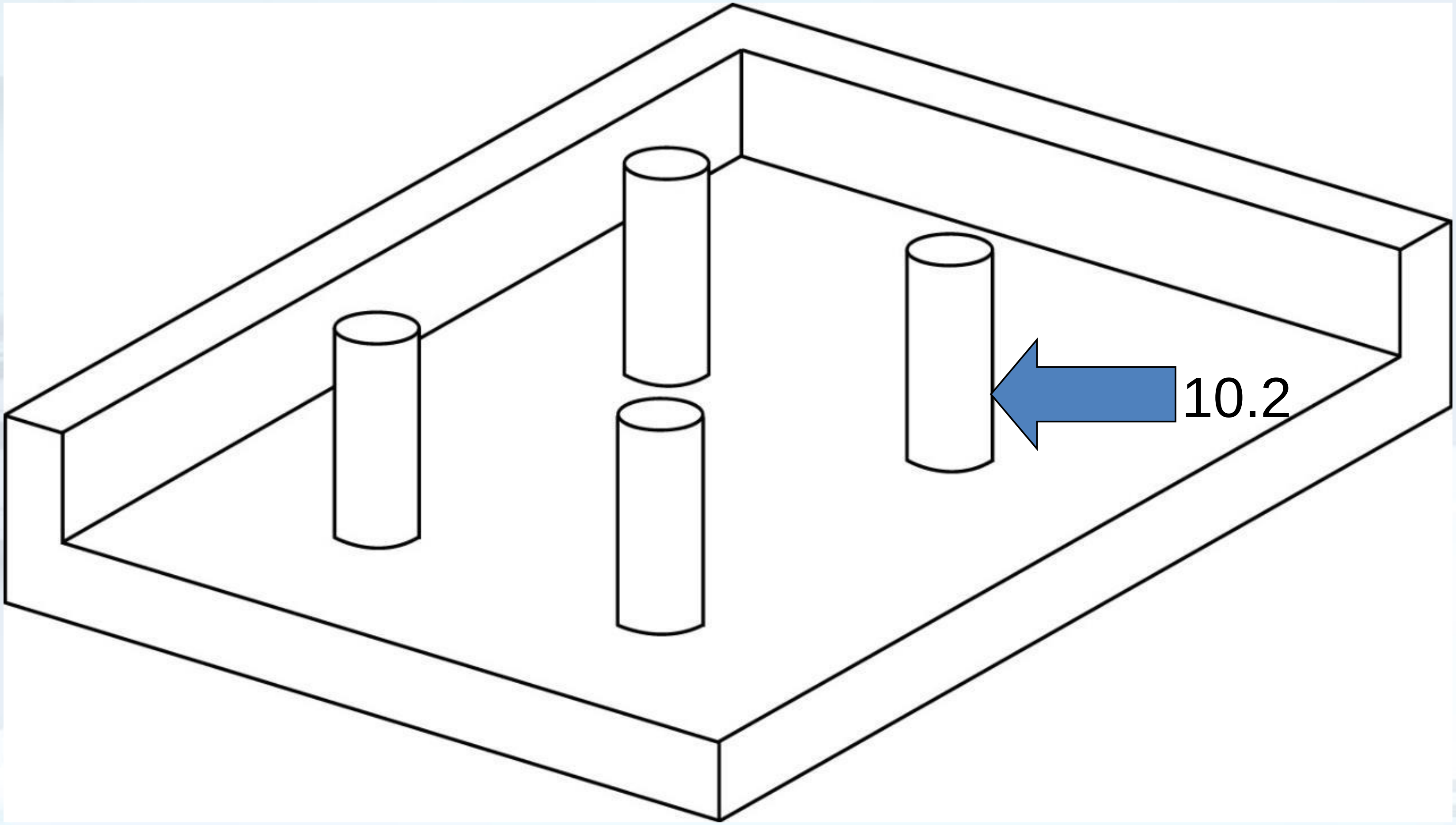
Bottom Frame Controls Orientation and Location to the Datums



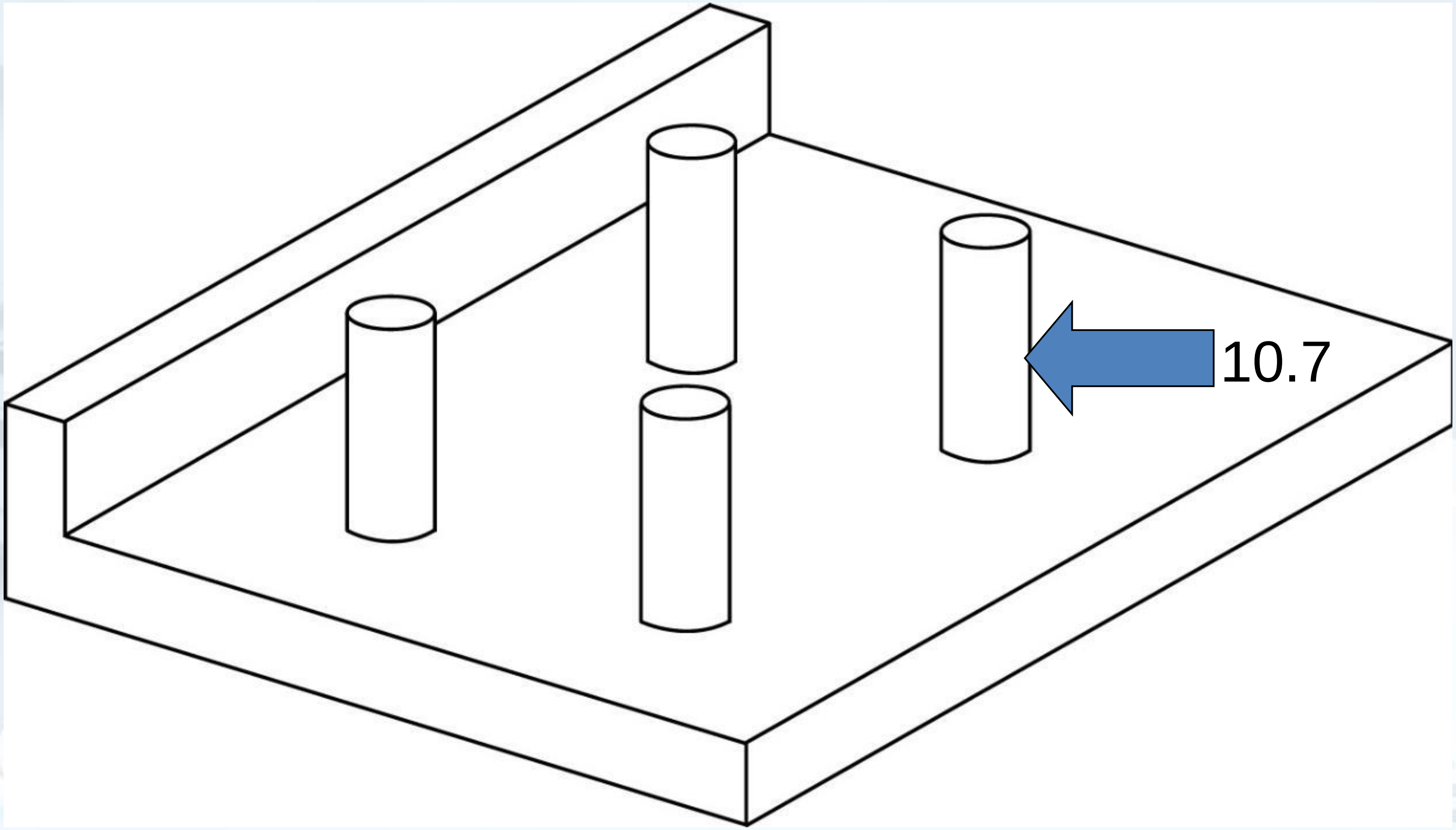
Bottom Frame Controls Orientation and Location to the Datums



Gage for Upper Frame



Gage for Lower Frame



Pop Quiz

- Which of the following callouts is incorrect (without seeing a drawing)?

a.

⌖	Ø 0.8 Ⓜ	A	B	C
	Ø 0.3 Ⓜ	A	B	

b.

⌖	Ø 1.2 Ⓜ	D	E	F
⌖	Ø 0.5 Ⓜ	D	E	F

c.

⌖	Ø .007 Ⓜ	X	Y
	Ø .002 Ⓜ		

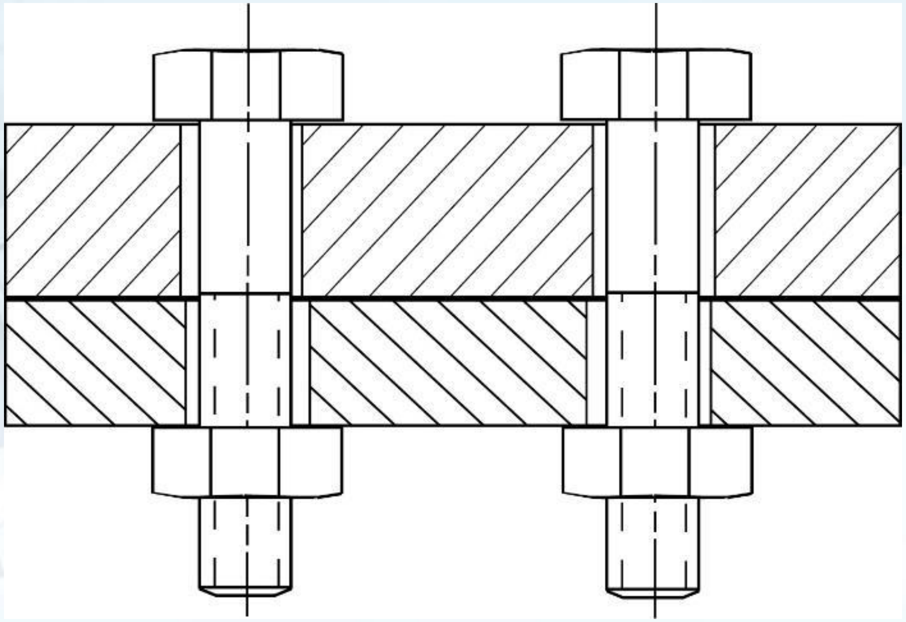
d.

⌖	Ø 1.2 Ⓜ	B	A	C
⌖	Ø 0.5 Ⓜ	F	D	

Floating Fasteners

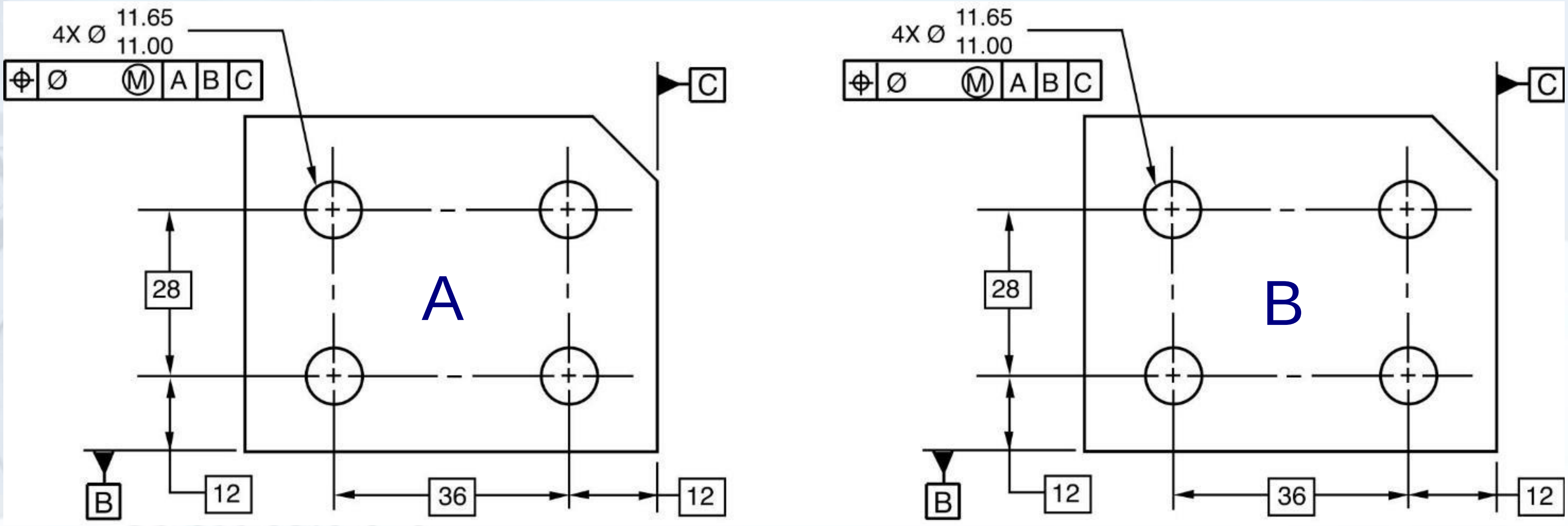
- How to select tolerance value?
- If using floating fasteners:

T = H - F



Floating Fasteners

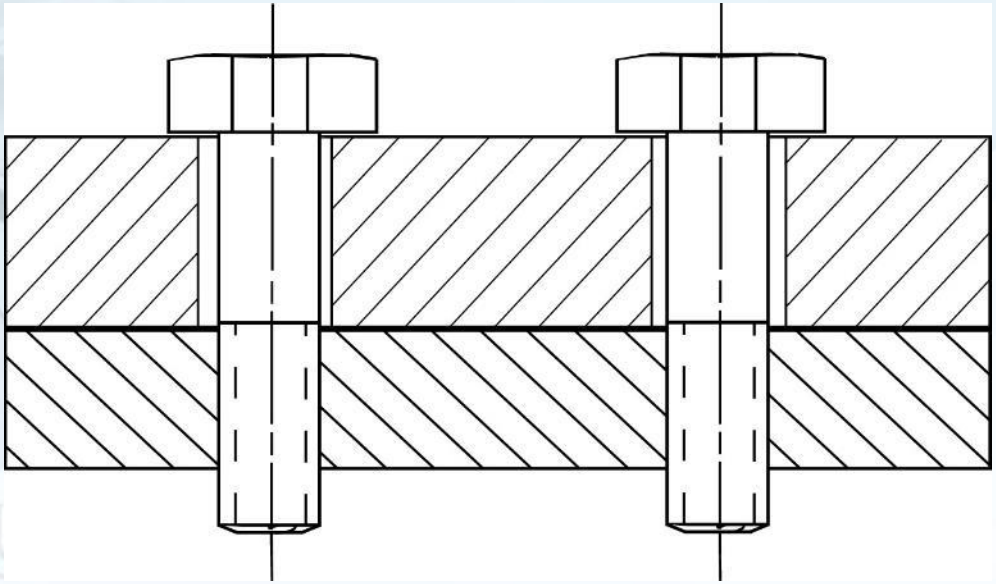
Assume we are using 10 mm bolts:



Fixed Fasteners

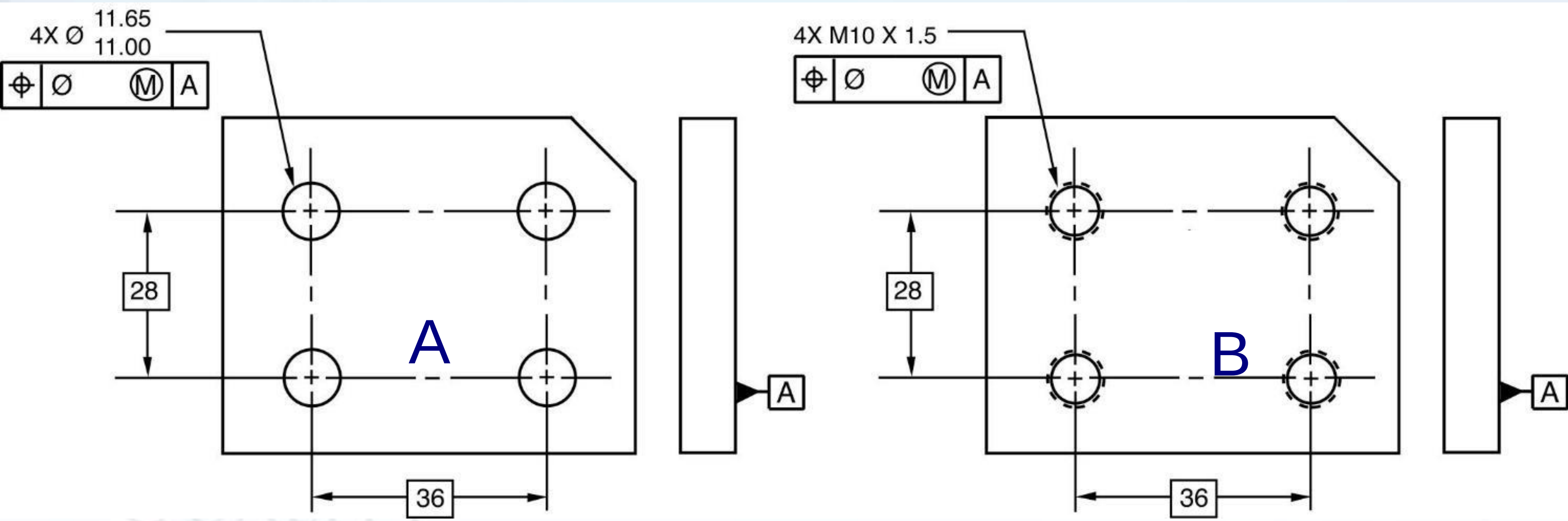
- If using fixed fasteners:

  Tolerance must be shared among parts in the assembly

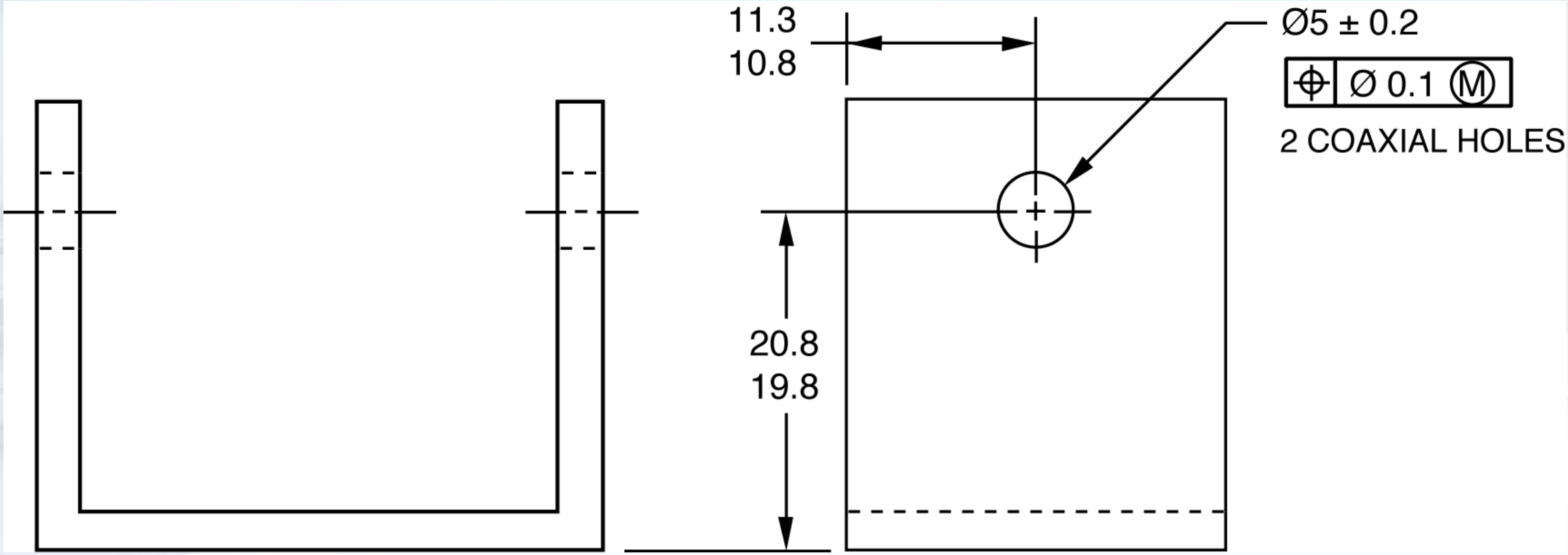


$$T = \frac{H - F}{2}$$

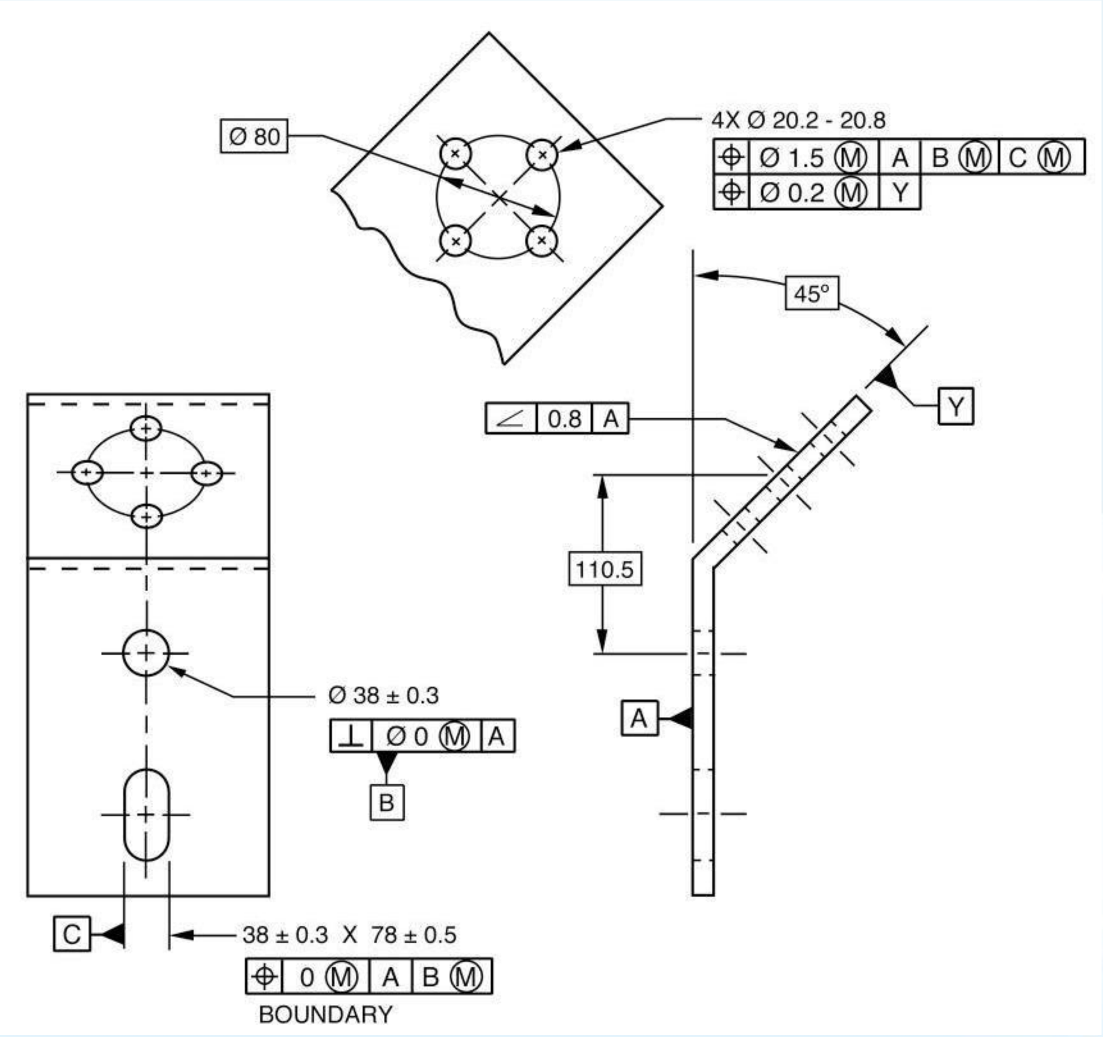
Fixed Fasteners



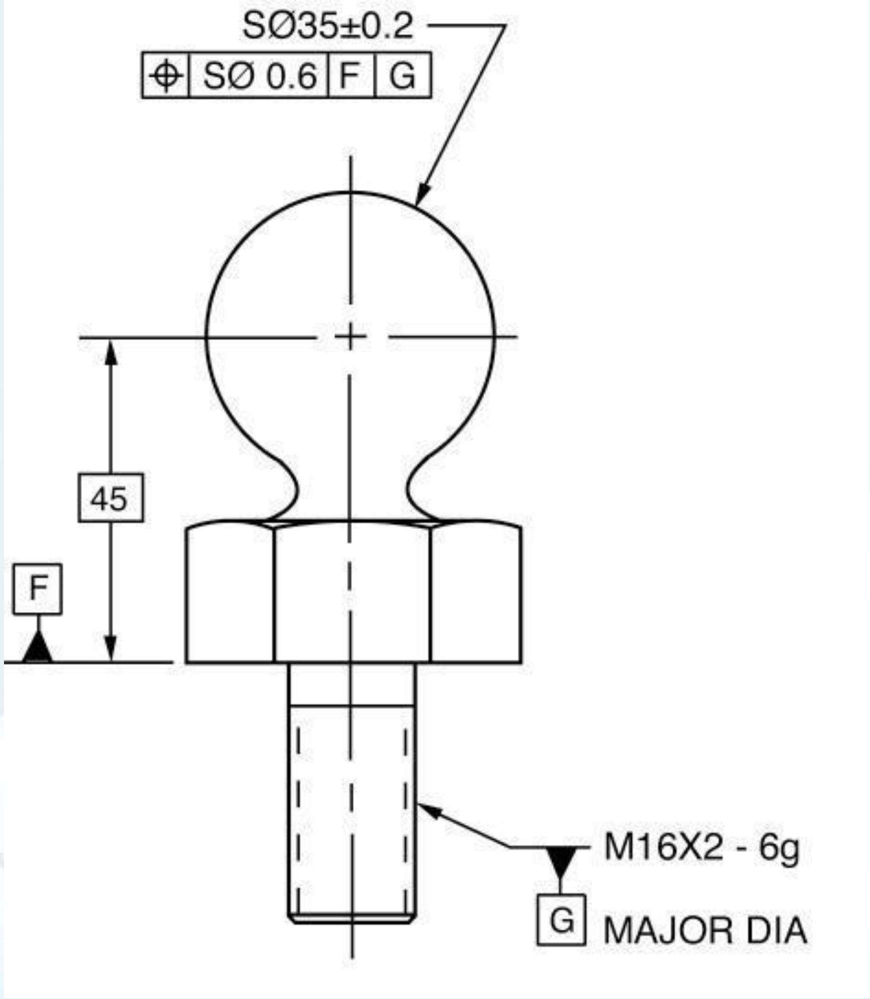
Other Examples



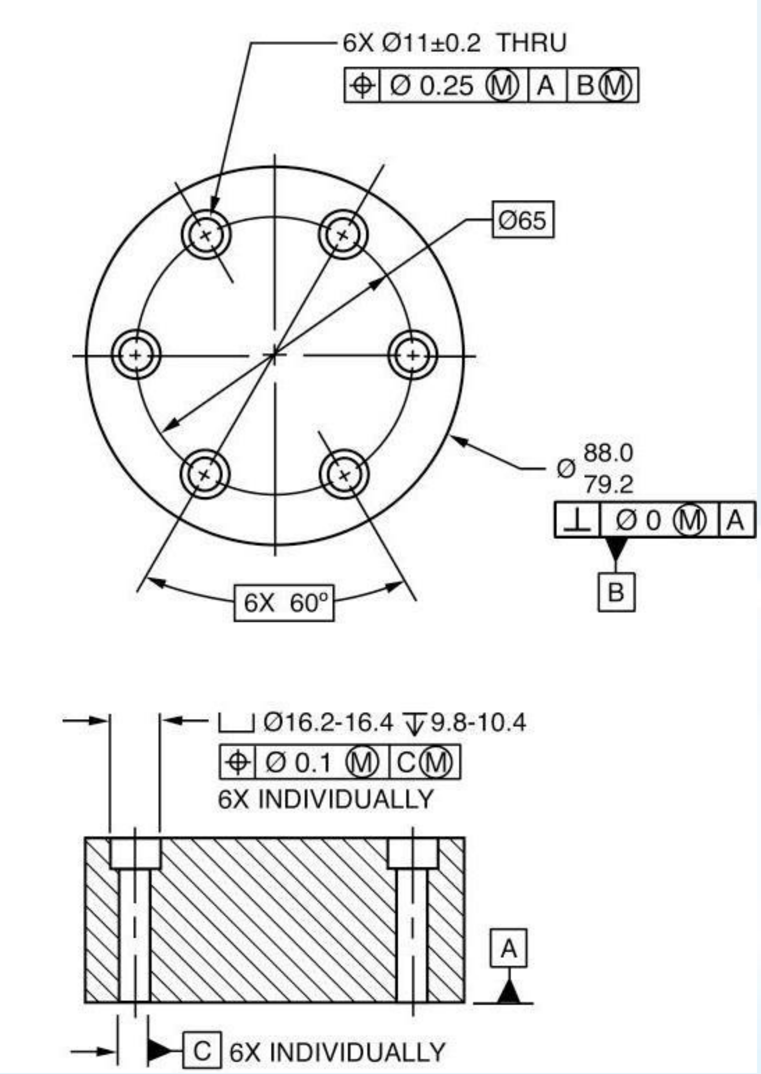
Other Examples



Other Examples



Other Examples



Chapter 7: Position – What We Covered

Learning Objectives

You should now be able to:

- Define true position
- Explain the effect of the MMC and LMC modifiers on a position tolerance
- Explain the pitch diameter rule
- Interpret a zero position tolerance and calculate the maximum position tolerance available
- For a composite position tolerance, explain what qualities are controlled by each number

Chapter Agenda

- Position
- Variation vs. Functional
- Boundary
- Tolerance Zones

Chapter 8

Other Types of Location

Chapter 8: Other Types of Location – What We Will Cover

Learning Objectives

At the end of this chapter, you will be able to:

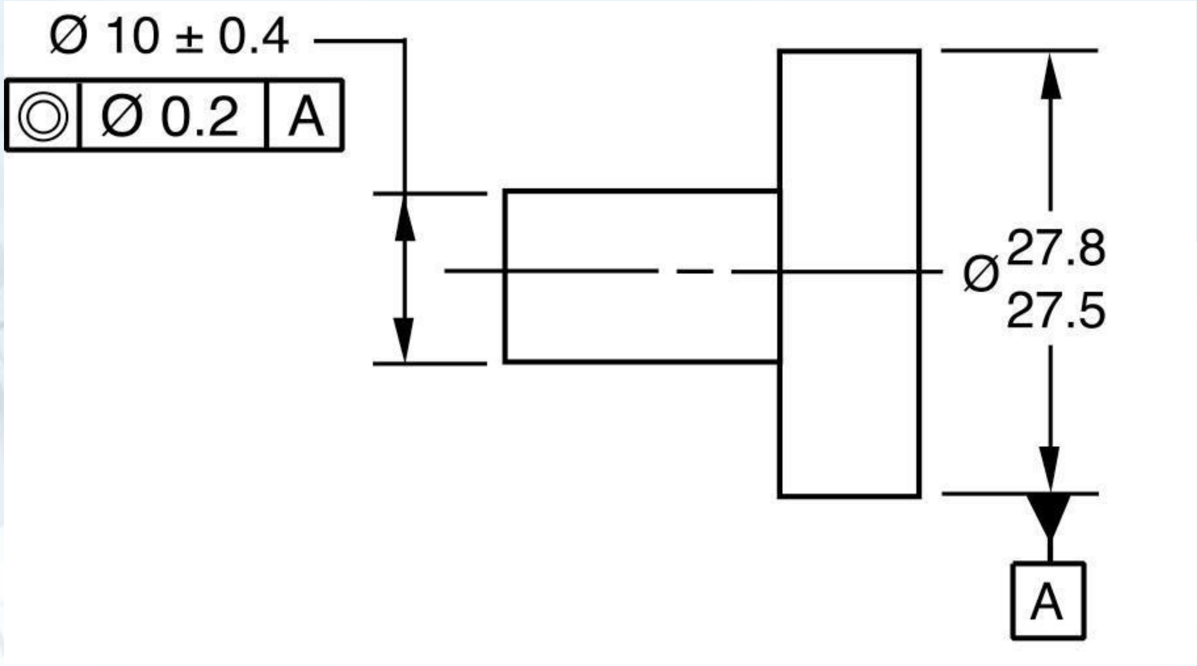
- Identify proper uses of concentricity, symmetry, circular runout, and total runout
- Explain why concentricity is difficult to measure
- Determine which characteristics are being controlled by circular runout vs. total runout
- Explain the free state rule, and when the free state modifier may be needed
- List three of the four parameters typically needed to restrain a part

Chapter Agenda

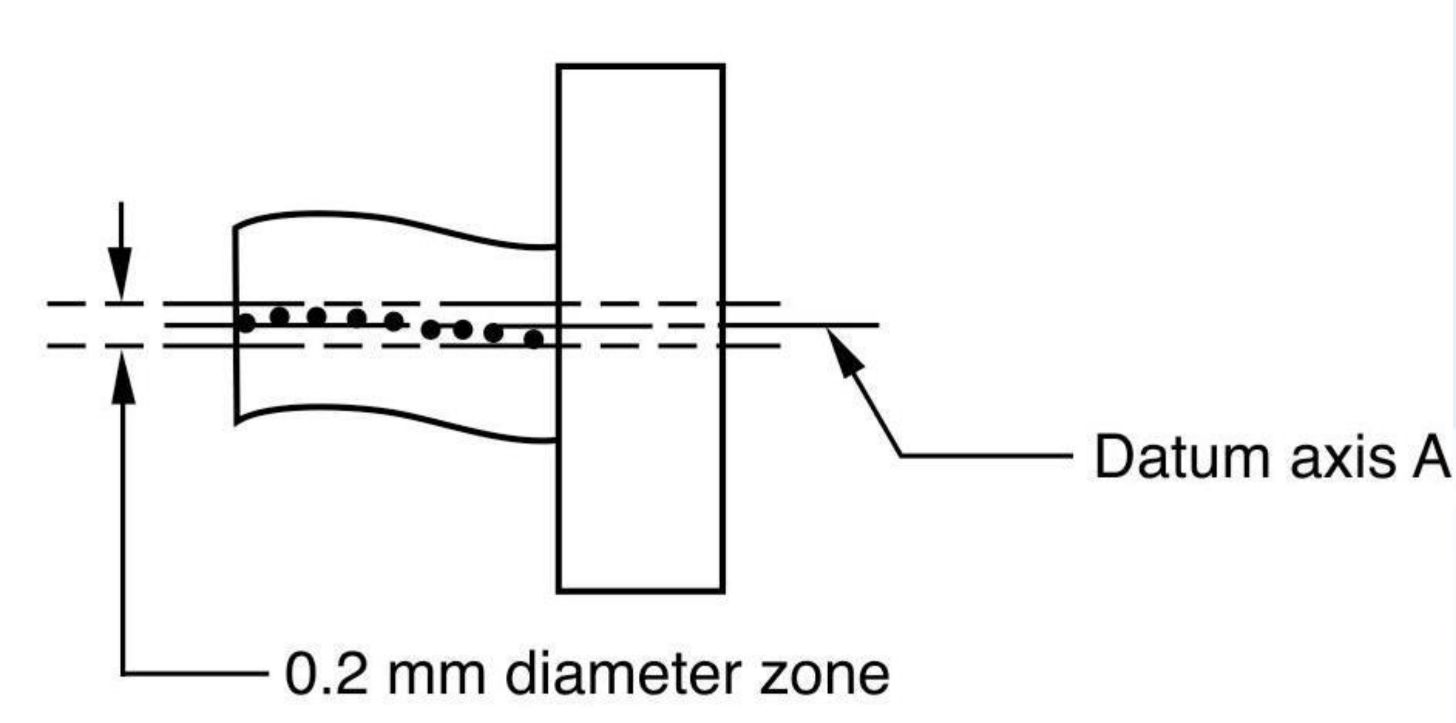
- Concentricity
- Symmetry
- Circular Runout
- Total Runout
- Free State Modifier

Concentricity

- **Definition:** The *median points* of all diametrically opposed elements must fall within the given tolerance zone.



Concentricity

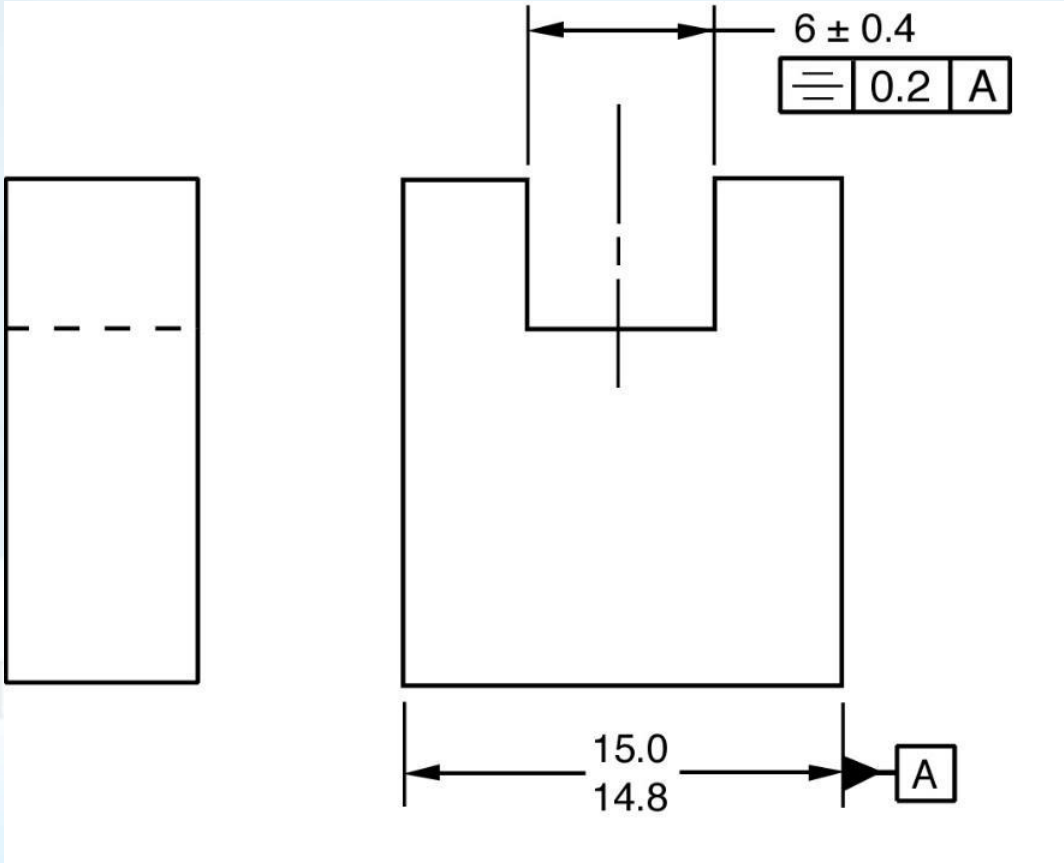


Things to Remember

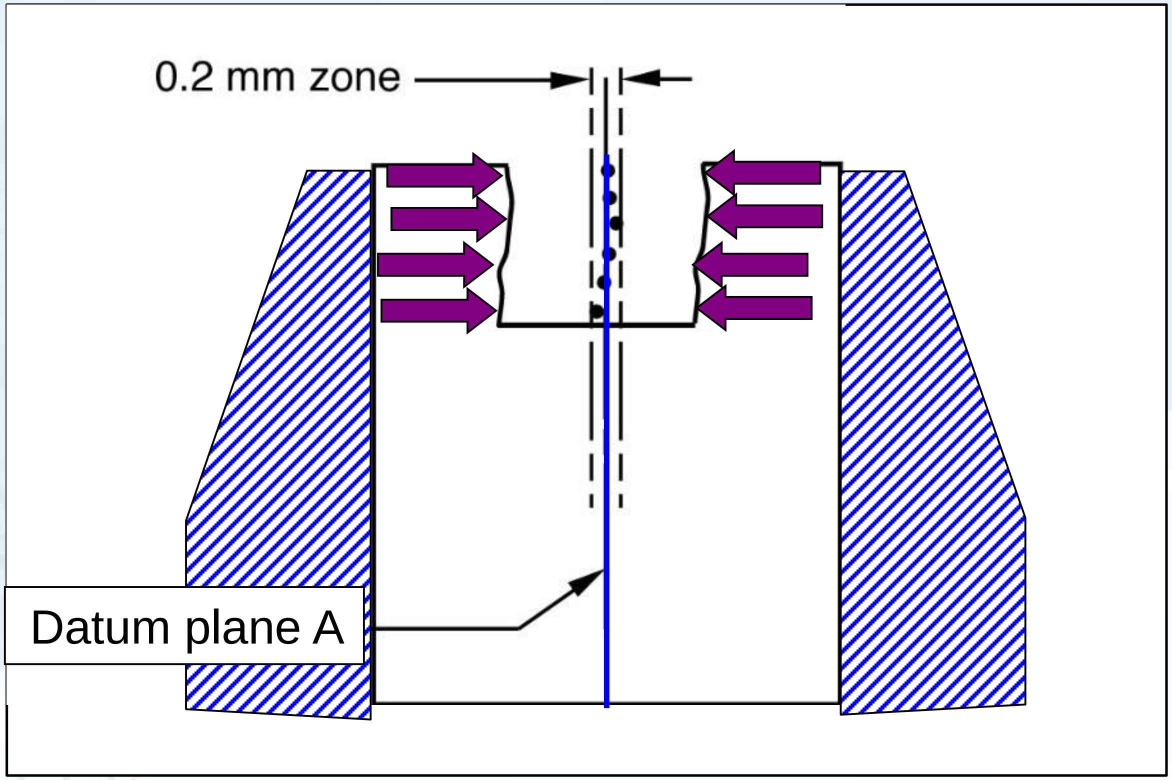
- It must reference a datum, which is diametrical
- Must be applied to diameters coaxial to datum
- Must show diameter symbol
- No modifiers are allowed other than diameter
- It is difficult to measure, and usually not the intended tolerance for the function

Symmetry

- **Definition:** The *median points* of all opposing elements of two surfaces must fall within the given tolerance zone.



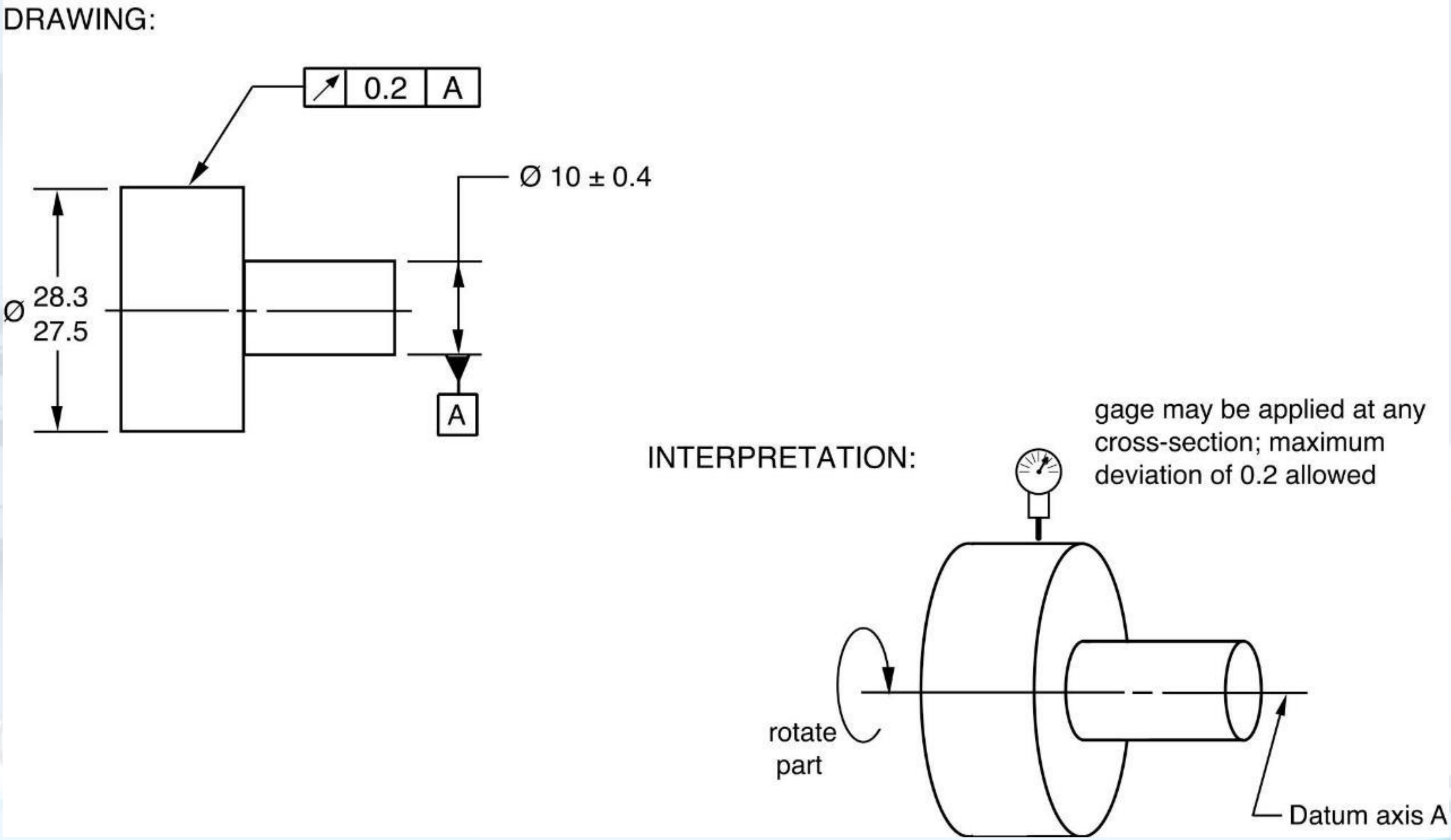
Symmetry



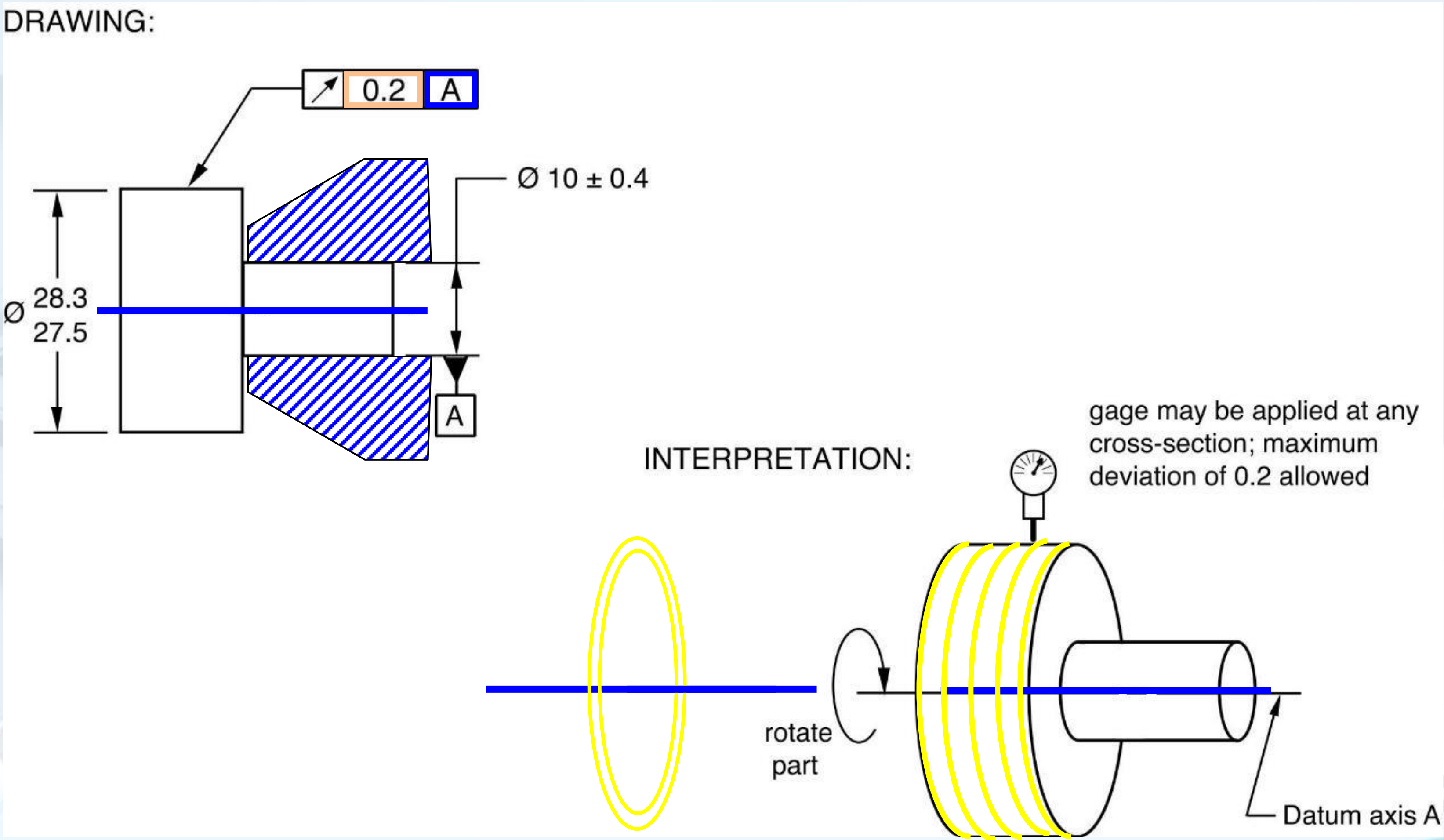
Things to Remember

- It must reference a datum which is a center plane
- For planar features to be centered on the datum
- No modifiers are allowed
- It is difficult to measure, and usually not the intended tolerance for the function

Circular Runout



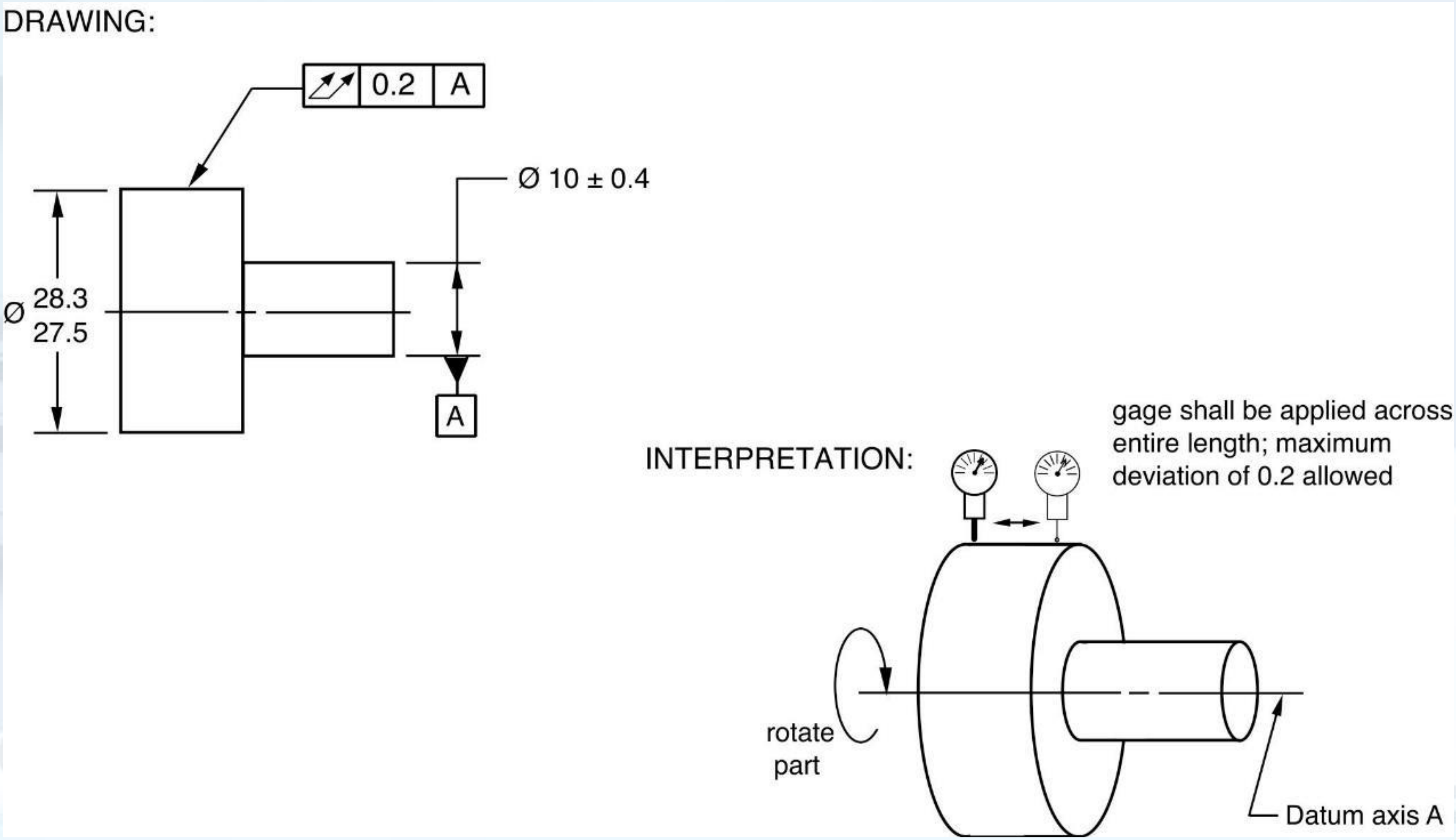
Circular Runout



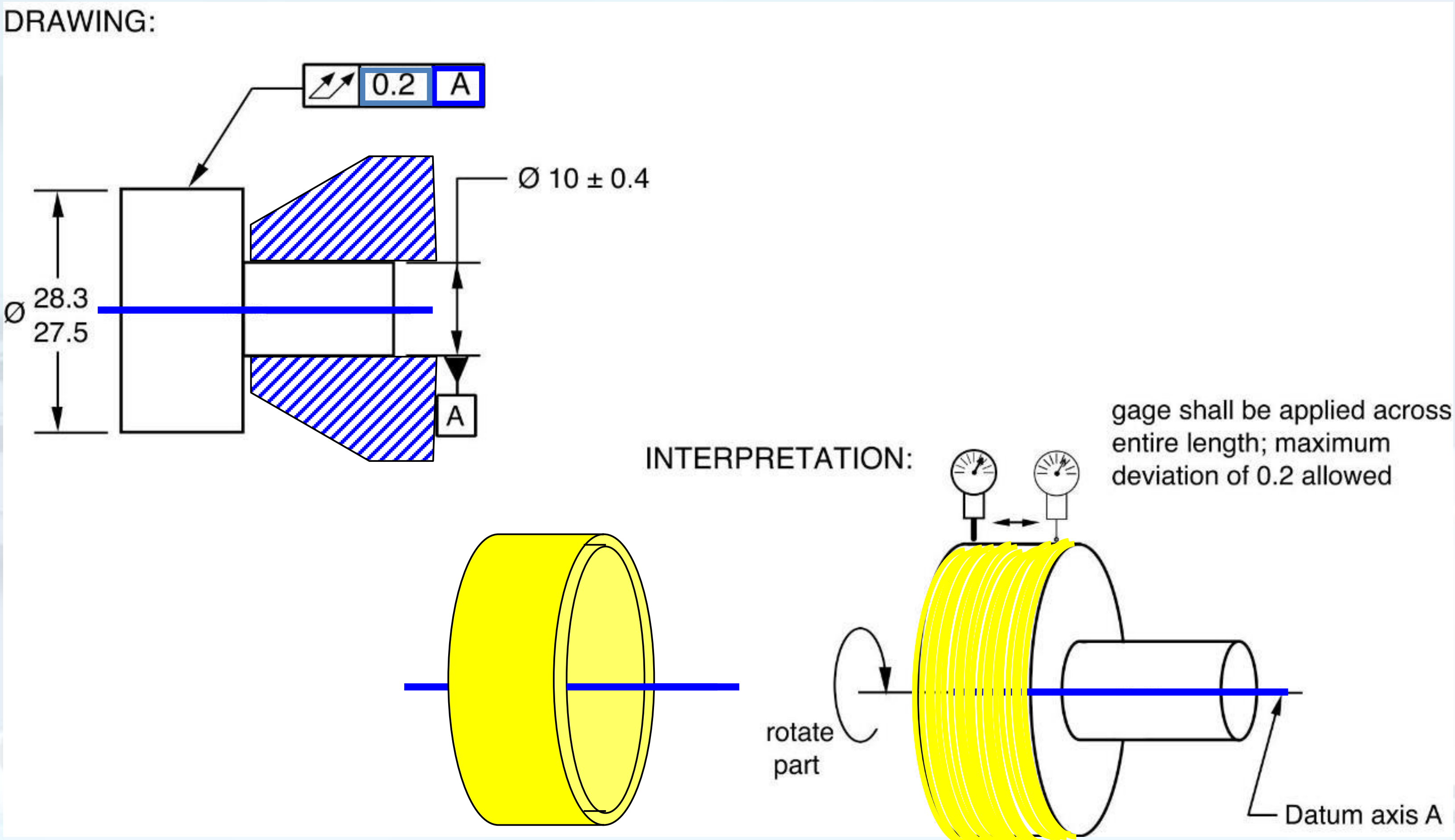
Things to Remember

- Sometimes called TIR or FIM
- It must reference a datum that is an axis (RMB)
- For surfaces around or 90° to the datum
- Does not control straightness/waviness
- No modifiers are allowed
- It can control location, orientation, and form
- The feature control frame can be placed under the size dimension or pointing to the surface

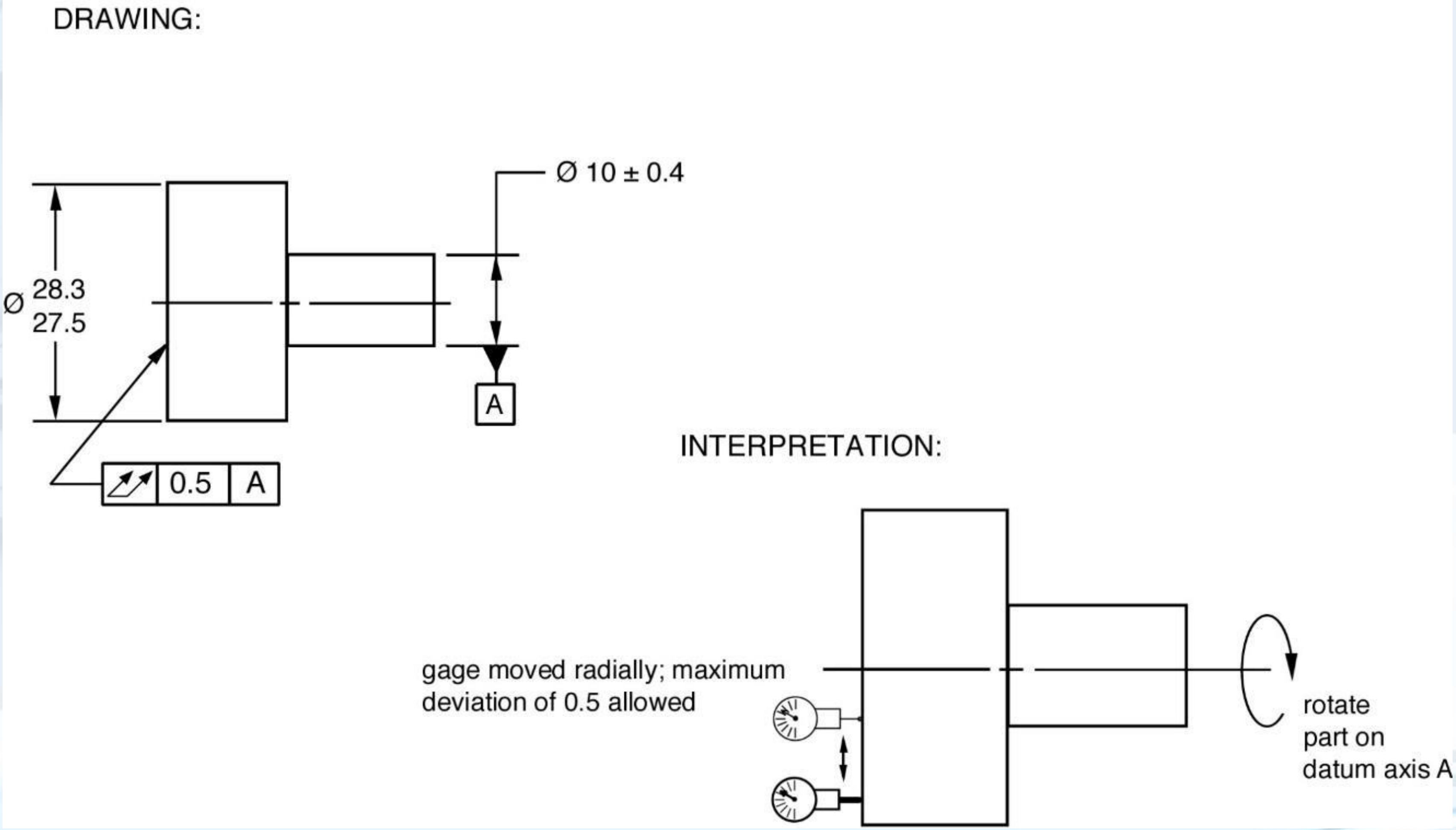
Total Runout



Total Runout



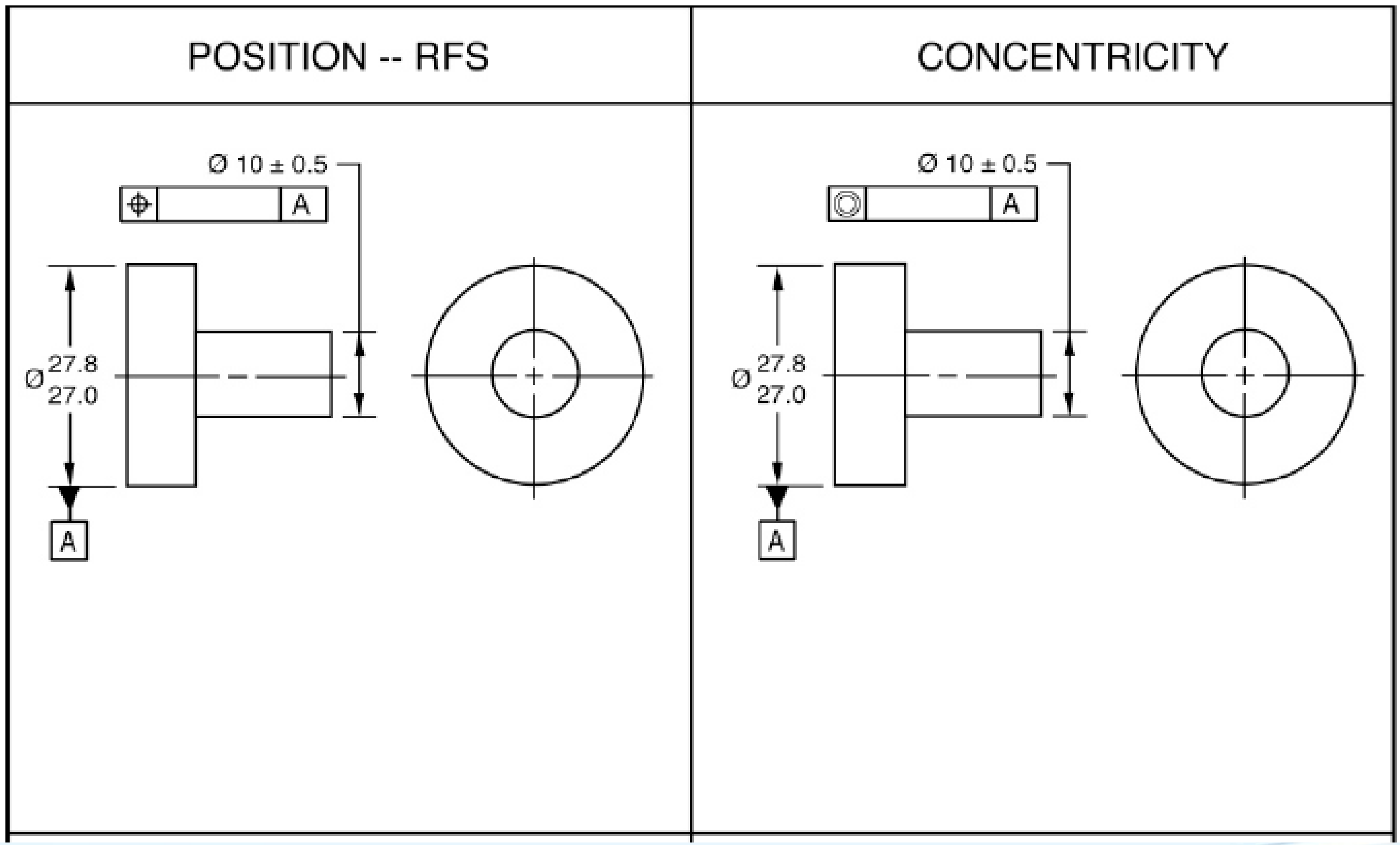
Runout for a Flat Surface



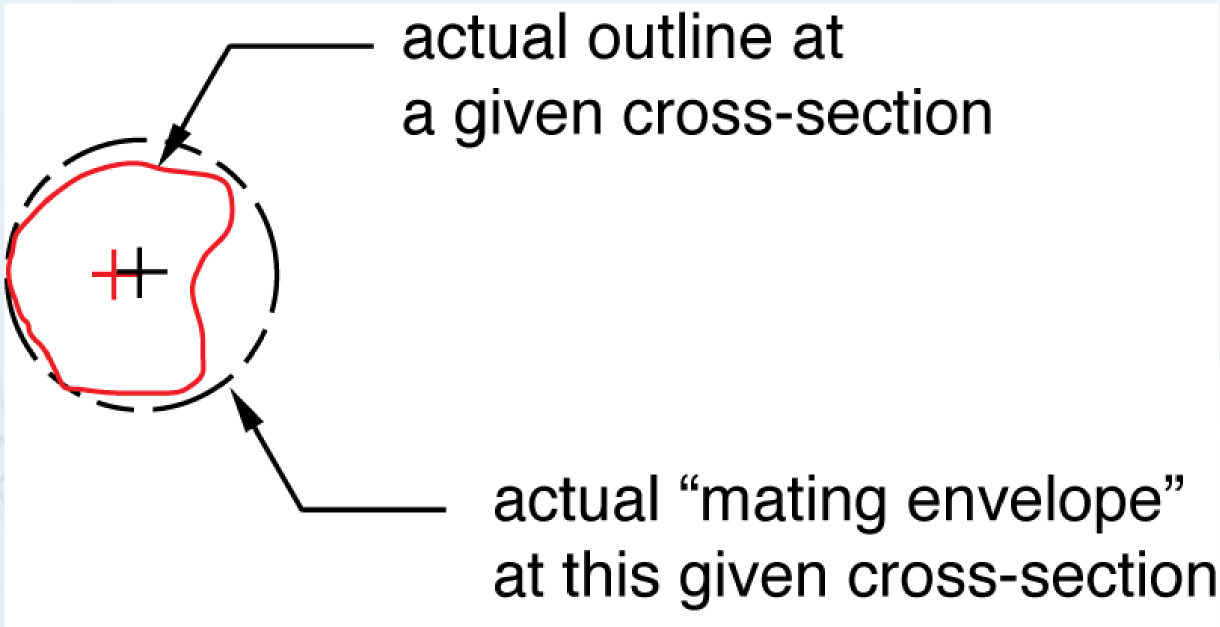
Things to Remember

- It must reference a datum that is an axis (RMB)
- For surfaces around or intersecting the datum
- It does control straightness/waviness
- No modifiers are allowed
- It can control location, orientation, and form
- The feature control frame can be placed under the size dimension or pointing to the surface

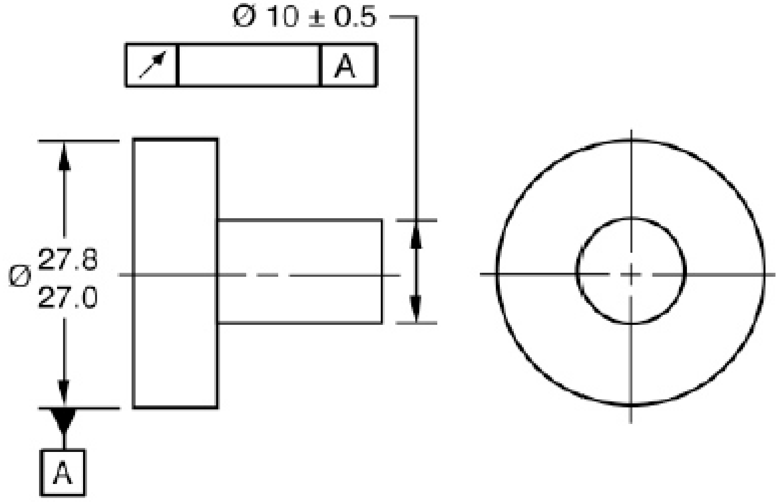
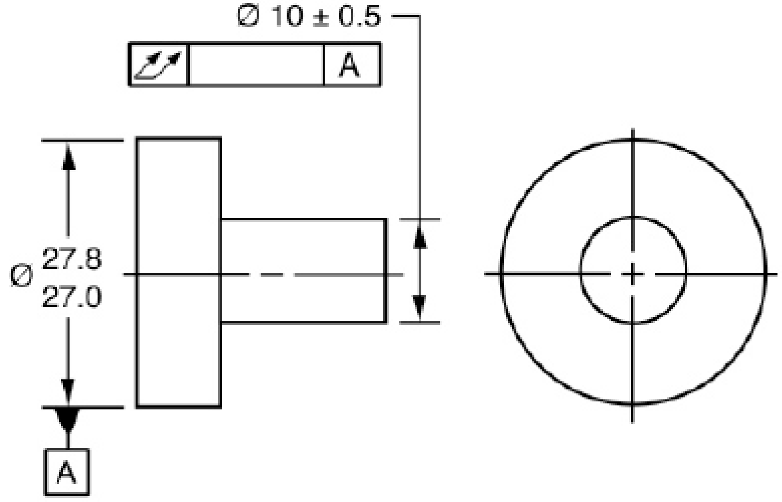
“Coaxial” Comparison



“Coaxial” Comparison



“Coaxial” Comparison

CIRCULAR RUNOUT	TOTAL RUNOUT
	

Free State

Check a part in its free state or restrained state?

Unless otherwise specified, we are to check parts in the ___________________________ state.

Restraining a Part

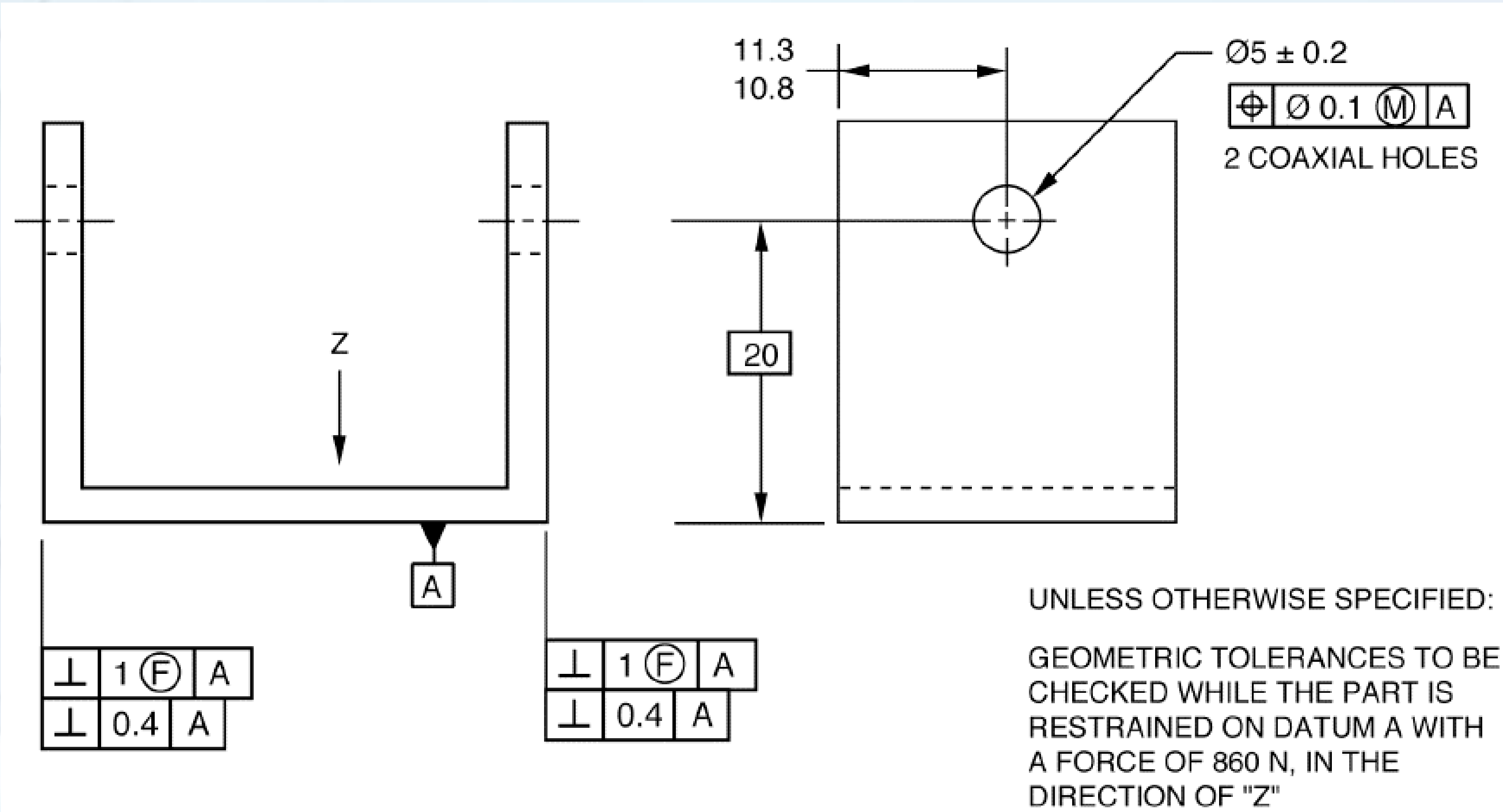
If a part needs to be restrained, then often a general note is given. Four things a general restraint note should specify:

1. ________________
2. ________________
3. ________________
4. ________________

Free State Modifier

Why do we need a *free state only* modifier symbol when free state is the default condition?

When to Use Ⓕ


11.3
10.8
Ø5 ± 0.2
⌖ | Ø 0.1 Ⓜ | A
2 COAXIAL HOLES
Z
20
A
⊥ | 1 Ⓕ | A
⊥ | 0.4 | A
⊥ | 1 Ⓕ | A
⊥ | 0.4 | A
UNLESS OTHERWISE SPECIFIED:
GEOMETRIC TOLERANCES TO BE CHECKED WHILE THE PART IS RESTRAINED ON DATUM A WITH A FORCE OF 860 N, IN THE DIRECTION OF "Z"

Dimensional Management

How do we often come up with tolerances?

Copy from an older drawing or similar part

Use standard textbooks or industry tables

Guess

Dimensional Management

How *should* you come up with tolerances?

TOL. BASED ON:	Product's function	The manufacturing process
Advantages	Provides flexibility for processing. Tolerances are dependent variables. The product will function as intended. The effects of a change request can be easily evaluated.	The part can be manufactured. Some stack calculations are eliminated.
Disadvantages	May create extra tolerance stacks. Does not give specific instructions to manufacturing. The process that is used may not be capable.	The product may not function correctly. Harder to evaluate change requests. Often yields tighter tolerances. The focus is removed from functionality.

Applying GD&T

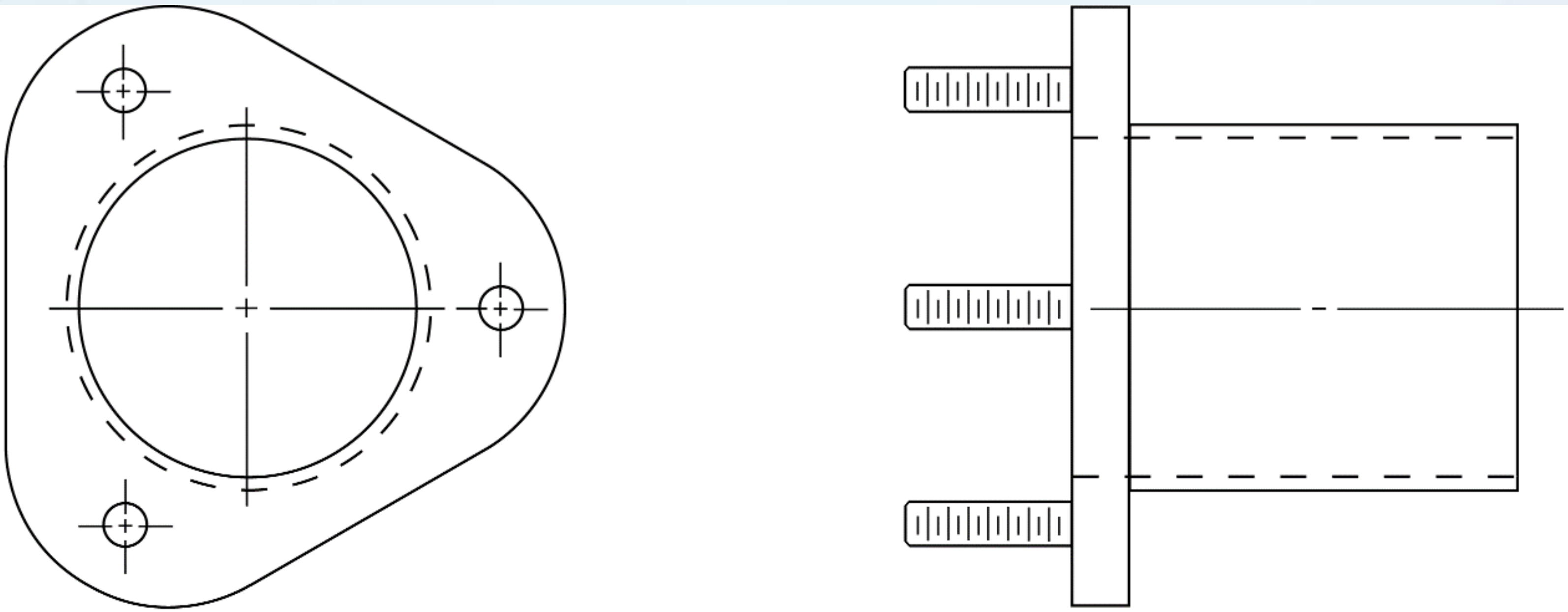
Step 1 – Identify the datum features

Step 2 – Select appropriate geometric symbols

Step 3 – Select tolerance values; do stacks

Step 4 – Get final sign-off from everyone

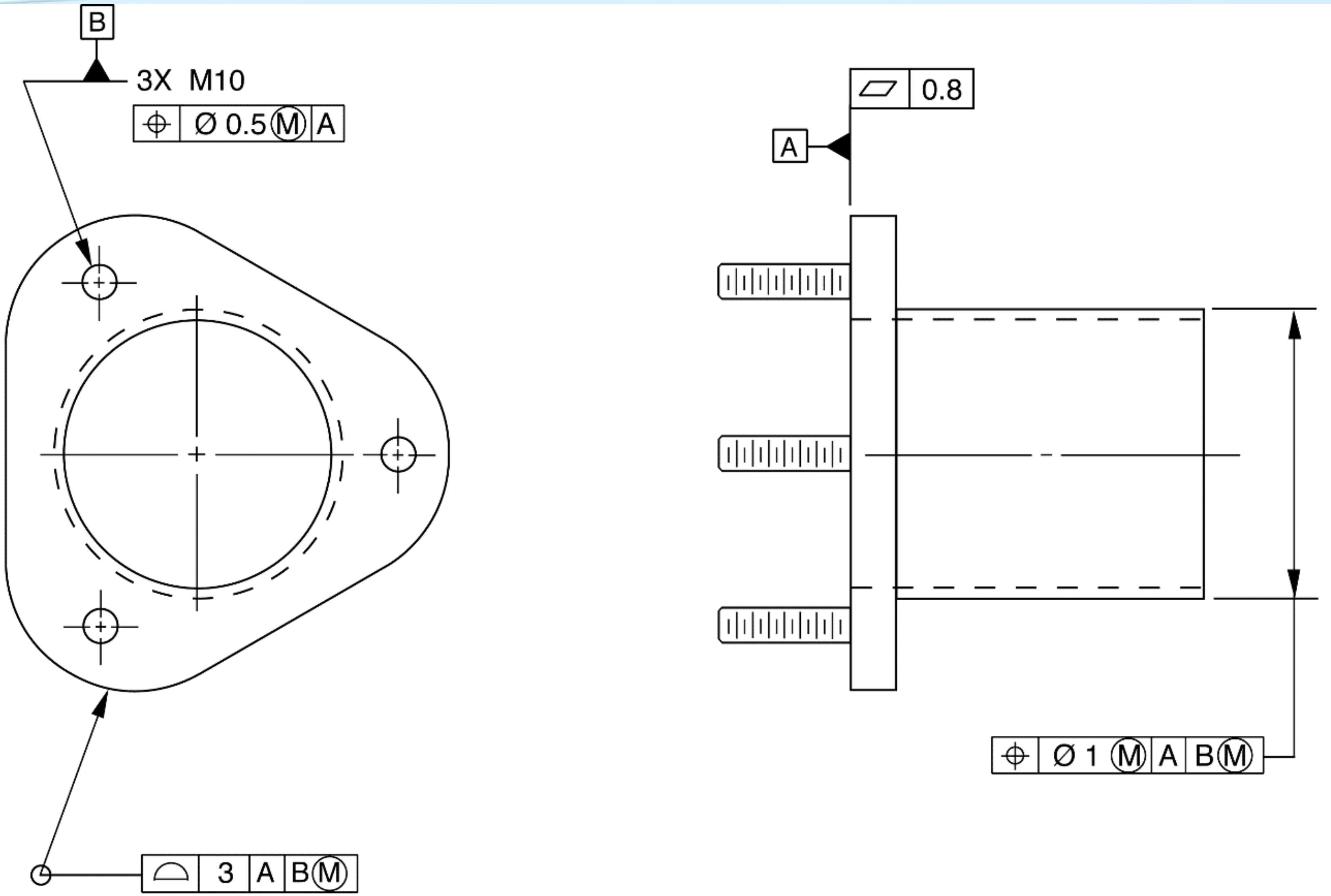
Applying GD&T



DIMENSIONS AND TOLERANCES
PER ASME Y14.5-2009

METRIC	ABC Manufacturing Co.
Third Angle Projection	EXHAUST FLANGE
	Part no. 64135-10

Applying GD&T



METRIC	ABC Manufacturing Co.
Third Angle Projection	EXHAUST FLANGE
	Part no. 64135-10

DIMENSIONS AND TOLERANCES
PER ASME Y14.5-2009

A Few Extra Thoughts…

GD&T doesn't automatically mean more expense

GD&T should be driven by function

Like any language, there may be more than one way to say something

…More Thoughts

- Don’t be intimidated by GD&T!
- Start with datums – how to select and simulate
- Interpret each feature control frame
- Watch for MMC and LMC symbols
- Stay on top with practice – perhaps get certified

Chapter 8: Other Types of Location – What We Covered

Learning Objectives

You should now be able to:

- Identify proper uses of concentricity, symmetry, circular runout, and total runout
- Explain why concentricity is difficult to measure
- Determine which characteristics are being controlled by circular runout vs. total runout
- Explain the free state rule, and when the free state modifier may be needed
- List three of the four parameters typically needed to restrain a part

Chapter Agenda

- Concentricity
- Symmetry
- Circular Runout
- Total Runout
- Free State Modifier

Thank You!

Questions?

info@omnex.com
734.761.4940