# 2023 Digital Manufacturing Challenge powered by SME's DDM Advisory Team

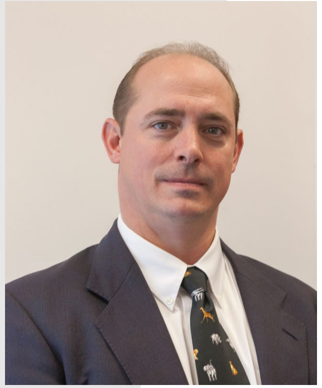
***The webinar titled***

**Fundamentals of Geometric Dimensioning and Tolerancing, Part II**

***will begin shortly***

# Fundamentals of Geometric Dimensioning and Tolerancing (GD&T)

## -Part II-

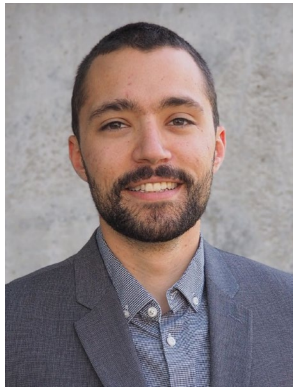


Host: Carl Dekker

President
Met-L-Flo and Chair of the Direct Digital Manufacturing Advisory Team

Moderator: Jason Fox

Mechanical Engineer
National Institute of Standards and Technology (NIST)



Speaker: Jaime Berez

Ph.D. Candidate, Instructor
Georgia Institute of Technology

Speaker: Maxwell Praniewicz

Mechanical Engineer
National Institute of Standards and Technology (NIST)

# 2023 Digital Manufacturing Challenge powered by SME's DDM Advisory Team

## Theme: AM to the Rescue: Digital Manufacturing Agility to Address Crises

## Deadline: February 27, 2023 (11:59 PM)

**NEW THIS YEAR:** High school and undergraduate students are highly encouraged to prepare a submission! Tiers have been added to separate High School, Undergraduate, and Graduate student submissions and a winner from each tier will be identified. Updated Submission Requirements - Geometric Dimensioning and Tolerancing included in Requirements (university students)

https://www.sme.org/aboutsme/awards/digital-manufacturing-challenge/

# Fundamentals of GD&T Seminar series overview

Georgia Tech

## Part 1

*February 3rd, 2023*

Speakers

Jaime Berez
Georgia Institute of Technology

Topics

- Introduction to imprecision in manufacturing
- Tolerancing systems (ASME Y14.5, etc.)
- Datums, form, orientation, location, and size
- The 'symbolic language' of GD&T– feature control frames & more

## Part 2

*February 17th, 2023*

Speakers

Jaime Berez
*Georgia Institute of Technology*

Maxwell Praniewicz
*National Institute of Standards and Technology*

Topics

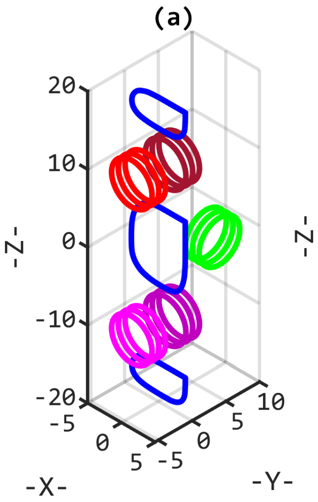
- Follow-ups from Part I
- Inspection
- Designer checklist for implementing GD&T
- Example implementation
- Case studies! (Focus on digital manufacturing)

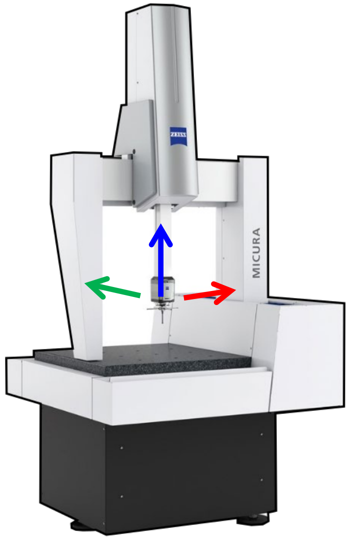
# Introductions

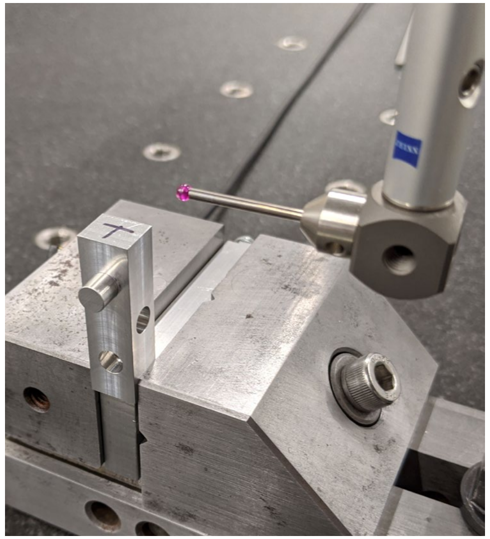
Georgia Tech

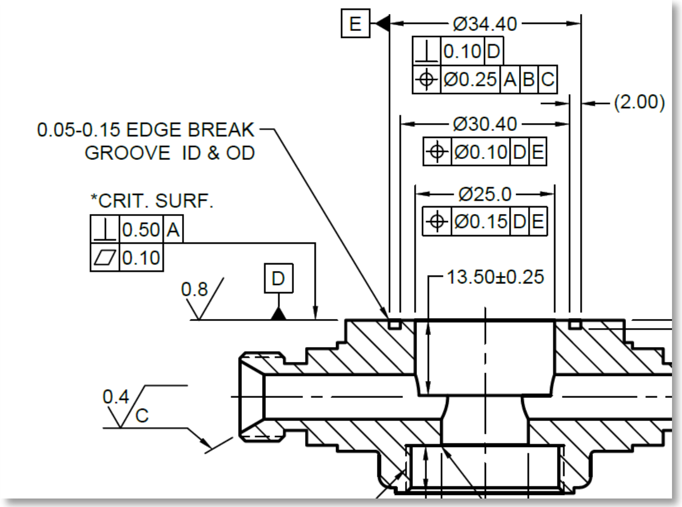

Jaime Berez
j.berez@gatech.edu


- Ph.D. Candidate, Georgia Institute of Technology
  - **Instructor**, ME 3210, Design Materials, and Manufacture
  - Research: Fatigue, manufacturing process monitoring, metal AM, **dimensional metrology**, NDE
- B.S. Mechanical Eng., University of Maryland, College Park
- Prior experience: Aerospace, automotive





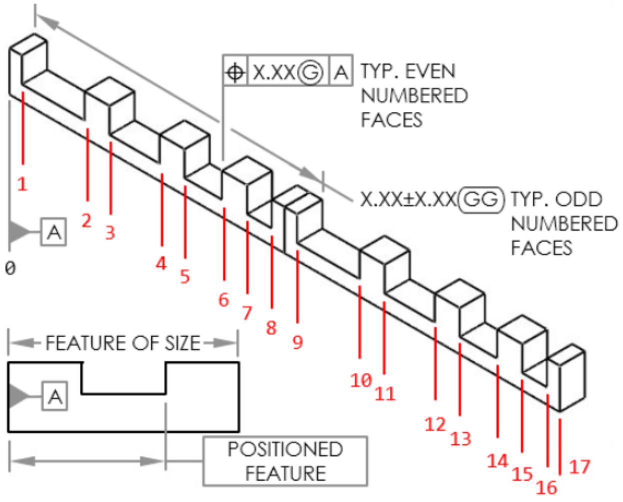
# Introductions

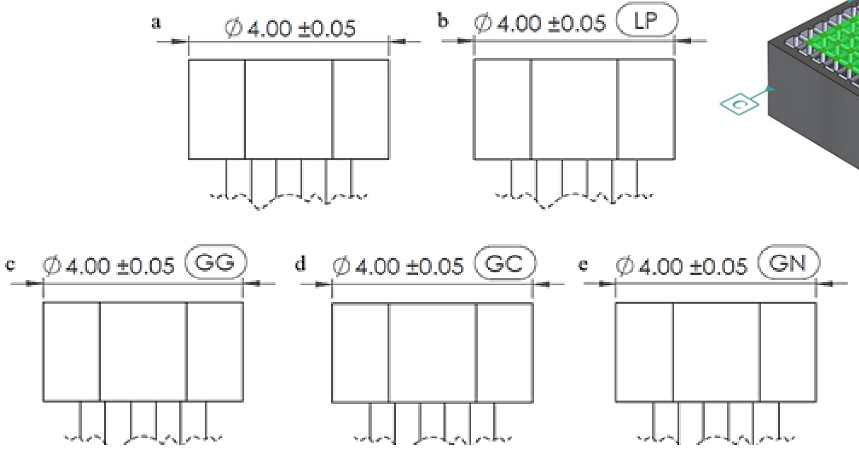
Georgia Tech

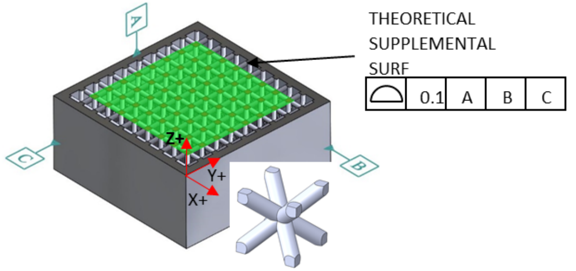
Maxwell Praniewicz*
maxwell.praniewicz@nist.gov

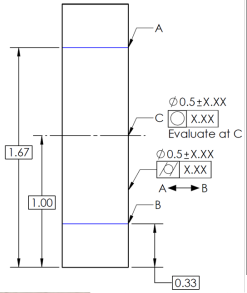
- Mechanical Engineer, National Institute of Standards and Technology
  - AM Component Qualification, NIST Measurement Science for Additive Manufacturing Program
  - Coordinate metrology, machine tool metrology, dimensional metrology on AM components
- Ph.D. & M.S. Mech. Eng., Georgia Institute of Technology, (2020, 2018)
- B.S. Mech. Eng., University of Pittsburgh, 2016

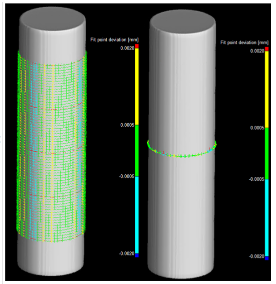


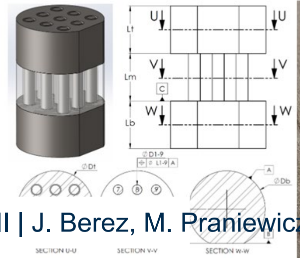


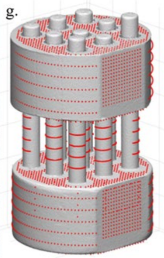







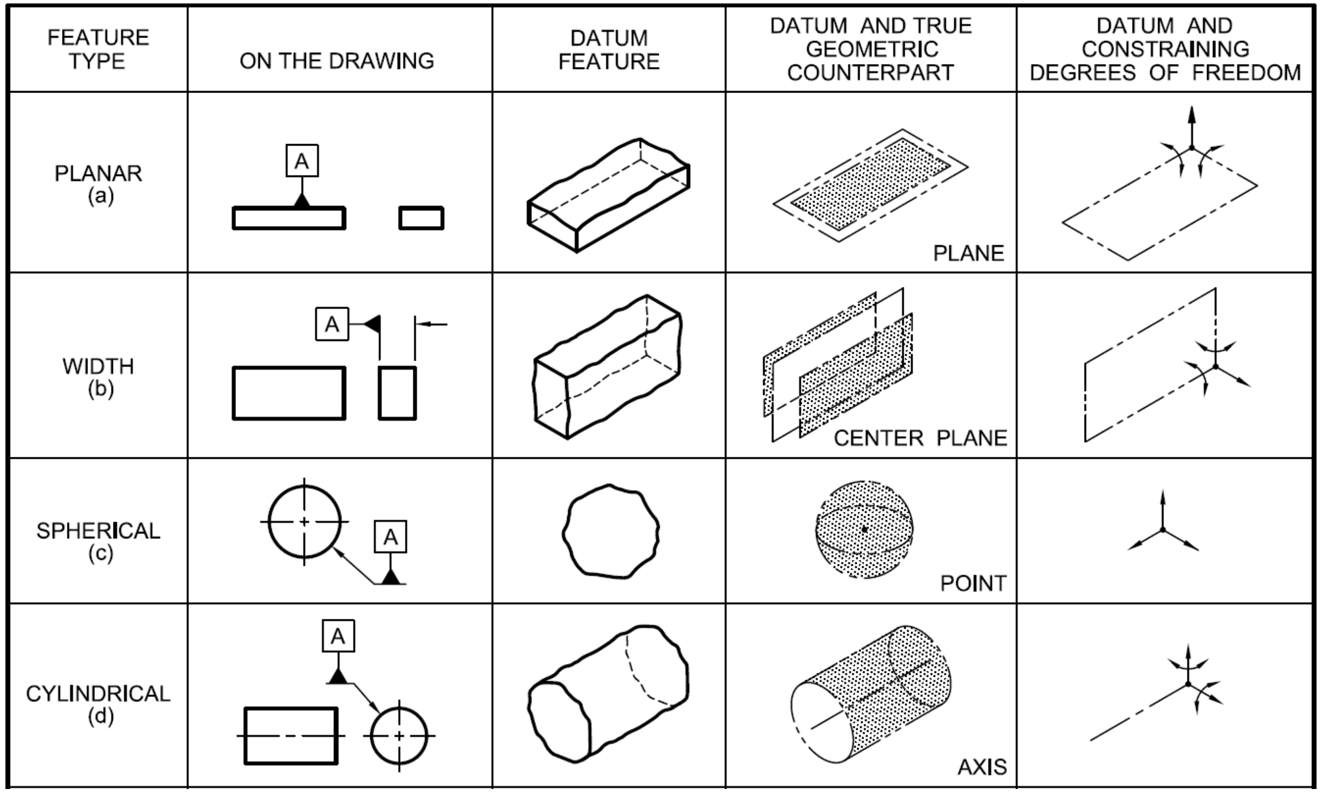
# Clarifications & review of GD&T fundamentals

# Clarifications to Part I

**Q:** Can a datum callout be attached to a feature axis, center line, or center plane?

**A:** No. ASME Y14.5 is clear on this.

- The ***true geometric counterpart's*** axis or center line or center plane is the datum.




*ASME Y14.52018, adapted*

# Clarifications to Part I

Q: Why were concentricity and symmetry removed from ASME Y14.5-2018?
How should we replace them?

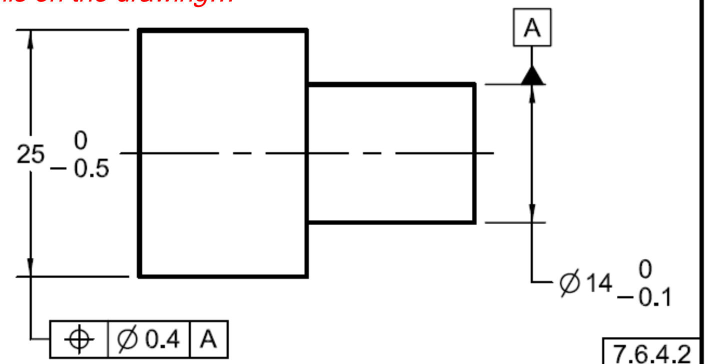
**Concentricity**

- Everyday definition ≠ GD&T definition.
- The GD&T definition was complex and often misunderstood.

**Instead…**

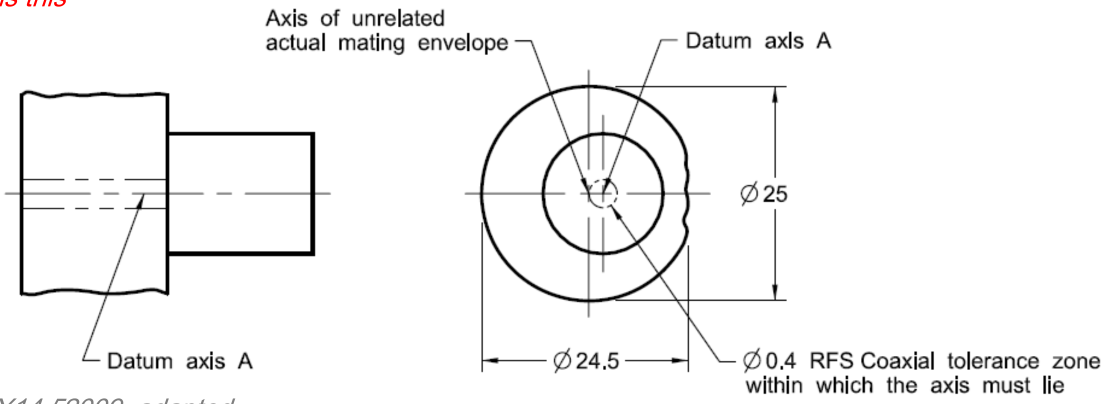
- Use position to control the feature's axis. A.k.a. "coaxiality."
- Use runout to control the feature's surface. A.k.a 'wobble.'

*This on the drawing…*




ASME Y14.5-2009, adapted

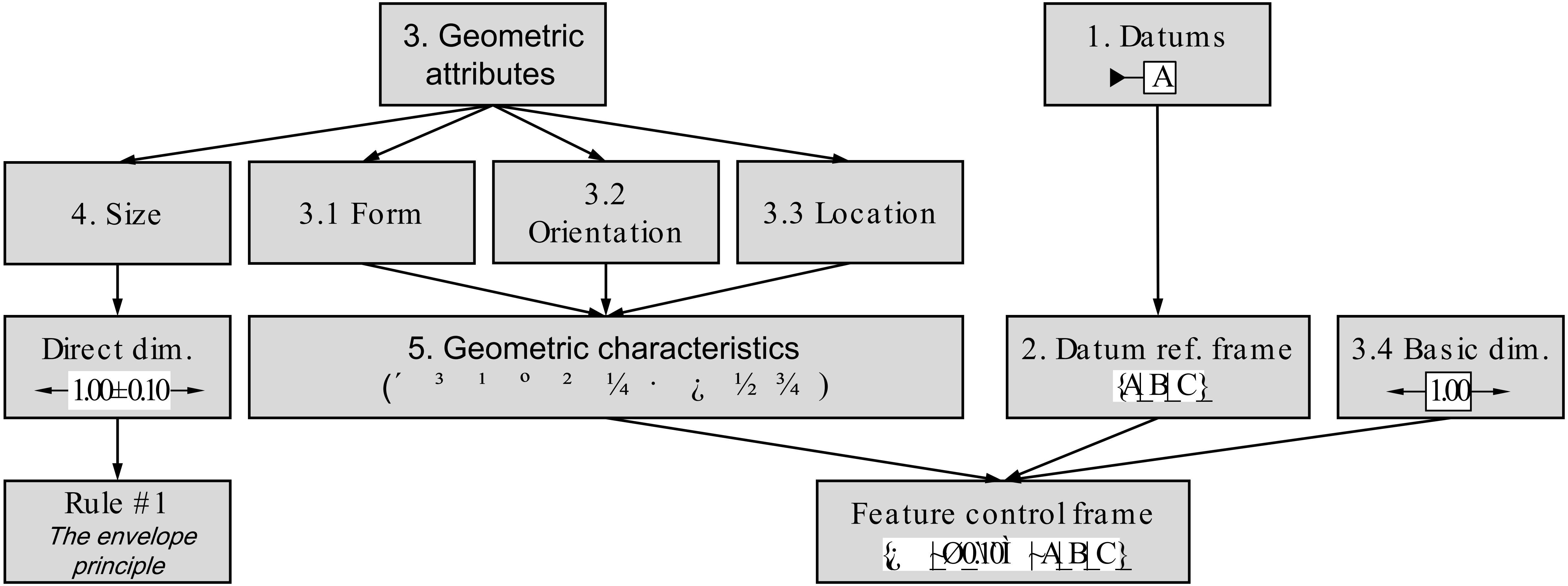
*…means this*




ASME Y14.5-2009, adapted

*__Symmetry__ was removed for similar reasons. Use position to control the location of a feature center line or center plane.*

# Map of GD&T


3. Geometric attributes
1. Datums
A
4. Size
3.1 Form
3.2 Orientation
3.3 Location
Direct dim.
1.00±0.10
5. Geometric characteristics
2. Datum ref. frame
A B C
3.4 Basic dim.
1.00
Rule #1
The envelope principle
Feature control frame
A B C

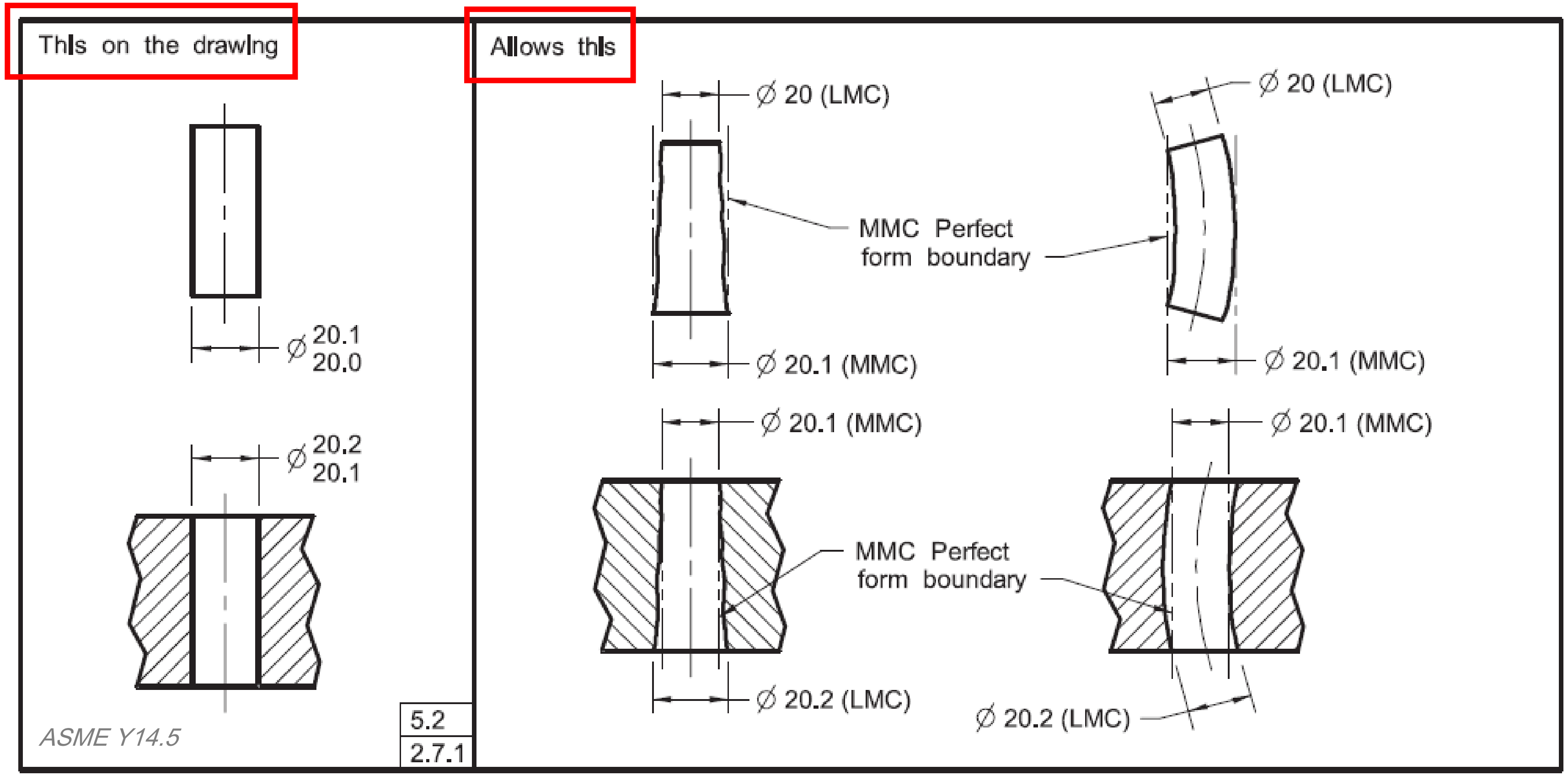
# Geometric characteristics

| Geometric characteristic | Symbol | Geometric attribute | Datum referencing? |
|---|---|---|---|
| Straightness | ¶ | Form | No |
| Flatness | ¸ | | |
| Circularity | ´ | | |
| Cylindricity | ³ | | |
| Profile of a line | ¹ | Profile (*location, orientation, size, & form*) | Sometimes datum referencing |
| Profile of a surface | º | | |
| Angularity | ² | Orientation | Datum referencing |
| Perpendicularity | ¼ | | |
| Parallelism | · | | |
| Position | ¿ | Location | Datum referencing |
| Circular runout | ½ | Runout (*location of a cylinder*) | Datum referencing |
| Total runout | ¾ | | |

# Rule #1– The envelope principle

"The form of an individual regular feature of size is controlled by its limits of size"

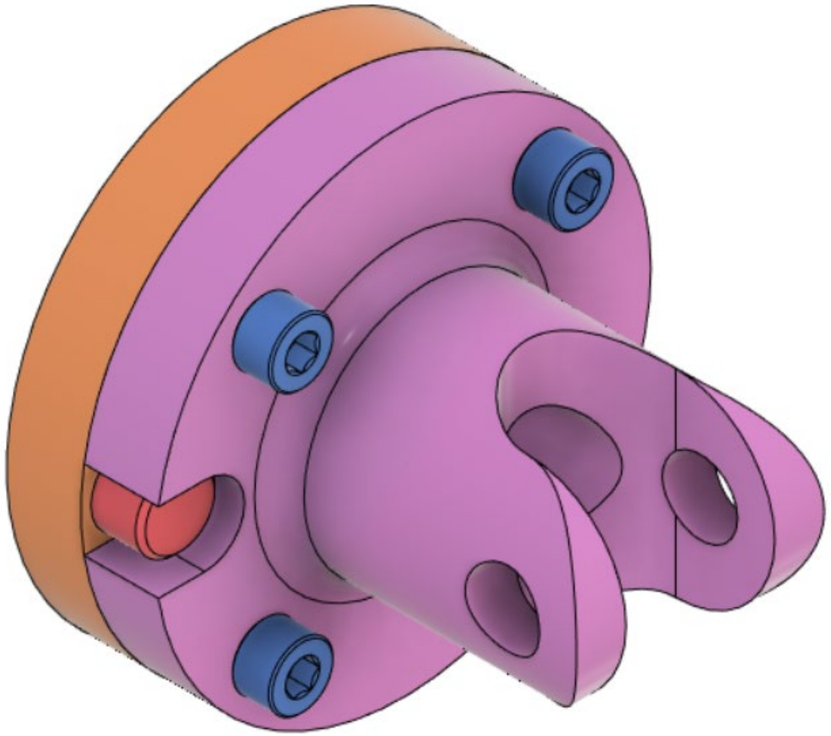
- *The MMC and LMC act like an envelope, therefore a feature of size inherently has form control.*
- *Form control can be additionally refined via*_, [, {, }, !,  ~

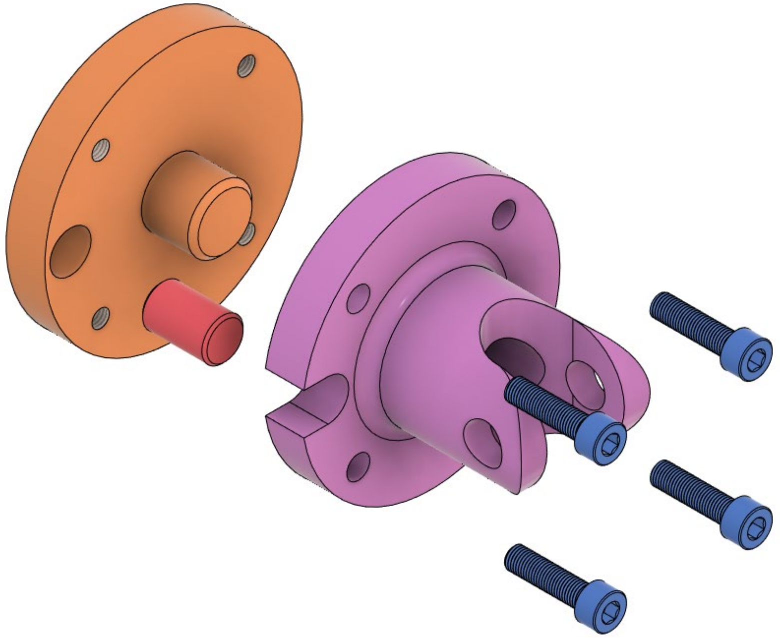


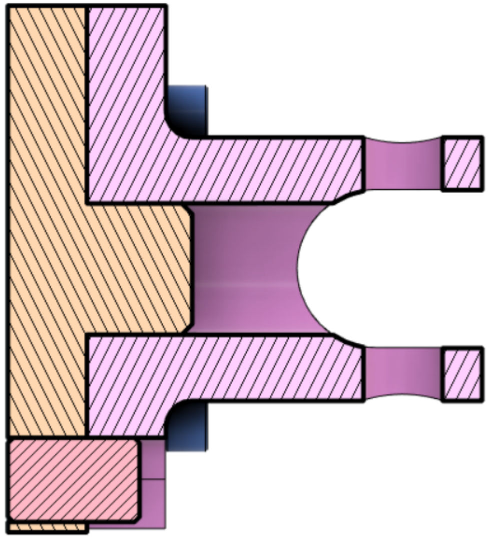
# Implementing GD&T:
## Checklist and walk-through

# GD&T How-to: Example

1. Understand the functionality of the part. Identify features that control function and assembly.







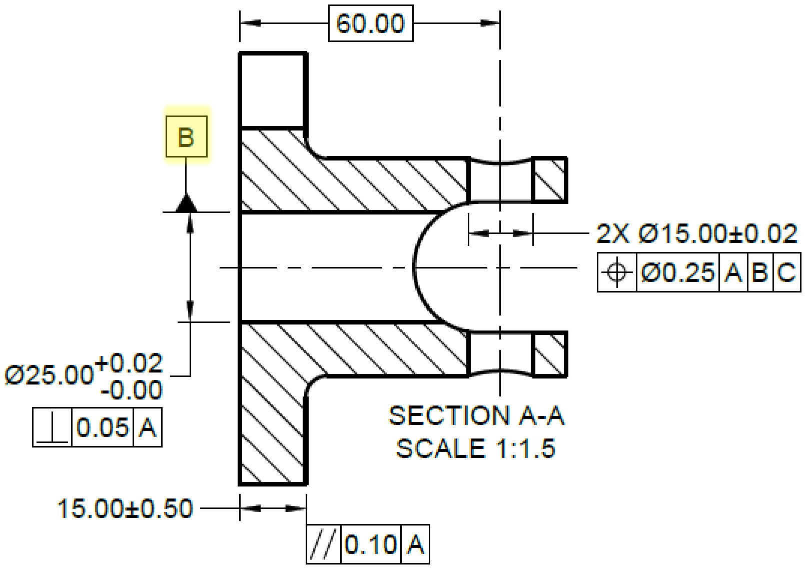
*Every part is different each requires special attention.*
*These guidelines are not definitive.*

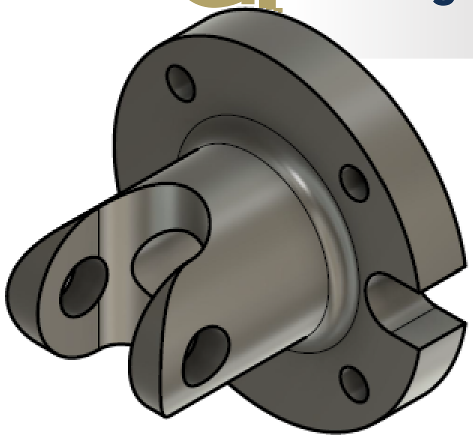
1. Understand the functionality of the part. Identify features that control function and assembly.
2. Based on (1), choose datums that mimic the functionality of the part
3. Control the form of datum features (normally [, }, ±*)

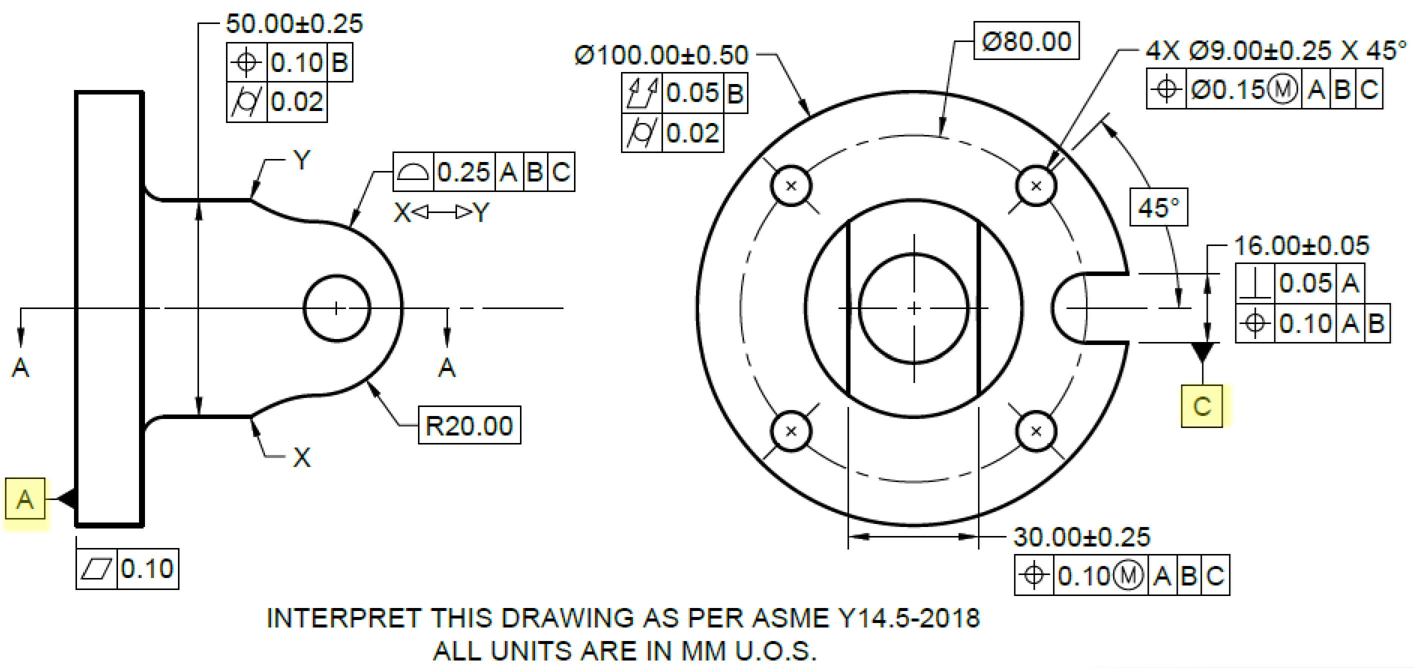
*Direct dimensioning controls form via the envelope principle.

1. Control the relation of datum features to each other (normally & and * )
2. Control features of size (±)
3. Control features of form that need no DRF
4. Control the position, orientation, profile, and/or runout of unconstrained features to a DRF**, apply basic dimensions.

**6 DoF not always required, DRF may vary for each feature





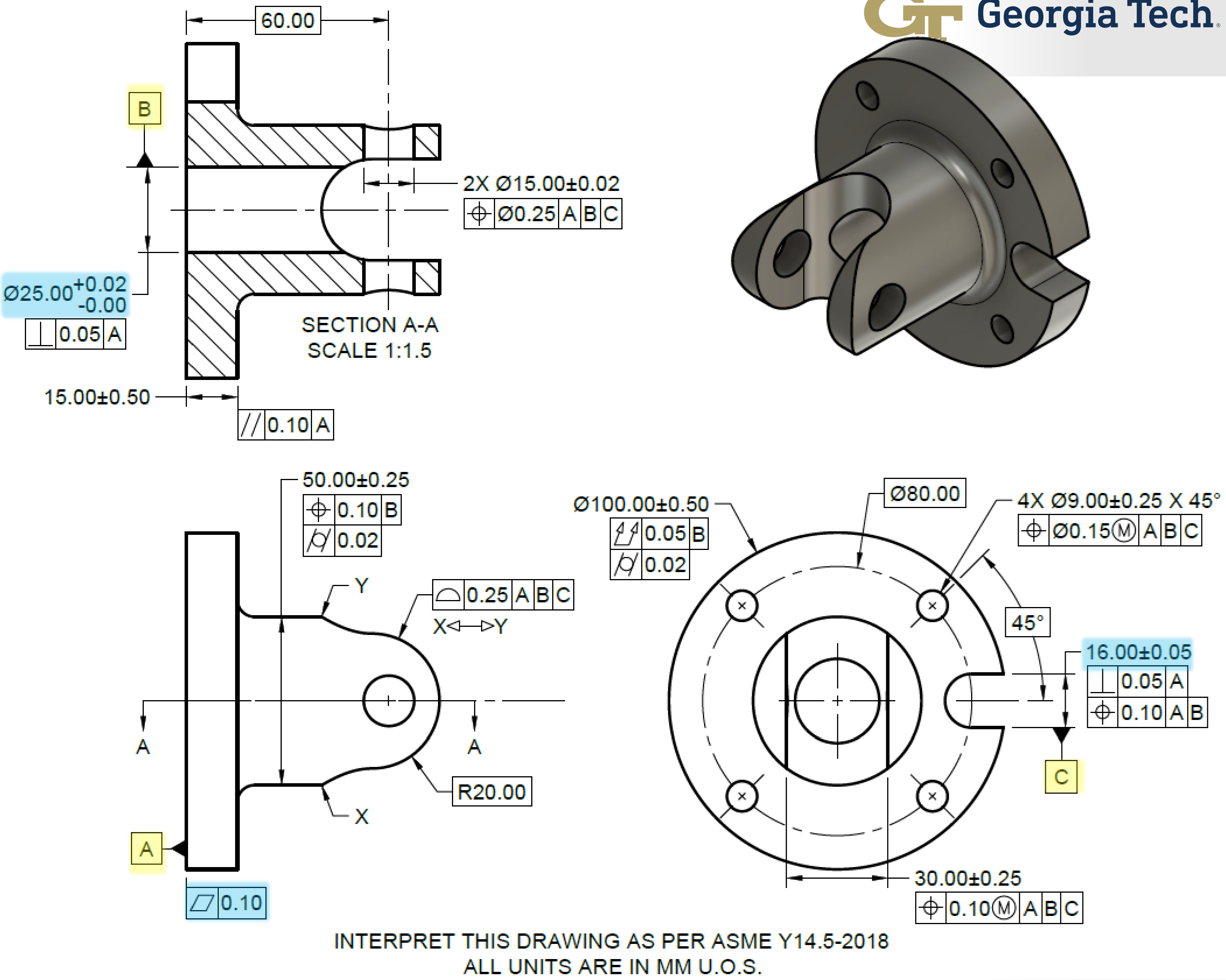
*Every part is different each requires special attention.*
*These guidelines are not definitive.*

1. Understand the functionality of the part. Identify features that control function and assembly.
2. Based on (1), choose datums that mimic the functionality of the part
3. Control the form of datum features (normally [, }, ±*)

*Direct dimensioning controls form via the envelope principle.

1. Control the relation of datum features to each other (normally & and * )
2. Control features of size (±)
3. Control features of form that need no DRF
4. Control the position, orientation, profile, and/or runout of unconstrained features to a DRF**, apply basic dimensions.

**6 DoF not always required, DRF may vary for each feature



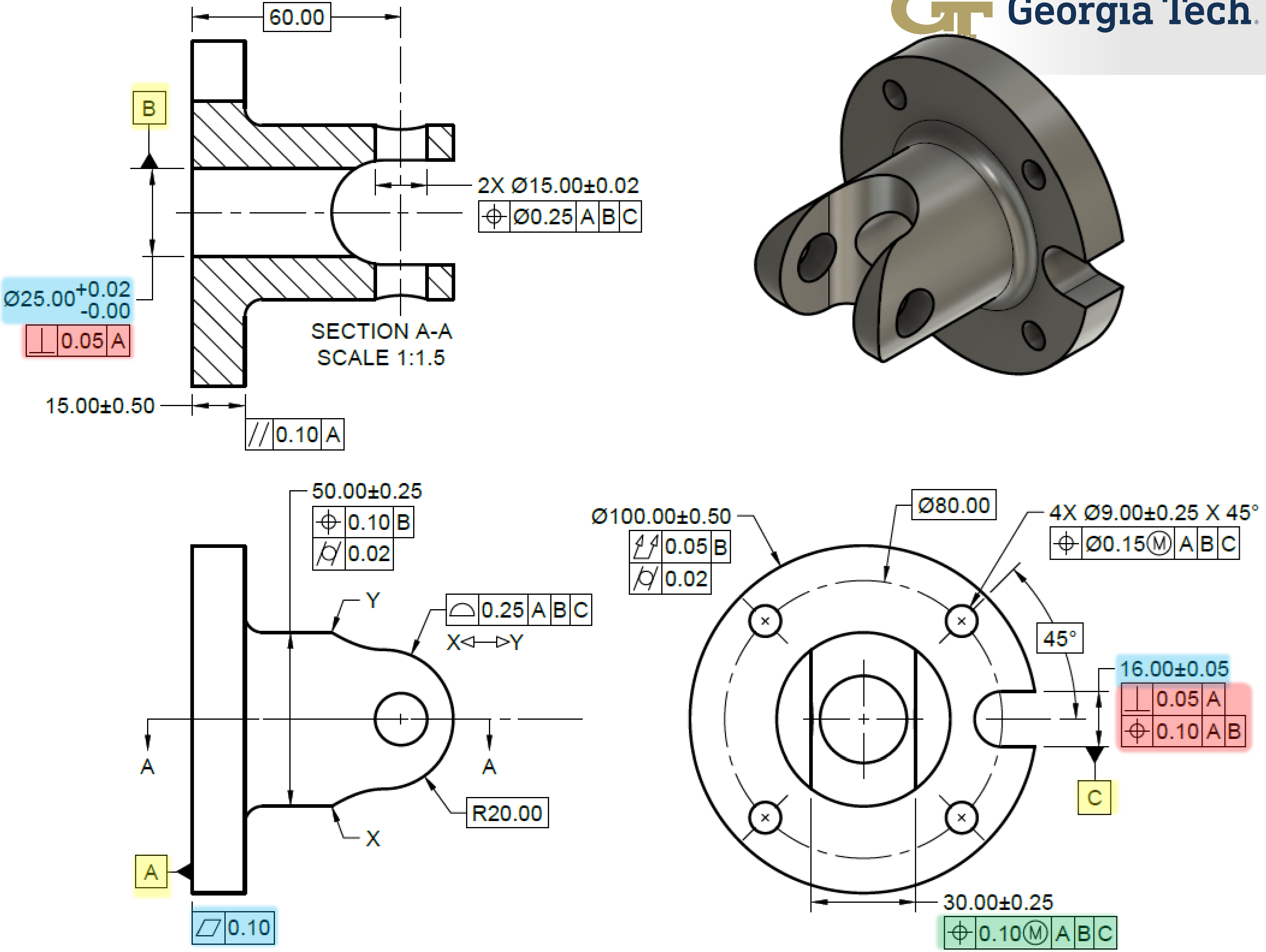
*Every part is different each requires special attention.*
*These guidelines are not definitive.*

1. Understand the functionality of the part. Identify features that control function and assembly.
2. Based on (1), choose datums that mimic the functionality of the part
3. Control the form of datum features (normally [, }, ±*)

*Direct dimensioning controls form via the envelope principle.

1. Control the relation of datum features to each other (normally & and * )
2. Control features of size (±)
3. Control features of form that need no DRF
4. Control the position, orientation, profile, and/or runout of unconstrained features to a DRF**, apply basic dimensions.

**6 DoF not always required, DRF may vary for each feature




INTERPRET THIS DRAWING AS PER ASME Y14.5-2018
ALL UNITS ARE IN MM U.O.S.

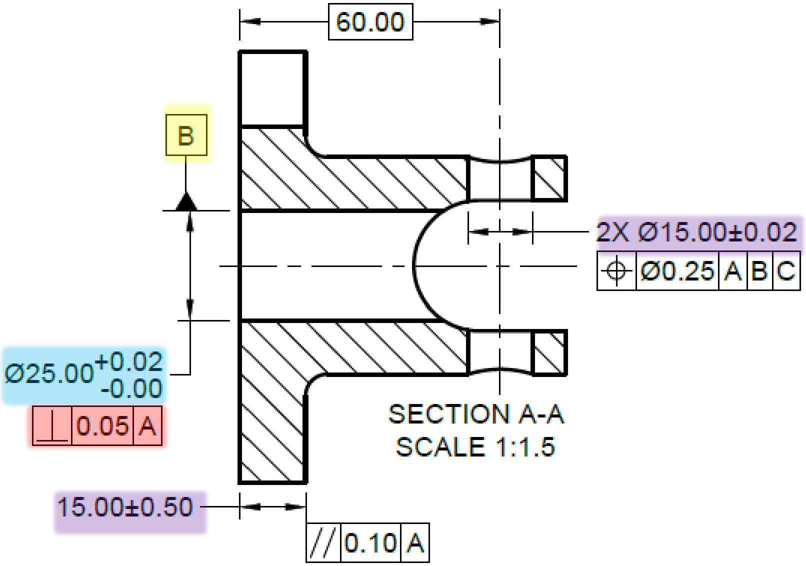
*Every part is different each requires special attention.*
*These guidelines are not definitive.*

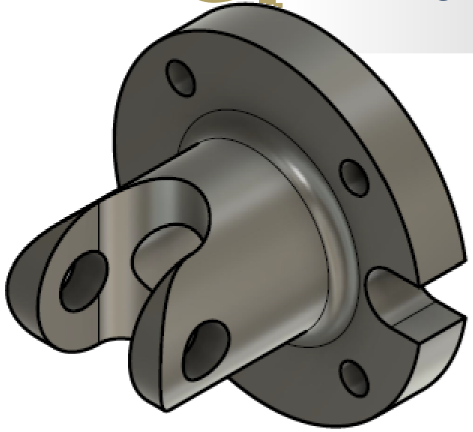
1. Understand the functionality of the part. Identify features that control function and assembly.
2. Based on (1), choose datums that mimic the functionality of the part
3. Control the form of datum features (normally [, }, ±*)

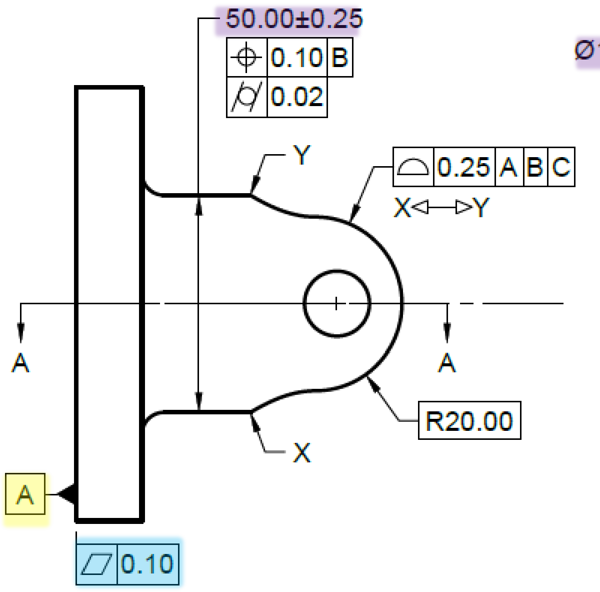
*Direct dimensioning controls form via the envelope principle.

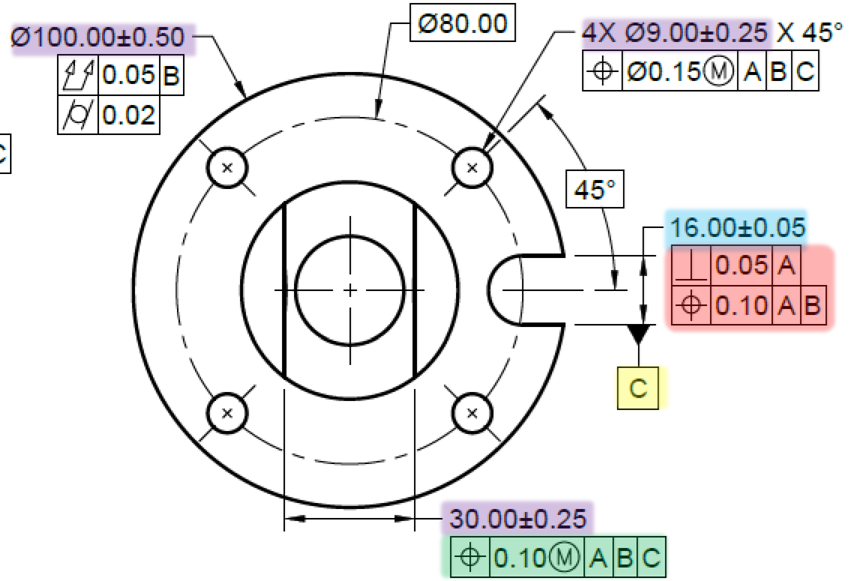
1. Control the relation of datum features to each other (normally & and * )
2. Control features of size (±)
3. Control features of form that need no DRF
4. Control the position, orientation, profile, and/or runout of unconstrained features to a DRF**, apply basic dimensions.

**6 DoF not always required, DRF may vary for each feature






INTERPRET THIS DRAWING AS PER ASME Y14.5-2018
ALL UNITS ARE IN MM U.O.S.

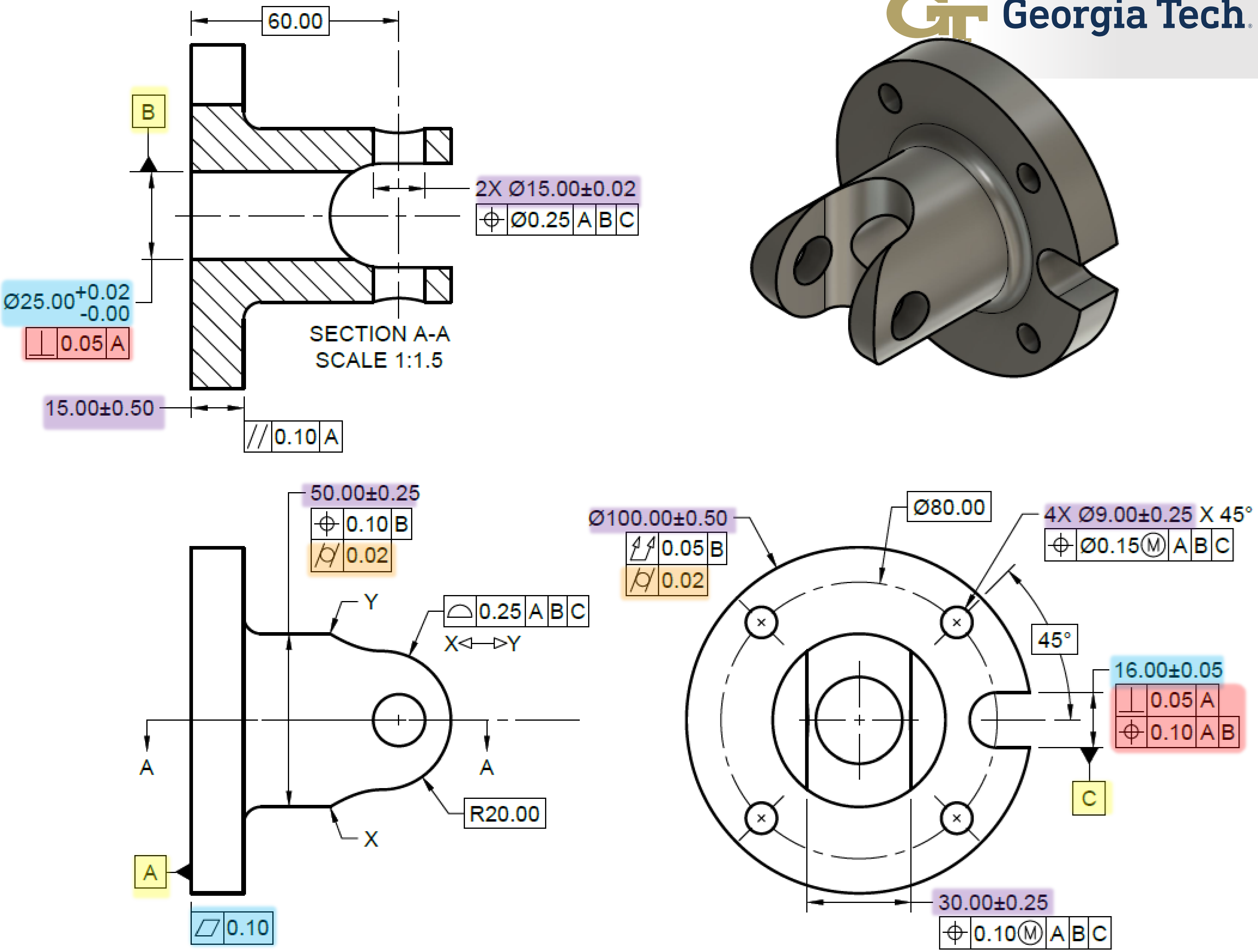
*Every part is different each requires special attention.*
*These guidelines are not definitive.*

1. Understand the functionality of the part. Identify features that control function and assembly.
2. Based on (1), choose datums that mimic the functionality of the part
3. Control the form of datum features (normally [, },          ±*)

*Direct dimensioning controls form via the envelope principle.

1. Control the relation of datum features to each other (normally &    and *    )
2. Control features of size (±)
3. Control features of form that need no DRF
4. Control the position, orientation, profile, and/or runout of unconstrained features to a DRF**, apply basic dimensions.

**6 DoF not always required, DRF may vary for each feature




INTERPRET THIS DRAWING AS PER ASME Y14.5-2018
ALL UNITS ARE IN MM U.O.S.

*Every part is different each requires special attention.*
*These guidelines are not definitive.*

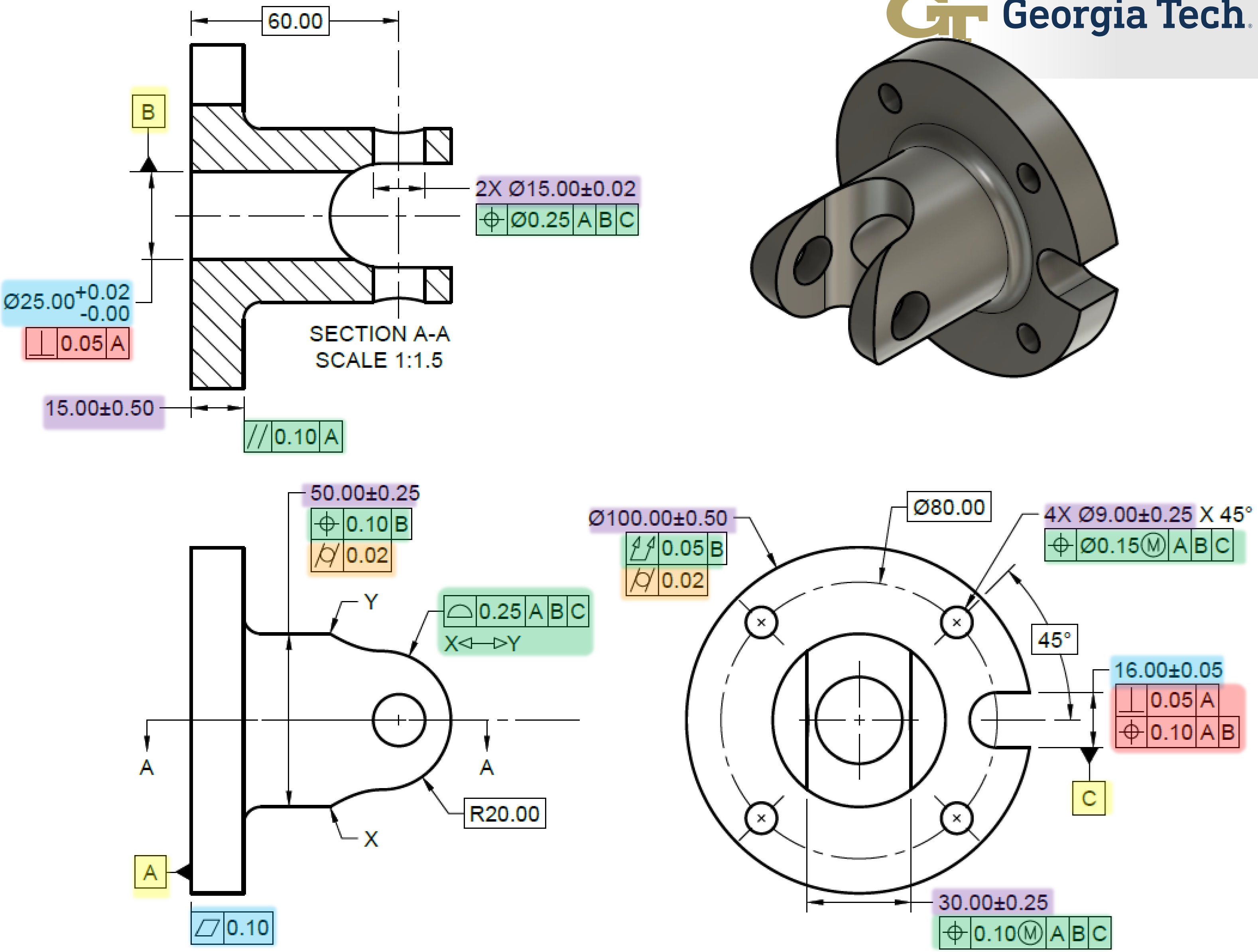
1. Understand the functionality of the part. Identify features that control function and assembly.
2. Based on (1), choose datums that mimic the functionality of the part
3. Control the form of datum features (normally [, },         ±*)

*Direct dimensioning controls form via the envelope principle.

1. Control the relation of datum features to each other (normally &    and *    )
2. Control features of size (±)
3. Control features of form that need no DRF
4. Control the position, orientation, profile, and/or runout of unconstrained features to a DRF**, apply basic dimensions.

**6 DoF not always required, DRF may vary for each feature

Georgia Tech



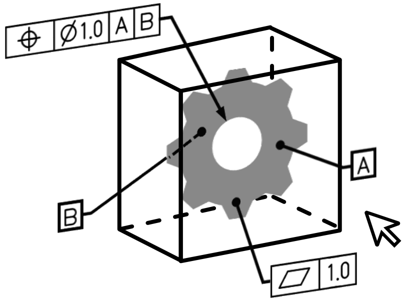

INTERPRET THIS DRAWING AS PER ASME Y14.5-2018
ALL UNITS ARE IN MM U.O.S.

# Digital product definition

## Model-based definition (MBD)

*Customer delivers a CAD file which includes GD&T**

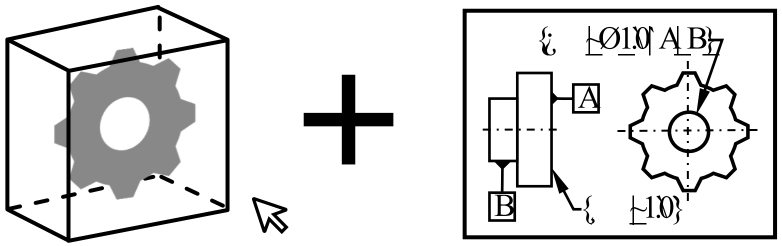
- Standard: ASME Y14.41-2019 Digital Product Definition
  - Not yet fully adopted
- MBD will very often be minimally dimensioned. Basic dimensions will not be automatically shown, but queried by the user as necessary.




## Minimally dimensioned drawings

*Customer delivers a minimally (a.k.a., partially, reduced, etc.) dimensioned drawing and CAD data*

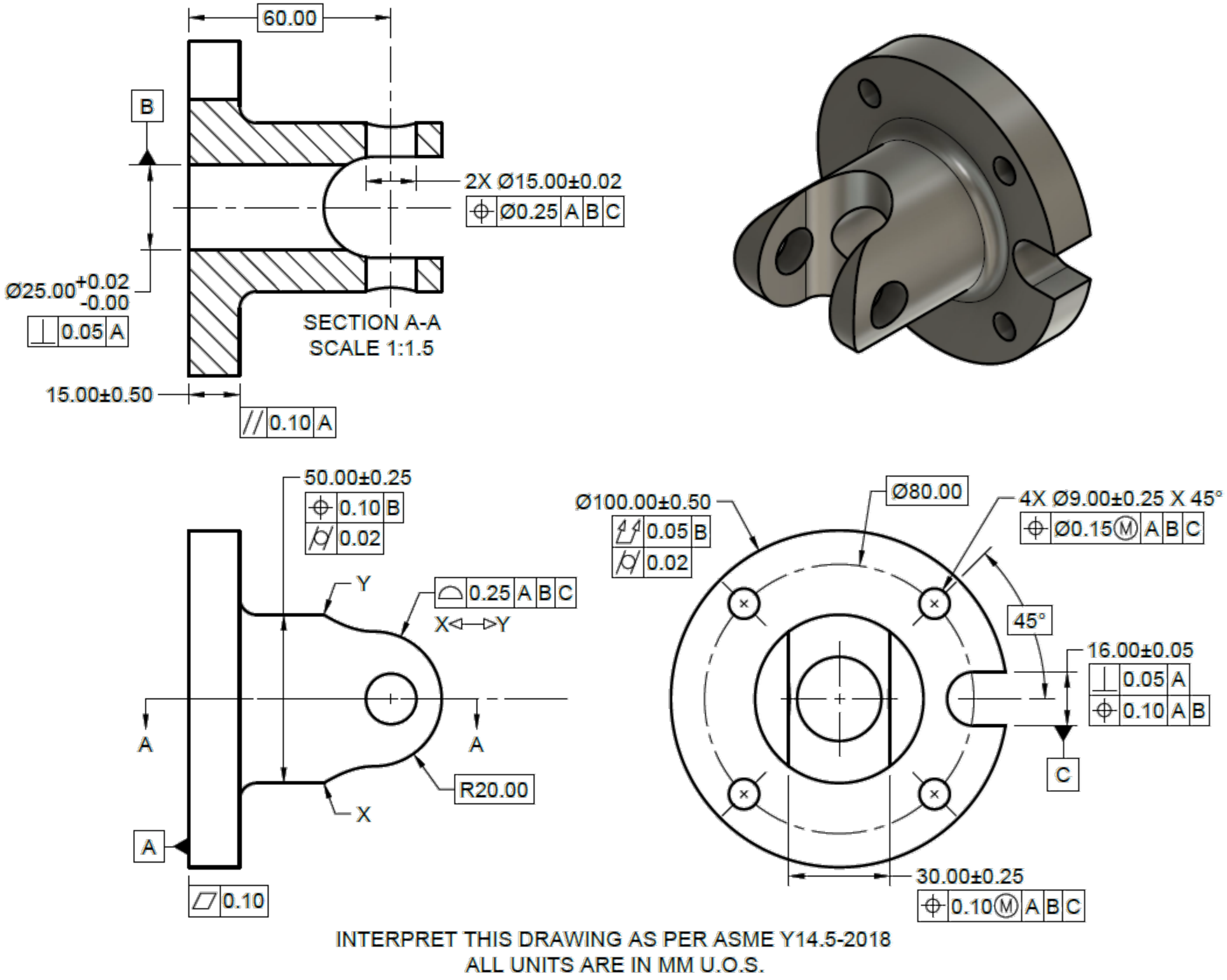
- It is acceptable practice to not fully-dimension drawings
- Ex: *Note: This drawing is minimally dimensioned. Refer to the provided CAD data for basic dimensions.*




**File management is complex and multiple standards may apply.*

# Example of a minimally dimensioned drawing

Georgia Tech




NOTES

1. THIS IS A MINIMALLY DIMENSIONED DRAWING. REFER TO THE PROVIDED CAD DATA, <PN HERE>, FOR BASIC DIMENSIONS.
2. THE FOLLOWING TOLERANCE APPLIES TO ALL UNDIMENSIONED FEATURES IN THIS DRAWING, UNLESS OTHERWISE SPECIFIED. {° ⌊1.0| A|B|C}

*This is an illustrative example. Drafters should use verbiage appropriate to their company and application.*

# Dimensional inspection for GD&T

# A brief introduction to dimensional inspection

*So, GD&T is used for specification… but how do we measure to ensure manufacturing met the specification?*

## ‘Simple’ measurement instruments

- Calipers, outside micrometers, etc. (used with features of size)
- Hard-gauging – gauge pins, etc. (used with features of size)
- Displacement instruments – dial indicators, test indicators, etc. (used for multiple functions)

## Coordinate measurements systems (CMS)

- A.k.a. coordinate measurement machines (CMMs)
- Modern CMS come use varying principles, commonly tactile measurement
- CMS instruments fundamentally measure samples of a surface in x,y,zdimensions

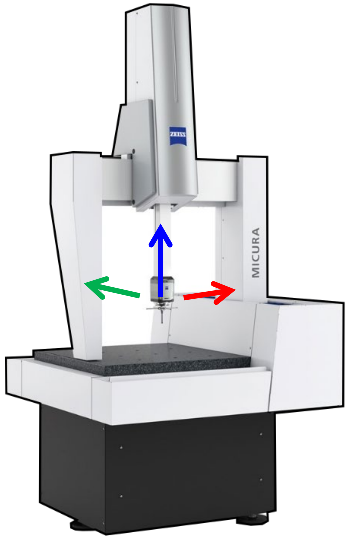
# Coordinate Measurement System (CMS) use… Georgia Tech

The best use cases for a CMS include…

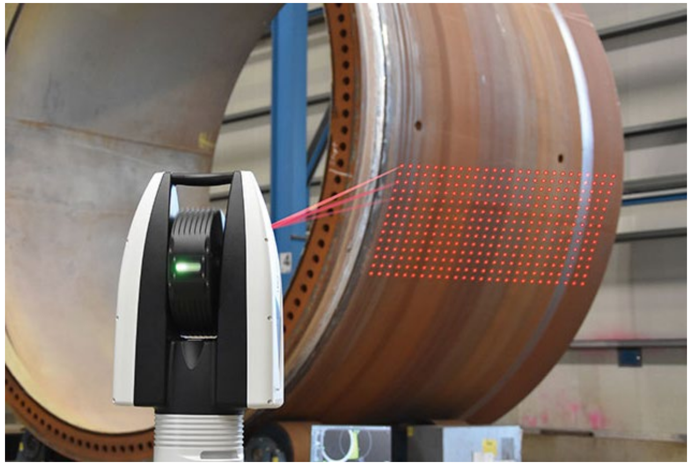
- Complex component surfaces
- Complex measurement tasks
- High degree of automation required

When might simpler instruments be appropriate?

- Simple measurement tasks
  - Feature-of-size (diameter, width, etc.)
  - Parallelism, squareness, flatness
- When inspection of a particular feature is required to be…
  - Inexpensive, high-volume, low-inspector expertise…

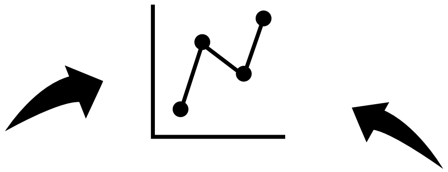



*Zeiss Industrial Metrology – Tactile CMM example*



*Hexagon AB– Laser theodolite example*

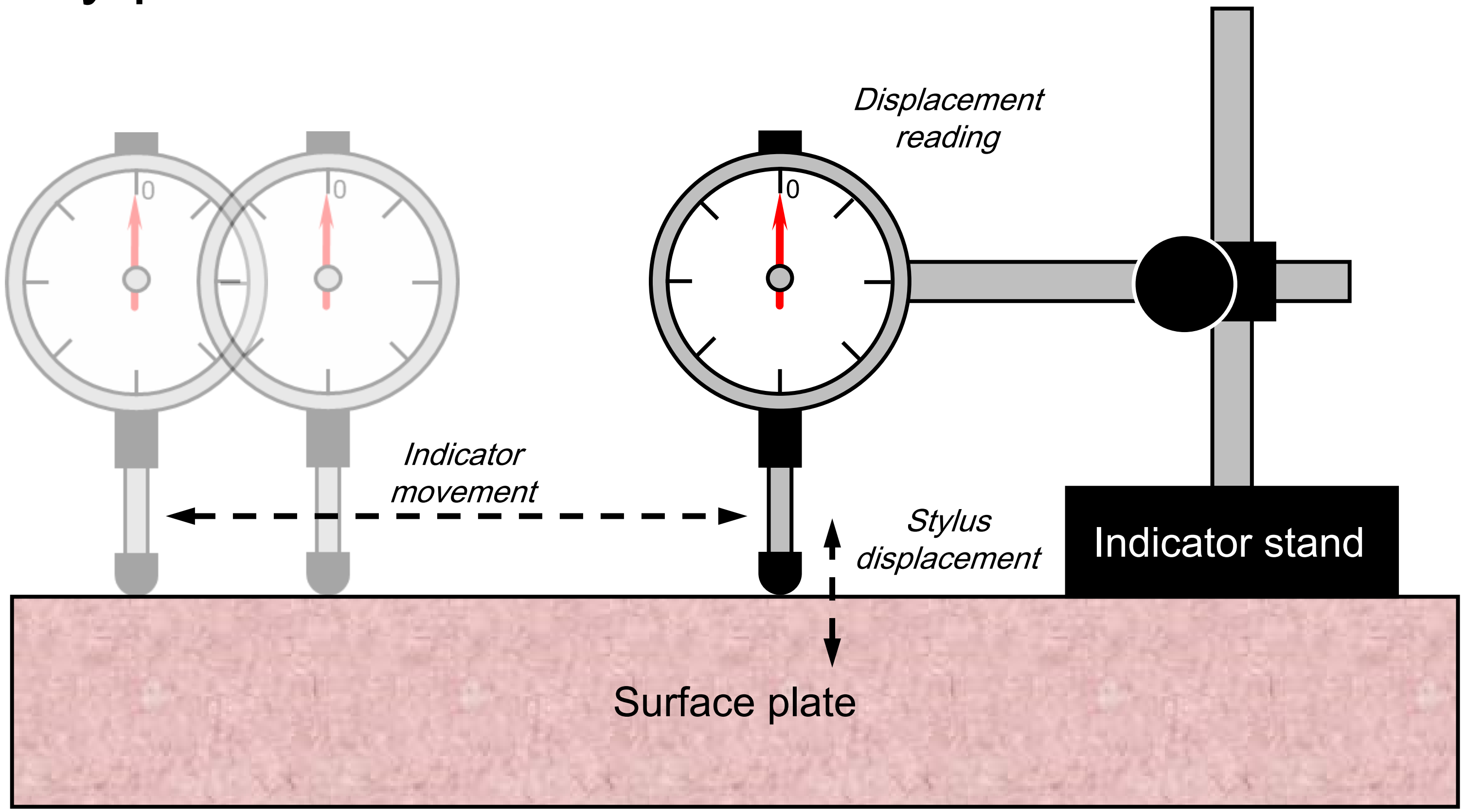

*Starrett – Digital indicator example*



*Mitutoyo America Corporation – Digital outside micrometer example*

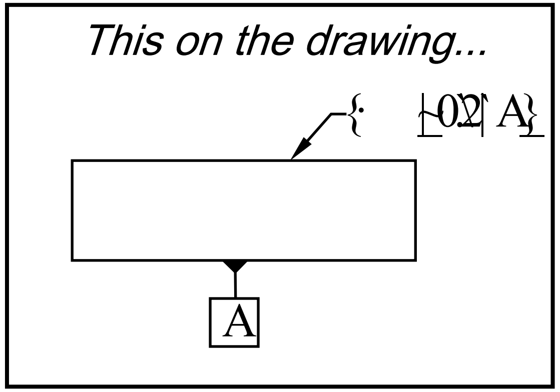
# Surface plate inspection principles

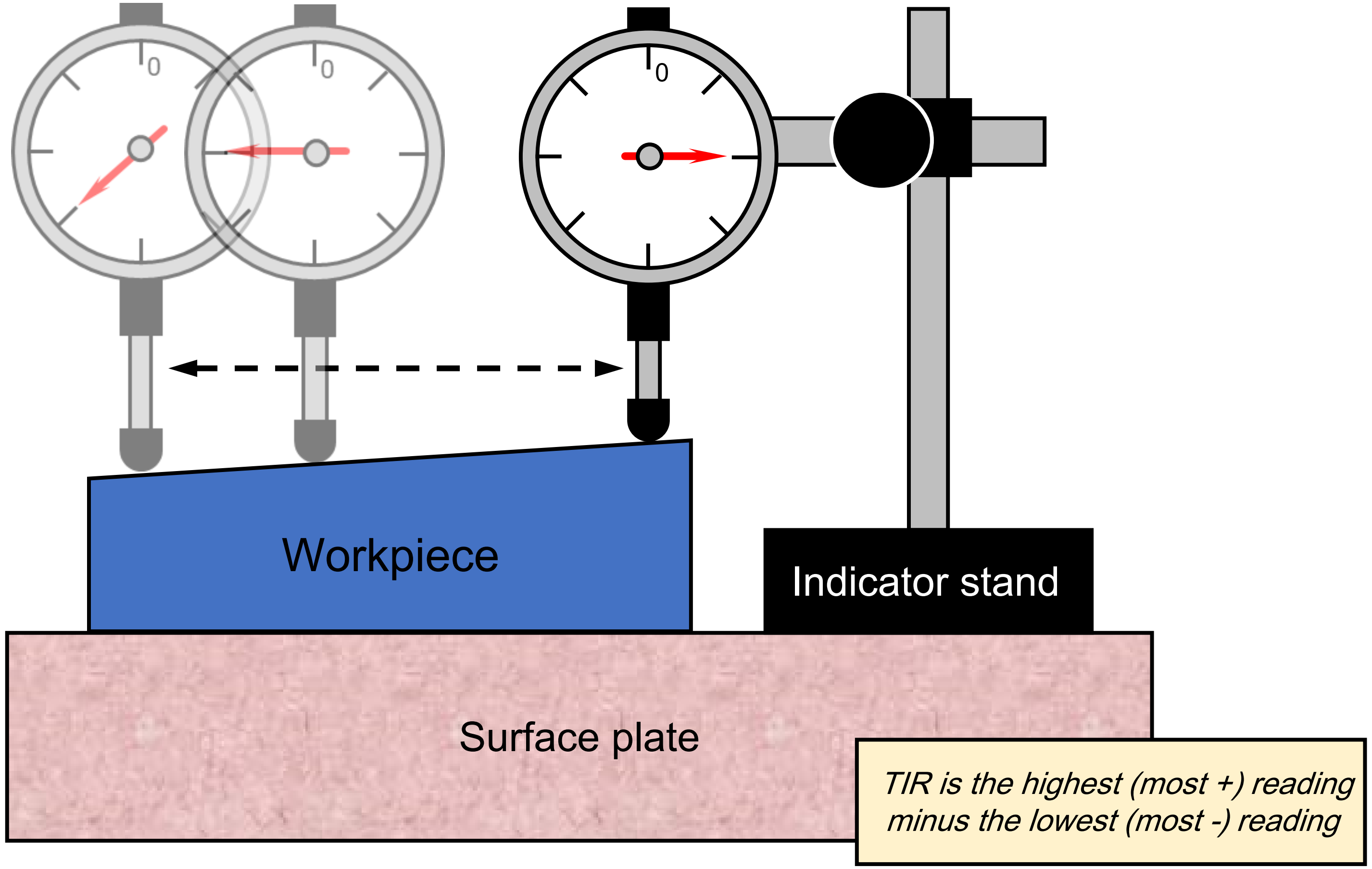
- Moving an indicator over a surface place should show zero dial movement – the stylus contact point and indicator stand base is ideally coplanar at any point of contact



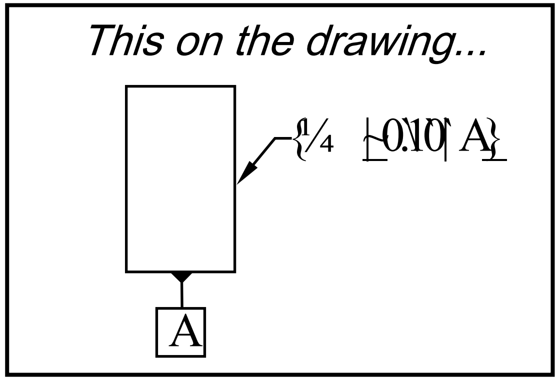
# Comparators- Parallelism measurement

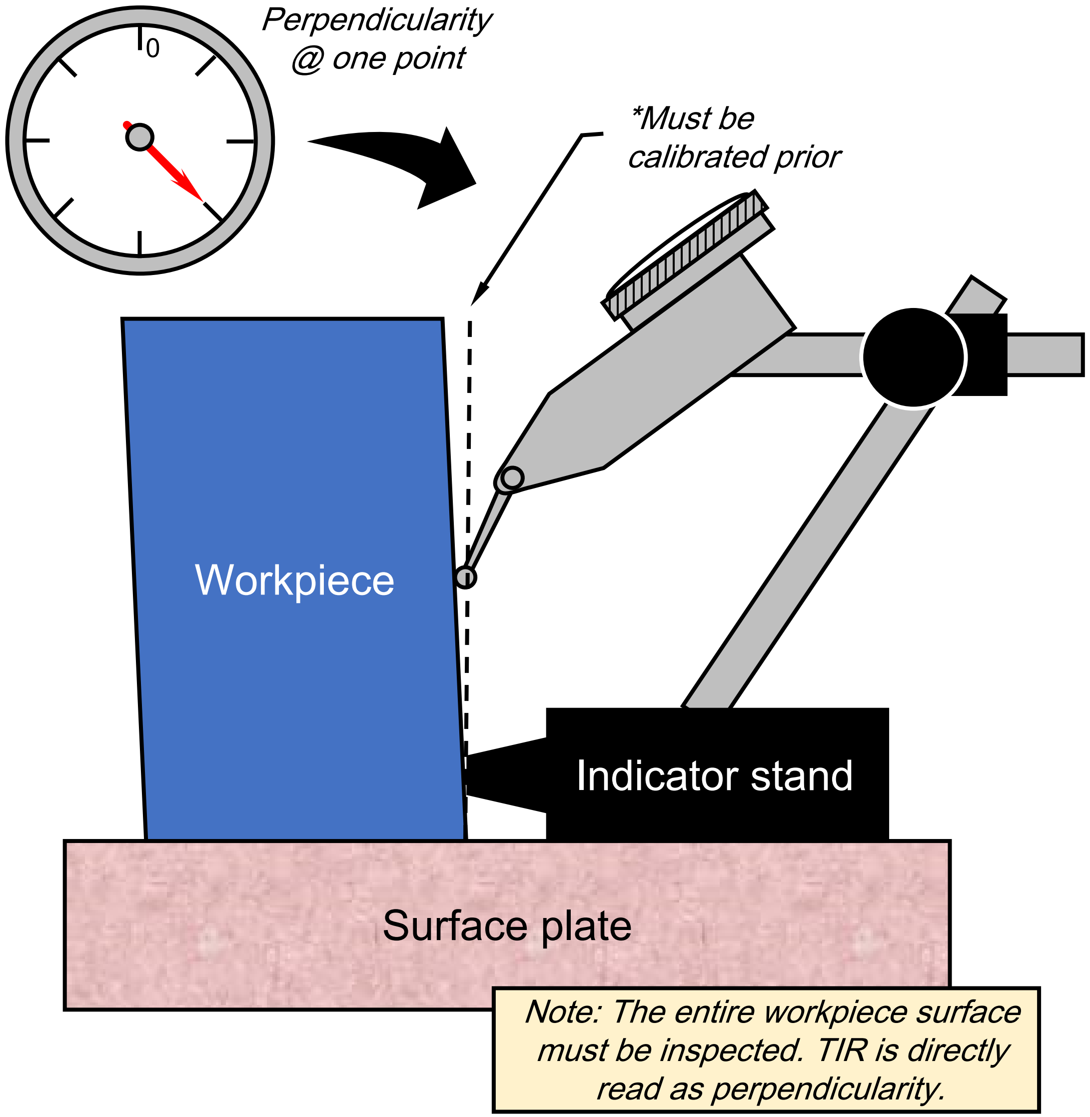
- The total indicator reading (TIR) is the maximum reading – the minimum reading
- TIR in over the workpiece is a direct reading of parallelism



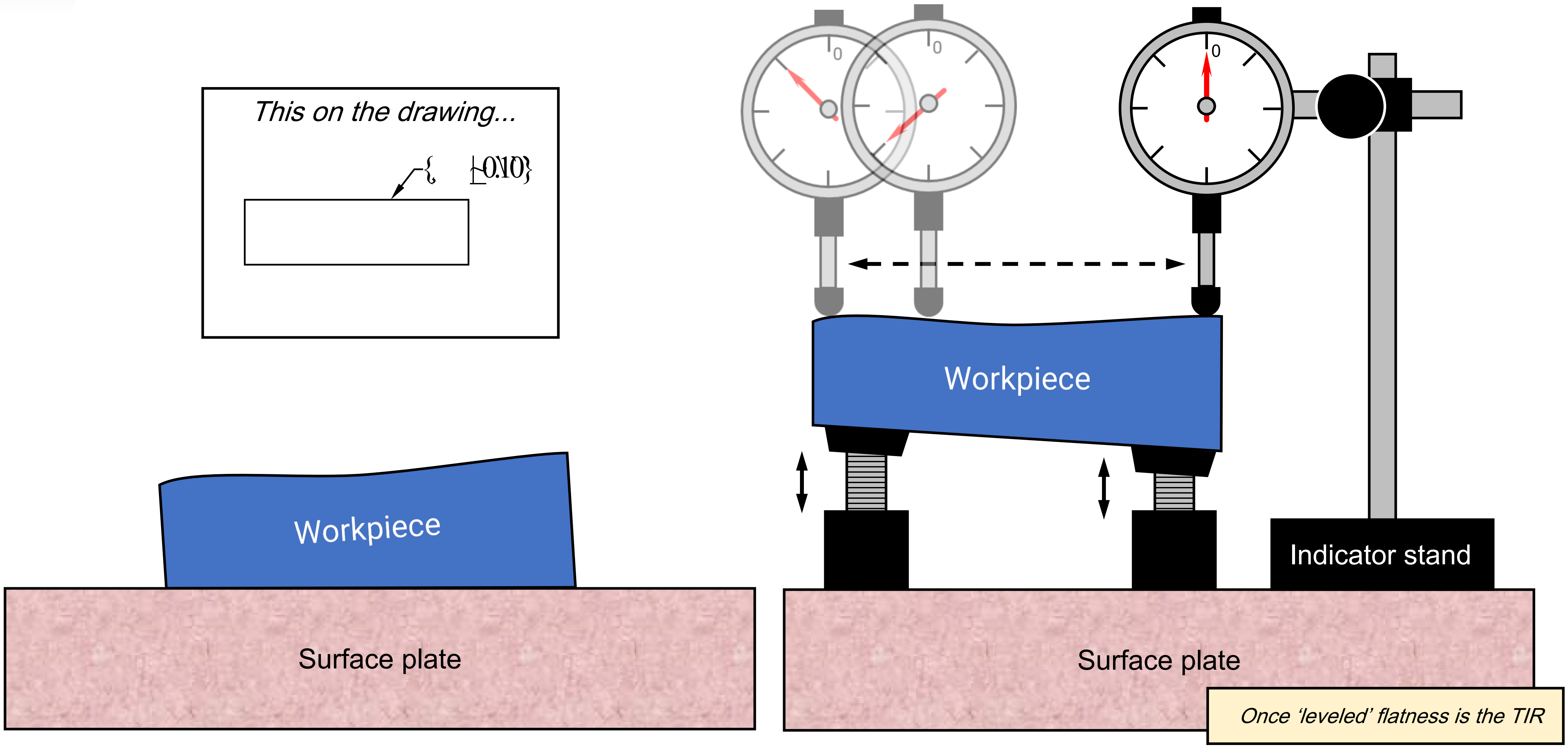



# Comparators- Perpendicularity measurement



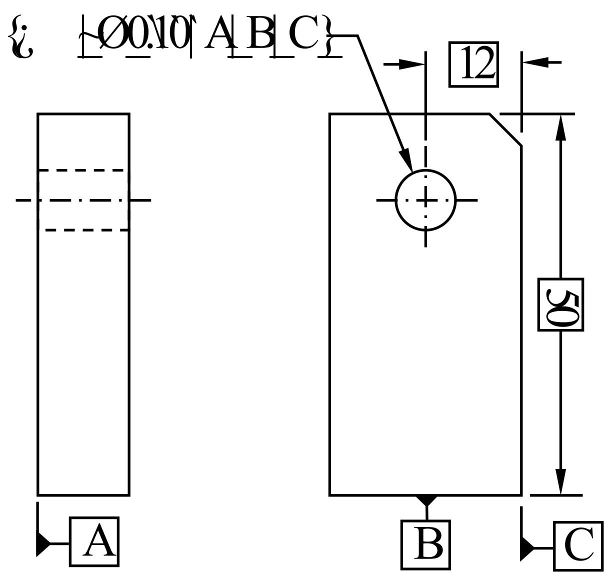
# Comparators- Flatness measurement

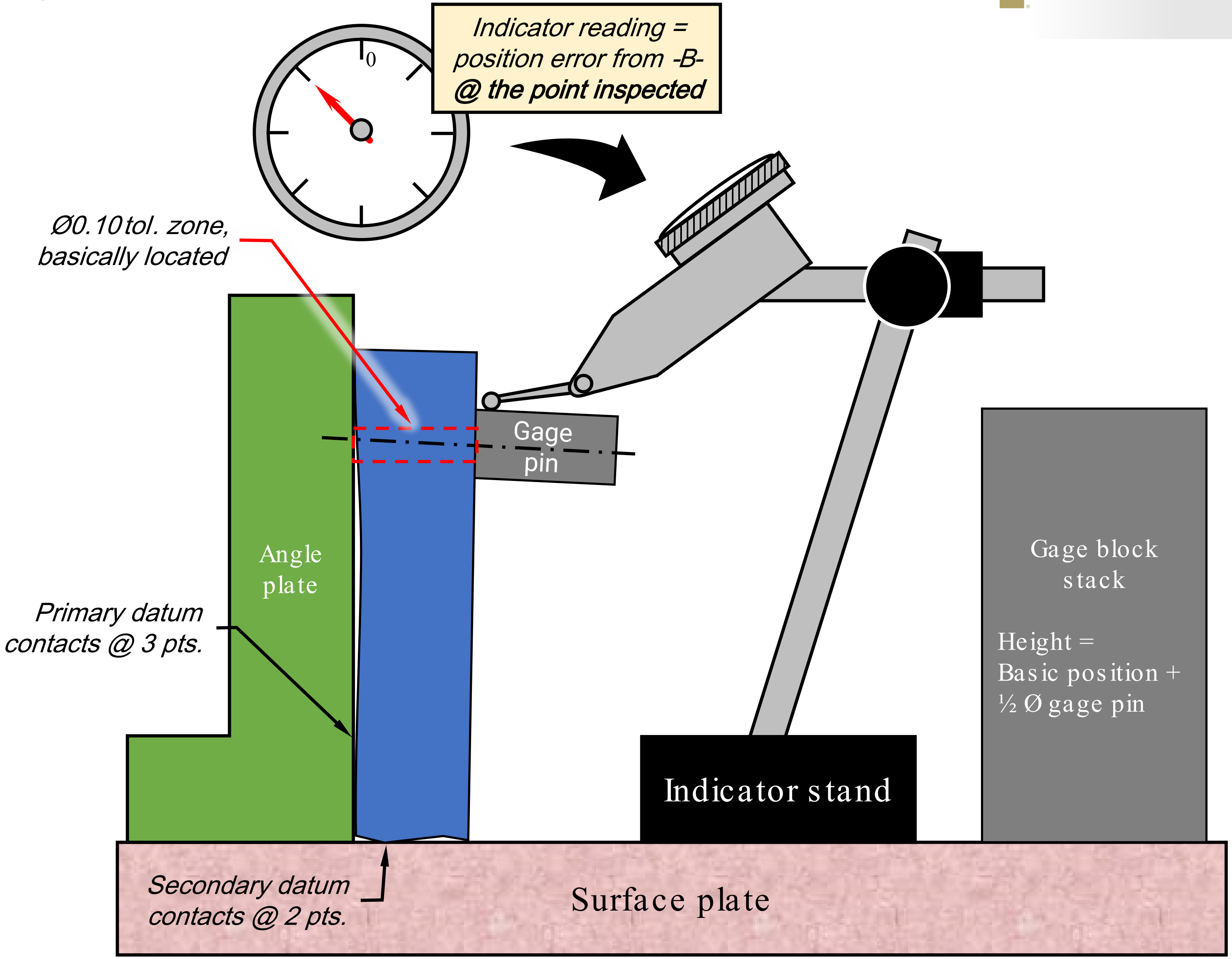


# Position measurement

1. Zero indicator on gage block stack.
2. Fit closest gage pin to hole.
3. Measure gage pin max. height *along its length.*
4. **Extrapolate measurements to the tolerance zone.**
5. Repeat for position from -C-
6. Use trig, combining measurements from steps 4 & 5, to find total position error

*This on the drawing...*



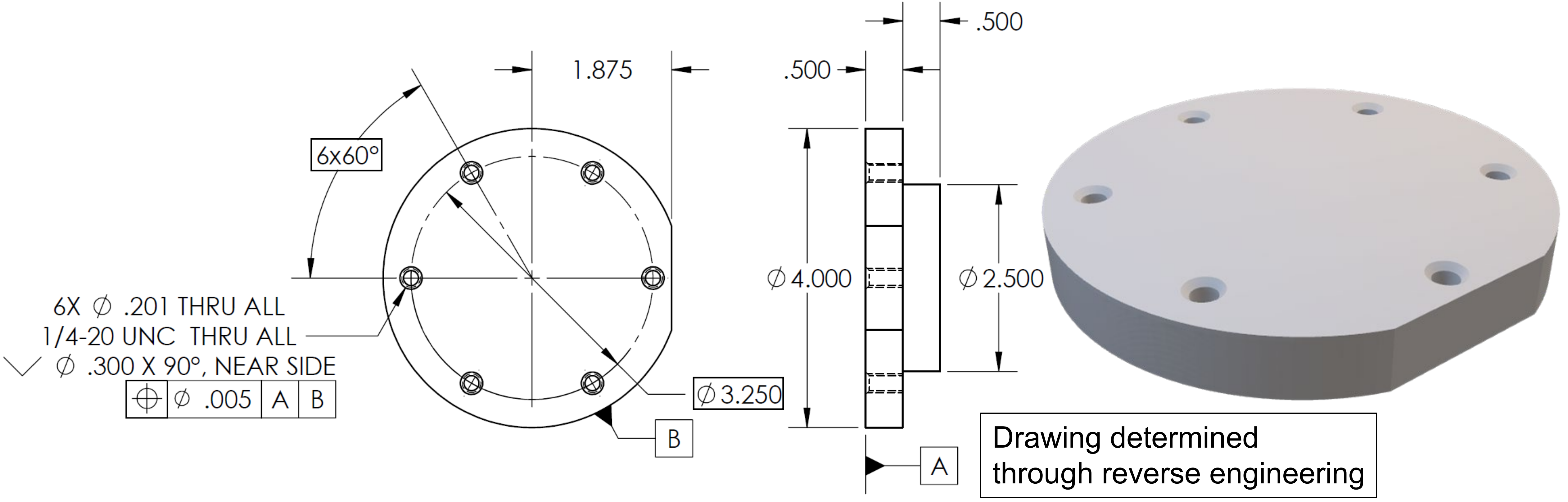



# Case studies

Study 1: Hole pattern tolerancing
Study 2: Imprecision in additive manufacturing
Study 3: CAD-to-actual comparison

# Case Study 1 Hole Tolerancing

Prompt: Your boss has asked you to design a component which bolts to the component shown below. Your drawing will be sent out to a manufacturing company to produce 1000 of your parts.

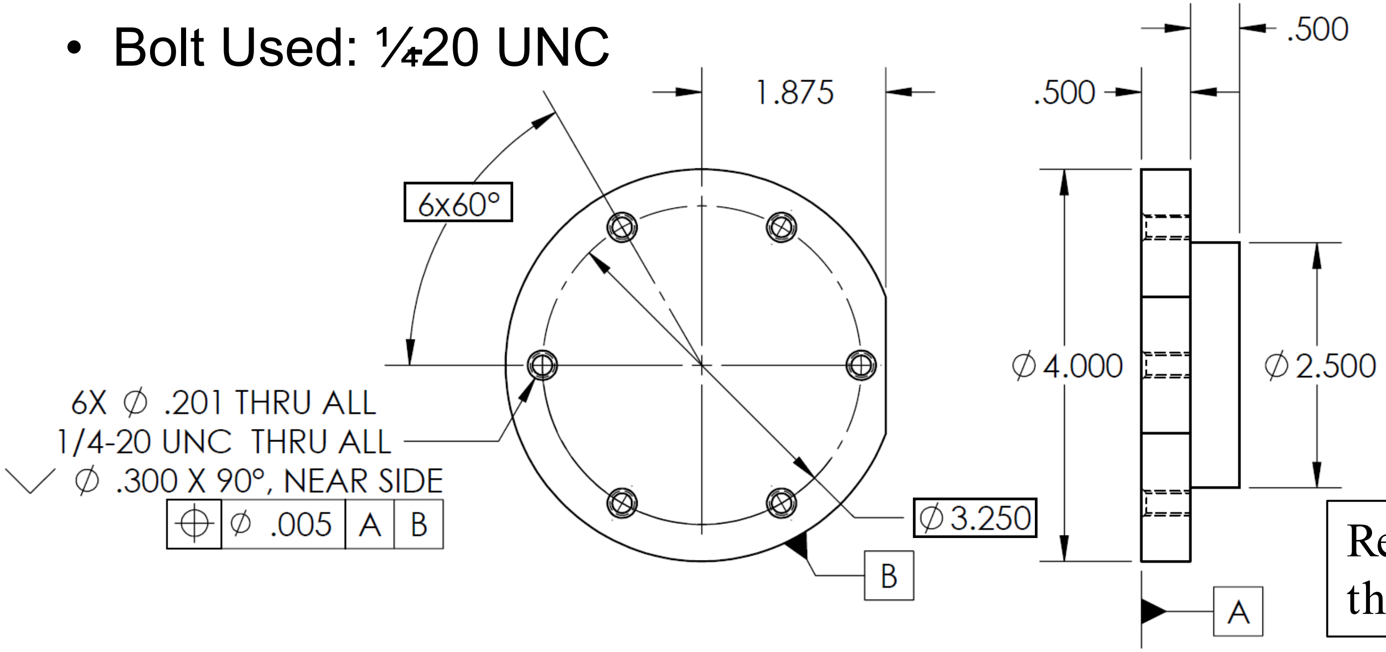


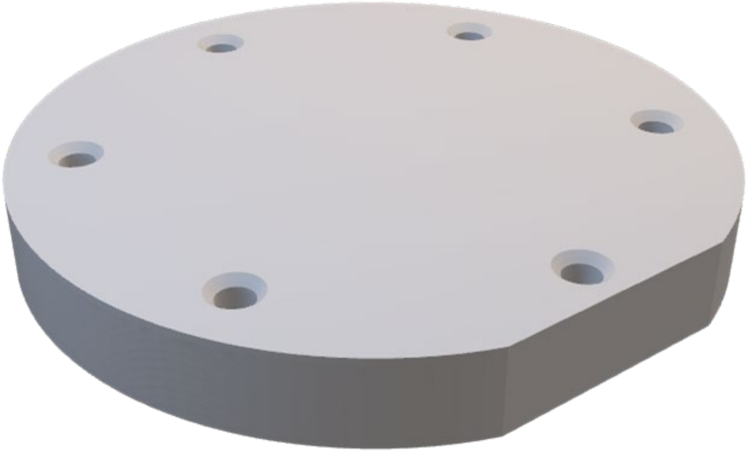
# Case Study: Hole Tolerancing

*UNC: Unifiednational-coarse thread*

## Understand the part:

- How will my part interface with this component?
  - Datums
    - Mating Surface, Outer Diameter
  - Bolt Pattern
    - Bolt Circle Diameter: 3.250", Spacing: 6 bolts evenly spaced, .005" position tolerance
    - Bolt Used: ¼-20 UNC



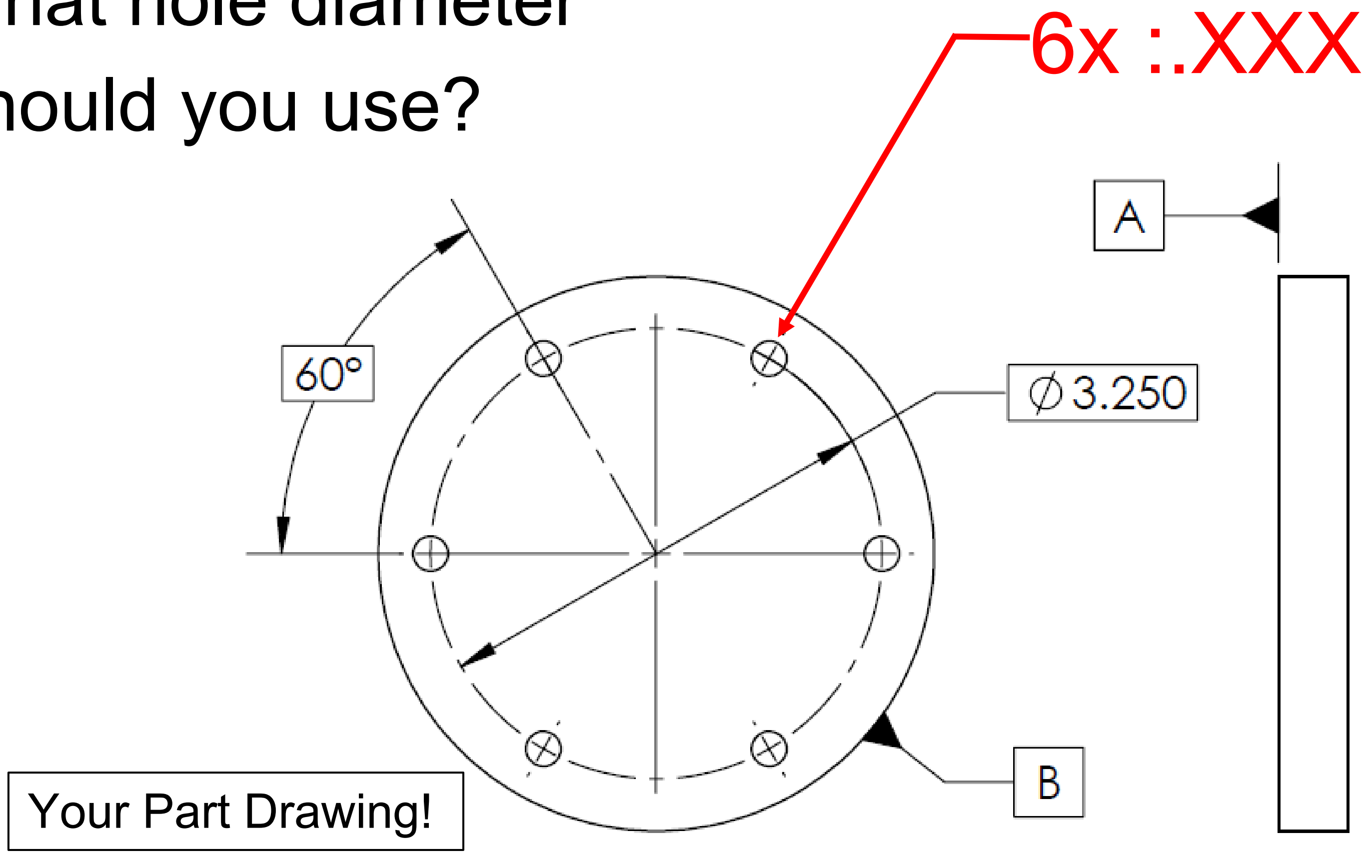



Reference drawing determined through reverse engineering

# Case Study: Hole Tolerancing

Basics established!

Next: What hole diameter
should you use?



# Case Study: Hole Tolerancing

Hole Diameter?

¼-20 UNC Bolt: .250 hole? No!

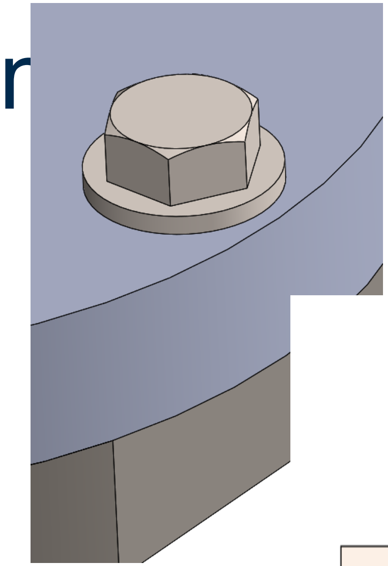
Class 2A ¼-20 UNC bolt:

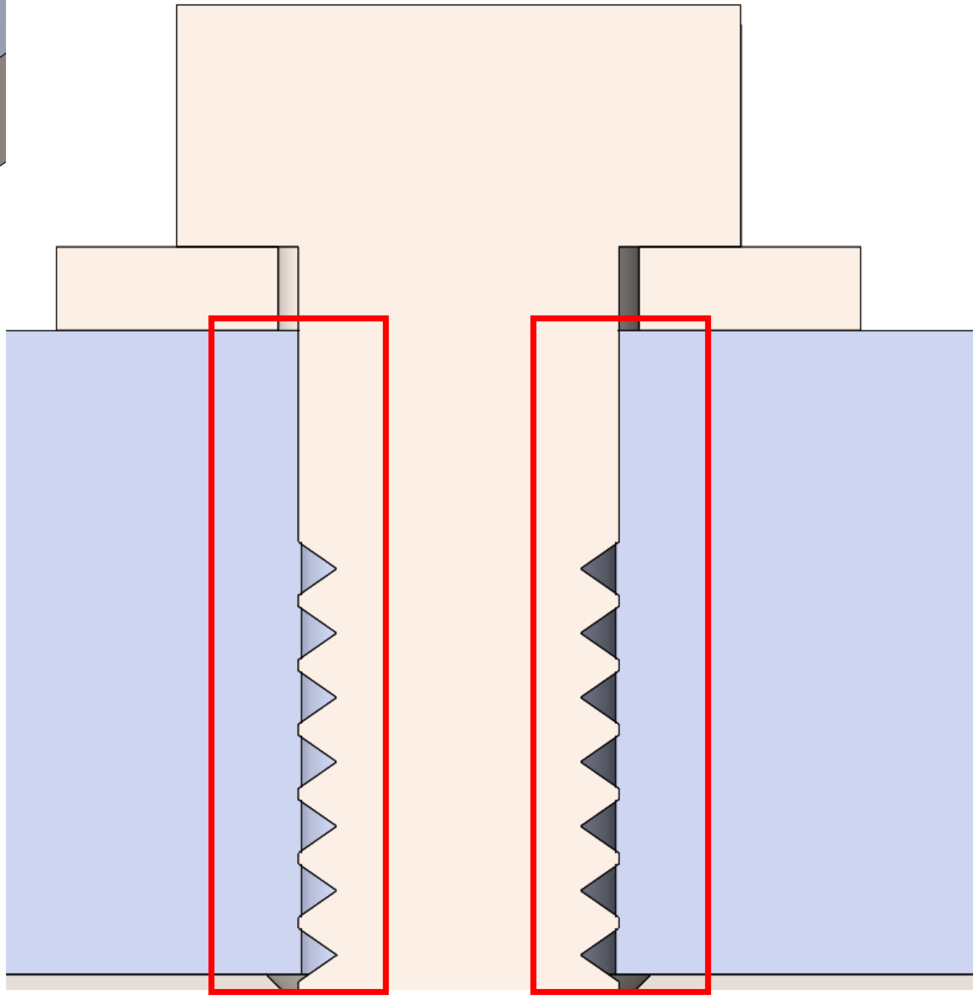
.2408 - .2489*

Drilling is imperfect...also needs tolerance!

.250 ± .005

This could lead to interference!





*Machinery's Handbook, 26th ed., pg. 1717*

# Case Study: Hole Tolerancing

Solution: Clearance hole sizes!

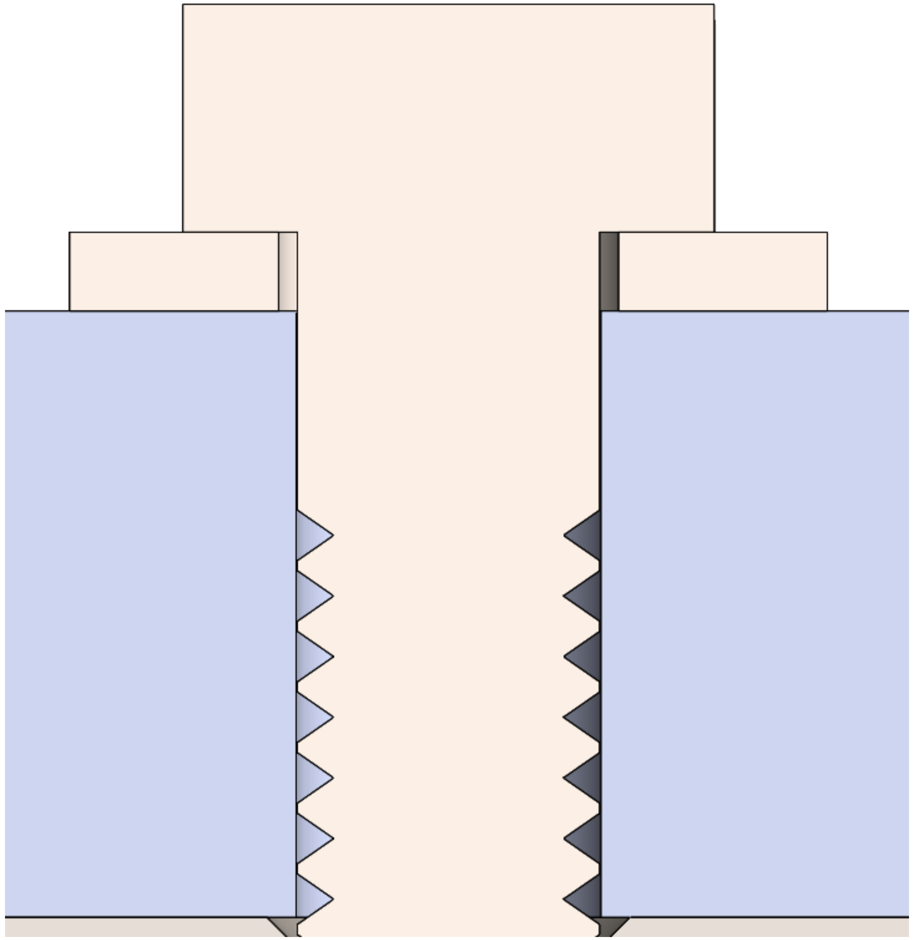
Class 2A ¼-20 UNC bolt:

.2408 - .2489

¼-20 clearance hole sizes*:

Close fit, .257

Free fit, .266



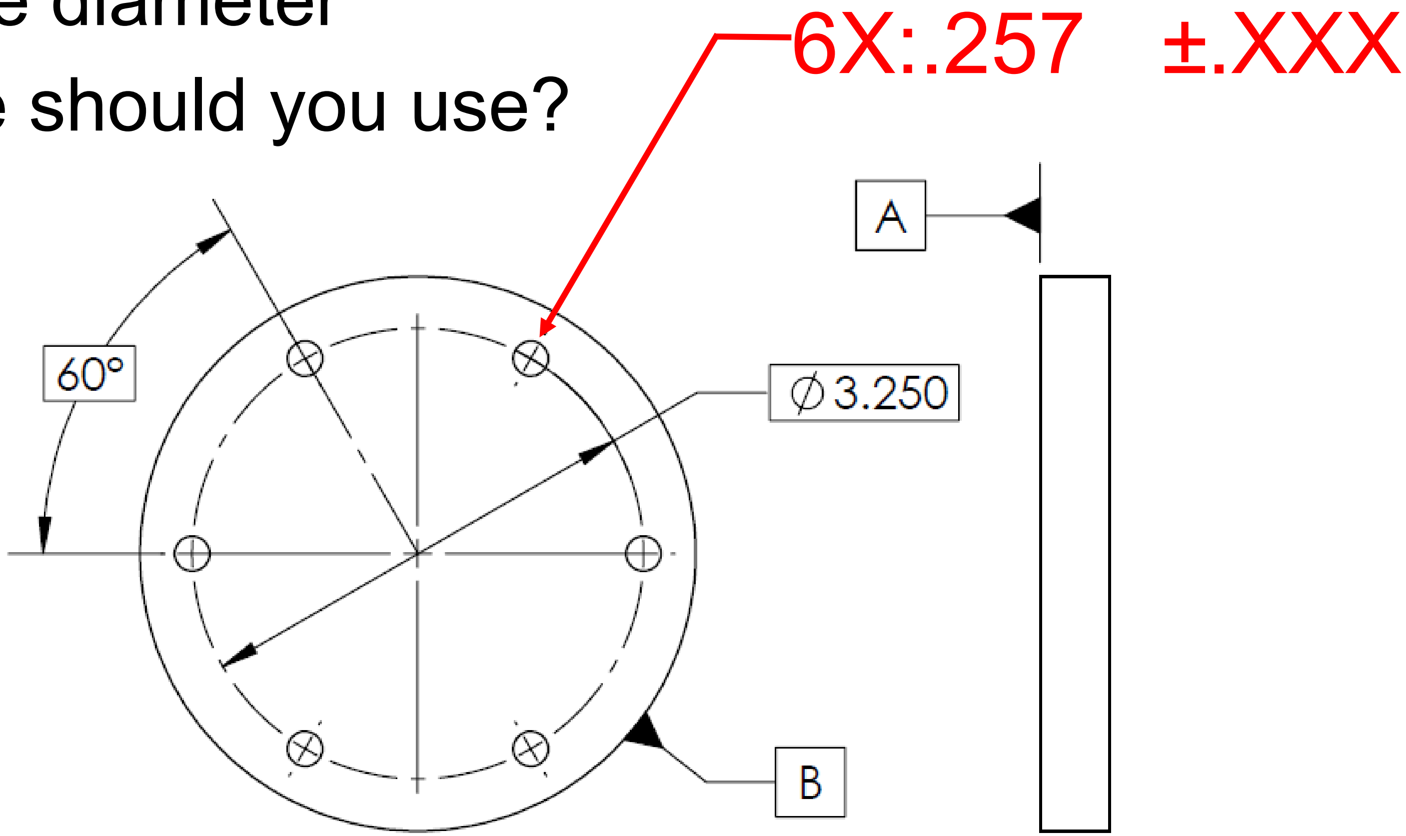
Even with tolerance .257±.005, bolts will always fit.

*Machinery's Handbook, 26th ed., pg. 1900

# Case Study: Hole Tolerancing

Hole diameter established!

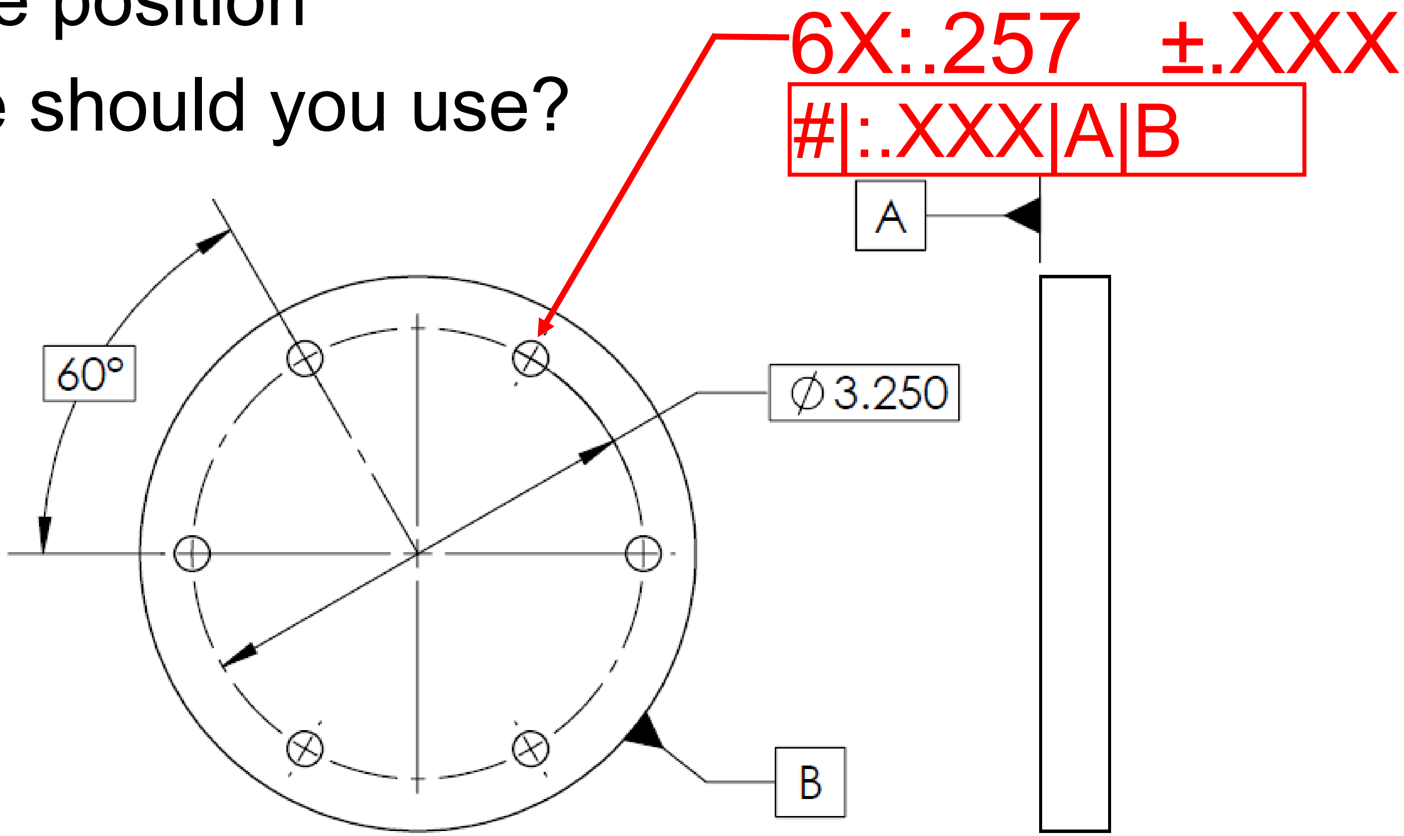
Next: What hole diameter tolerance should you use?



# Case Study: Hole Tolerancing

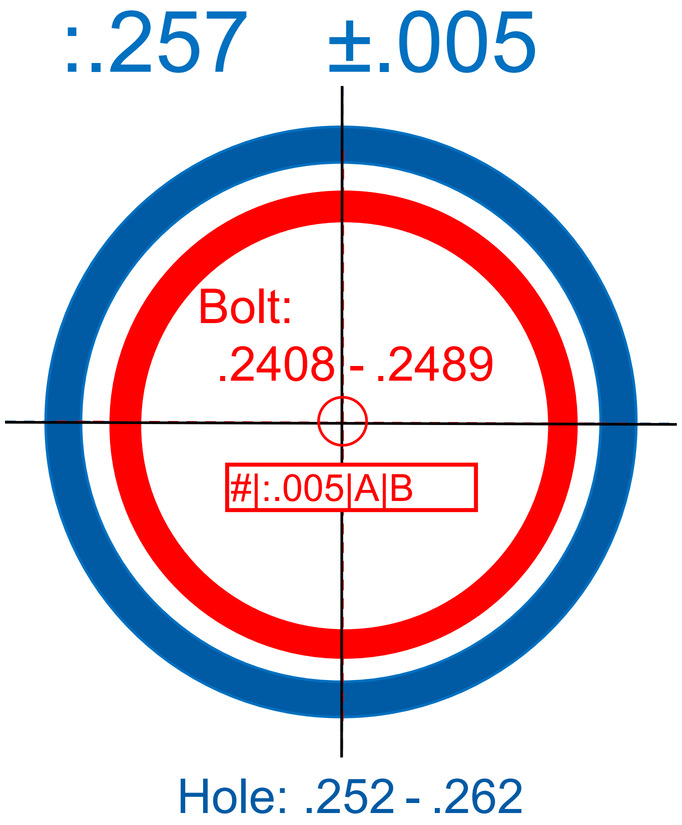
Closely related:

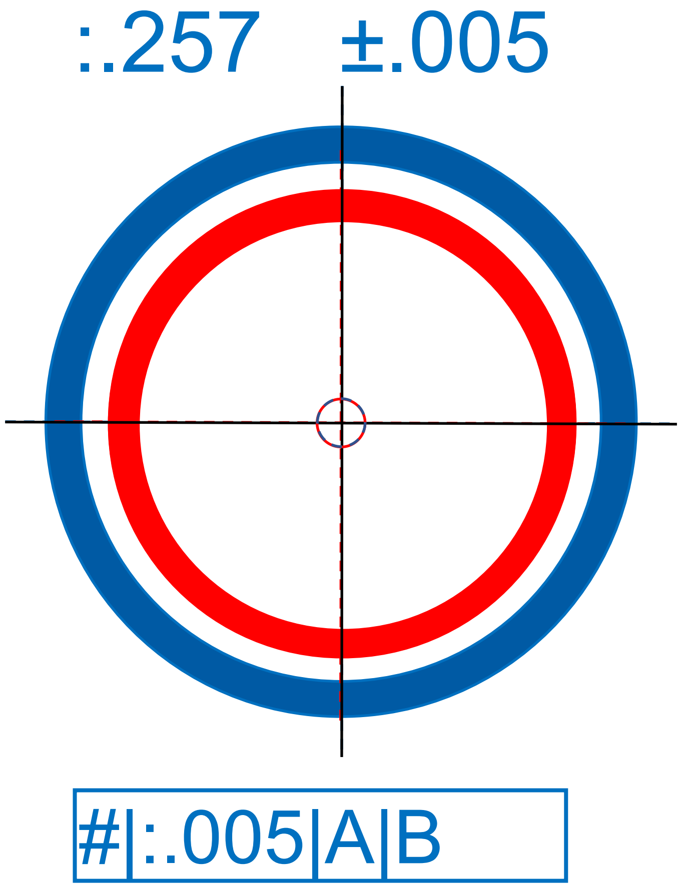
Next: What hole position
tolerance should you use?

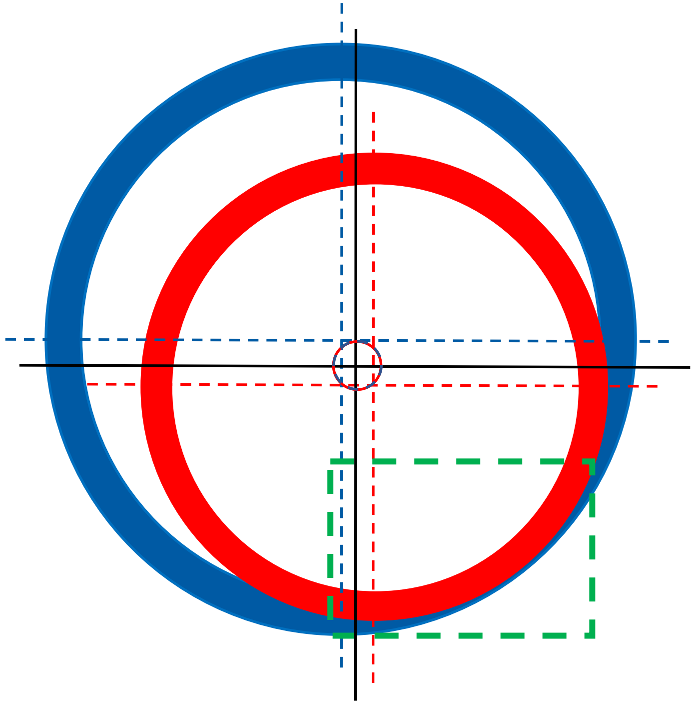


# Case Study: Hole Tolerancing

- Hole diameter and position are interrelated to function



½[(.252-.005) - (.2489+.005)] = -.0035

Radial Interference!

# Case Study: Hole Tolerancing

Georgia Tech

## Tolerance Components

Nominal Diameter

Diameter Tolerance

$$\tfrac{1}{2}[(\{.257 - .005\} - .005) - (.2489 + .005)] = -.0035$$

Smallest Hole Diameter

Largest Bolt Diameter

Hole Position Tolerance

Bolt Position Tolerance

Radial Interference!

## Changing tolerances

$$\tfrac{1}{2}[(\{.257 - .001\} - .001) - (.2489 + .005)] = .0006$$

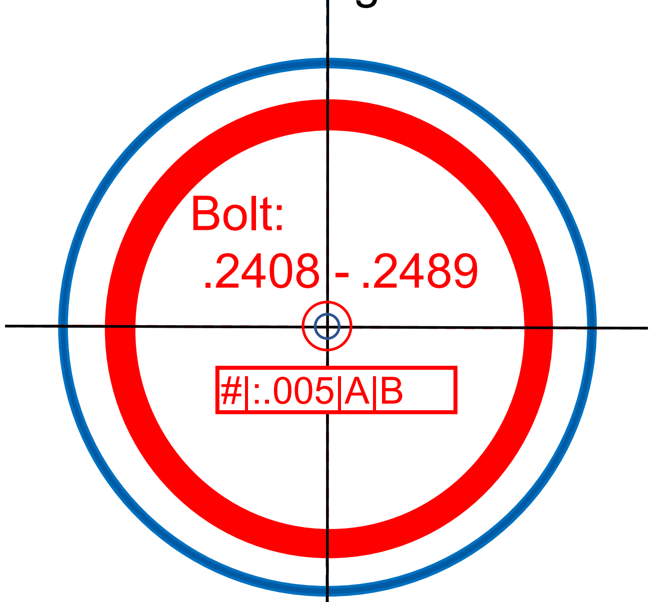
Radial Clearence!

Tighter Tolerances = More Cost

# Case Study: Hole Tolerancing

- Three choices (which is the most cost efficient?)

Decrease Hole Diameter Tolerance Range

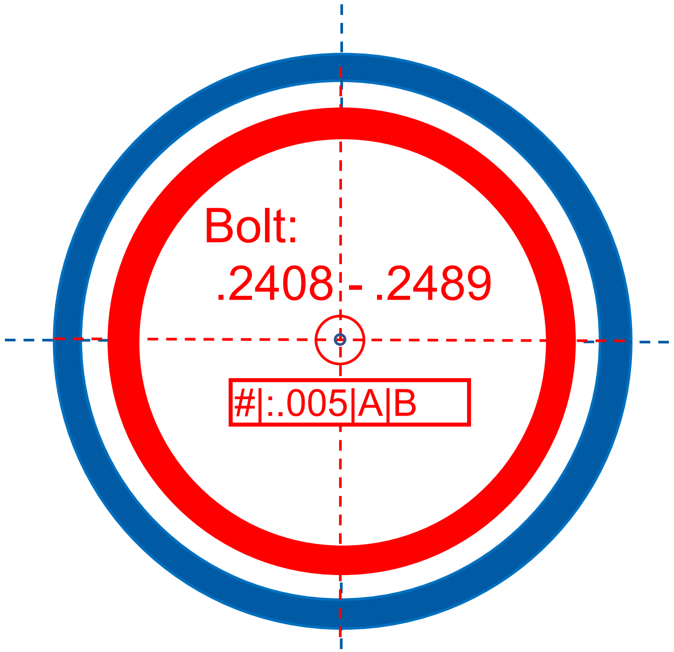



Hole: .252 - .262

:.257 ±.0005

#|: .0025|A|B

Decrease Hole Position Tolerance

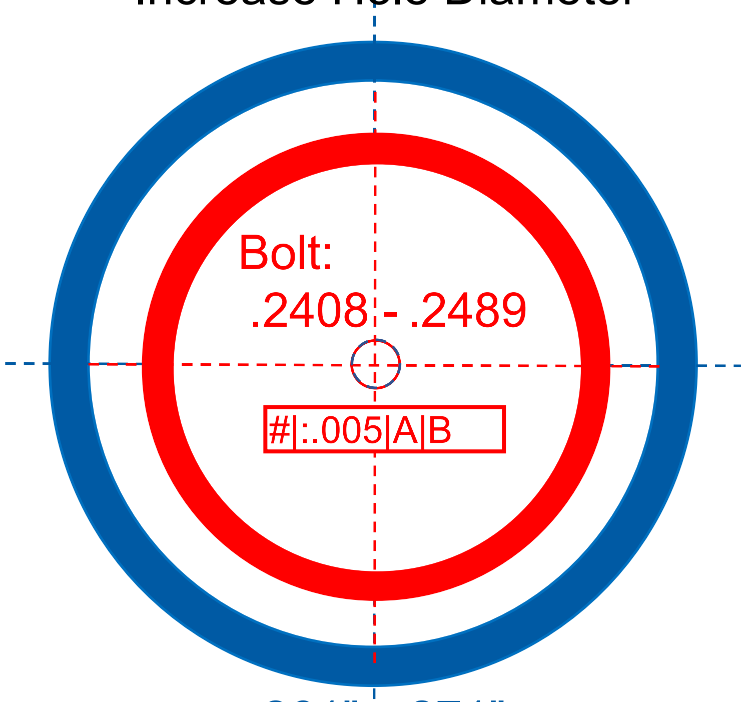



.252" - .262"

:.257 ±.0025

#|: .0005|A|B

Increase Hole Diameter



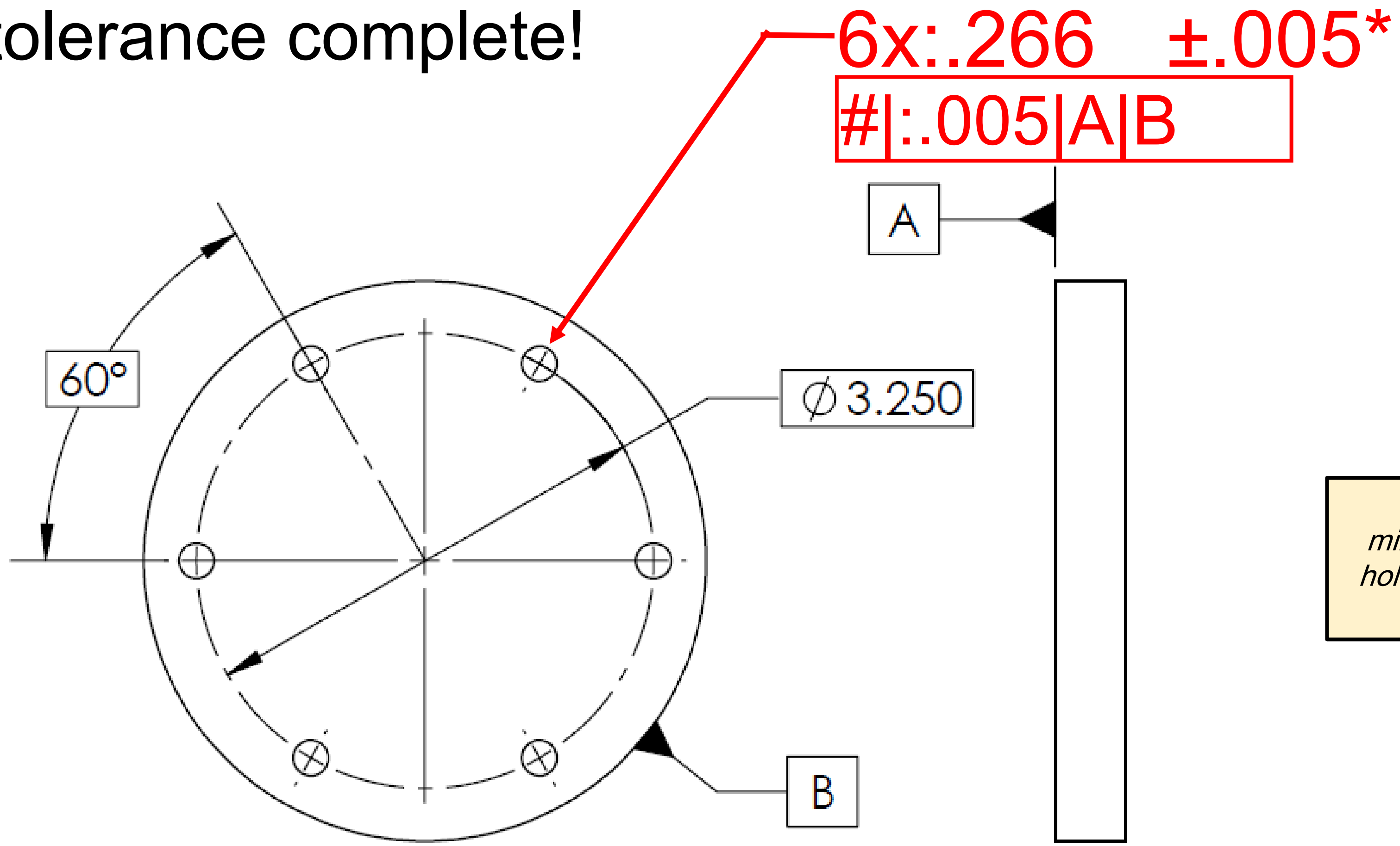

.261" - .271"

: .266±.005

#|:.005|A|B

# Case Study: Hole Tolerancing

## Bolt pattern tolerance complete!

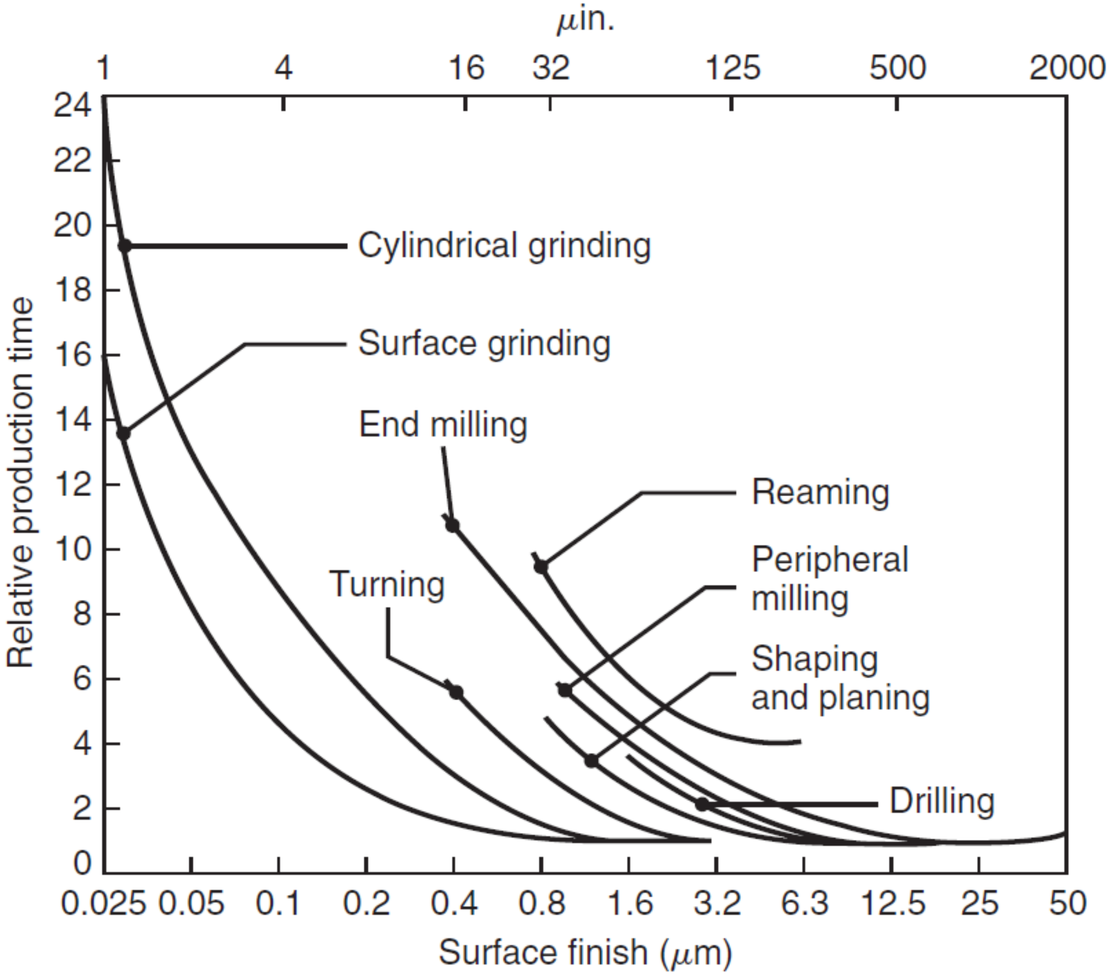



** .266 is listed as the minimum close fit clearance hole for a ¼fastener in ASME B18.2.8*

½[({.266 - .005} - .005) - (.2489 + .005)] = .0011 Radial Clearence!

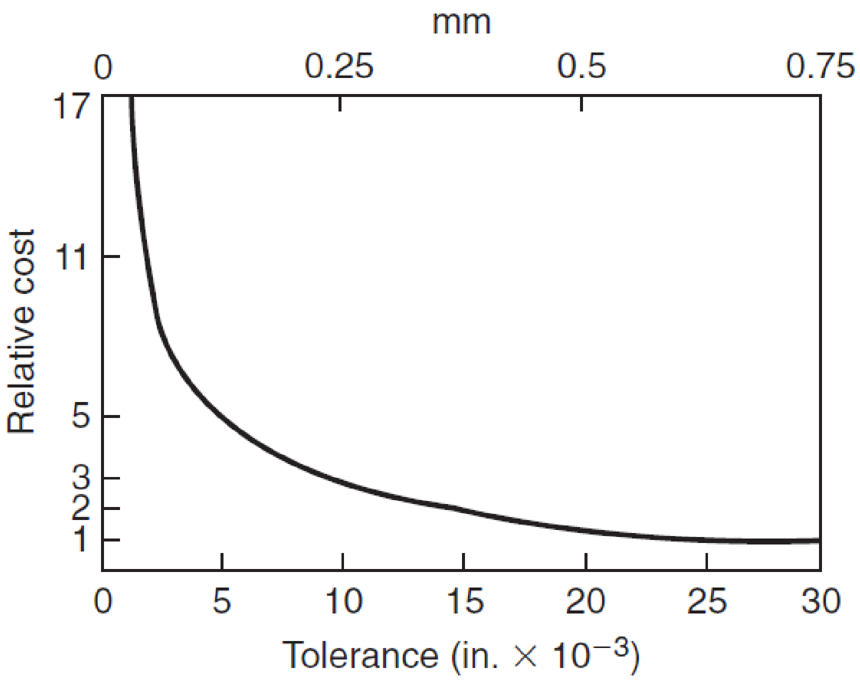
# Impacts of geometric specification on cost

<u>*Production time scales with surface finish*</u>




*Kalpakjian & Schmid/American Machinist*

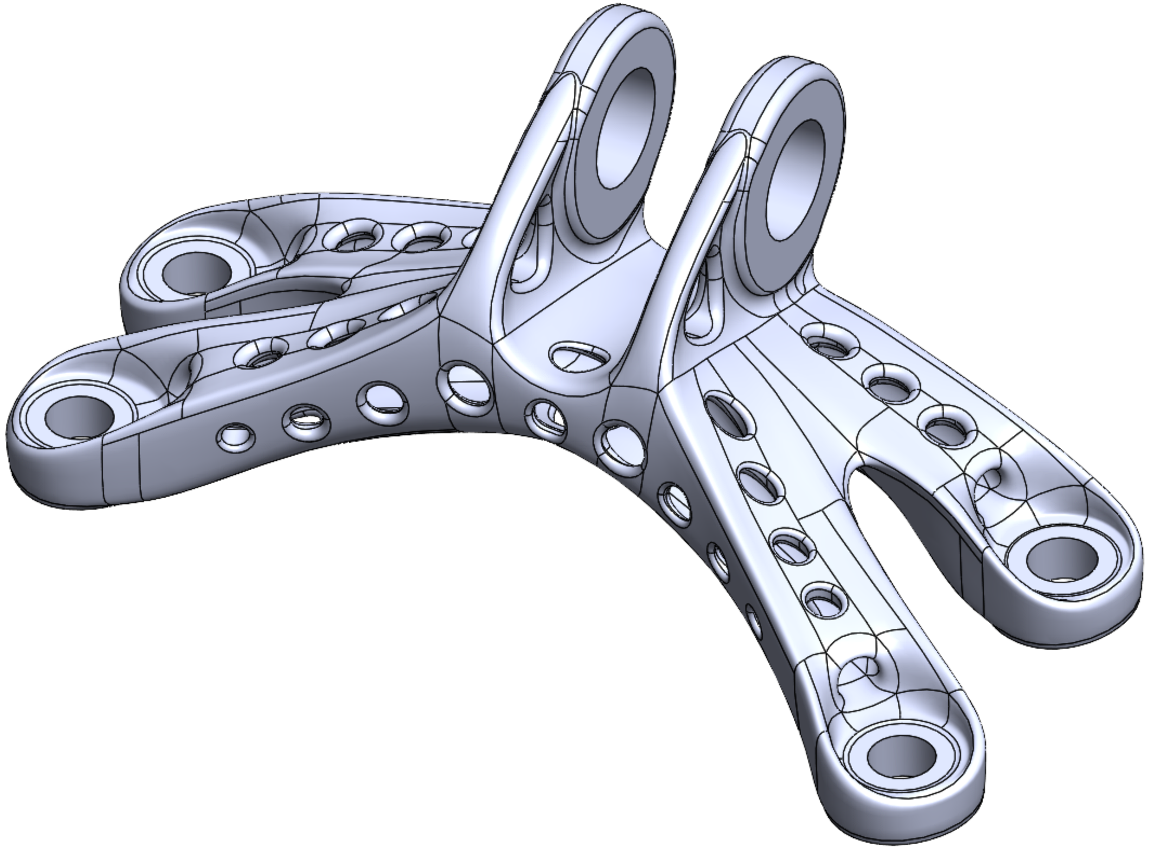
<u>*Cost scales with tolerance*</u>




*Kalpakjian & Schmid*

# Casestudy 2– Complex AM components

Georgia Tech

https://grabcad.com/library/spacehugger -1

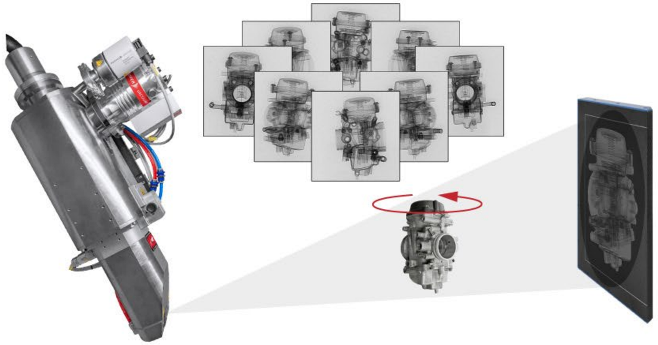
# High-Density Coordinate Measurement Systems

Georgia Tech

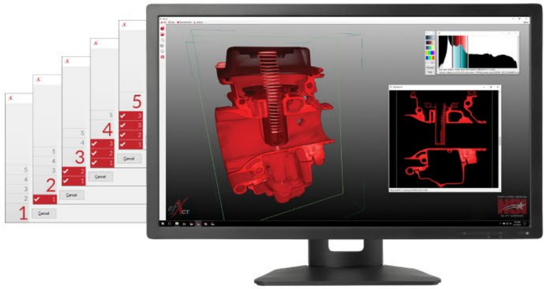
## Structured Light Scanning


*Zeiss – GOM Metrology*

## X-ray Computed Tomography

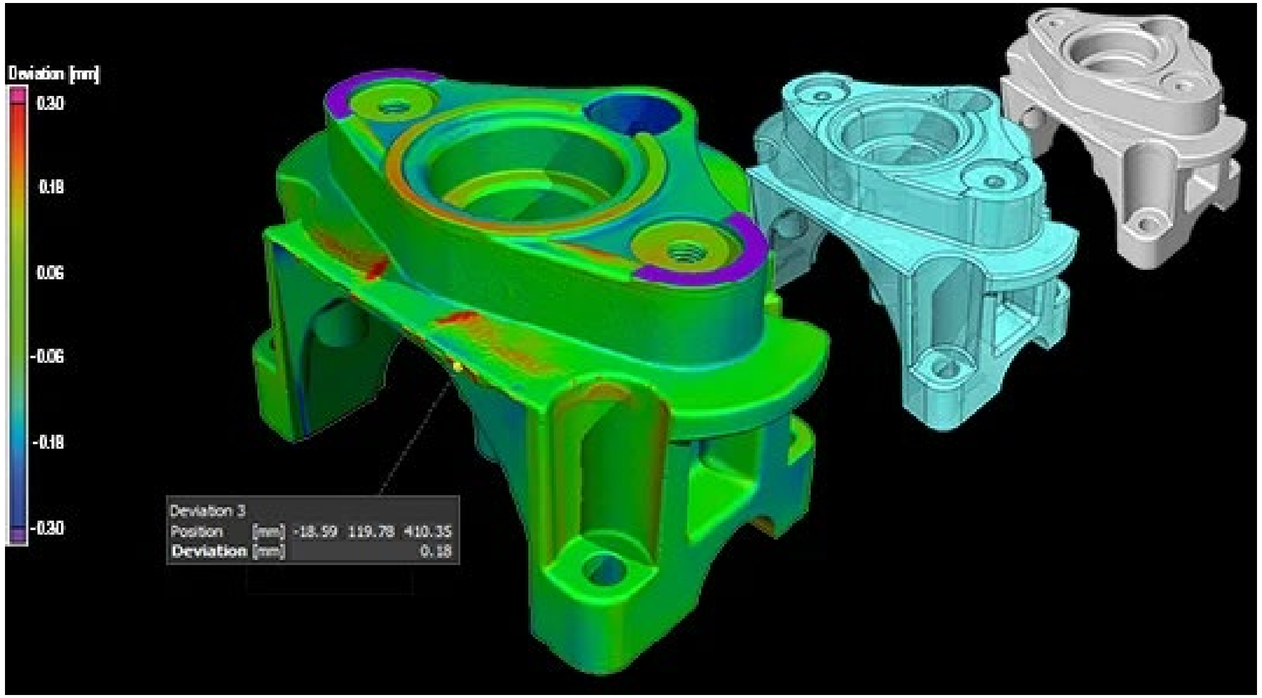
Computed Tomography Scan



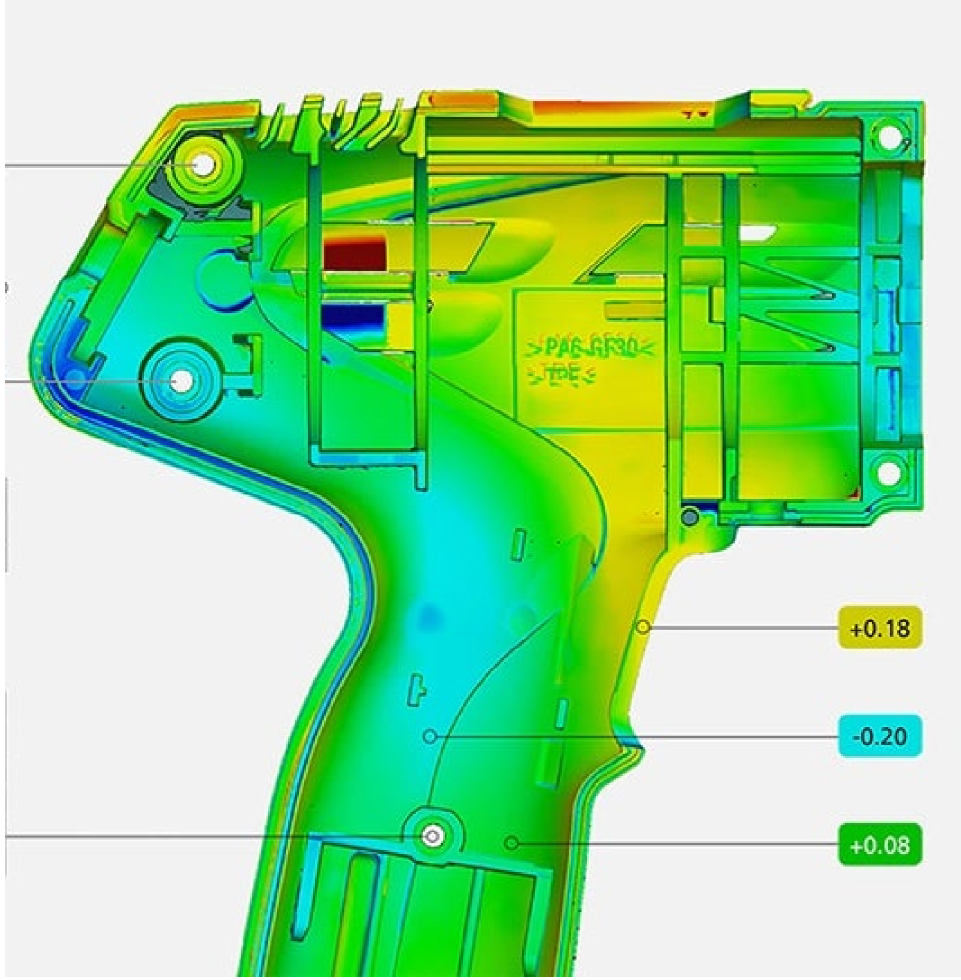
3D Model Reconstruction

*North Star Imaging*

# Nominal / Actual Comparisons



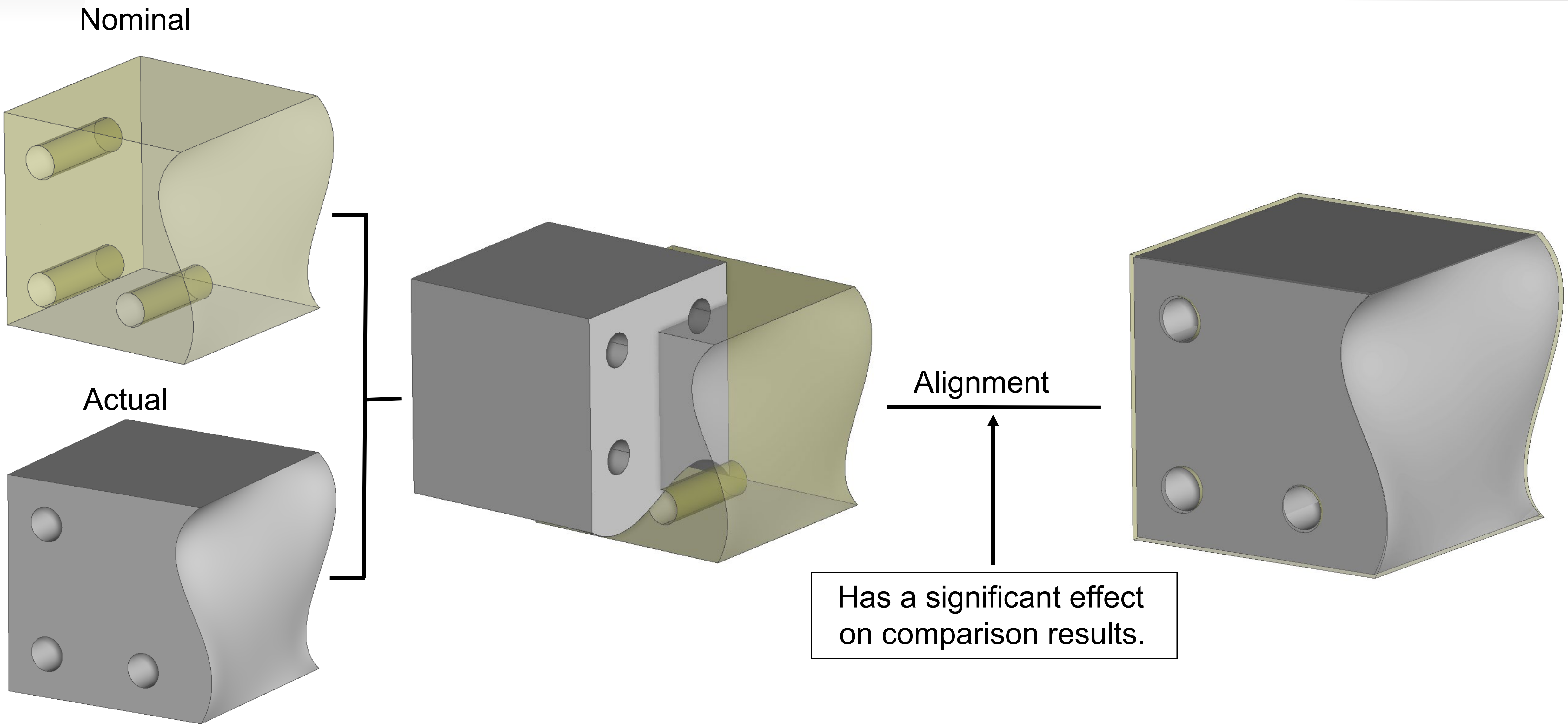

*Volume Graphics*




*Zeiss – GOM Metrology*

Pretty pictures, but are they the measurements you want?

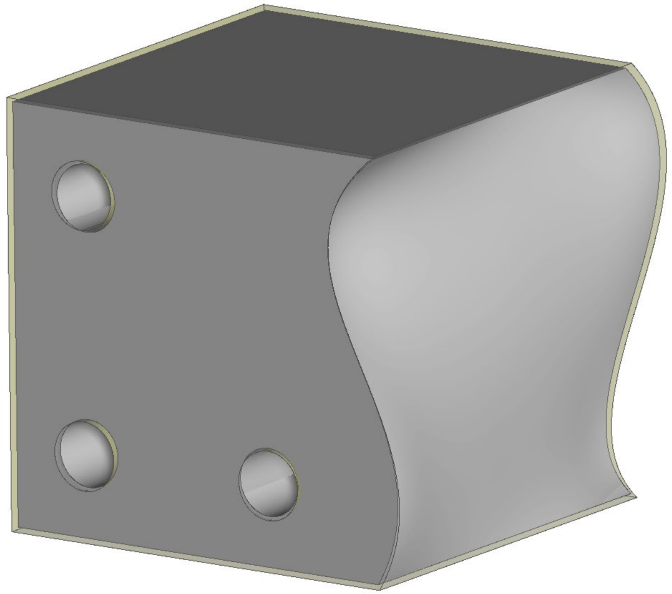
# Nominal / Actual Alignment



# Nominal / Actual Alignment

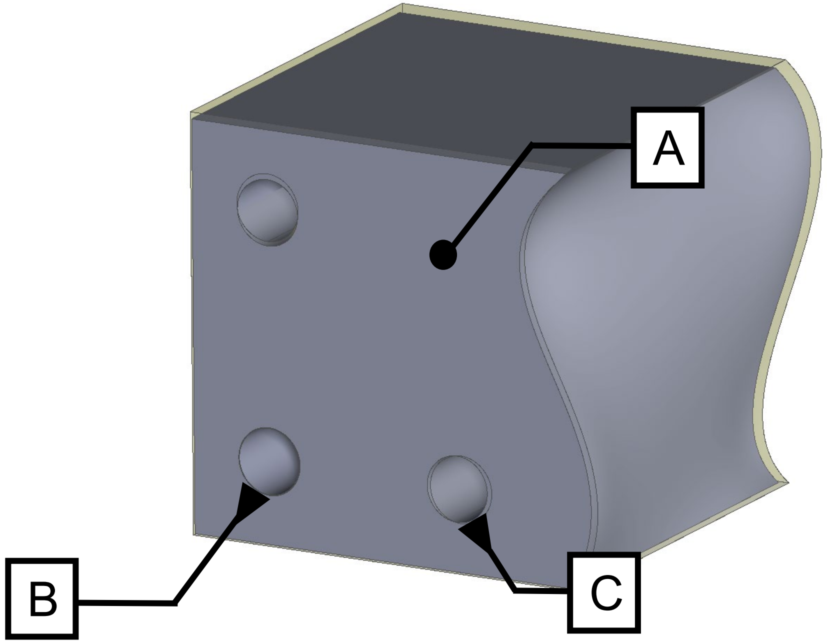
## Best Fit Alignment

- Minimizes total deviation between two models
- Highly subject to settings



## Datum Based Alignment

- Utilizes datums for alignment
- Provides meaningful geometric data



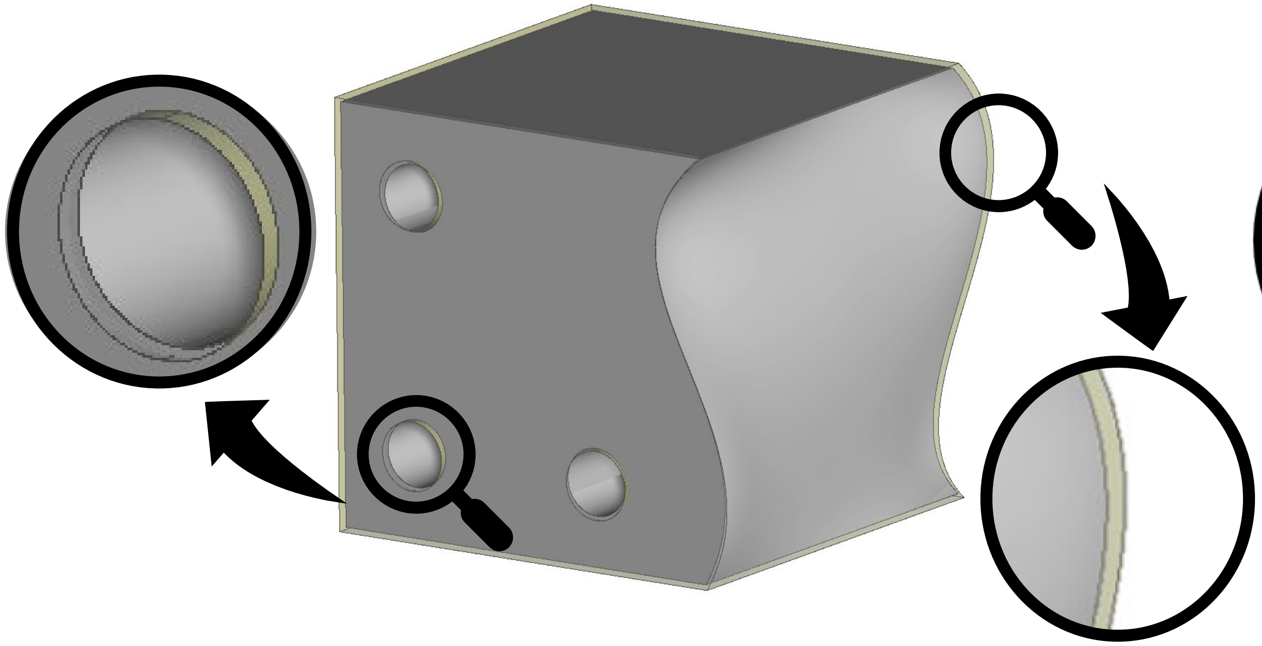
# Nominal / Actual Alignment

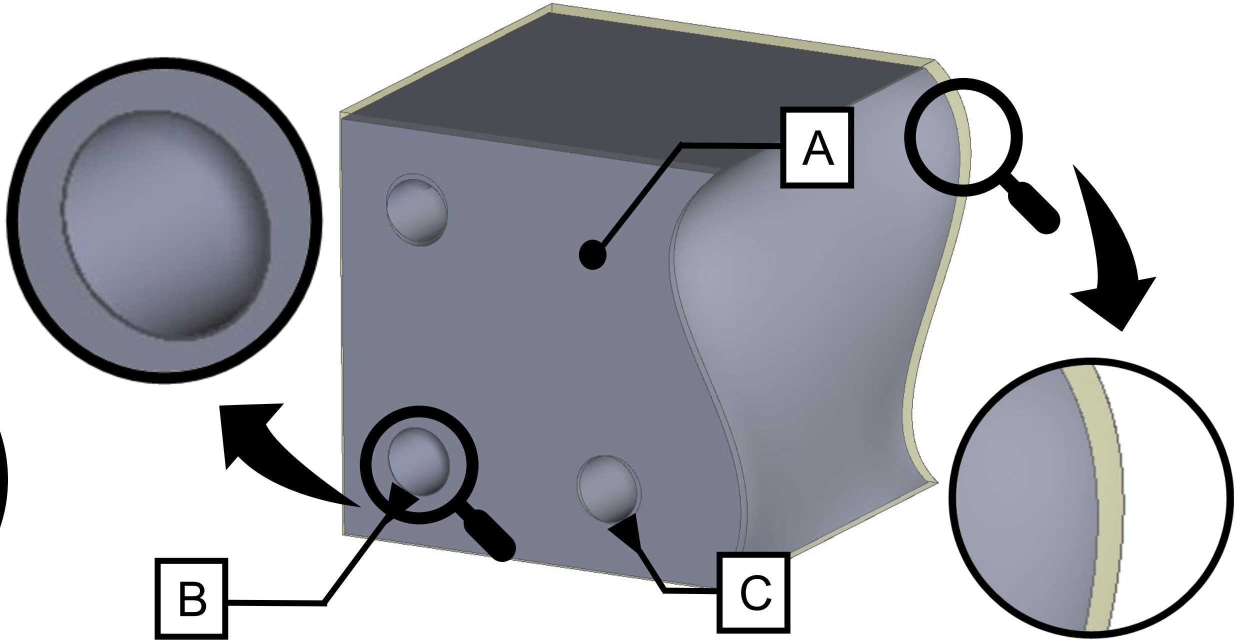
Best Fit Alignment

- Minimizes total deviation between two models
- Highly subject to settings

Datum Based Alignment

- Utilizes datums for alignment
- Provides meaningful geometric data

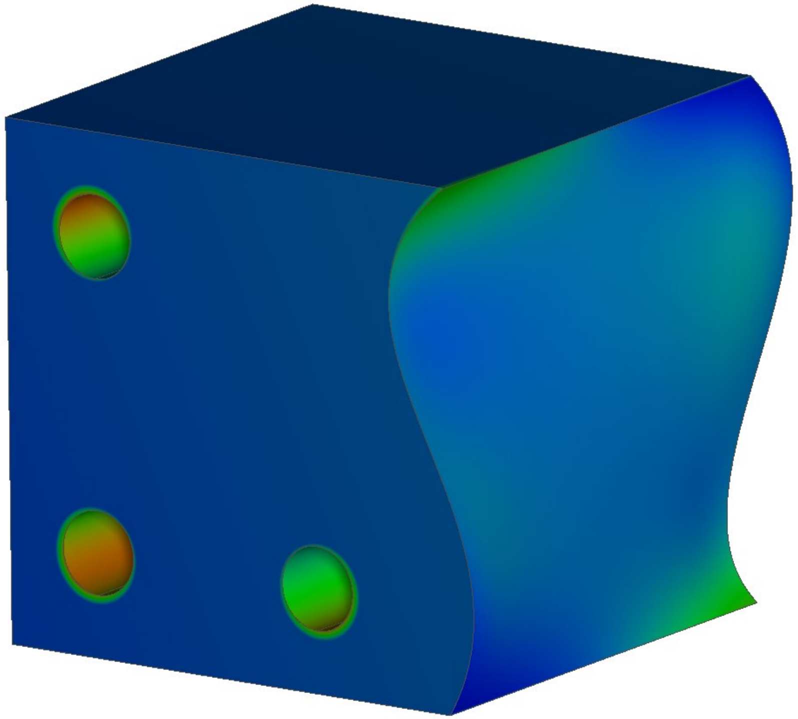




# Nominal / Actual Alignment

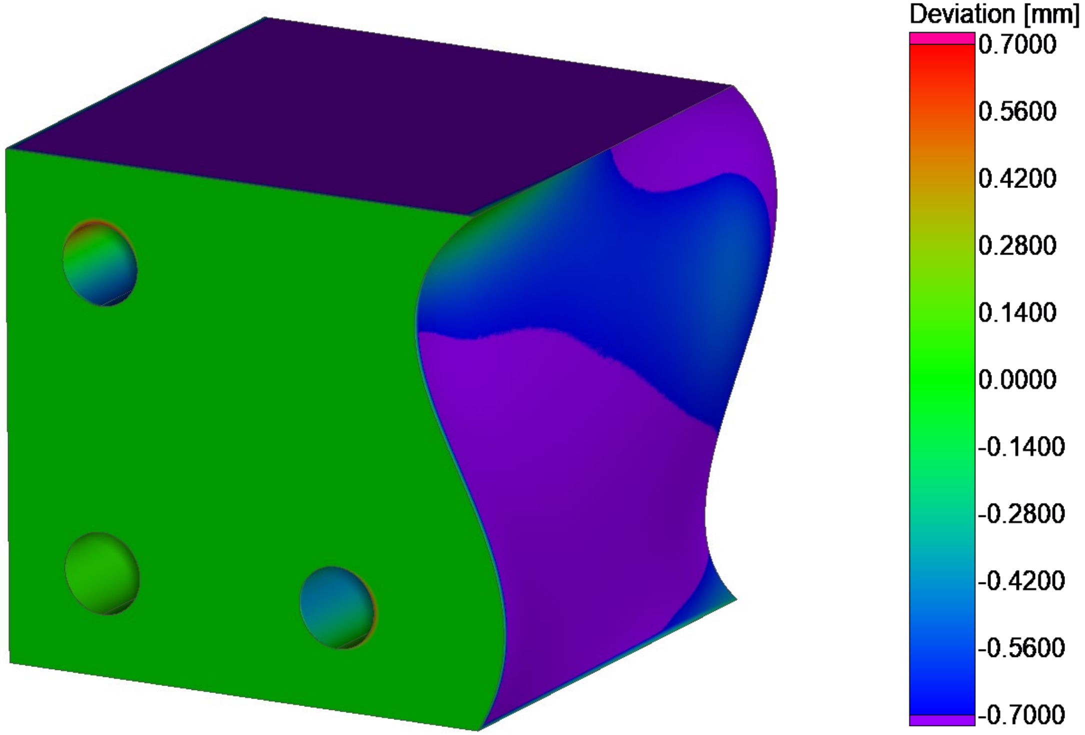
## Best Fit Alignment

- Minimizes total deviation between two models
- Highly subject to settings

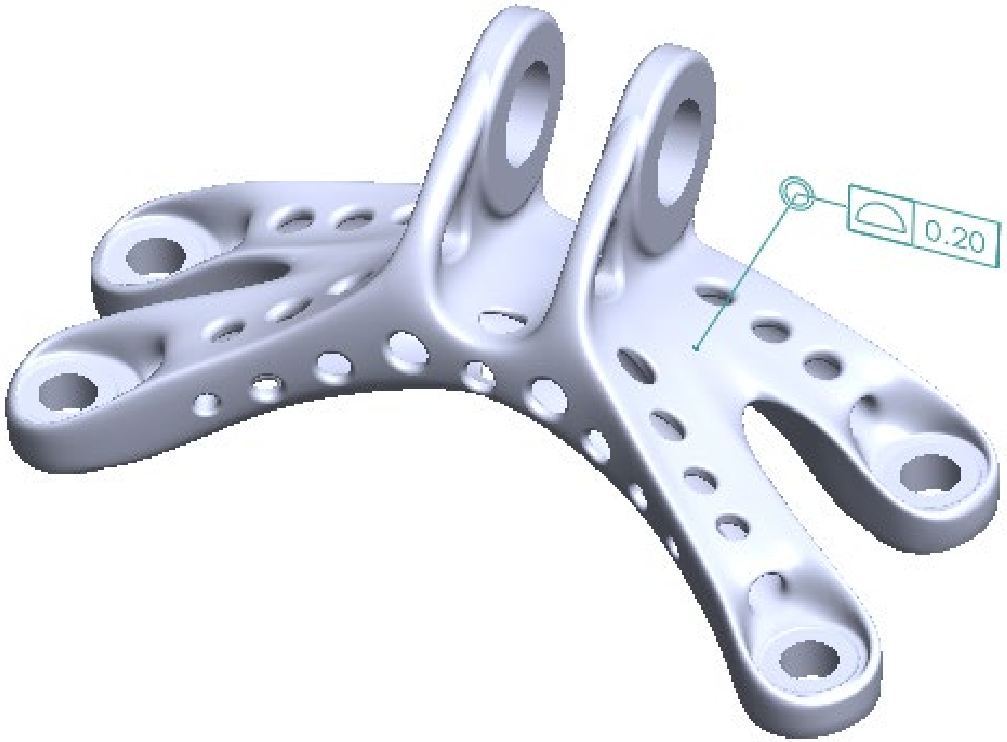


## Datum Based Alignment

- Utilizes datums for alignment
- Provides meaningful geometric data

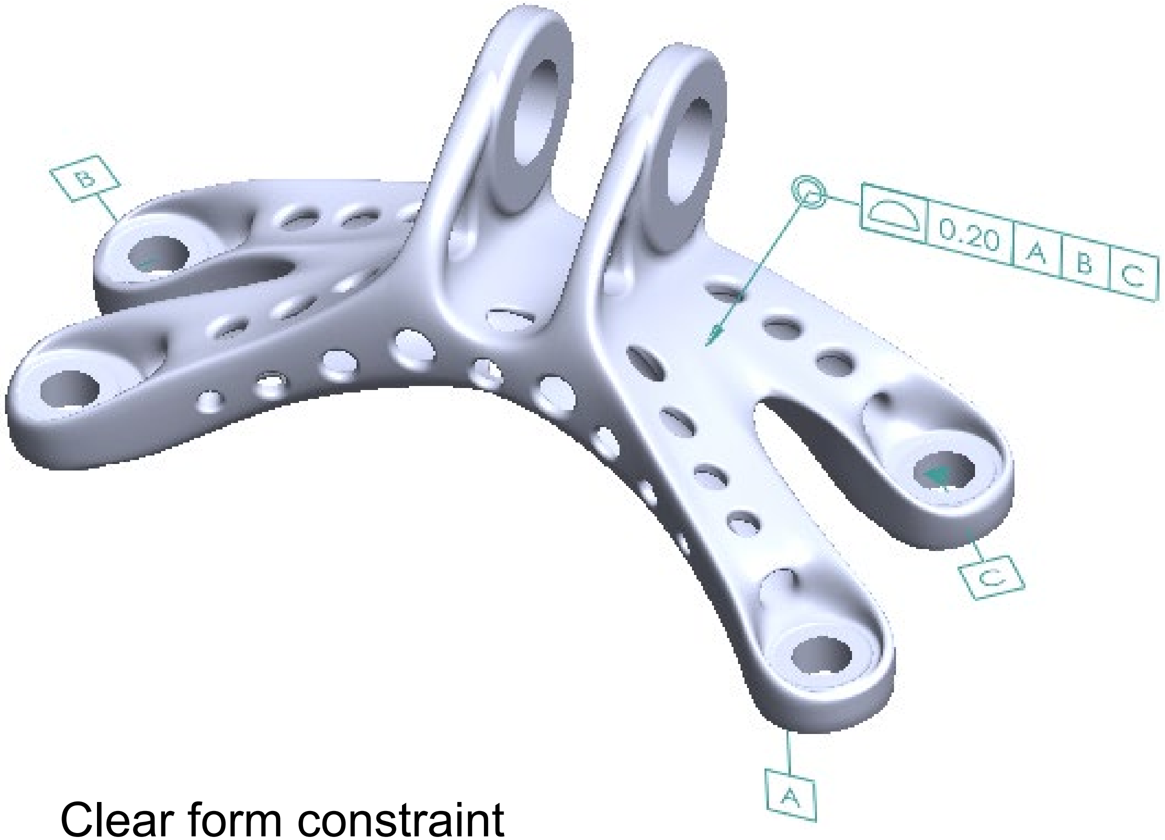


# Model-based definition (MBD)




Ambiguous form constraints

- No clear functional requirements

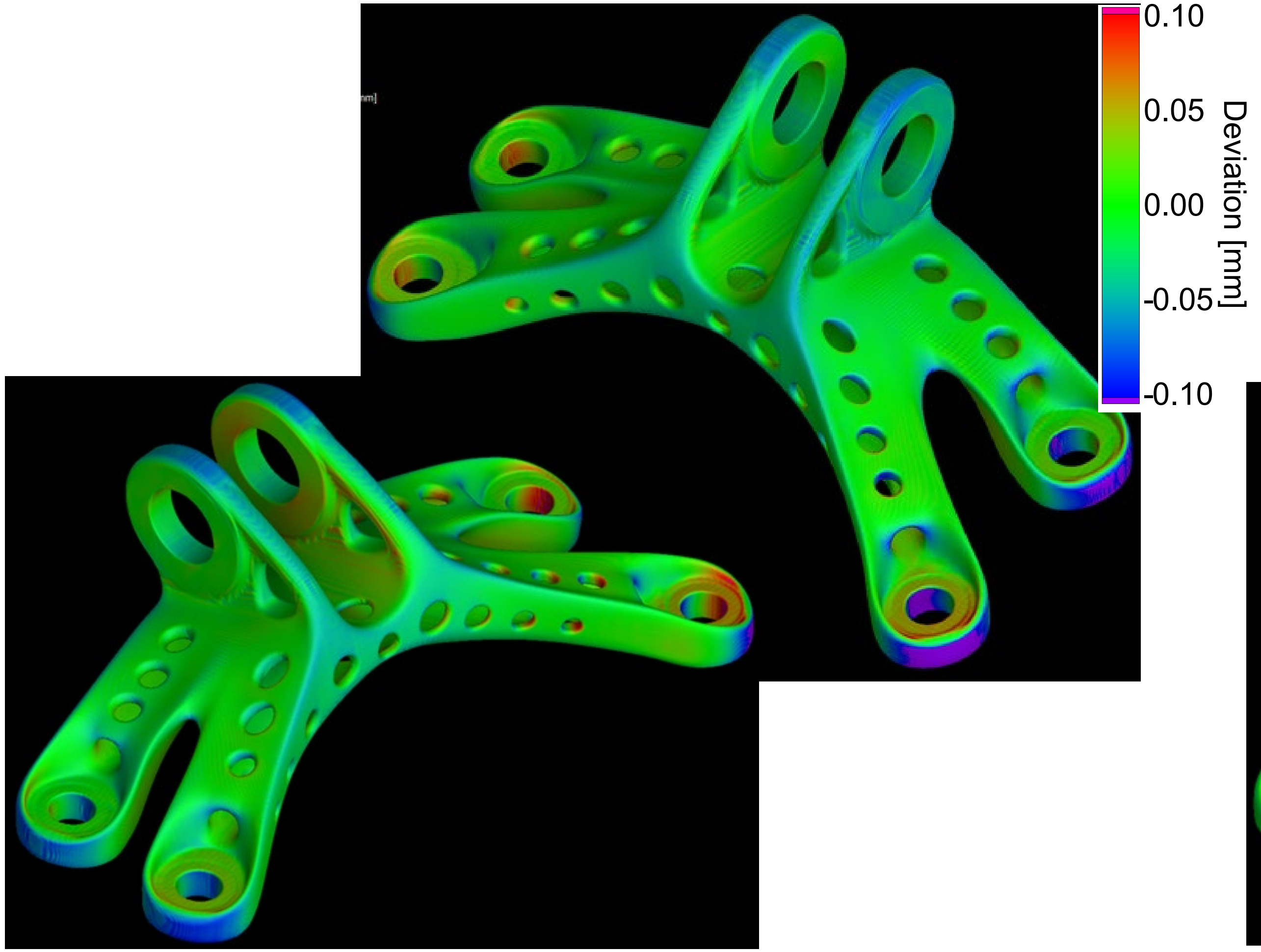



Clear form constraint

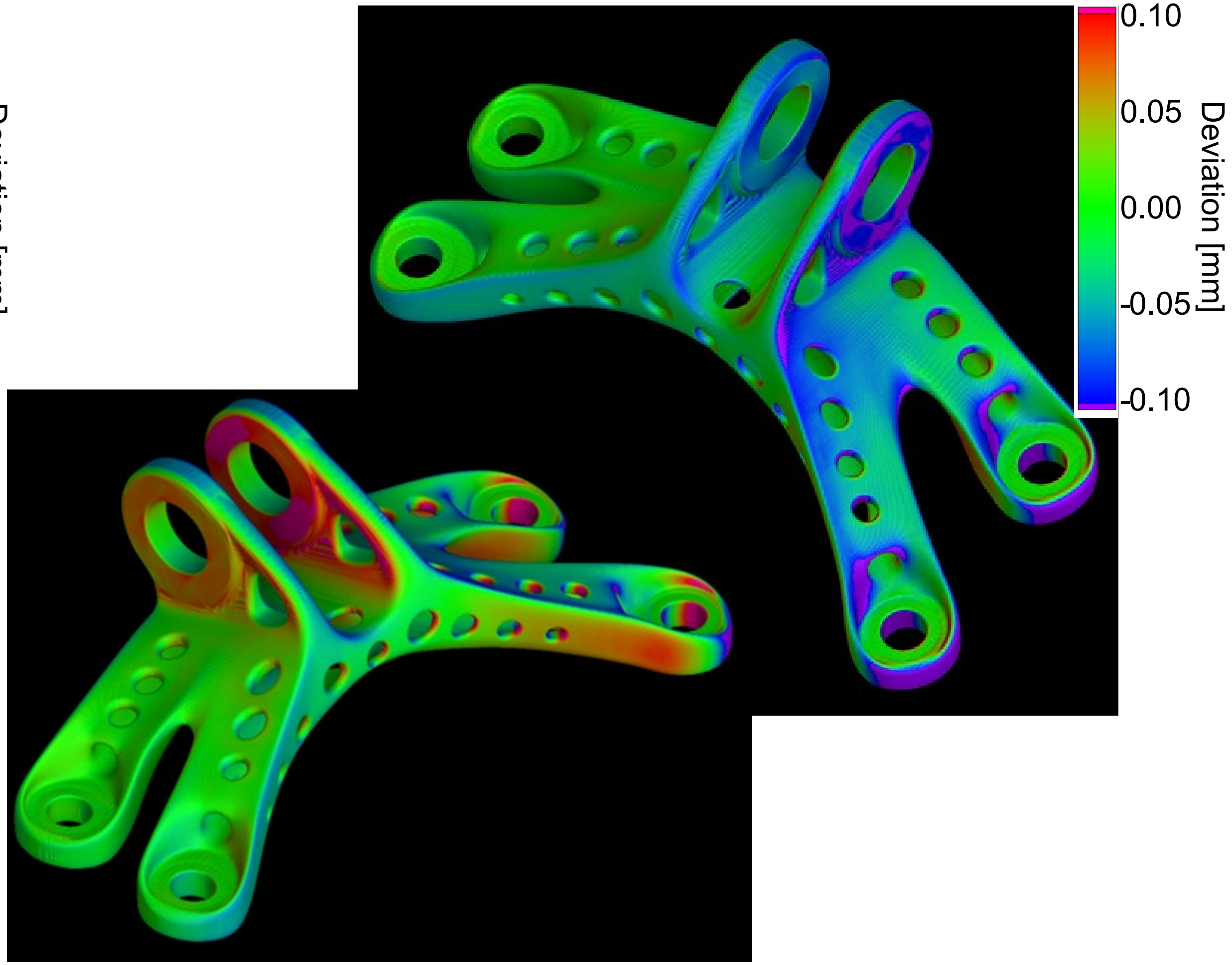
- Datum precedence indicates function

# Nominal / Actual Alignment

- Best Fit




- Datum Based

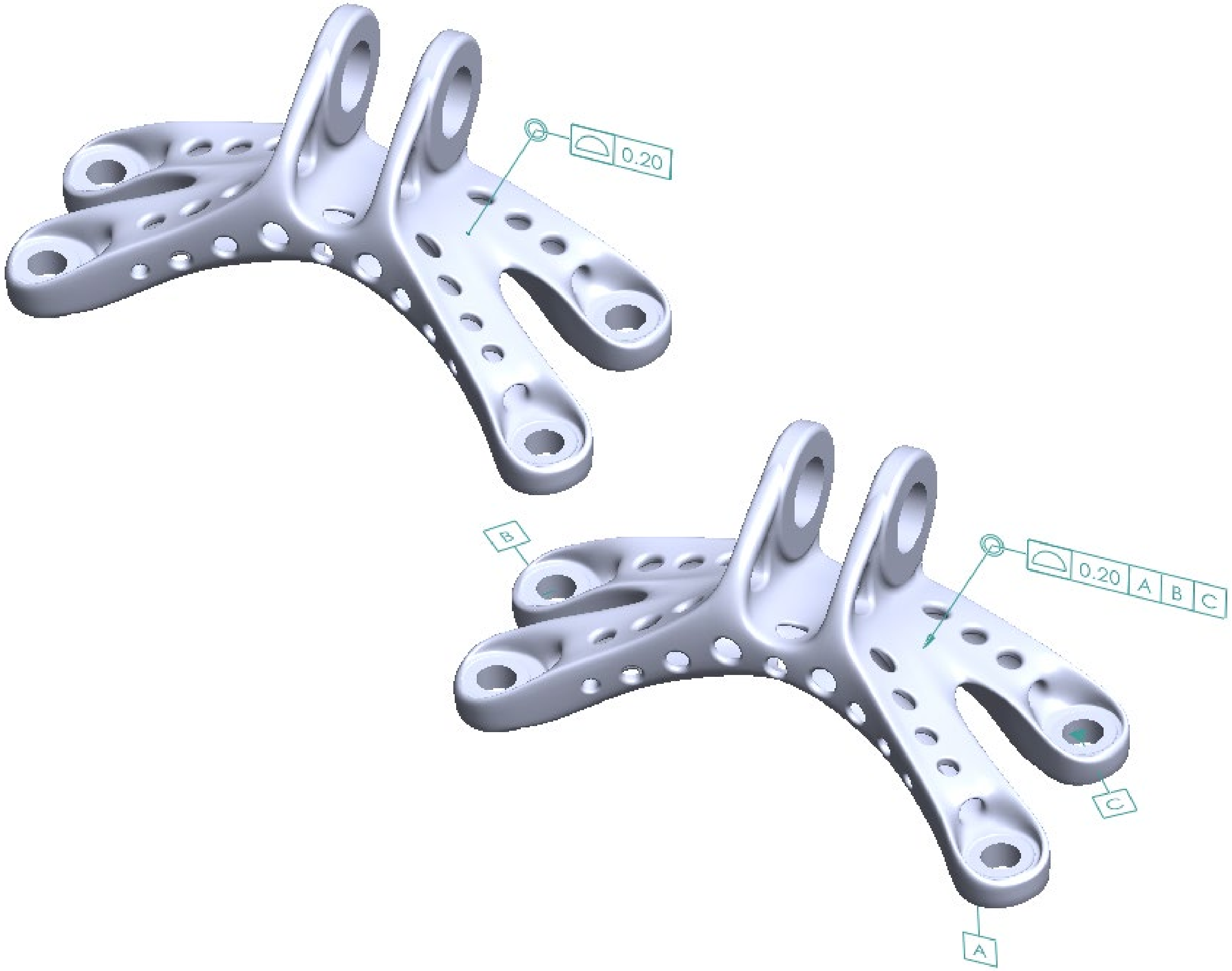



Nominal/Actual can be useful...if defined well!

# Dimensioning & tolerancing systems

Why use GD&T/GPS?

- **Functional** – related to component functionality
- **Unambiguous** – clearly defined and standardized
- **Inspectable** – specifications relate to inspection methods

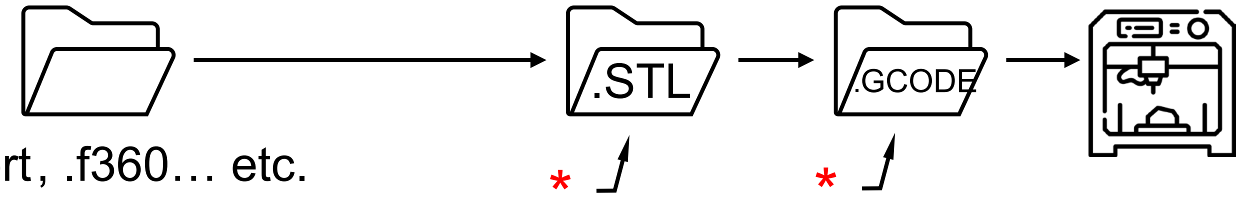


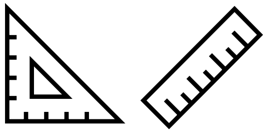
# Case study ~~3~~ Imprecision in digital manufacturing

Georgia Tech

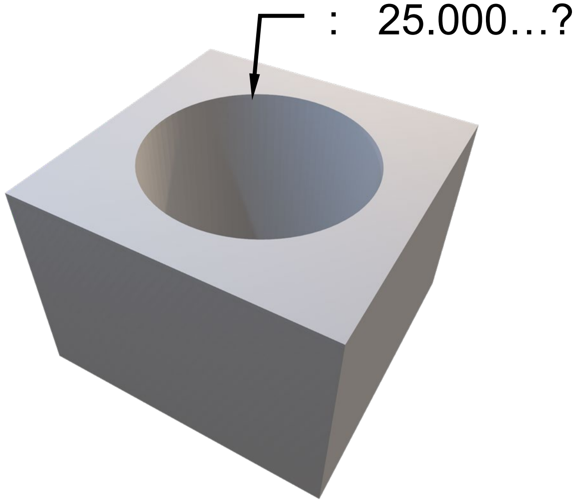
*Just because it's <u>digital</u> doesn't mean the manufacturing process is perfect*

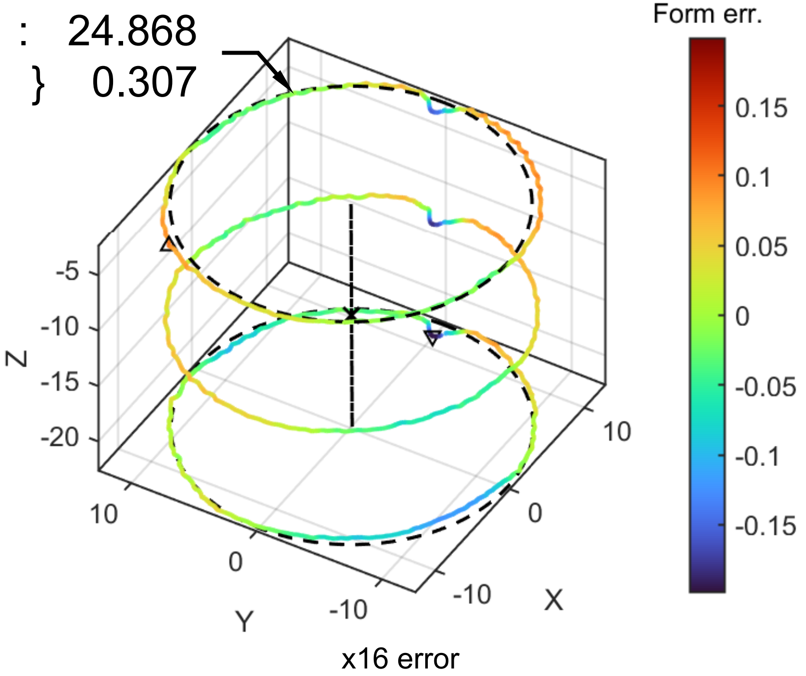
CAD → FFF additive manufacturing → Dimensional inspection



.sldprt, .f360… etc.

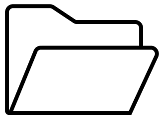


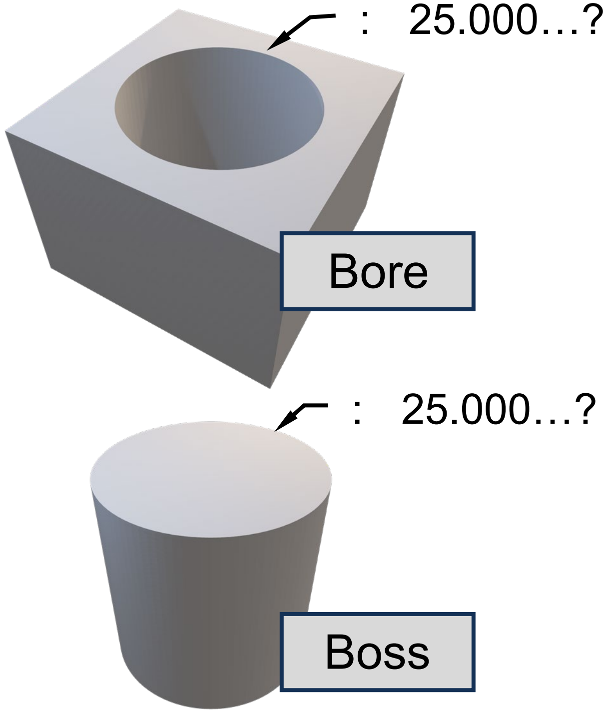
# Additive manufacturing example

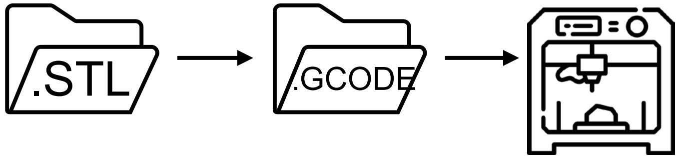
CAD

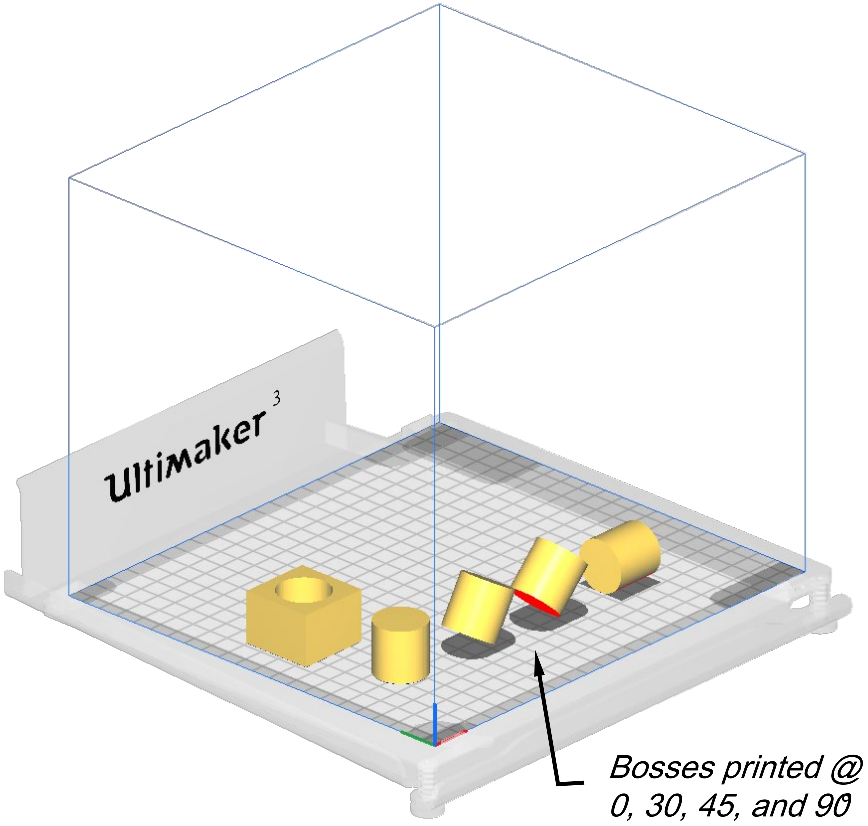
FFF additive manufacturing

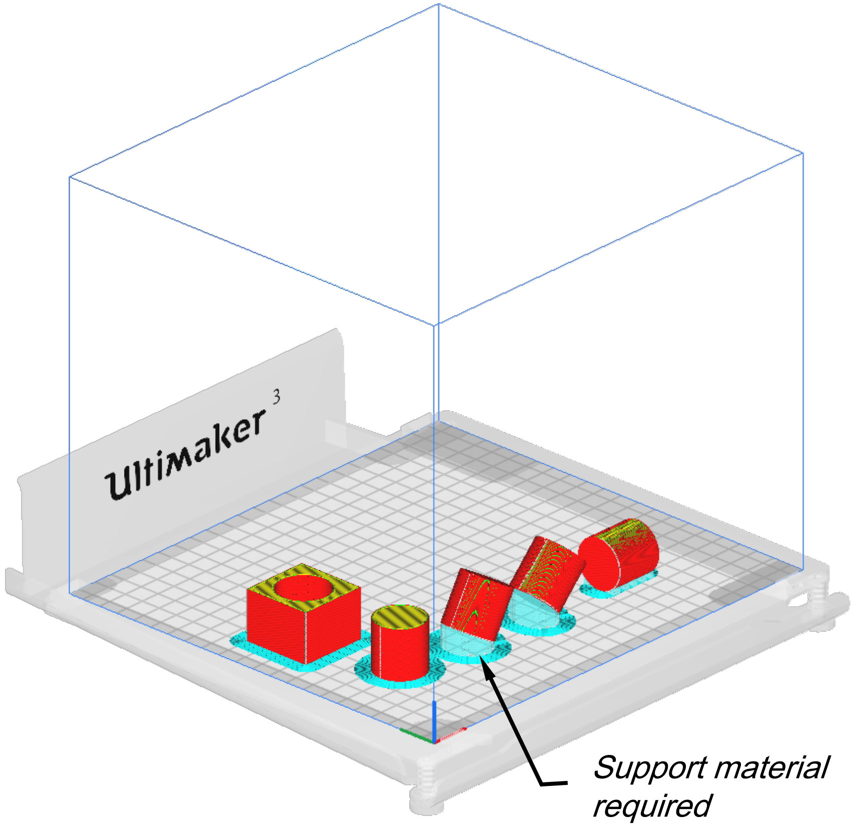


.sldprt, .f360… etc.



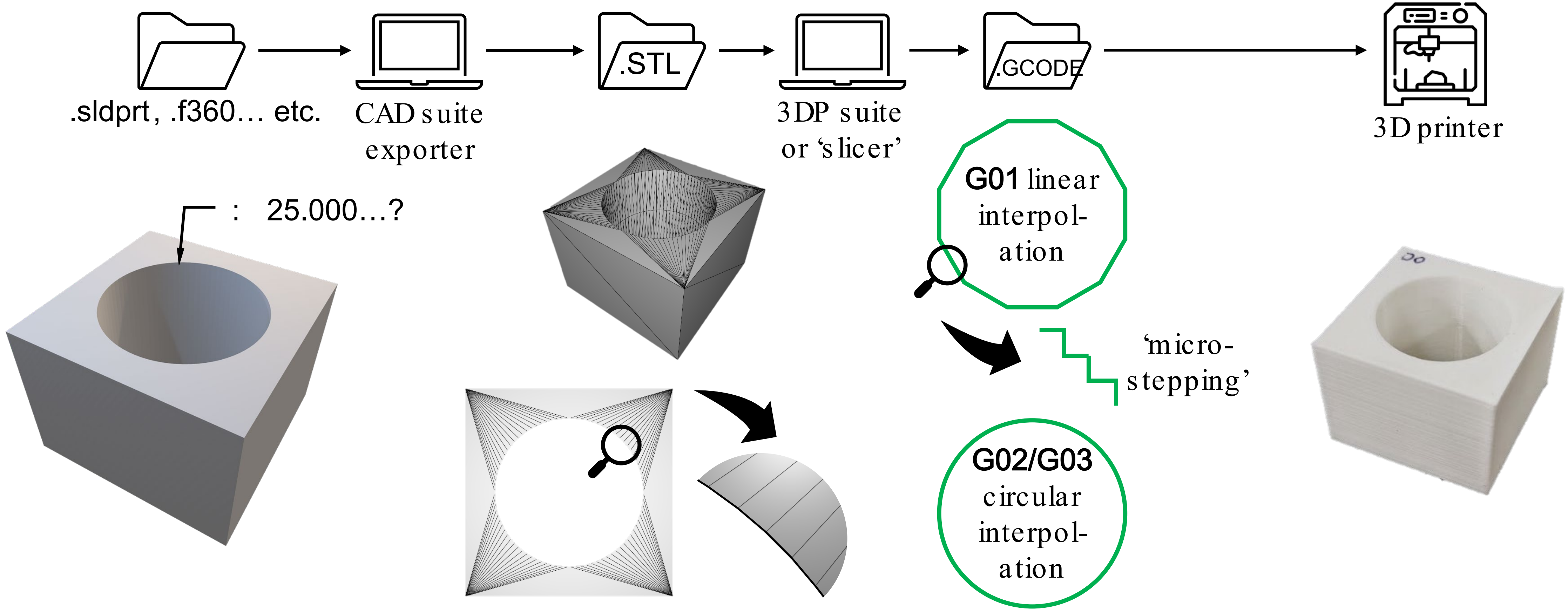



# Sources of error in digital manufacturing

CAD ⇨ FFF additive manufacturing




*CAD normally uses ‘brep’ or ‘boundary representation’ that models shapes precisely with math.*

*.STL files are tessellations, and only contain flat faces with straight edges between them.*

*G02/G03 circular interpolation is used in machine tools, where .STL files are not the driving model.*

*Physical error: Thermal expansion, tool/nozzle offset, lead screws, axes kinematics…*

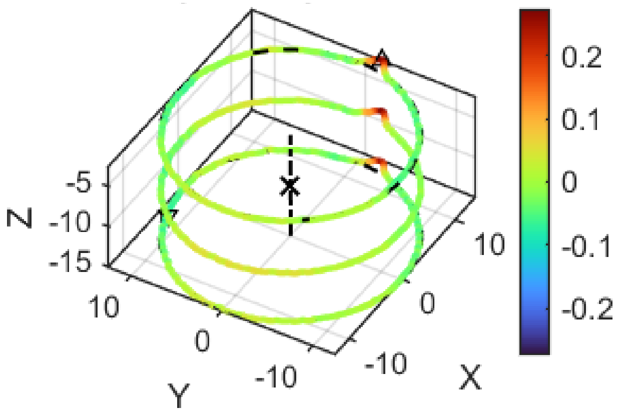
# Manufacturing approach determines dimensional error

Designer specification informs manufacturing approach!

Ø24.87, 0.31

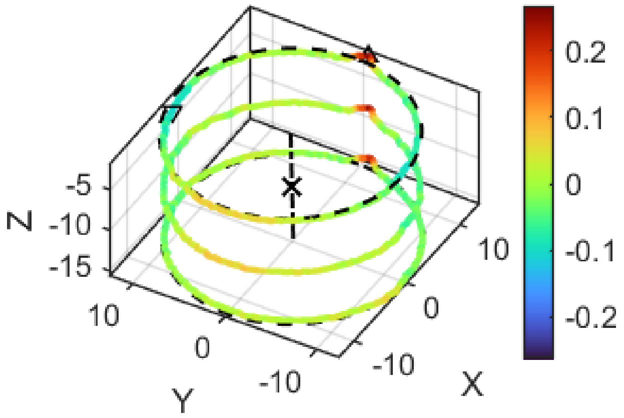

Axis @ 0° to B dir.

Ø24.86, 0.36



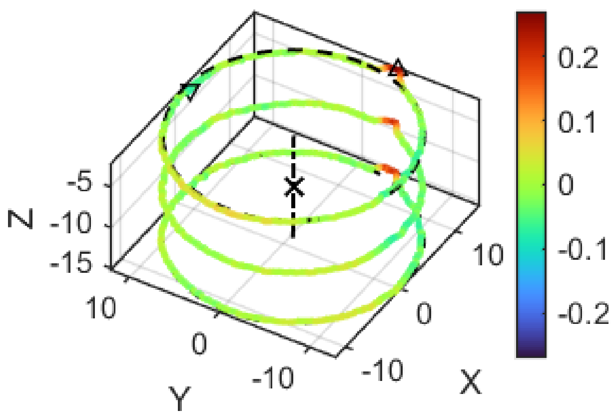

Axis @ 30° to B dir.

Ø24.87, 0.38



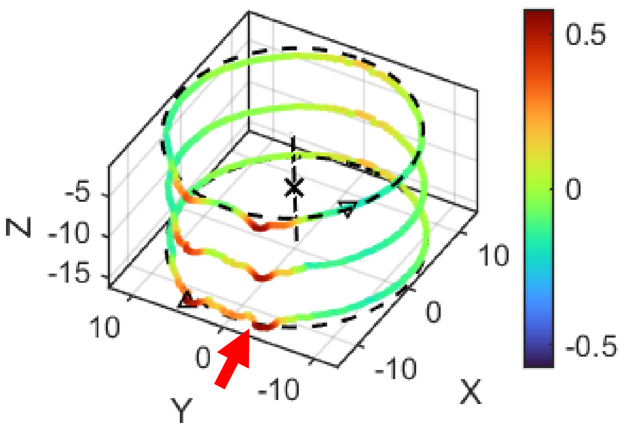

Axis @ 45° to B dir.

Ø24.87, 0.35




Axis @ 90° to B dir.

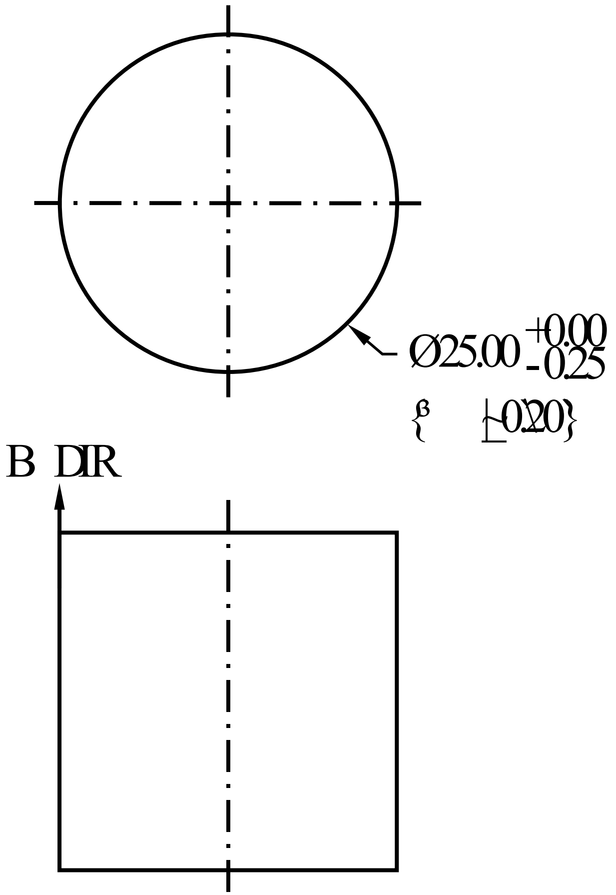
Ø25.11, 0.79



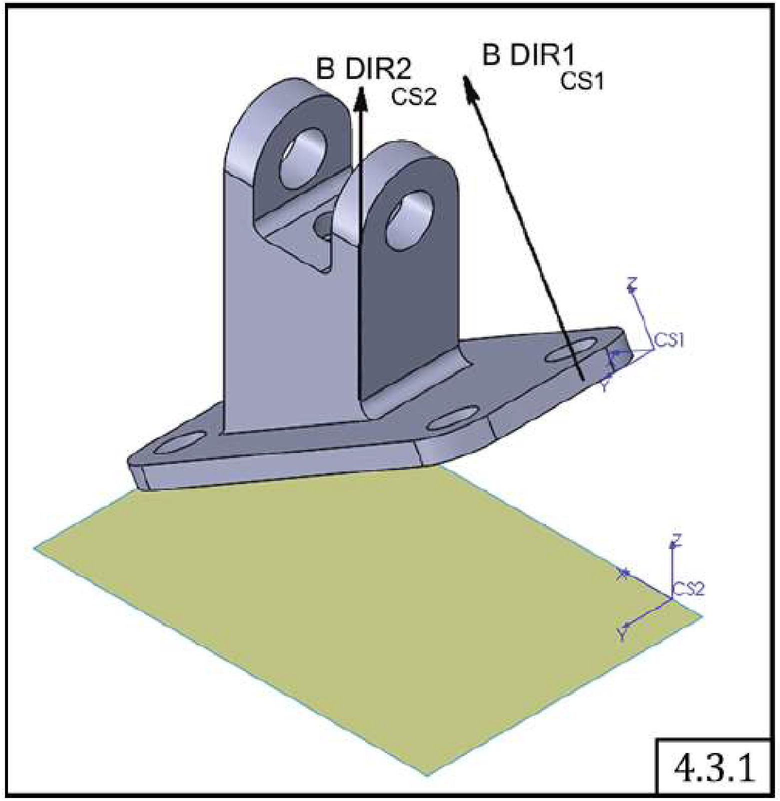
# GD&T practices for AM components (a brief look)

Georgia Tech

*This product specification ensures the desired result*




*Applied to a more complex design*



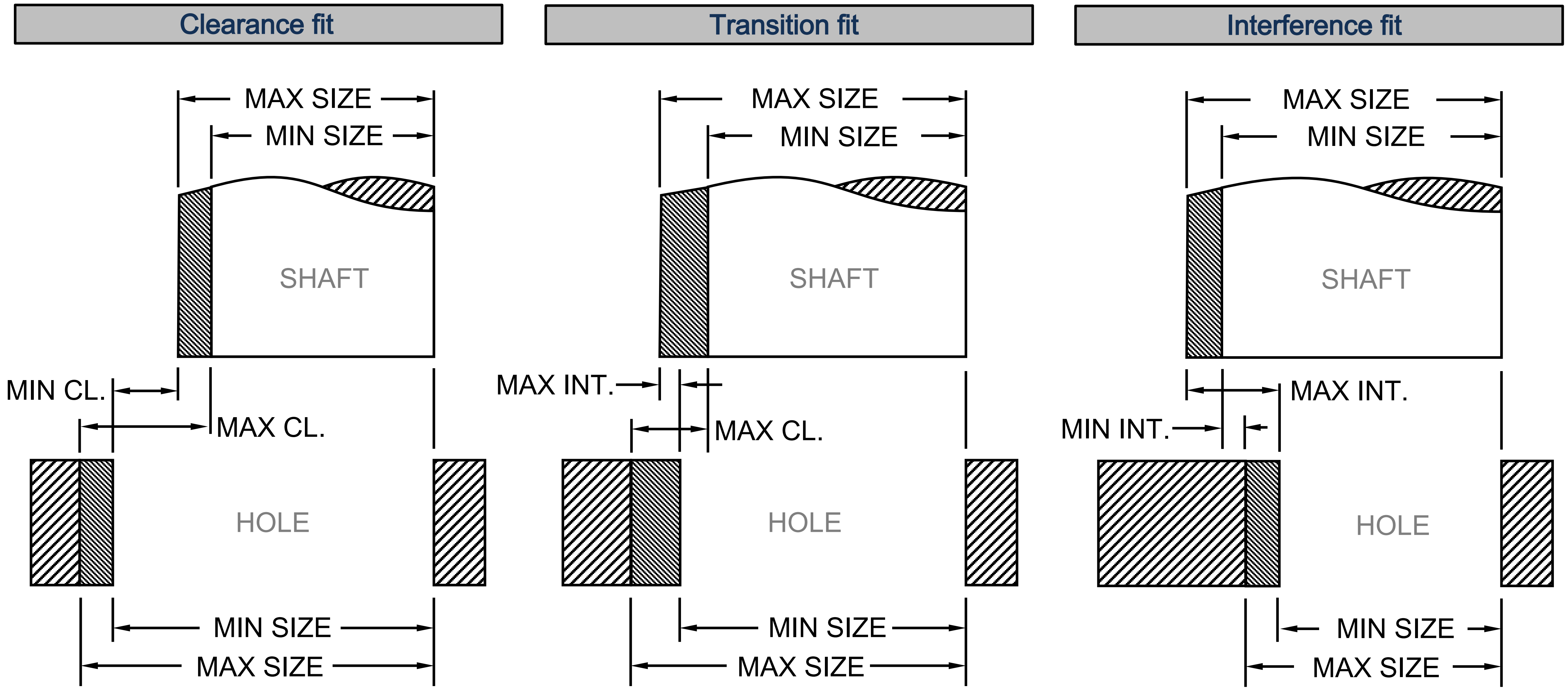

ASME Y14.46-2022

*Practices shown are as per ASME Y14.46-2022*

# Limits and fits: A brief review

# ASME B4.2 terminology

*Fit types apply to 'external' and 'internal' features, not just shafts and holes*



# Limits & Fits – Standardization

*ASME B4.1-1978(R2020) Preferred Limits and Fits for Cylindrical Parts*

- Designed for inch units
- Uses running fit (RC), locational fit (LC/LT/LN) and force fit (FN) classes

*ASME B4.2-1978(R2020) Preferred Metric Limits and Fits*

- Mimics ISO 286-1 for mm units in U.S. (most popular system in U.S.)
- Uses clearance, transition, and interference fit classes

*ISO 286-1:2010 Geometric product specifications (GPS) – ISO code system for tolerances on linear sizes – Part 1: Basis of tolerances, deviations, and fits*

# Preferred fits

| Type | Hole Basis | Shaft Basis | Description |
|---|---|---|---|
| Clearance | H11/c11 | C11/h11 | Loose running fit for wide commercial tolerances or allowances on external members. |
| | H9/d9 | D9/h9 | Free running fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures. |
| | H8/f7 | F8/h7 | Close running fit for funning on accurate machines for accurate location at moderate speeds and journal pressures. |
| | H7/g6 | G7/h6 | Sliding fit not intended to run freely, but ot move and turn freely and locate accurately. |
| Transition | H7/k6 | K7/h6 | Locational clearance fit for accurate location, a compromise between clearance and interference. |
| | H7/n6 | N7/h6 | Locational transition fit for more accurate location where greater interference is permissible. |
| Interference | H7/p6 | P7/h6 | Locational interference fit for parts requiring rigidity an alignment with prime accuracy of location but without special bore pressure requirements |
| | H7/s6 | S7/h6 | Medium drive fit for ordinary steel parts or shrink fits on light sections, the tightest fit usable with cast iron. |
| | H7/u6 | U7/h6 | Force fit suitable for parts which can be highly stress or shrink fits where the heavy pressing forces required are impractical. |

*Based on ASME B4.21994*

# References and continuing education

Georgia Tech

Standards

- ASME Y14.5-2018: Geometric Dimensioning and Tolerancing
- ASME Y14.5.1-2019: Mathematical Definition of Dimensioning and Tolerancing Principles
- ASME Y14.41-2019 Digital Product Definition Data Practice
- ASME Y14.46-2022 Product Definition for Additive Manufacturing
- Clearance holes for fasteners
  - Machinery's Handbook Tables (*not a standard)
  - ASME B18.2.8-1999 (R2017)
- Standard limits and fits
  - ASME B4.1 (inch) and 4.2 (metric)

*The slide deck from part I of this seminar series may be found at: [https://doi.org/10.5281/zenodo.7647256](https://doi.org/10.5281/zenodo.7647256)*

Texts


- R. Bundynas, K. Nisbett, *Shigley's Mechanical Engineering Design, 10th Ed., McGraw-Hill*, 2014.
- S. Kalpakjian & S. Schmid, *Manufacturing Processes for Engineering Materials*, 6th Ed., Pearson Education Inc., 2017.

# 2023 Digital Manufacturing Challenge powered by SME's DDM Advisory Team

***Thank you for joining us for…***

**Fundamentals of Geometric Dimensioning and Tolerancing, Part II**

***…Questions?***