Fundamentals of
Geometric Dimensioning and Tolerancing (GD&T)
-Part II-

Jaime Berez, Ph.D.*

Asst. Professor
Center for Precision Metrology
Mechanical Engineering and Engineering Science
University of North Carolina at Charlotte

j.berez@charlotte.edu

Maxwell Praniewicz, Ph.D.**

Mechanical Engineer
Production Systems Group, Intelligent Systems Division
National Institute of Standards and Technology

maxwell.praniewicz@nist.gov

*An earlier version of this presentation was developed when the author was affiliated with the Georgia Institute of Technology.

Fundamentals of GD&T – Seminar series overview

Part 1

Authors

Jaime Berez
UNC Charlotte

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

Authors

Jaime Berez
UNC Charlotte

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)
- Limits & fits: A brief review

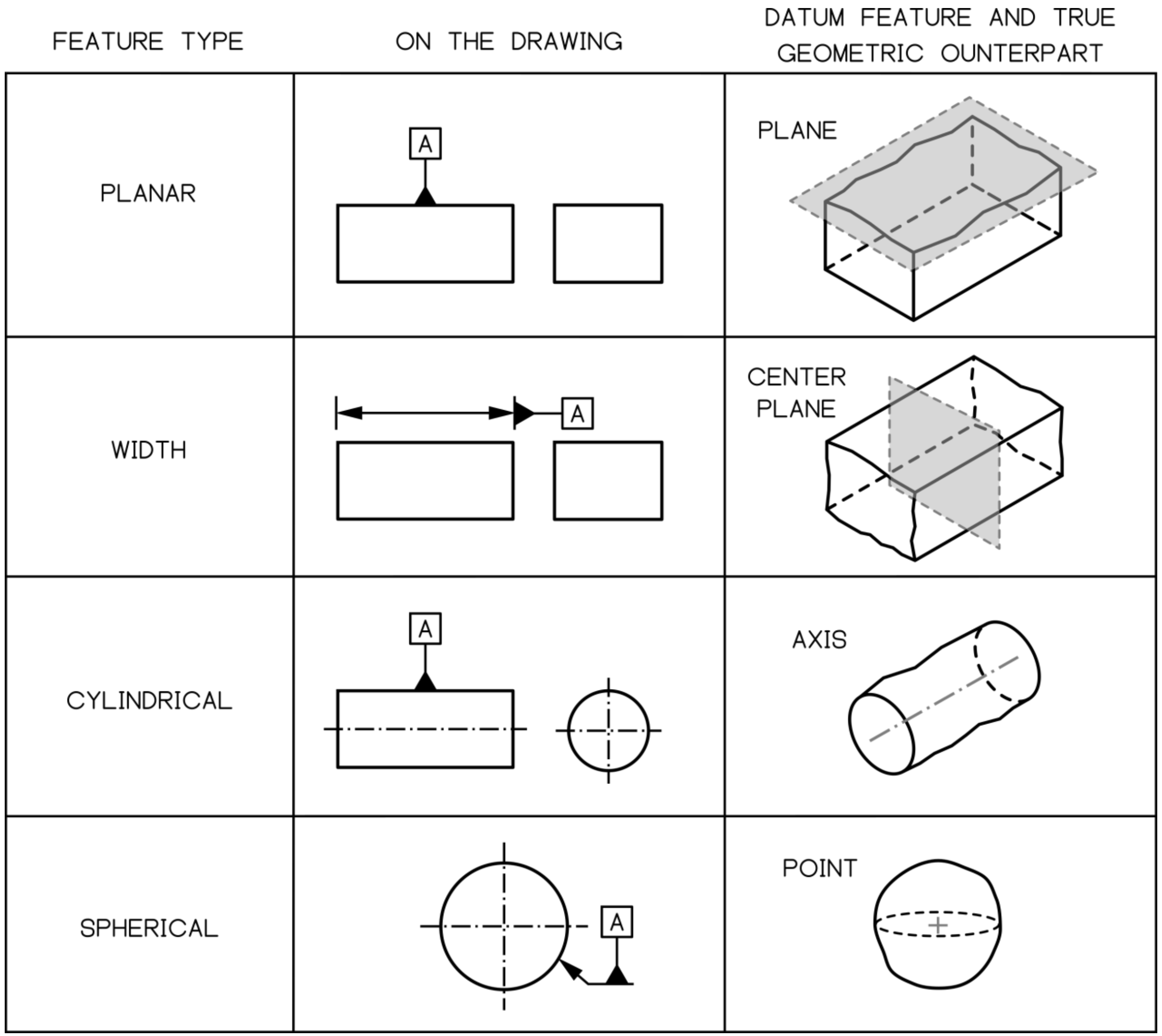
Clarifications & review of GD&T fundamentals

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.



Clarifications to Part I

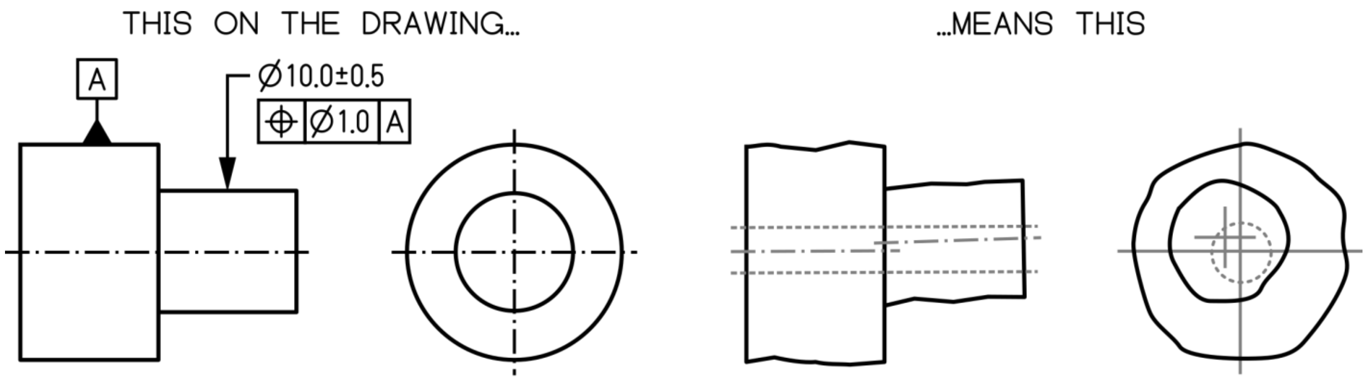
Q: Why were concentricity and symmetry removed from ASME Y14.5-2018?
How should we replace them?

Concentricity

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

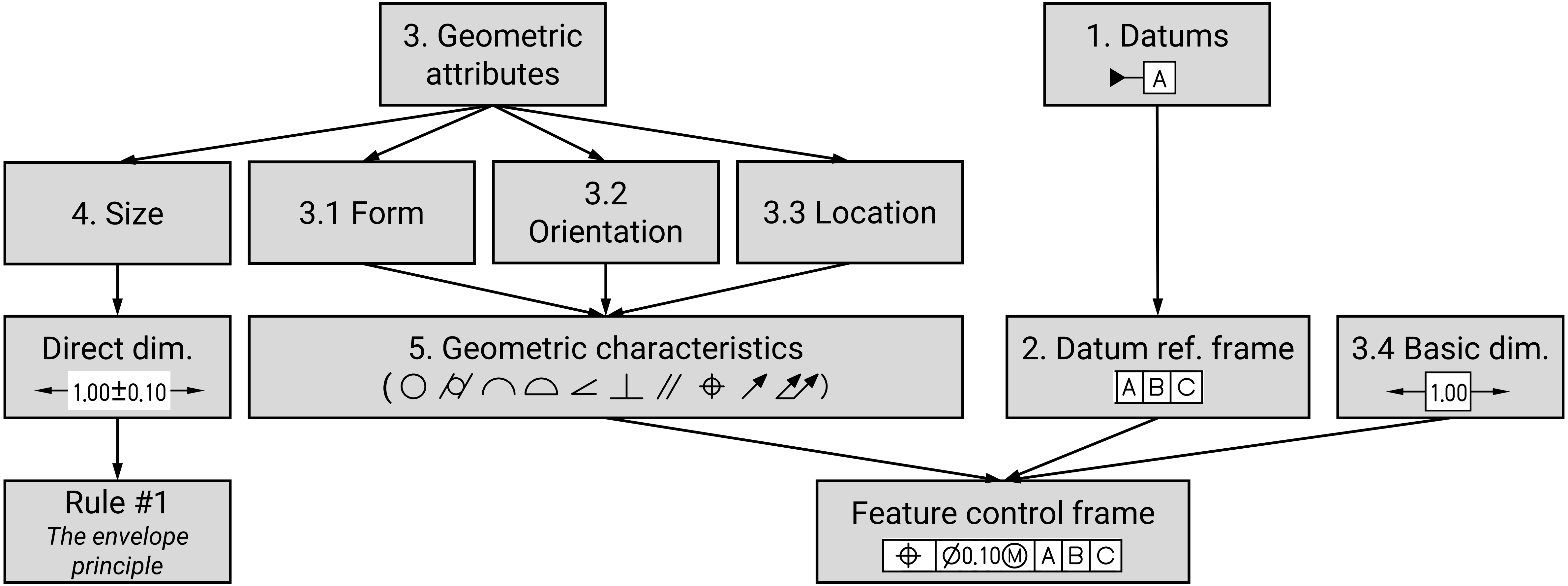
Instead…

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



__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
Ø0.10 Ⓜ A B C

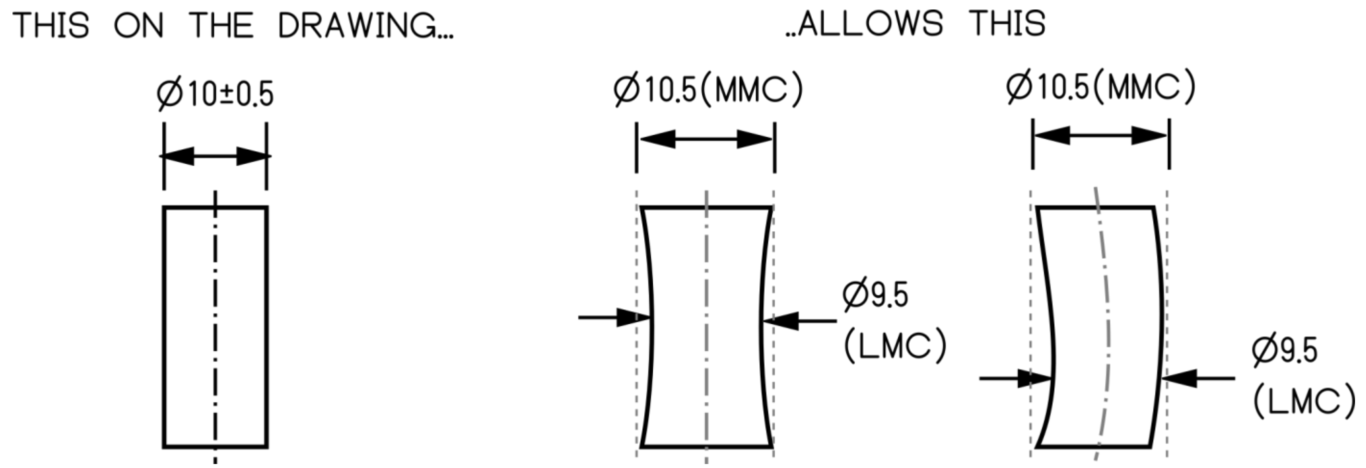
Geometric characteristics

Geometric characteristic	Symbol	Geometric attribute	Datum referencing?
Straightness	⏤	Form	No
Flatness	⏥		
Circularity	○		
Cylindricity	⌭		
Profile of a line	⌒	Profile (*location, orientation, size, & form*)	Sometimes datum referencing
Profile of a surface	⌓		
Angularity	∠	Orientation	Datum referencing
Perpendicularity	⊥		
Parallelism	//		
Position	⌖	Location	Datum referencing
Circular runout	↗	Runout (*location of a cylinder*)	Datum referencing
Total runout	⌰		

Rule #1 – The envelope principle

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

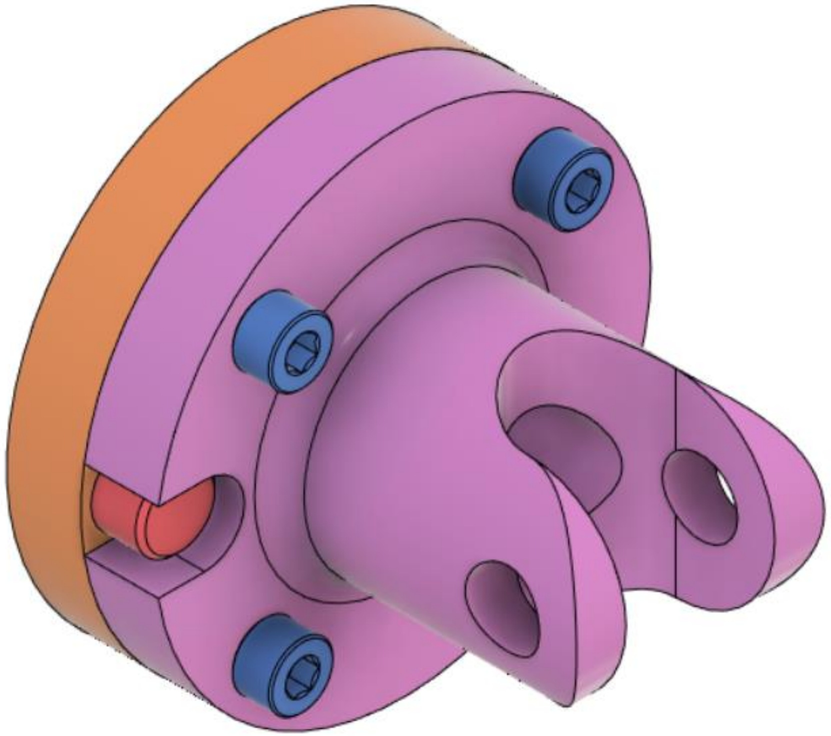
- *The MMC and LMC act like an envelope, therefore a feature of size inherently has form control. This applies to positive AND negative features.*
- *Form control can be additionally refined via* ⏤ ⏥ ⌭ ⌒ ⌓

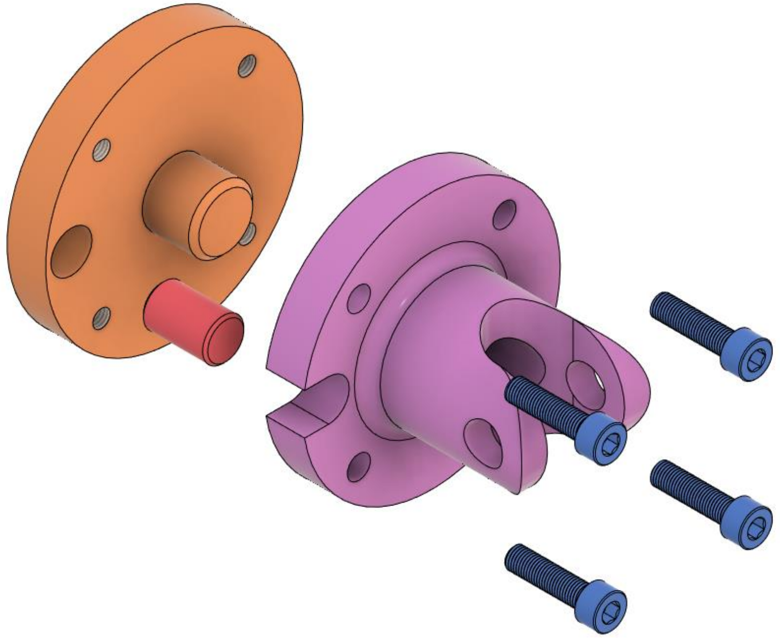


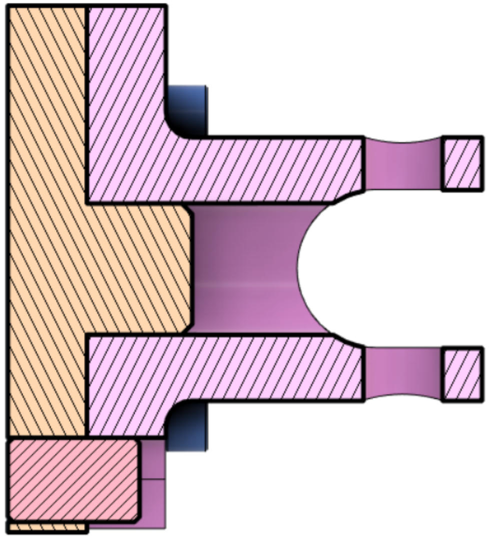
Implementing GD&T: Checklist and walk-through

GD&T How-to: Example

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







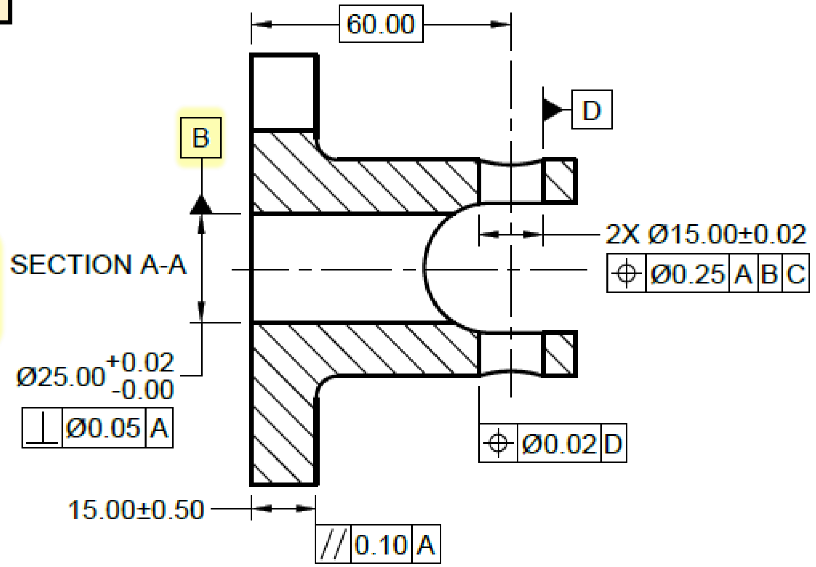
Every part is different: each requires special attention.
These guidelines are not definitive.

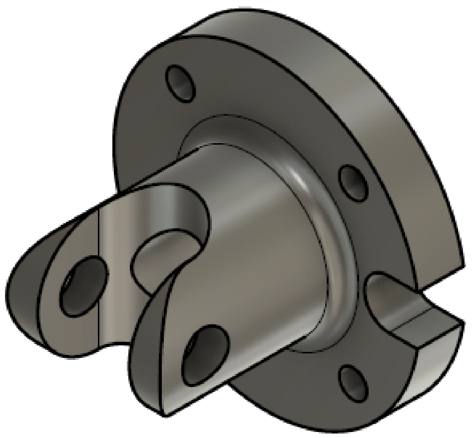
1. Understand the functionality of the part. Identify features that control function and assembly.
2. Based on (1), choose datums that mimic the functionality of the part
3. Control the form of datum features (normally ⏥, ⌭, ±*)

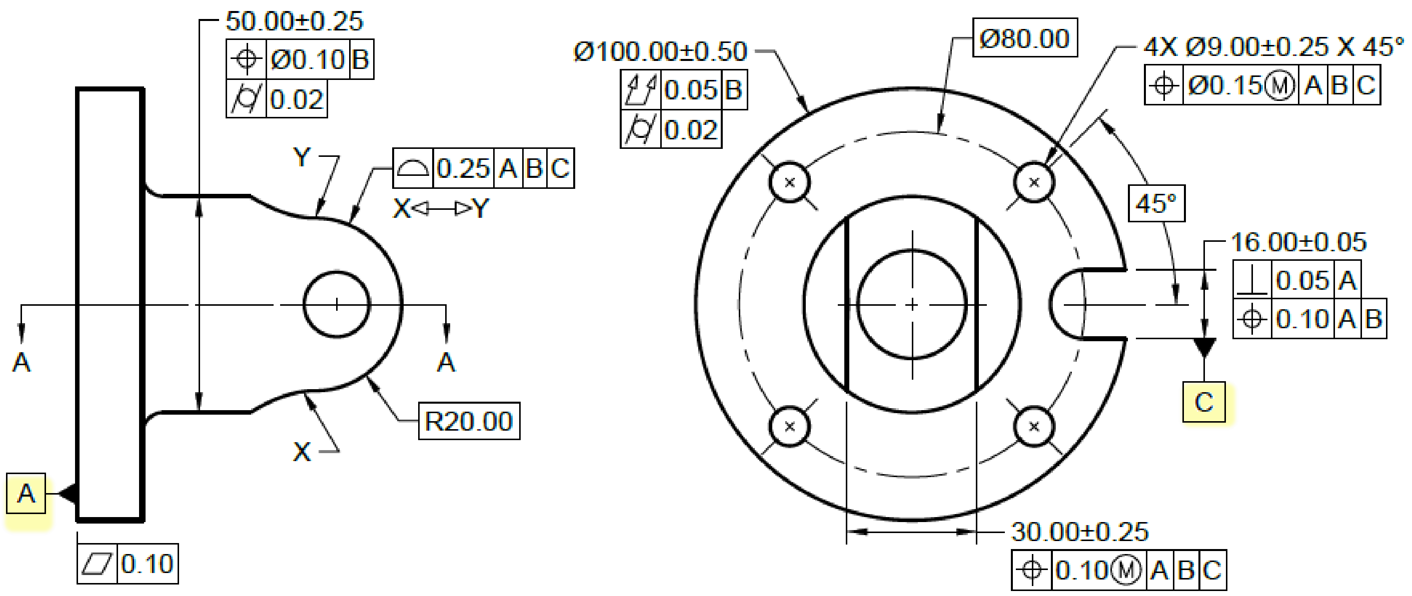
   *Direct dimensioning controls form via the envelope principle.

4. Control the relation of datum features to each other (normally ⊥ and //)
5. Control features of size (±)
6. Control features of form that need no Datum Reference Frame (DRF)
7. Control the position, orientation, profile, and/or runout of unconstrained features to a DRF**, apply basic dimensions.

   **6 Degrees of Freedom (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.

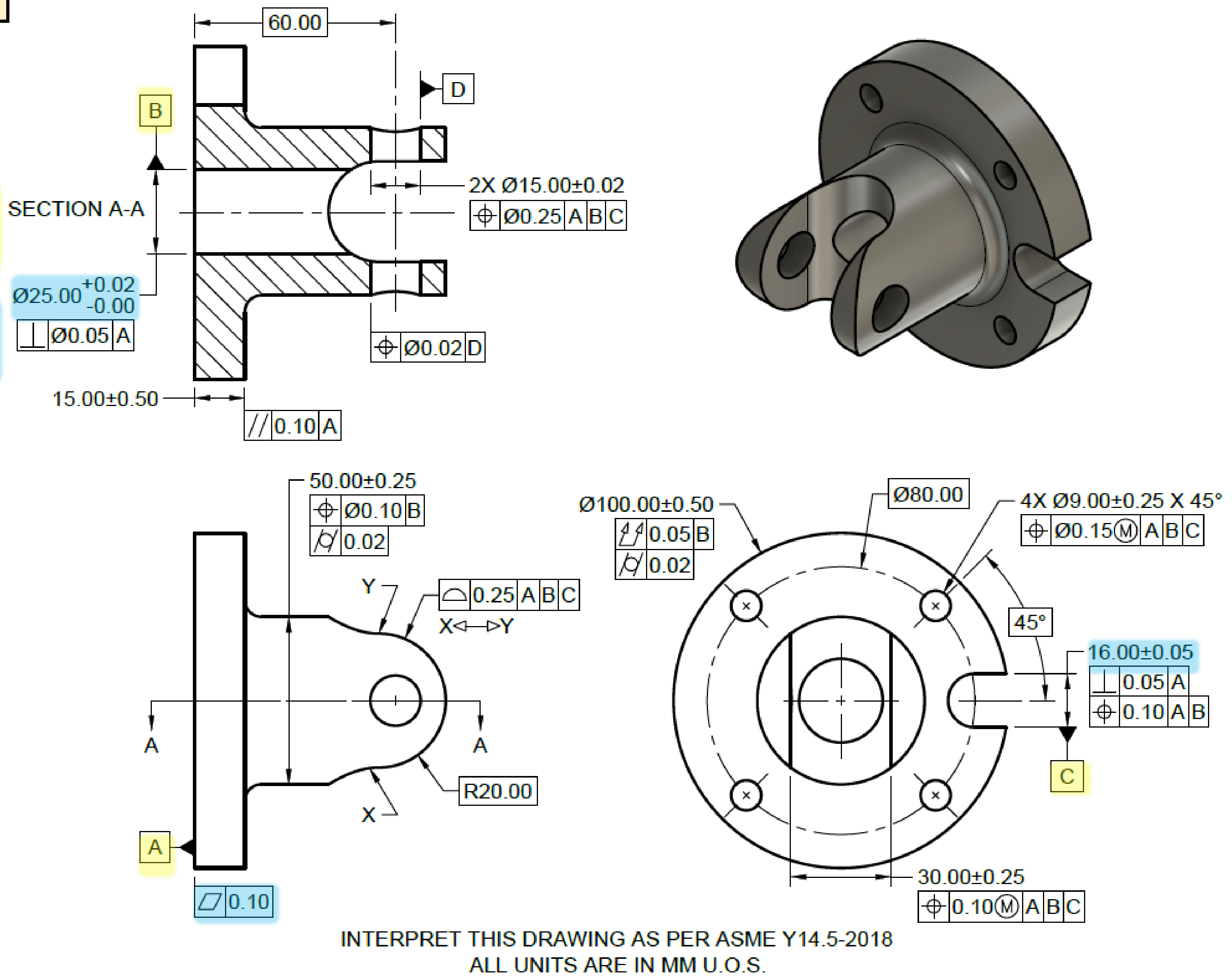
Every part is different: each requires special attention.
These guidelines are not definitive.

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.

4. Control the relation of datum features to each other (normally ⊥ and //)
5. Control features of size (±)
6. Control features of form that need no DRF
7. 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



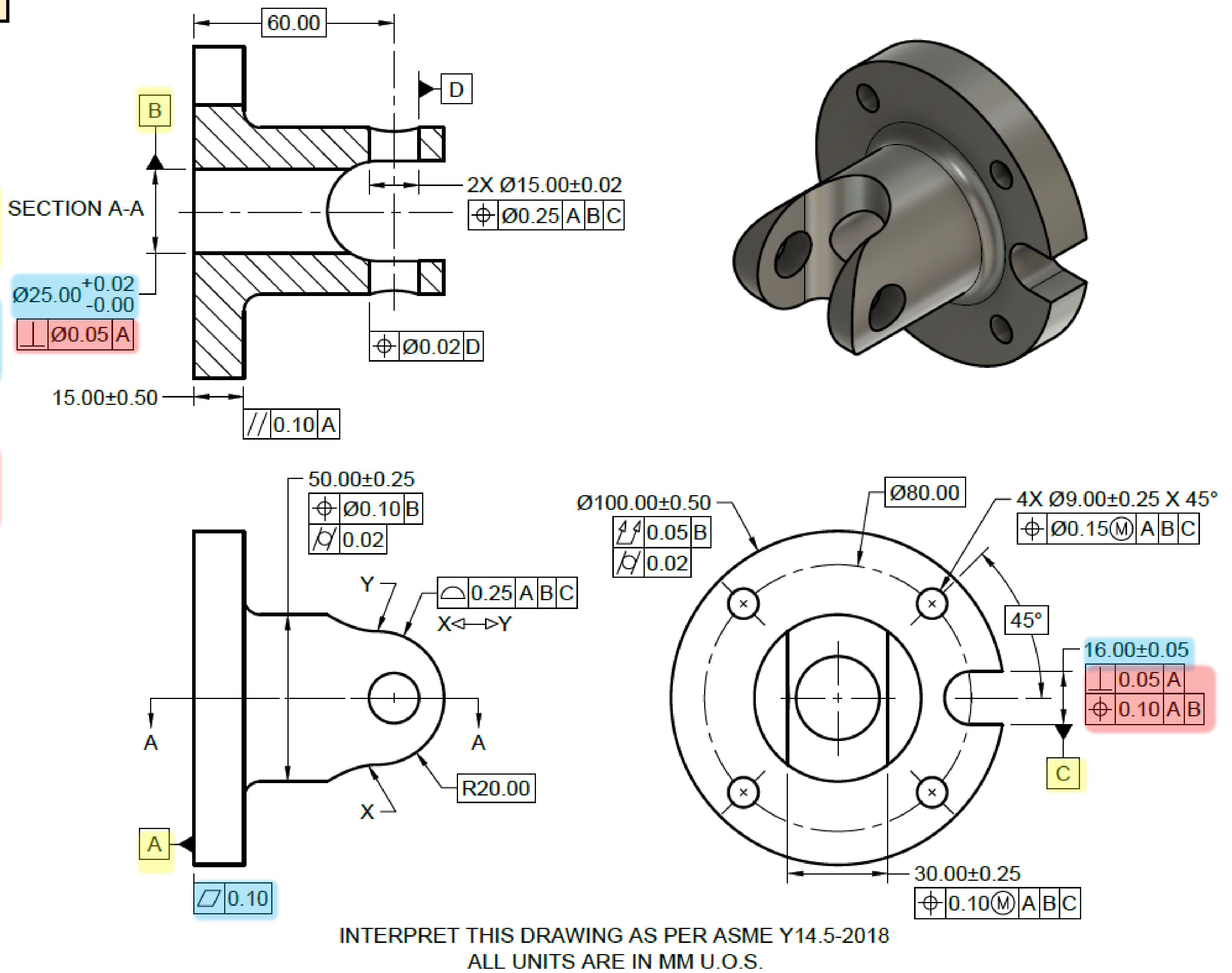
Every part is different: each requires special attention.
These guidelines are not definitive.

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.

4. Control the relation of datum features to each other (normally ⊥ and //)
5. Control features of size (±)
6. Control features of form that need no DRF
7. 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



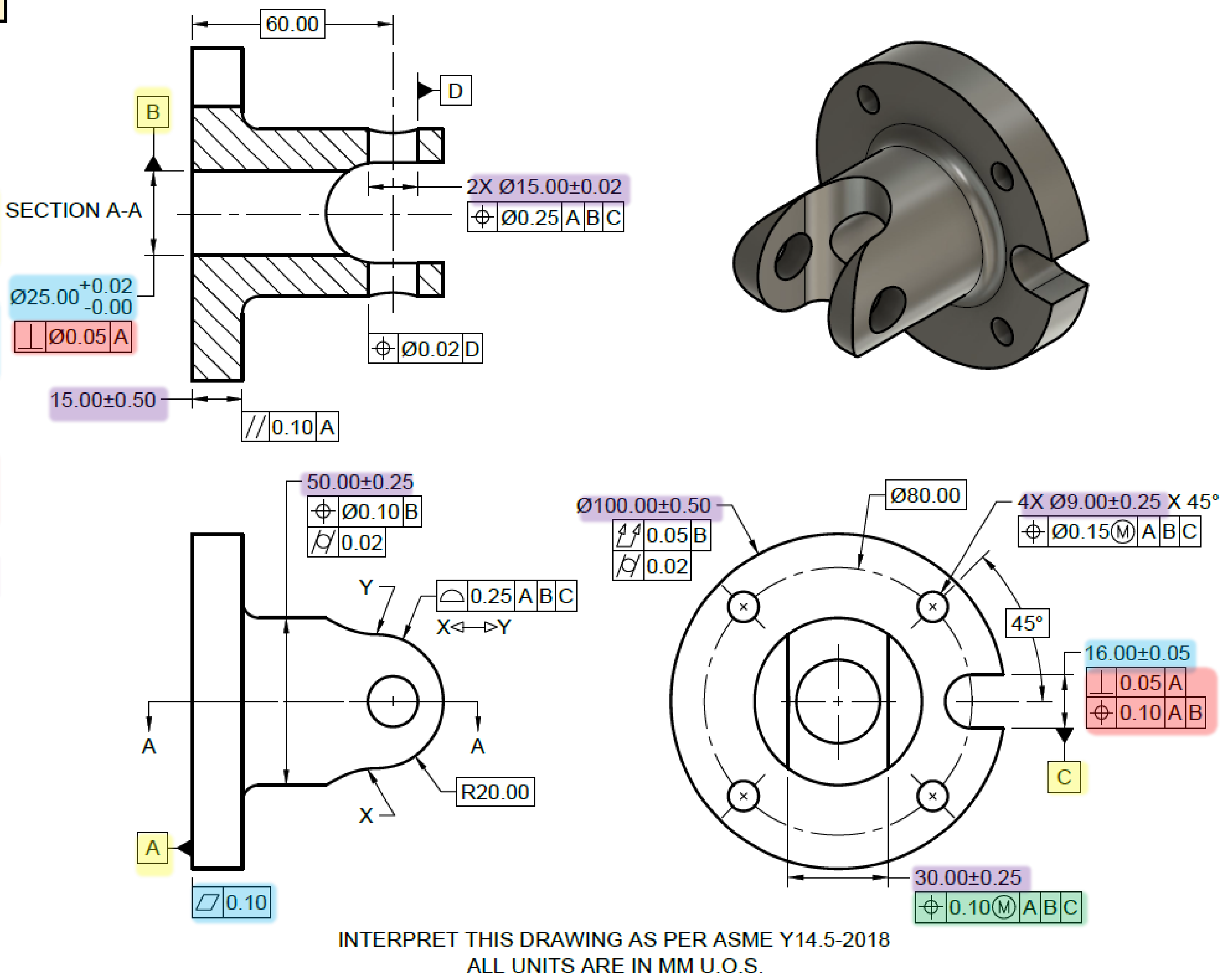
Every part is different: each requires special attention.
These guidelines are not definitive.

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.

4. Control the relation of datum features to each other (normally ⊥ and //)
5. Control features of size (±)
6. Control features of form that need no DRF
7. 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



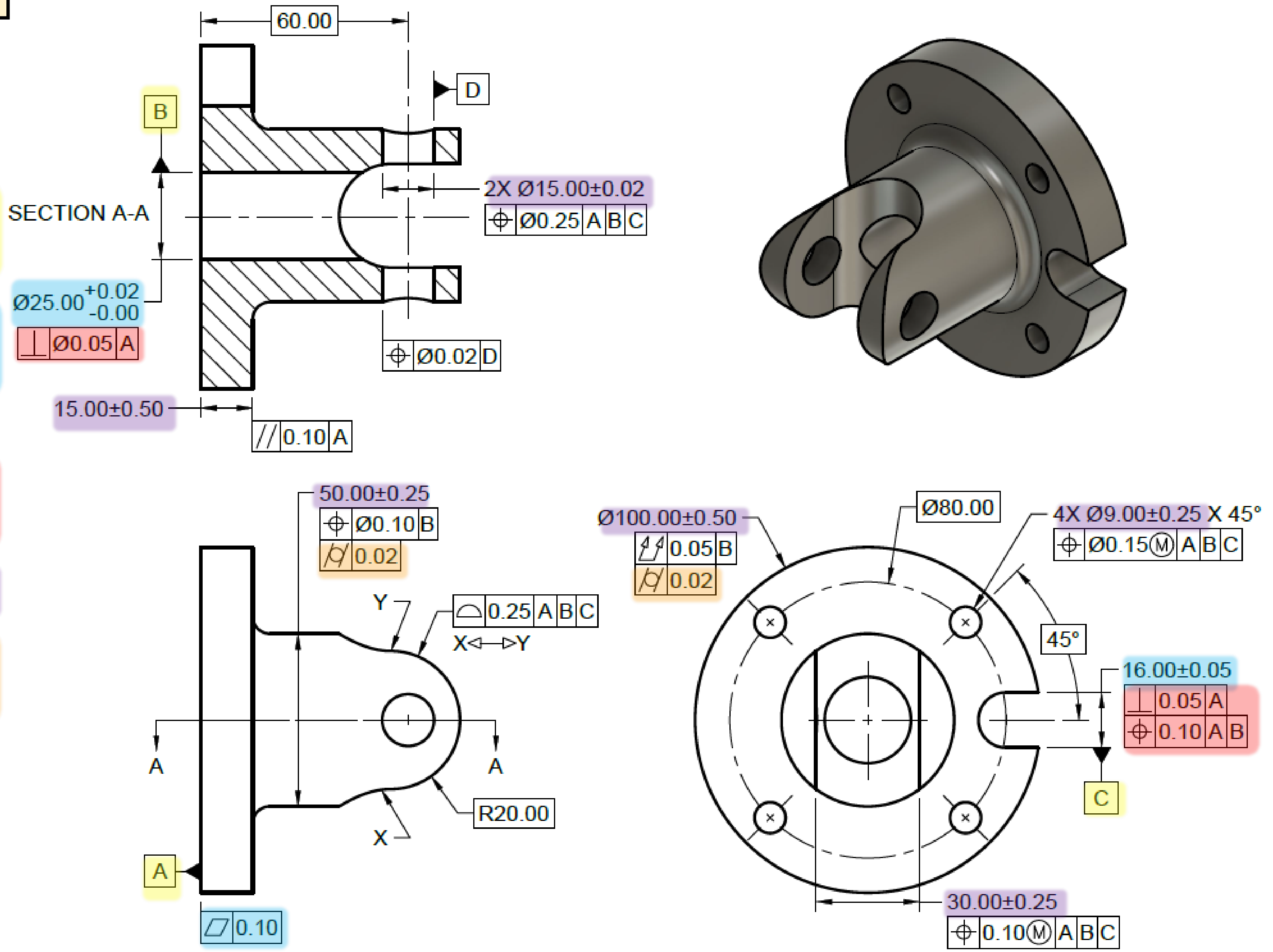
Every part is different: each requires special attention.
These guidelines are not definitive.

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.

4. Control the relation of datum features to each other (normally ⊥ and //)
5. Control features of size (±)
6. Control features of form that need no DRF
7. 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.

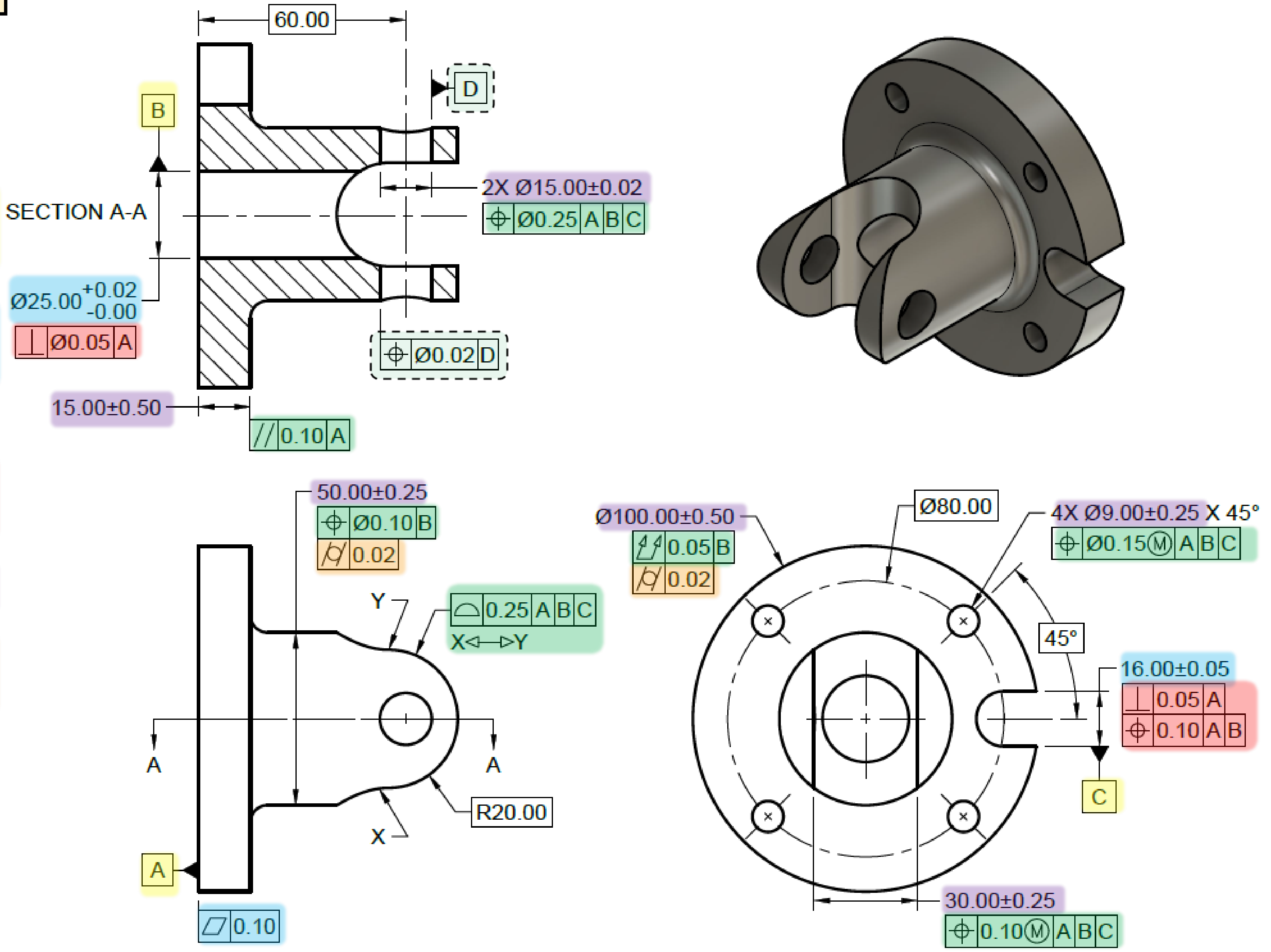
Every part is different: each requires special attention.
These guidelines are not definitive.

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.

4. Control the relation of datum features to each other (normally ⊥ and //)
5. Control features of size (±)
6. Control features of form that need no DRF
7. 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



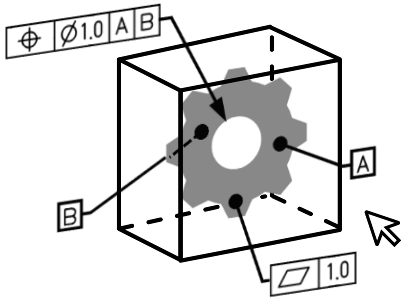
INTERPRET THIS DRAWING AS PER ASME Y14.5-2018
ALL UNITS ARE IN MM U.O.S.

Digital product definition

Model-based definition (MBD)

*Customer delivers a Computer-aided Design (CAD) file which includes GD&T**

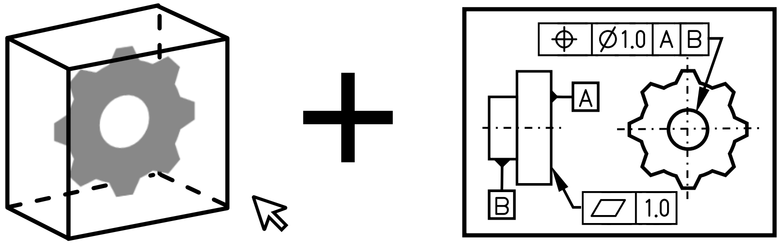
- Standard: ASME Y14.41-2019 Digital Product Definition
  - Not yet fully adopted
- MBD will very often be minimally dimensioned. Basic dimensions will not be automatically shown, but queried by the user as necessary.



Minimally dimensioned drawings

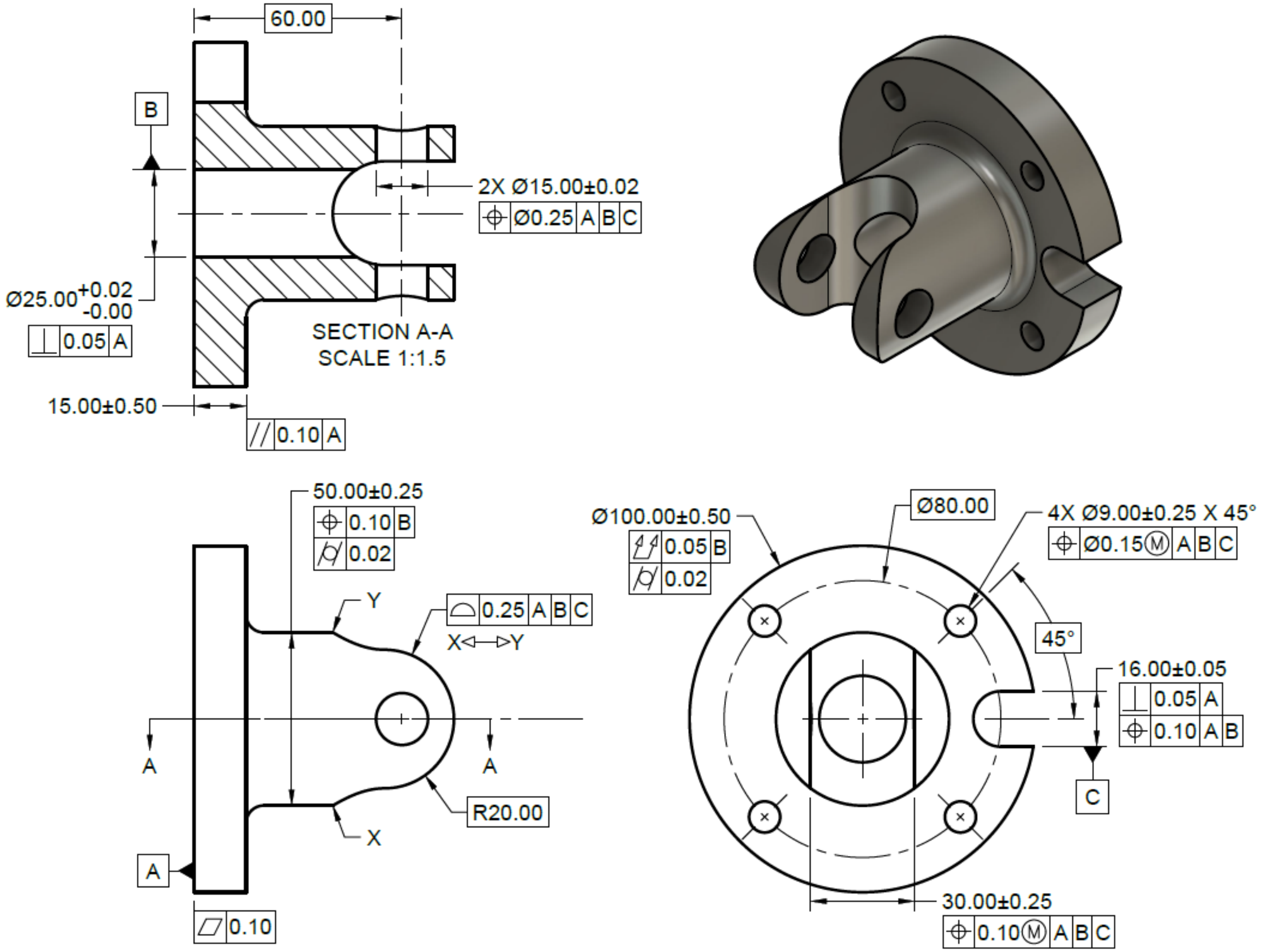
Customer delivers a minimally (e.g., partially, reduced, etc.) dimensioned drawing ***and*** *CAD data*

- 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



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

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’ and ‘manual’ 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 can use varying principles, commonly tactile measurement
- CMS instruments fundamentally measure samples of a surface in x,y,z dimensions

Coordinate Measurement System (CMS) use…

The best use cases for a CMS include…

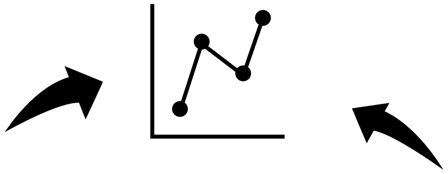
- Complex component surfaces
- Complex measurement tasks
- High degree of automation required

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…

Tactile CMM

Non-contact metrology systems
Reproduced from D.A. Maisano et al. (CC 4.0)

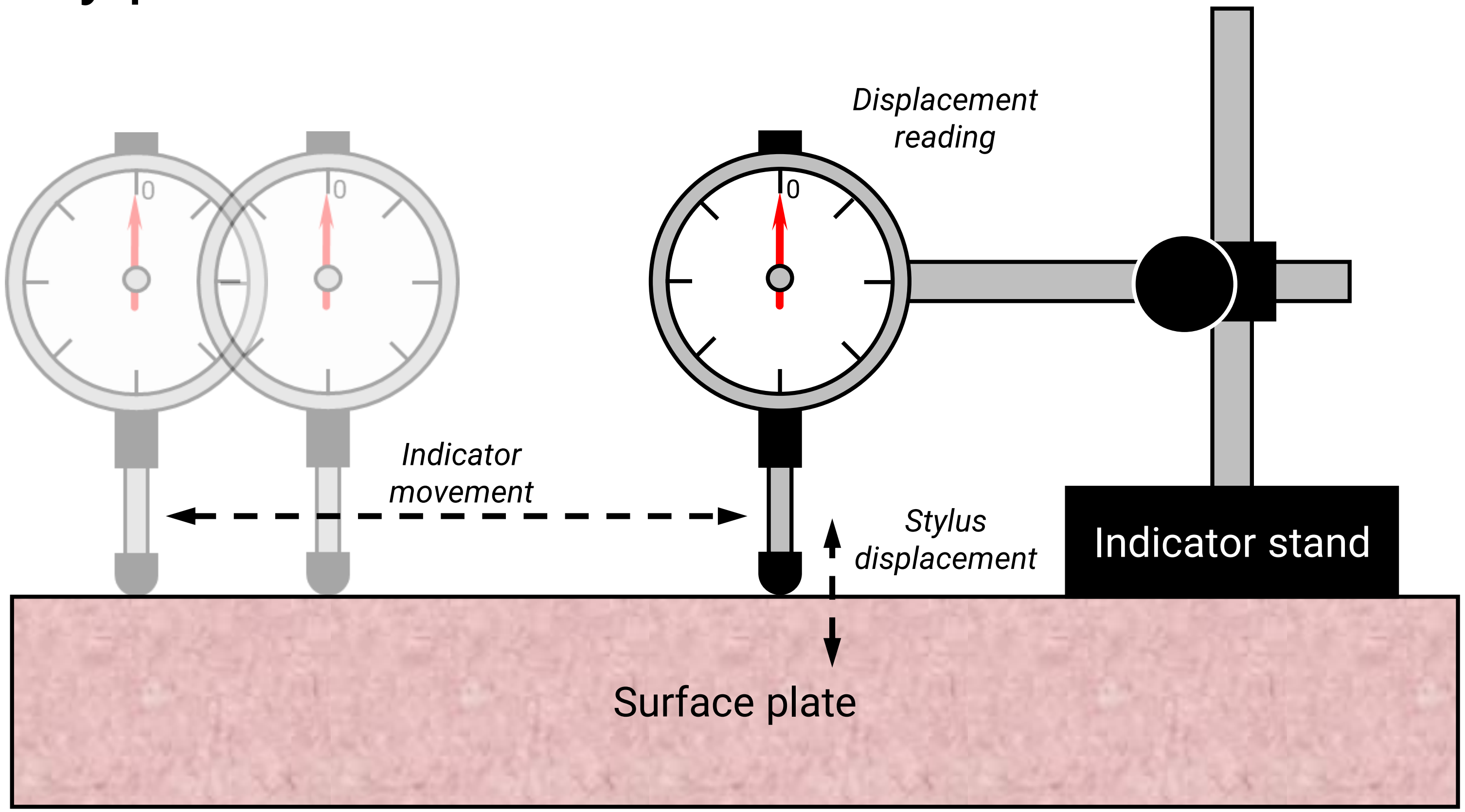


Starrett – Digital indicator example

Mitutoyo America Corporation – Digital outside micrometer example

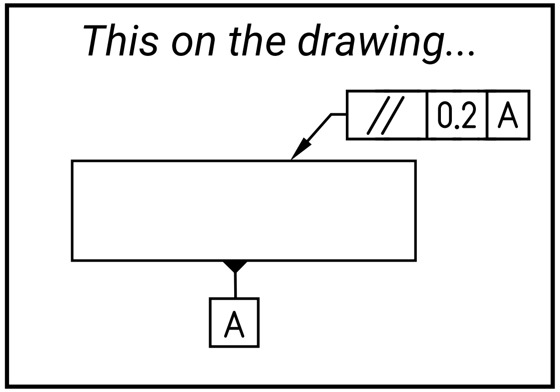
Surface plate inspection principles

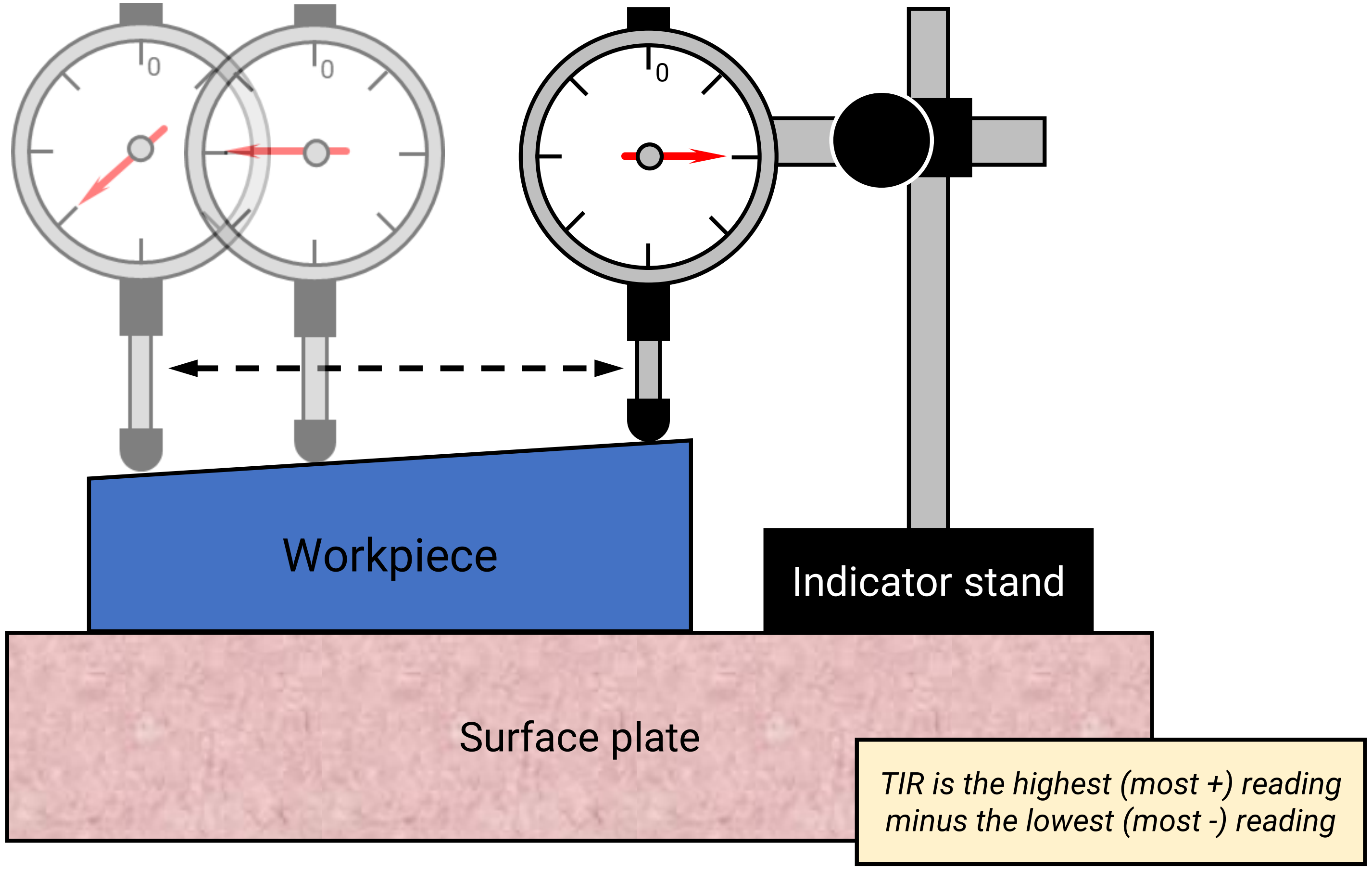
- Moving an indicator over a surface place should show zero dial movement – the stylus contact point and indicator stand base is ideally coplanar at any point of contact



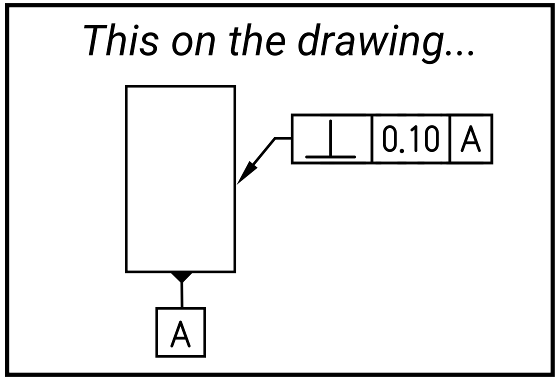
Comparators – Parallelism measurement

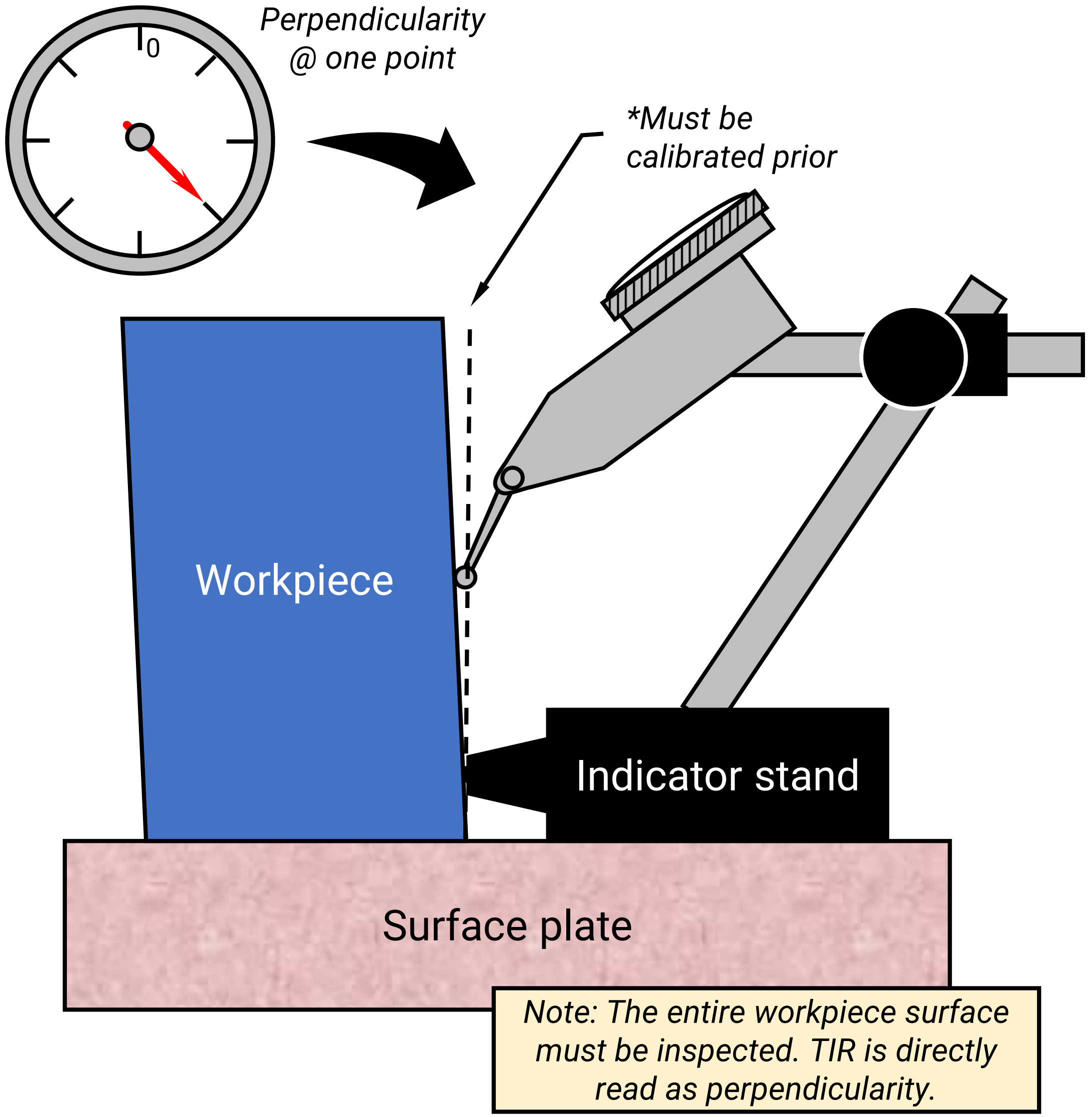
- The total indicator reading (TIR) is the maximum reading – the minimum reading
- TIR over the workpiece is a direct reading of parallelism



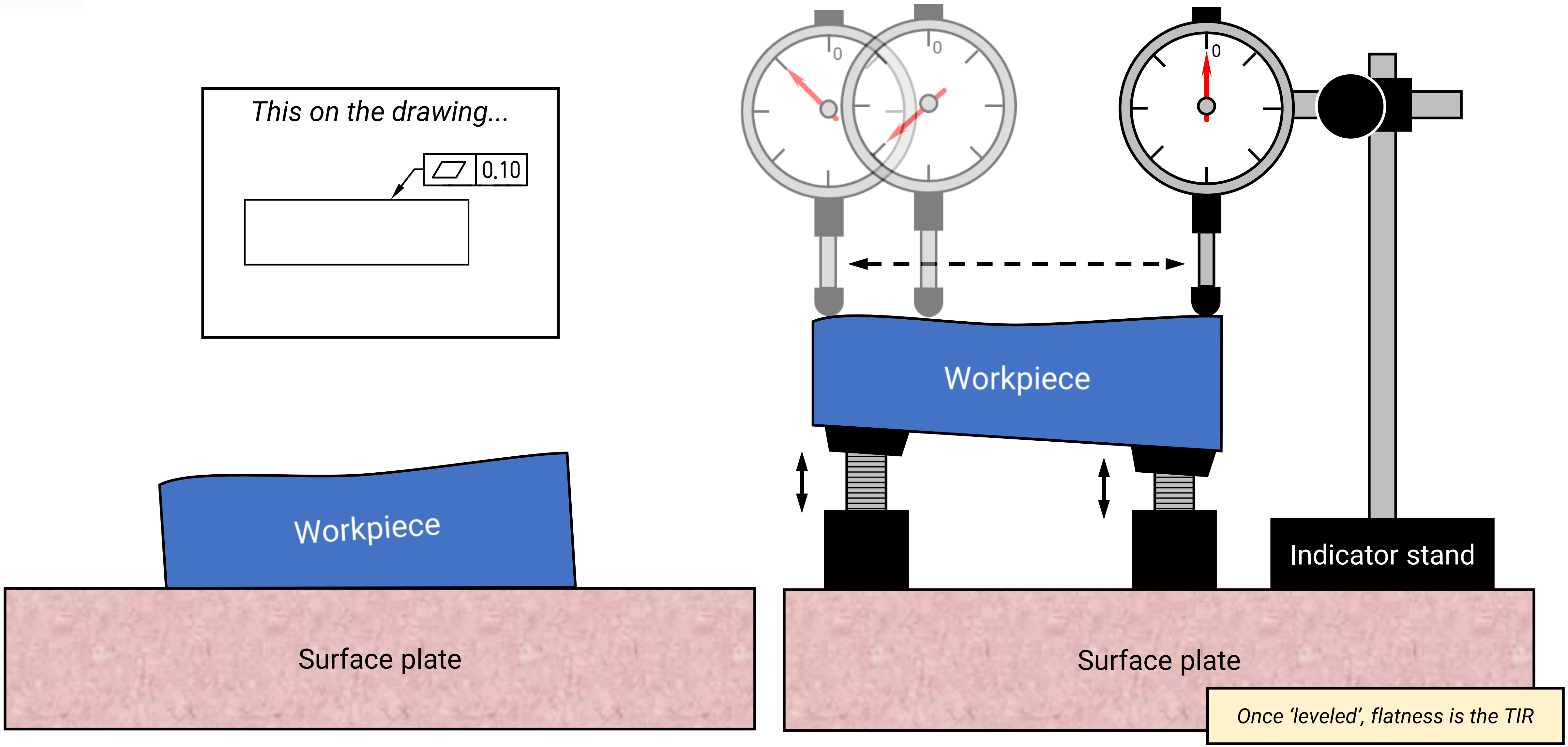


Comparators – Perpendicularity measurement





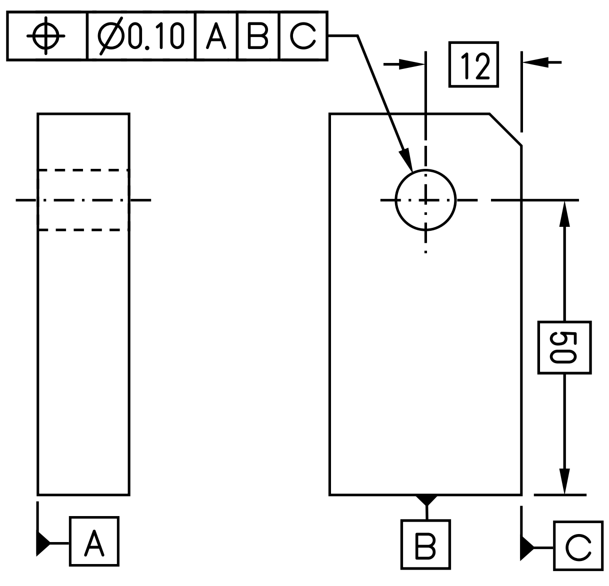
Comparators – Flatness measurement

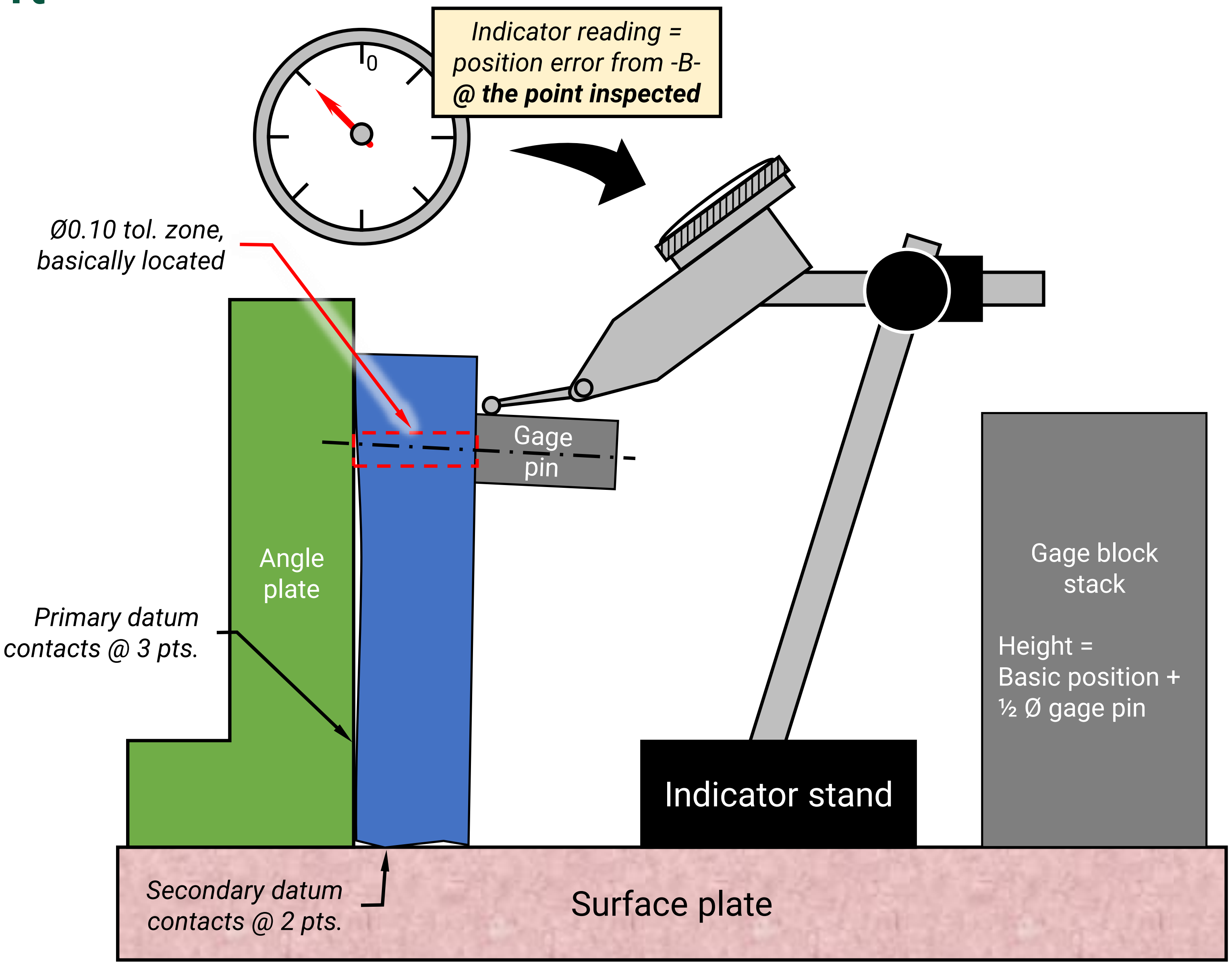


Position measurement

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...



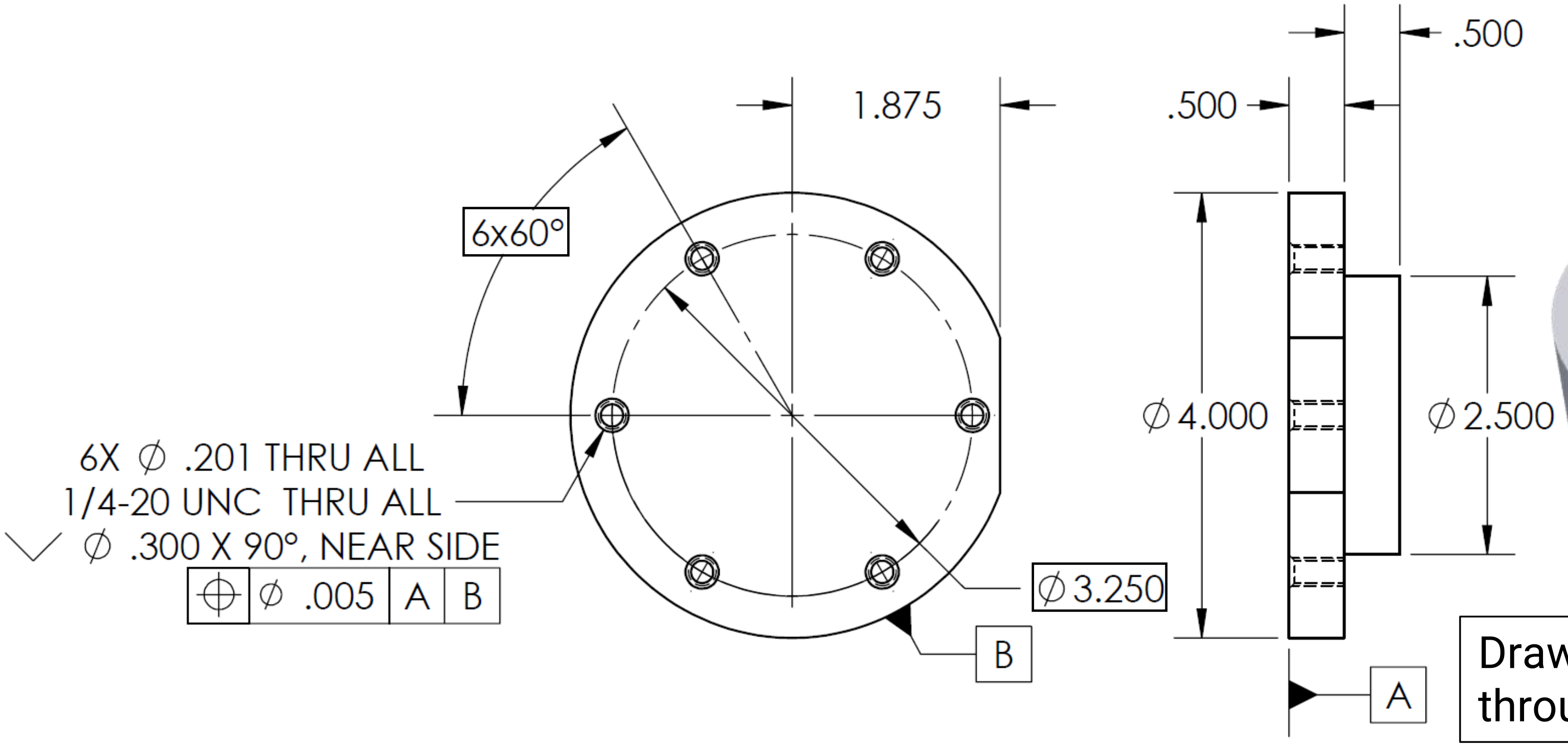


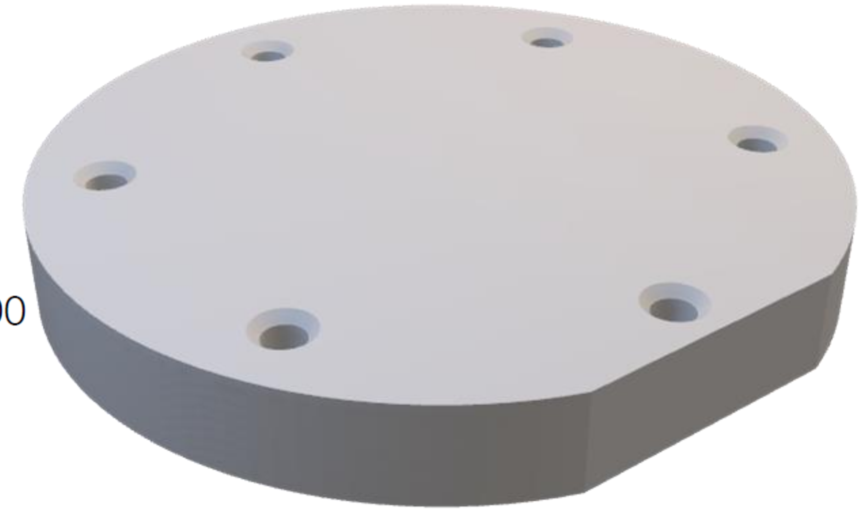
Case studies

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

Case Study 1 – Hole Tolerancing

<u>Prompt:</u> 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.





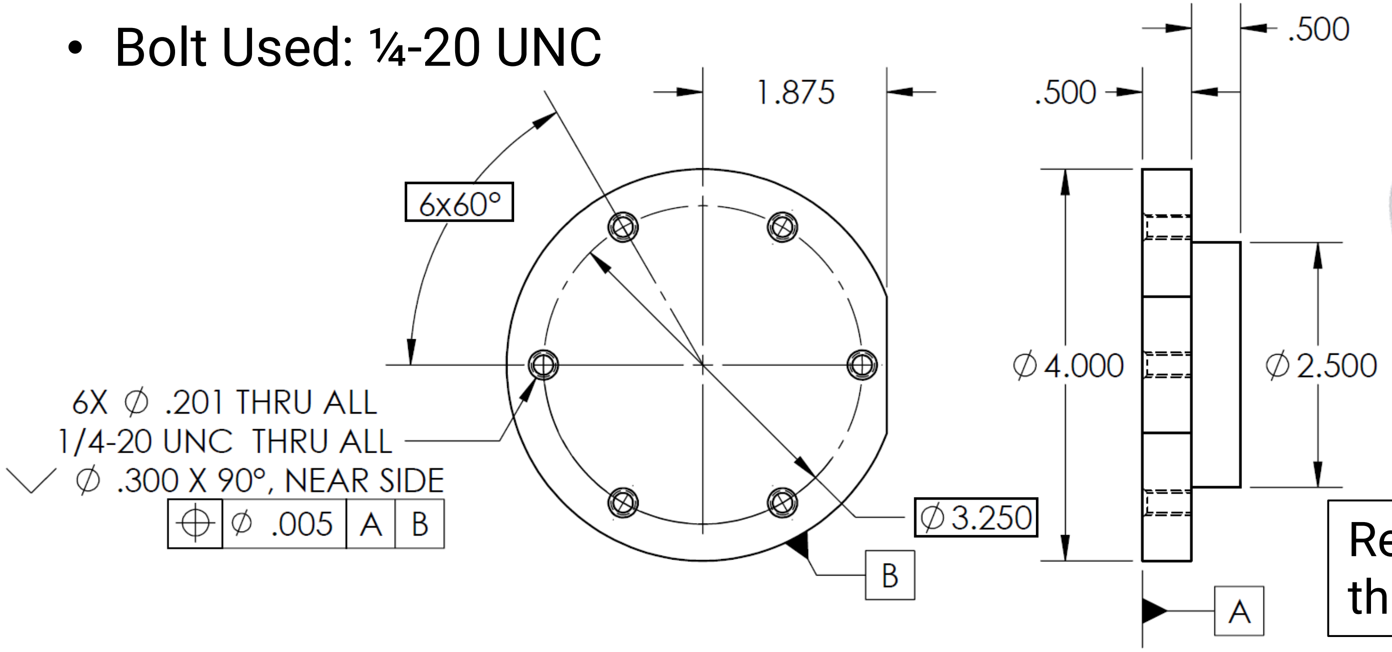
Drawing determined through reverse engineering

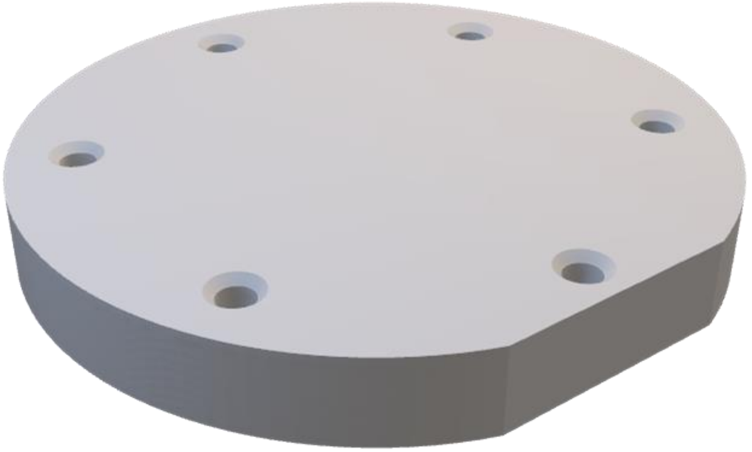
Case Study: Hole Tolerancing

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

UNC: Unified-national-coarse thread



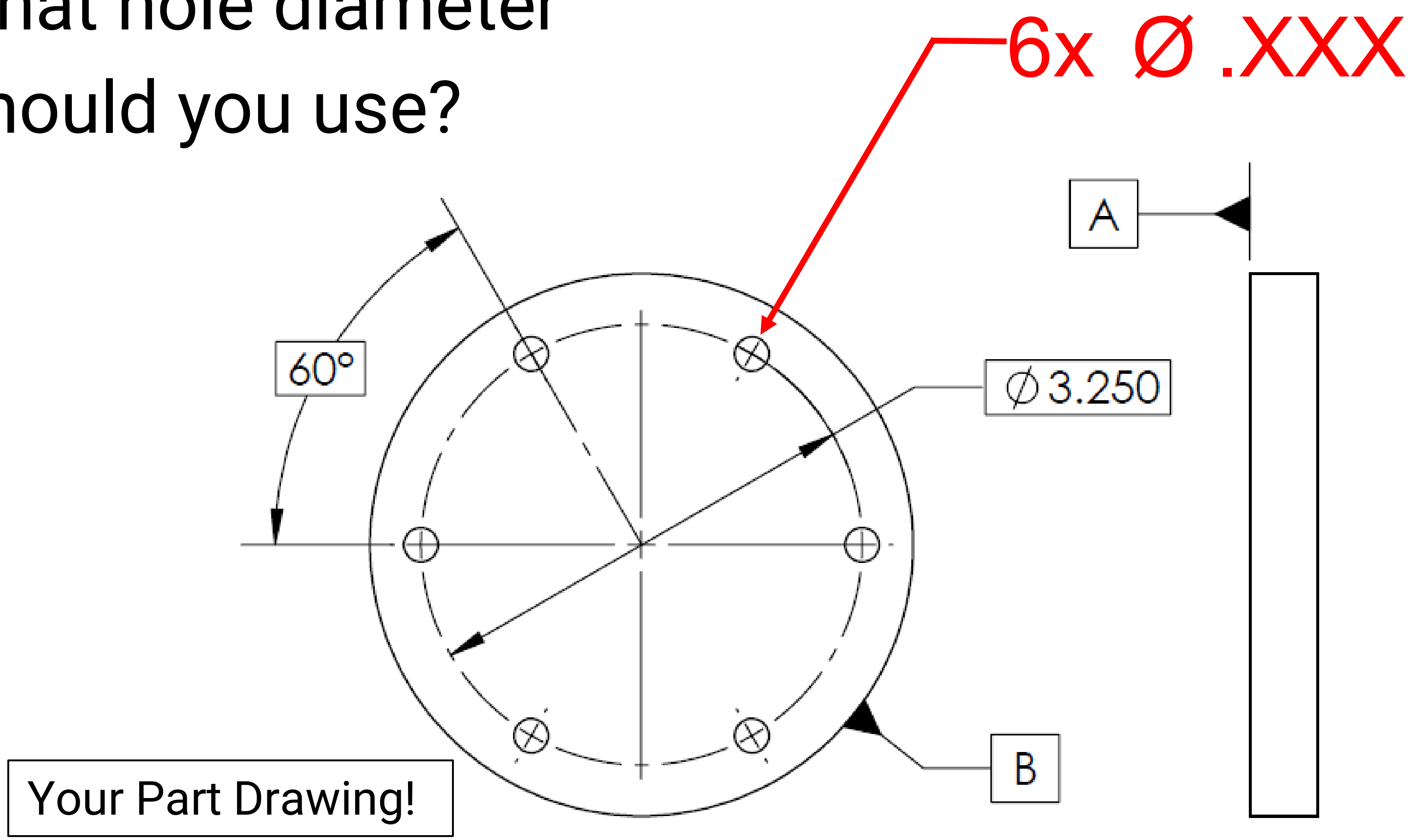


Reference drawing determined through reverse engineering

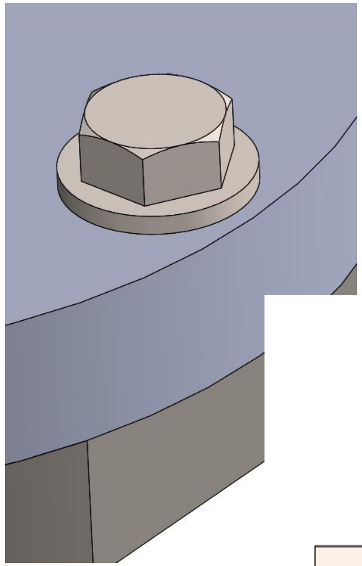
Case Study: Hole Tolerancing

Basics established!

Next: What hole diameter should you use?



Case Study: Hole Tolerancing



Hole Diameter?

¼-20 UNC Bolt: .250 hole?   No!

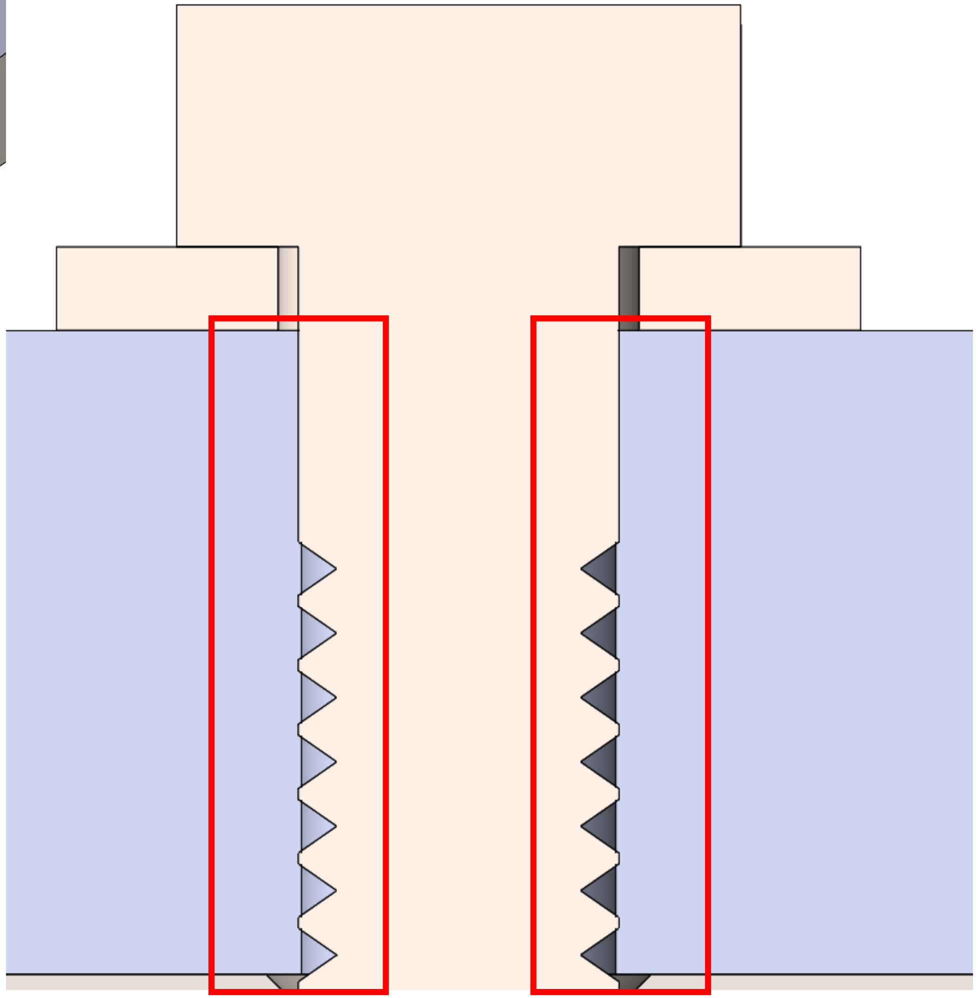
Class 2A ¼-20 UNC bolt:

.2408 - .2489*

Drilling is imperfect...also needs tolerance!

.250 ± .005

This could lead to interference!



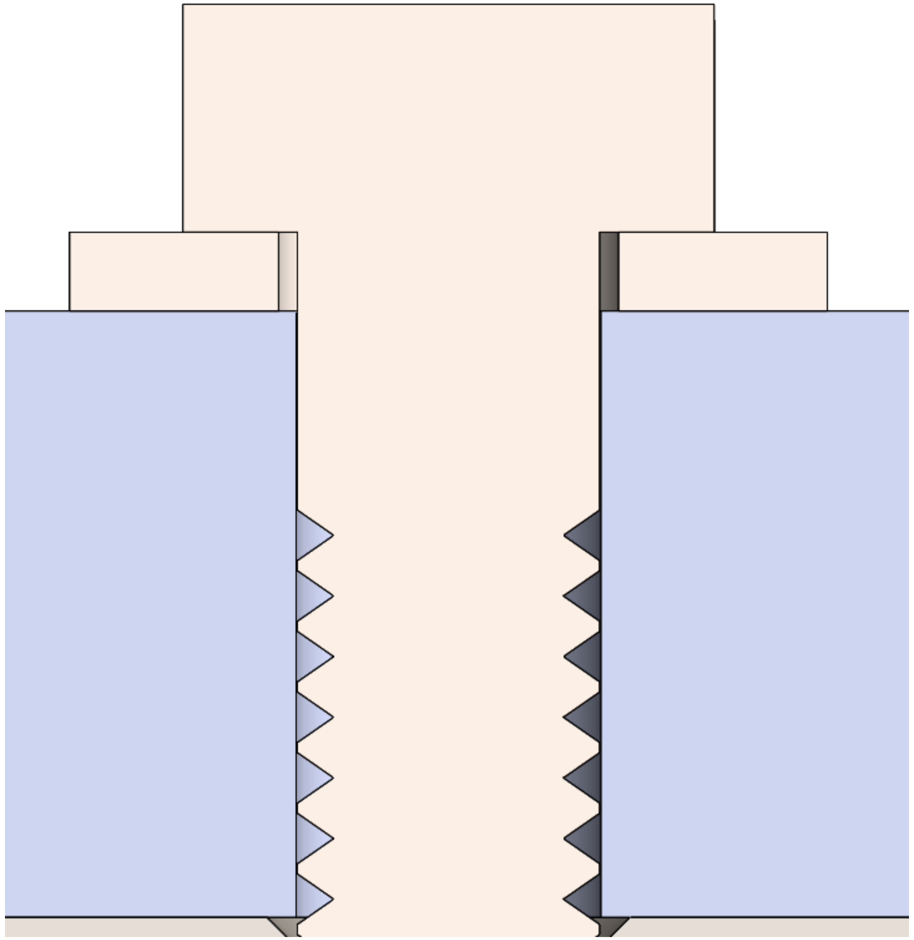
**Machinery's Handbook, 26th ed., pg. 1717*

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



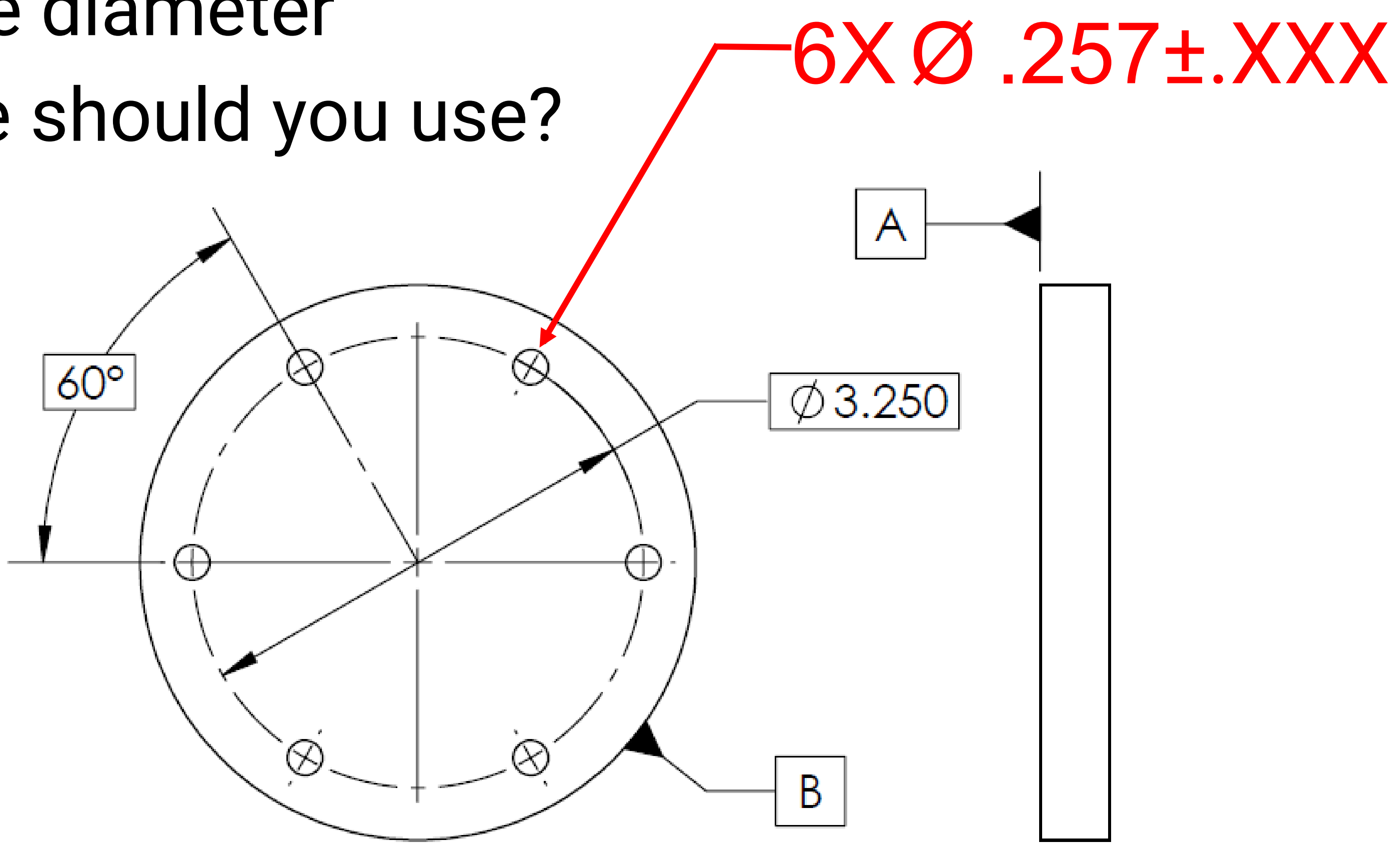
Even with tolerance .257±.005, bolts will always fit.

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

Case Study: Hole Tolerancing

Hole diameter established!

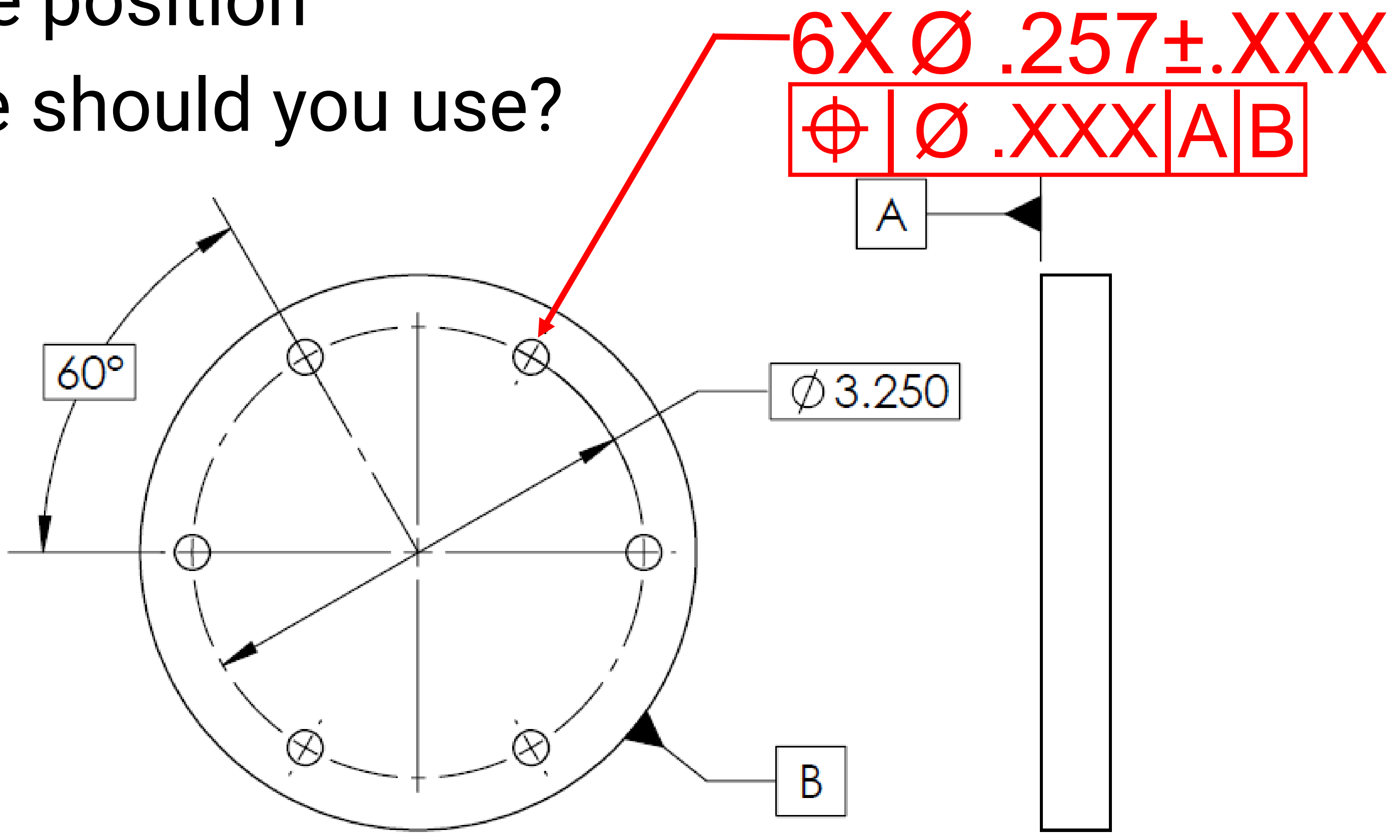
Next: What hole diameter tolerance should you use?



Case Study: Hole Tolerancing

Closely related:

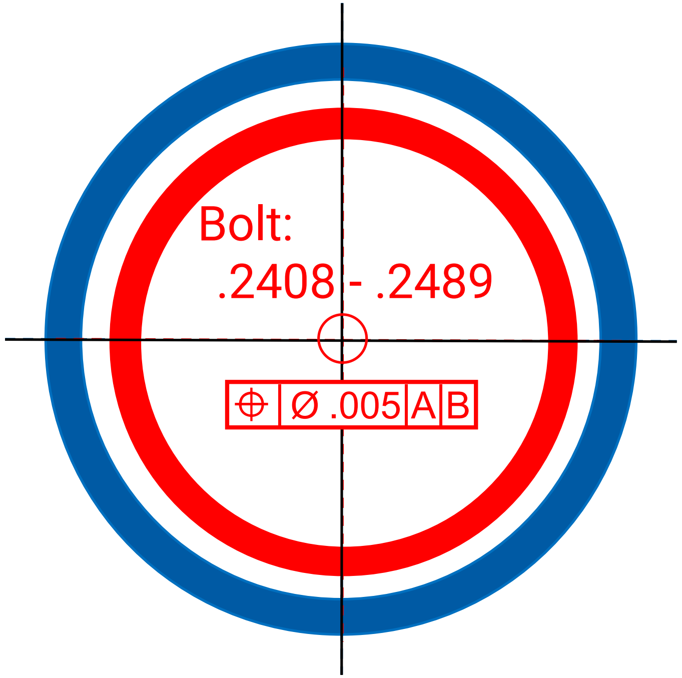
Next: What hole position tolerance should you use?



Case Study: Hole Tolerancing

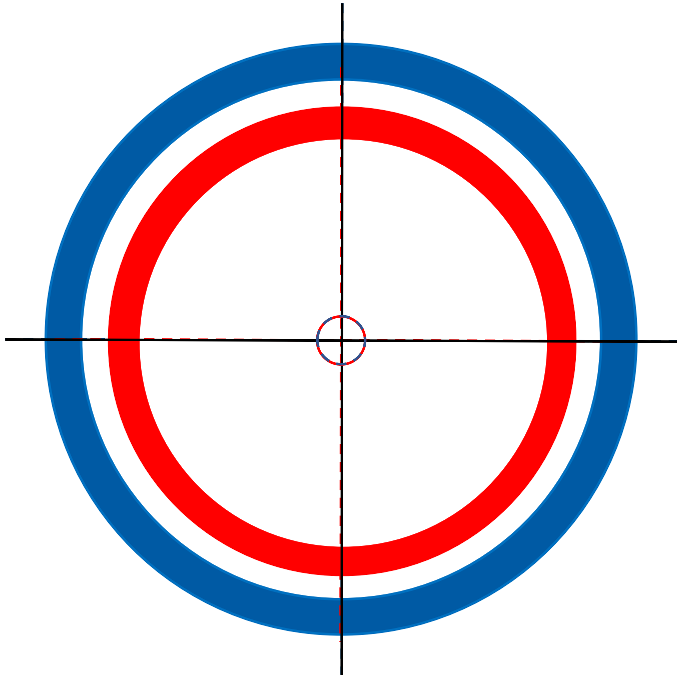
- Hole diameter and position are interrelated to function

Ø .257±.005

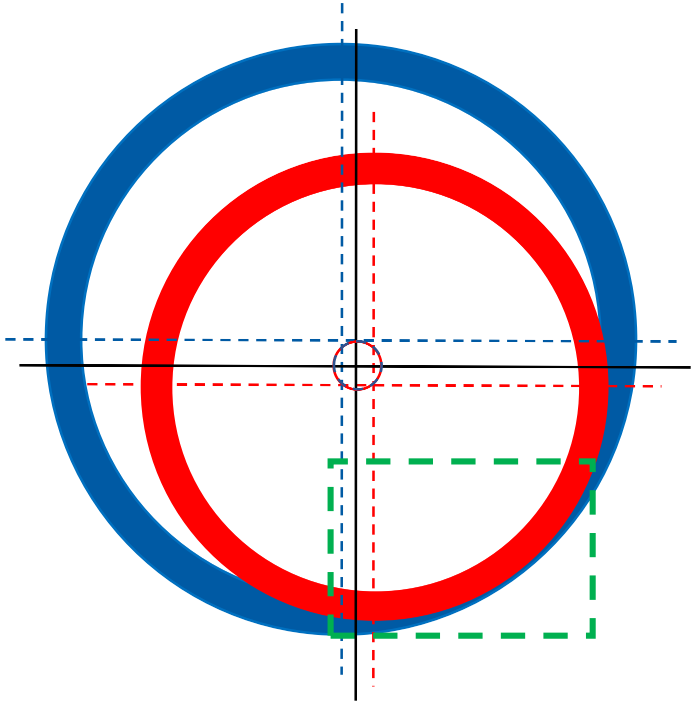


Hole: .252 - .262

Ø .257±.005



⌖ | Ø .005 | A | B



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

Radial Interference!

Case Study: Hole Tolerancing

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

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

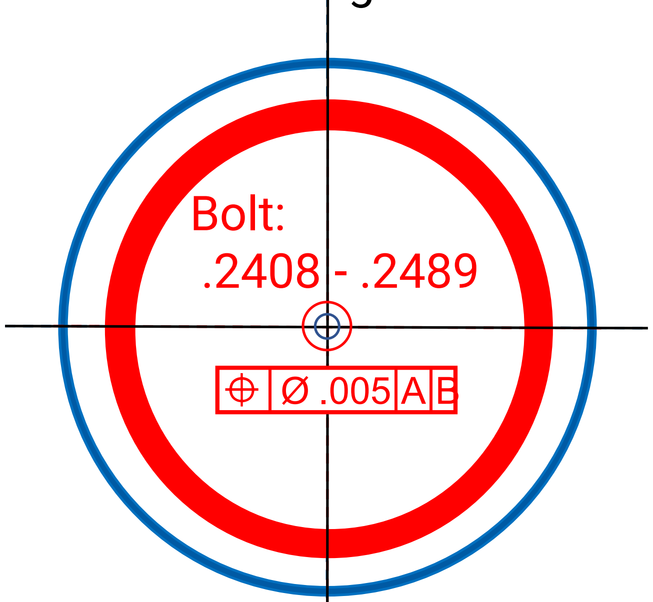
Radial Clearence!

Tighter Tolerances = More Cost

Case Study: Hole Tolerancing

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

Decrease Hole Diameter Tolerance Range

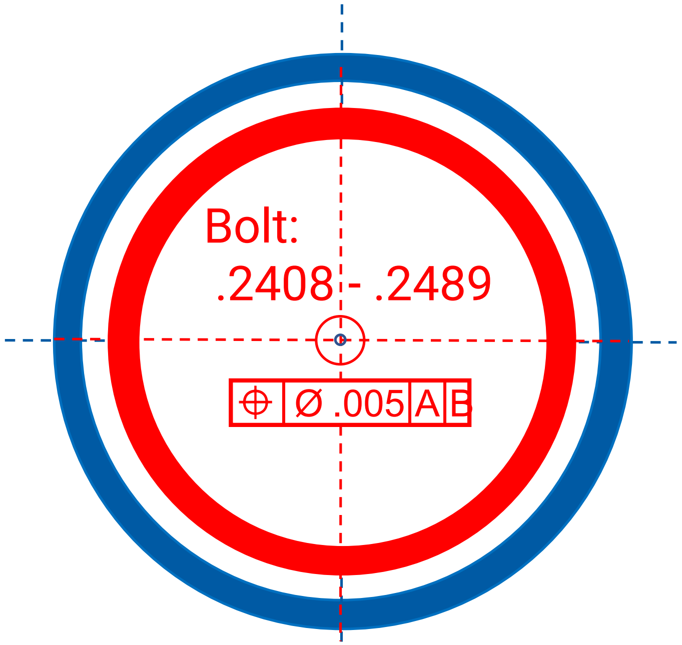


Hole: .252 - .262

Ø .257±.0005

⌖	Ø .0025	A	B

Decrease Hole Position Tolerance

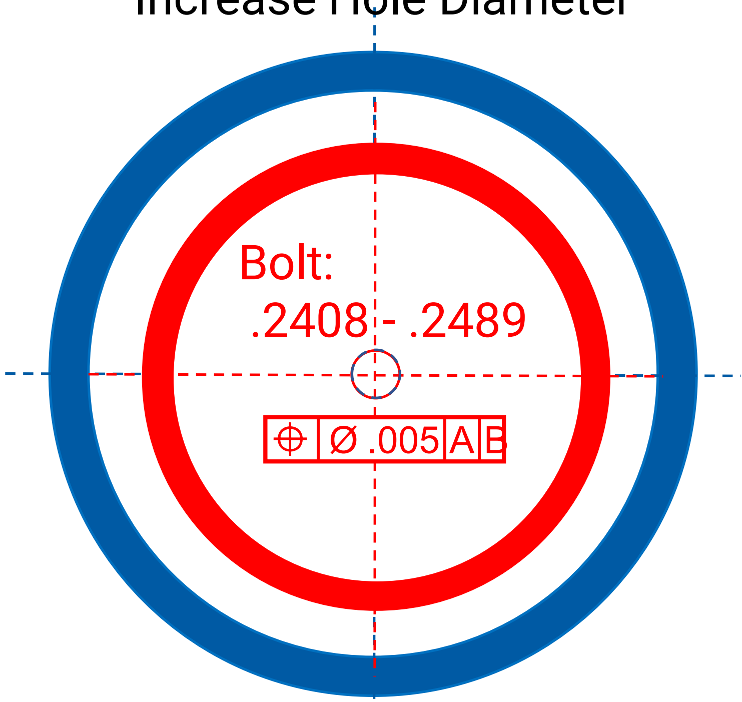


.252" - .262"

Ø .257±.0025

⌖	Ø .0005	A	B

Increase Hole Diameter



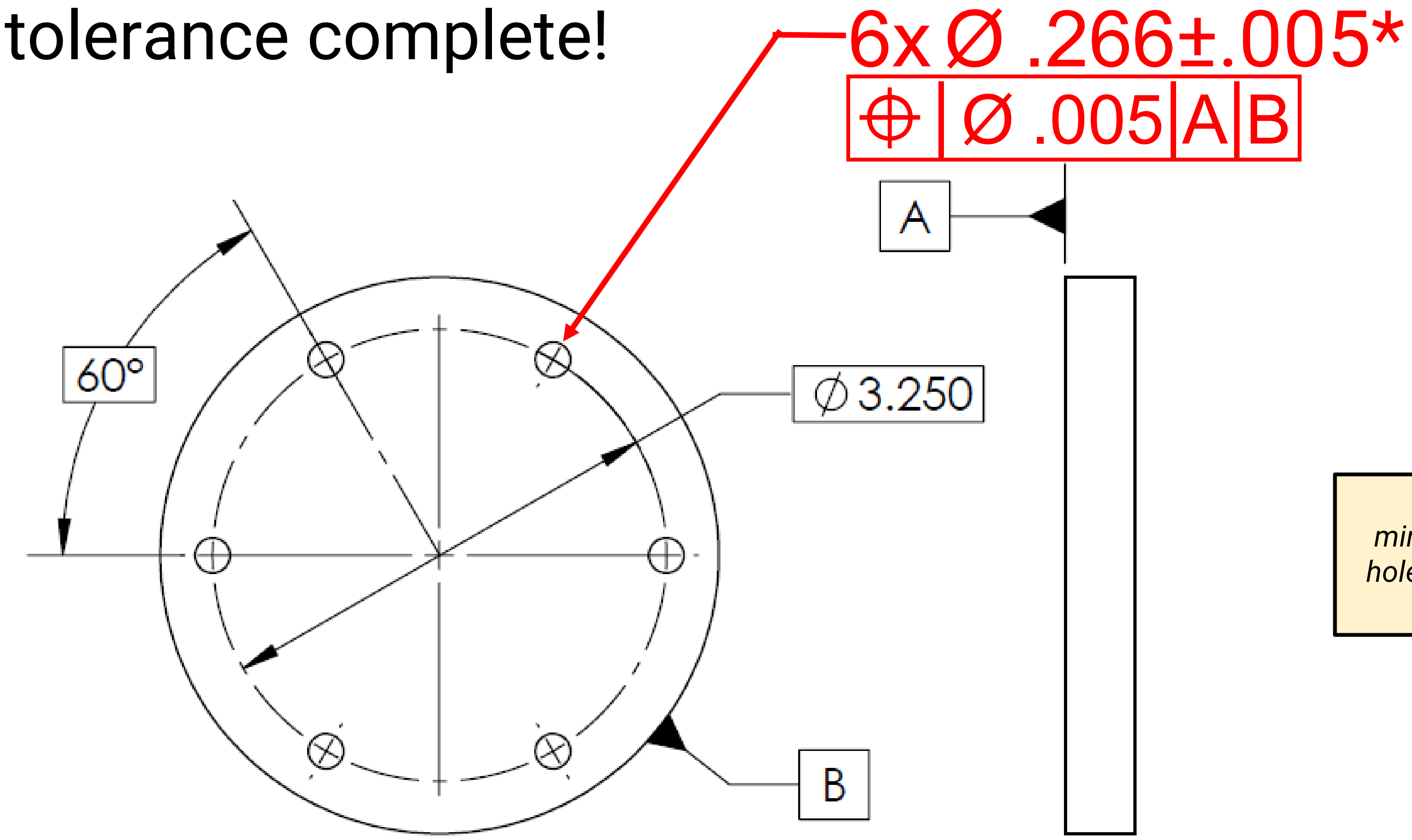
.261" - .271"

Ø .266±.005

⌖	Ø .005	A	B

Case Study: Hole Tolerancing

Bolt pattern tolerance complete!

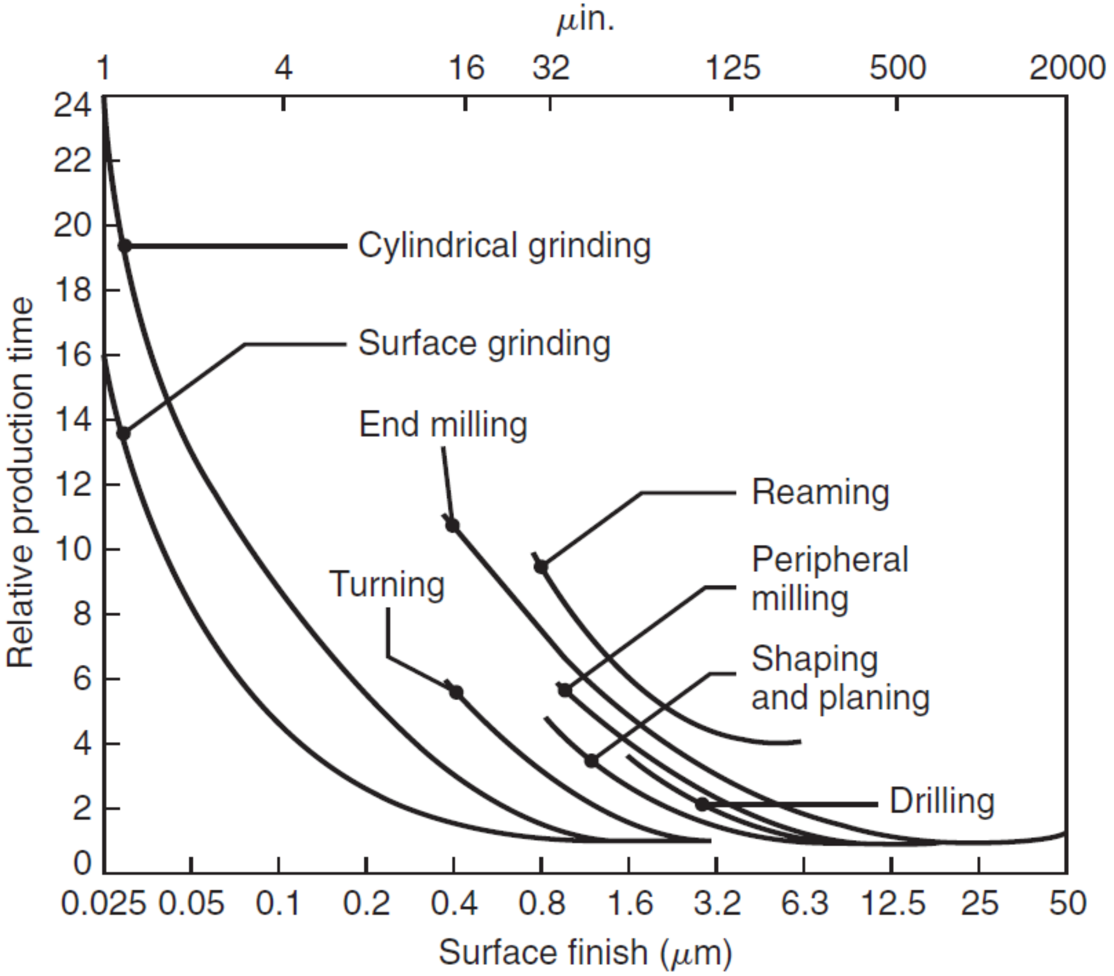


** .266 is listed as the minimum close fit clearance hole for a ¼ fastener in ASME B18.2.8*

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

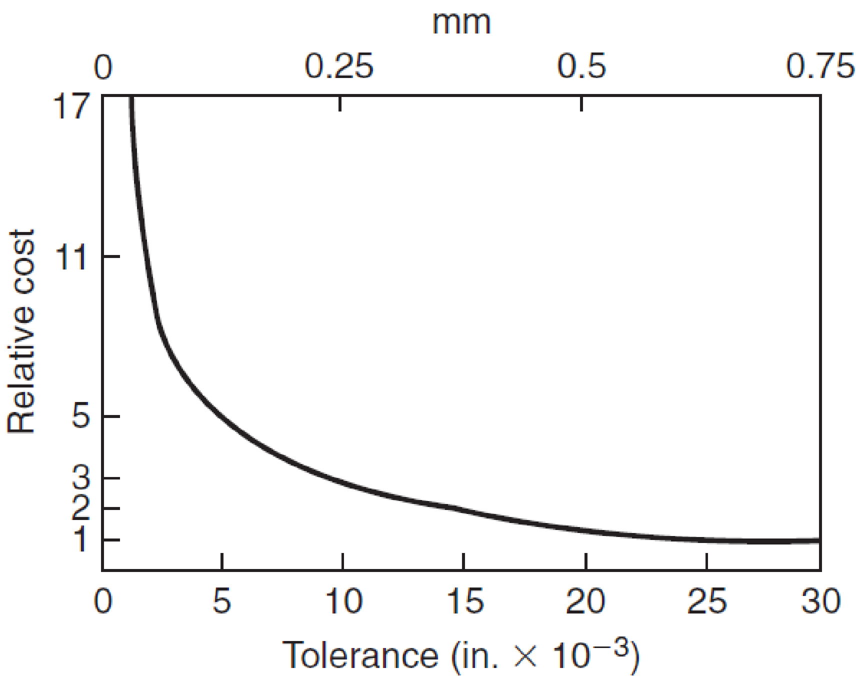
Impacts of geometric specification on cost

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



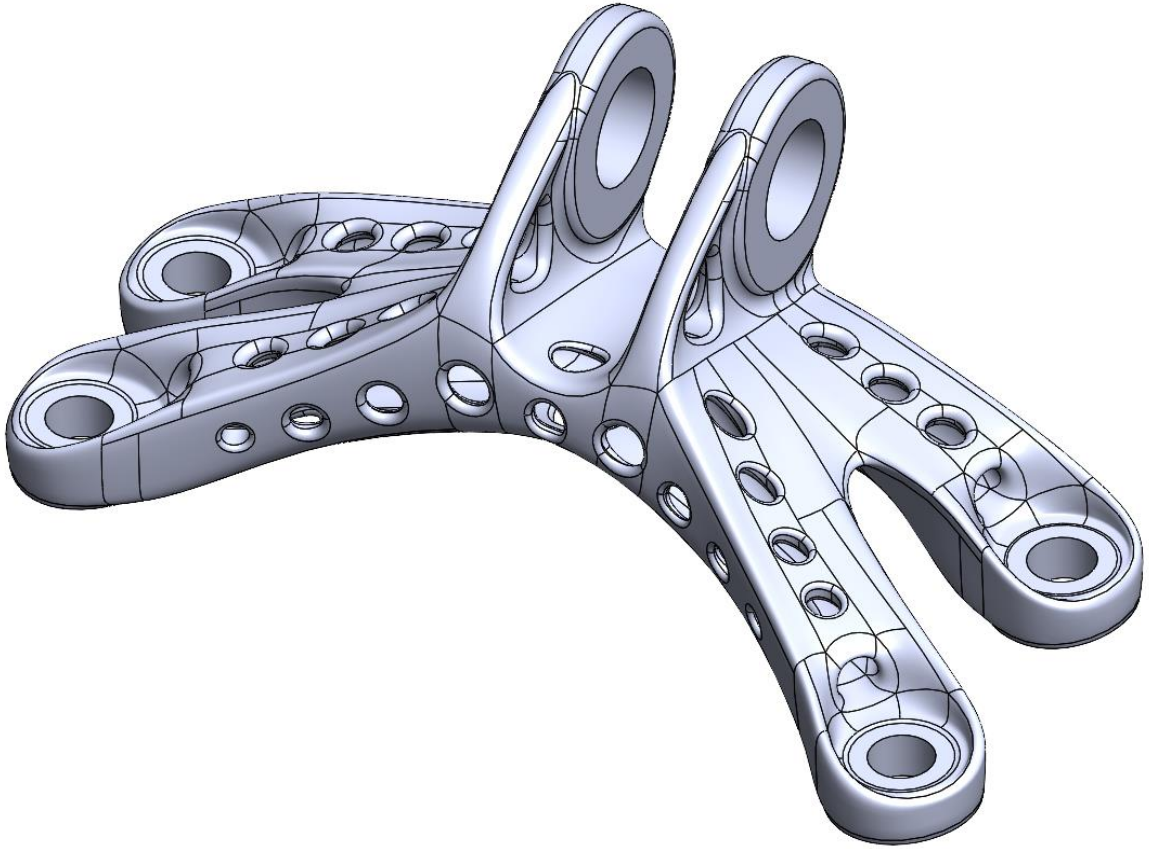
Kalpakjian & Schmid/American Machinist

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



Kalpakjian & Schmid

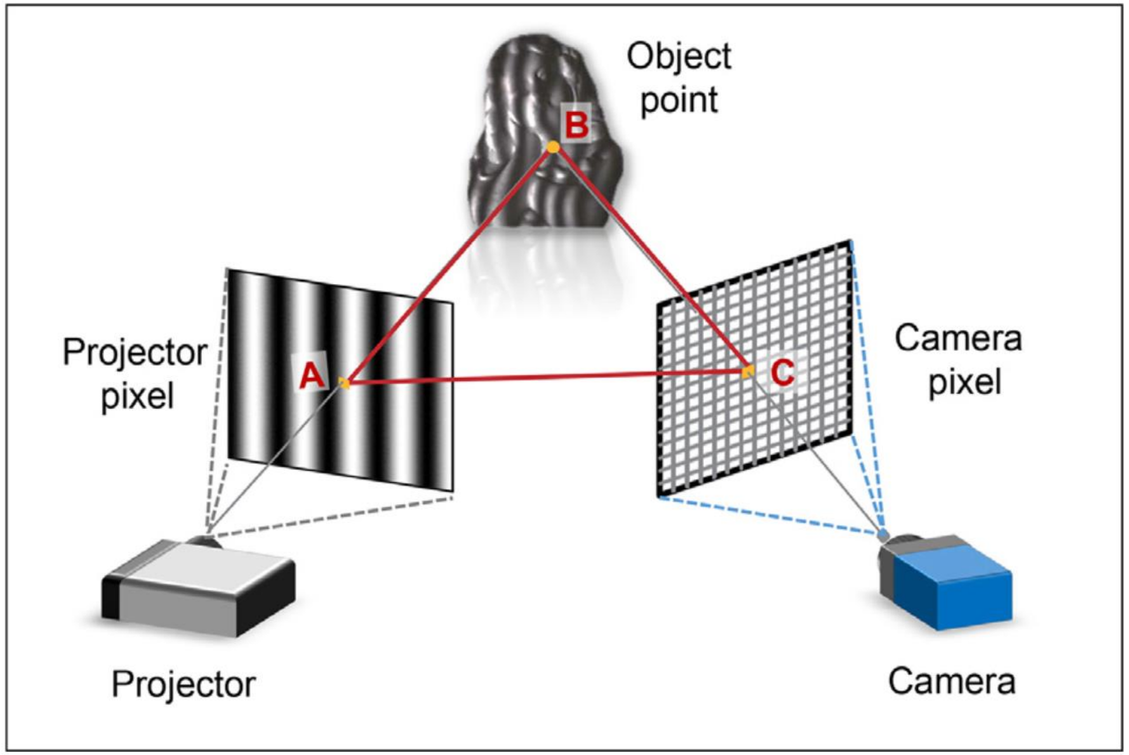
Case-study 2 – Complex AM components



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

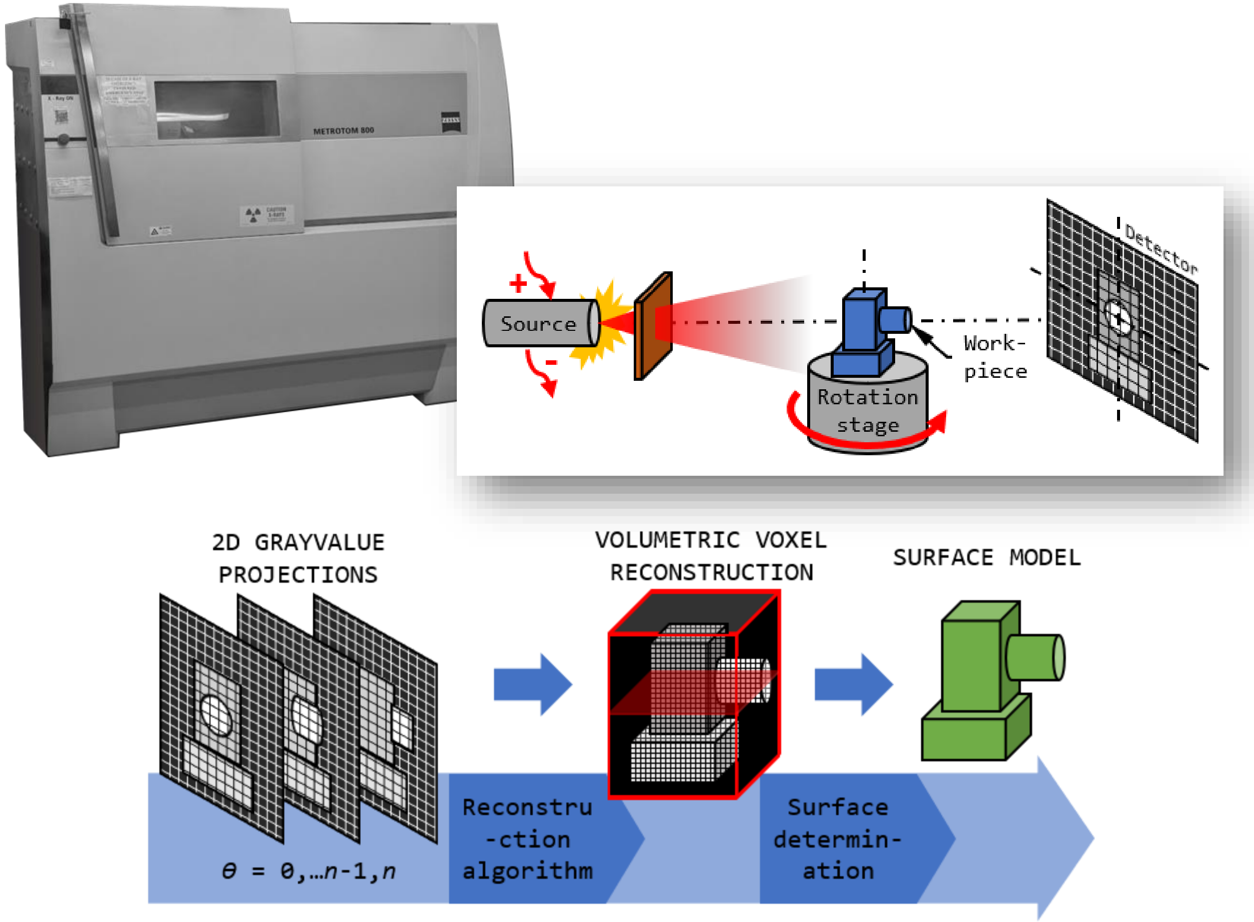
High-Density Coordinate Measurement Systems

Structured Light Scanning (Fringe Projection)



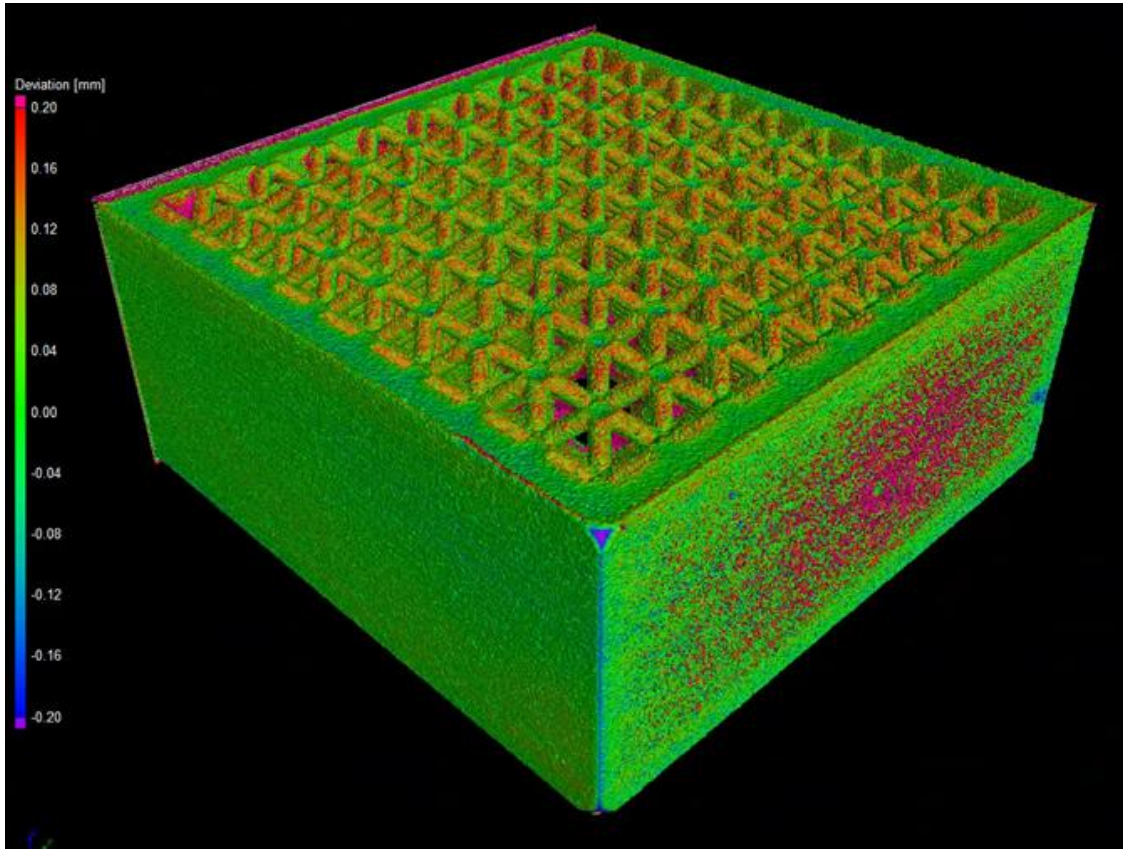
Reproduced from S. Feng et al. (CC BY-NC-ND 4.0)

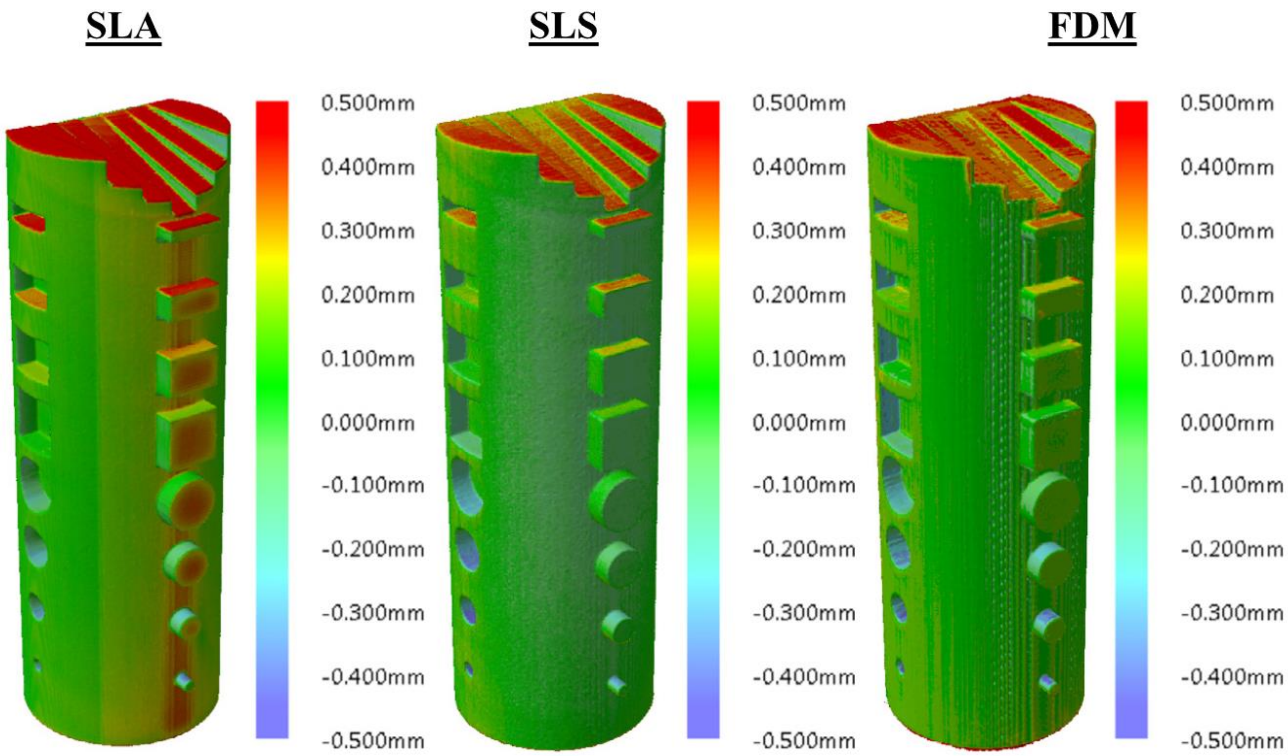
X-ray Computed Tomography



Data processing

Nominal / Actual Comparisons

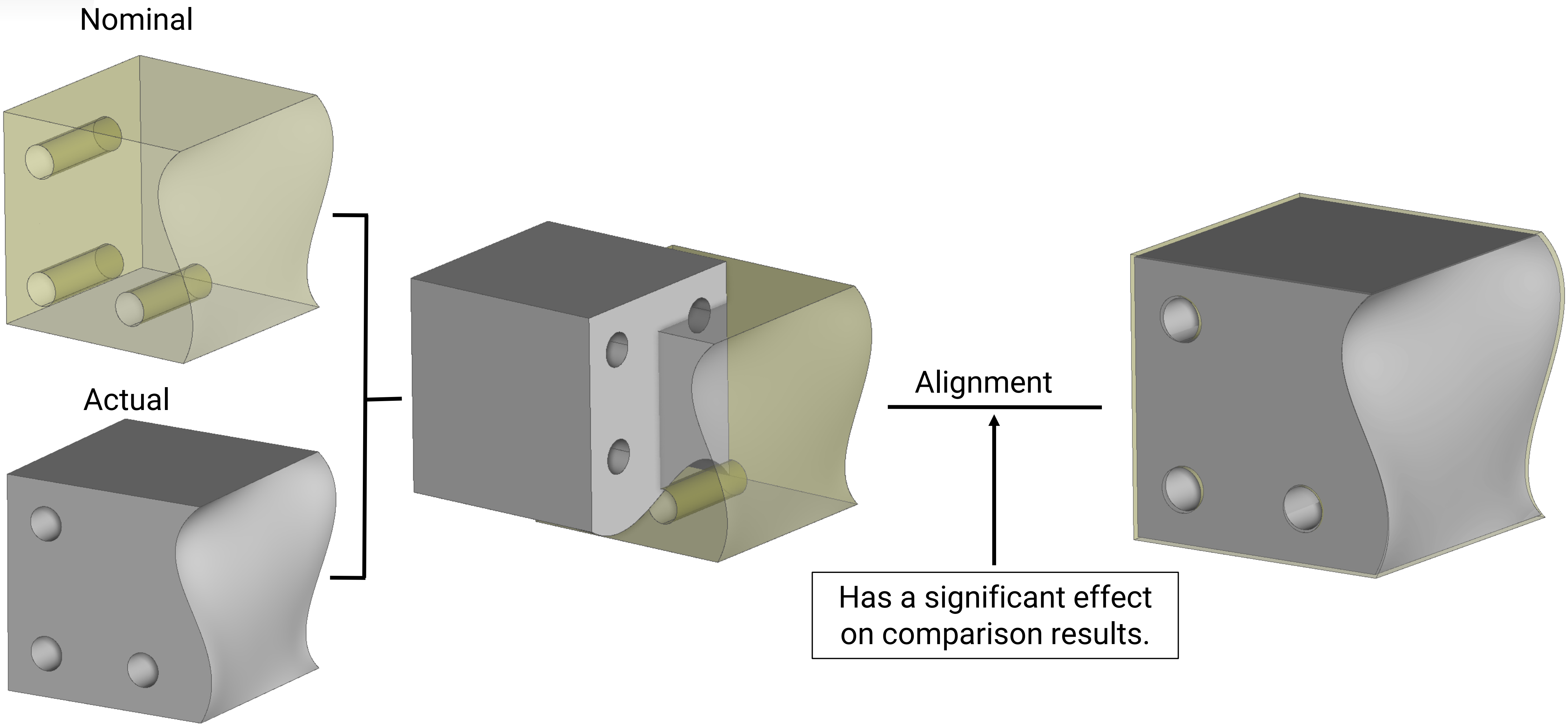




Reproduced from P. Shah et al. (CC BY-NC-ND 4.0)

Lots of dense data… but are they the measurements you want or need?

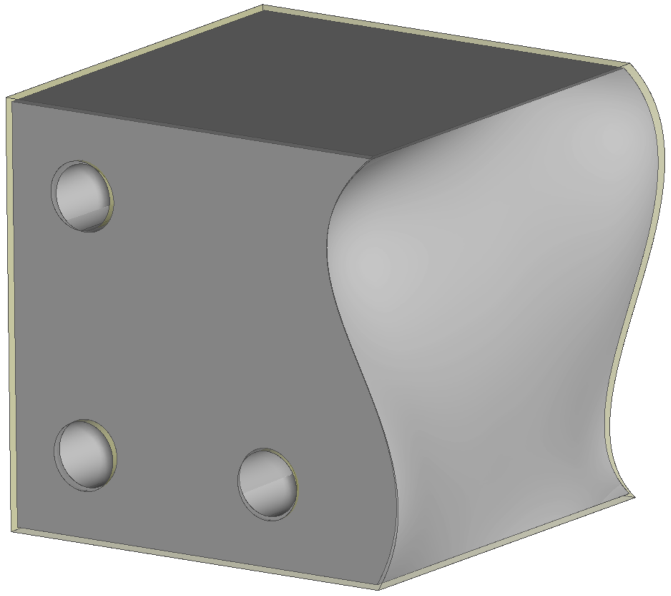
Nominal / Actual Alignment



Nominal / Actual Alignment

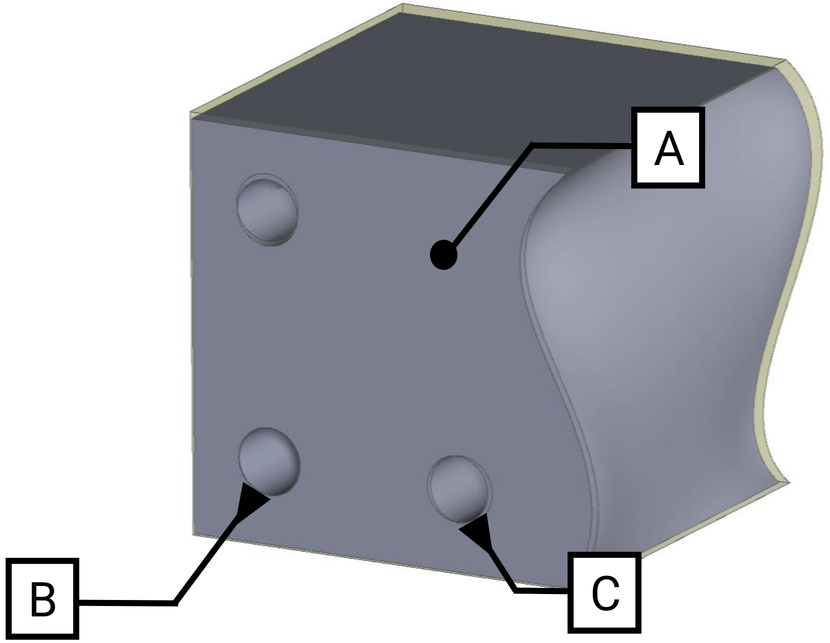
Best Fit Alignment

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



Datum Based Alignment

- Utilizes datums for alignment
- Provides meaningful geometric data



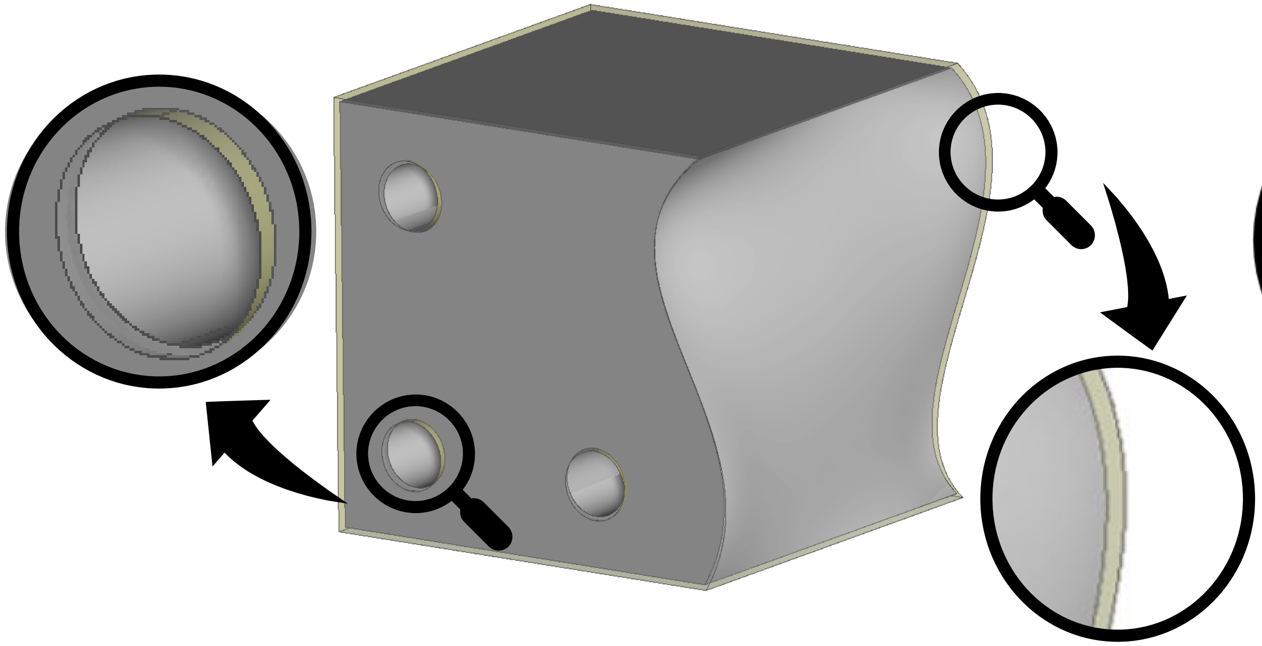
Nominal / Actual Alignment

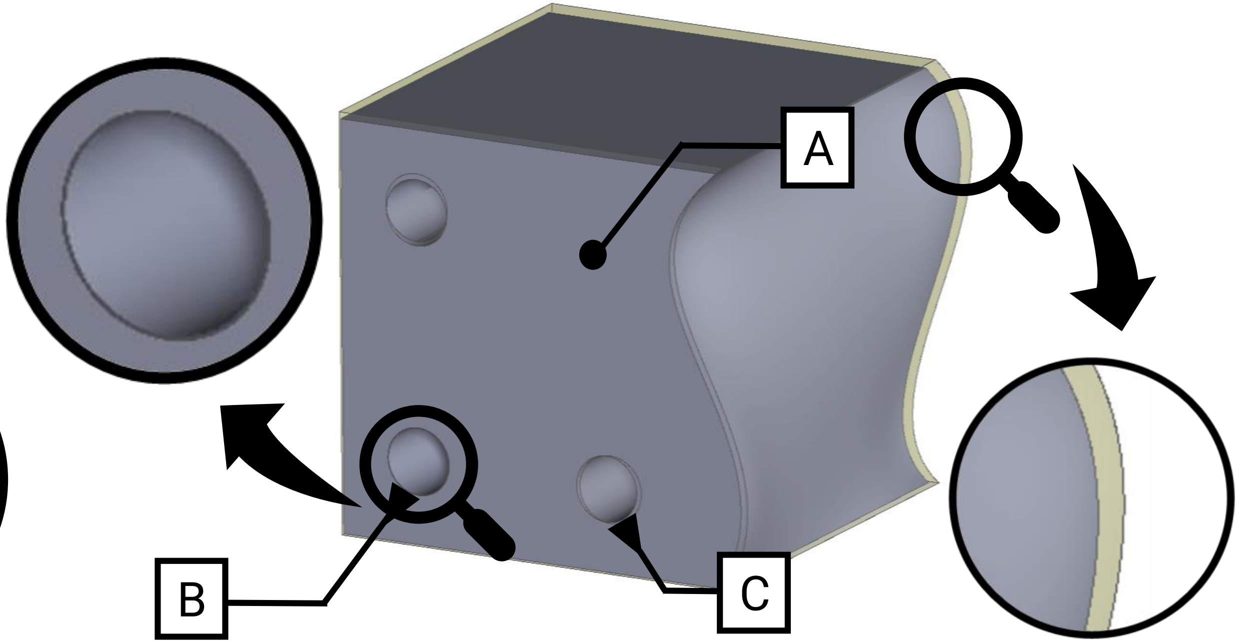
Best Fit Alignment

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

Datum Based Alignment

- Utilizes datums for alignment
- Provides meaningful geometric data

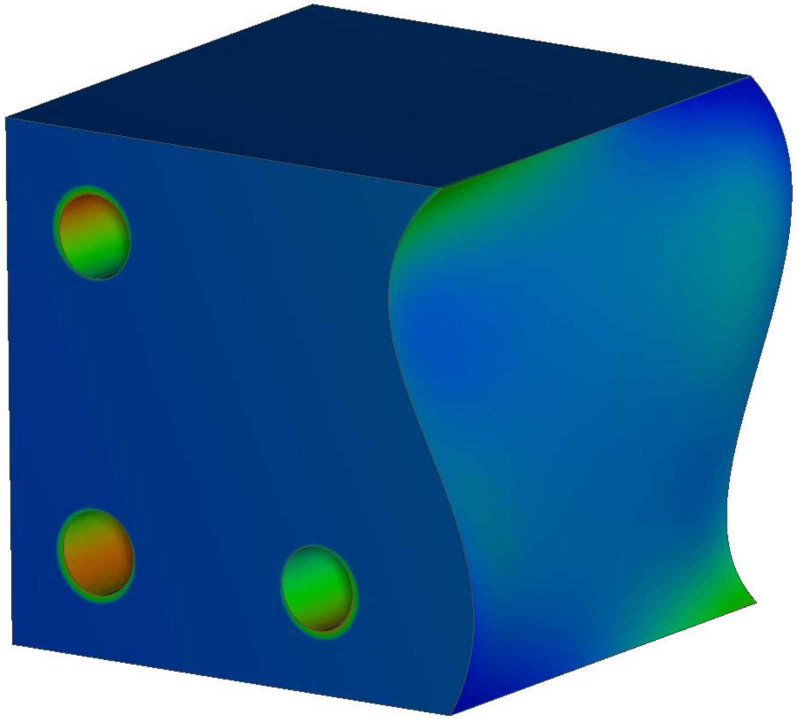




Nominal / Actual Alignment

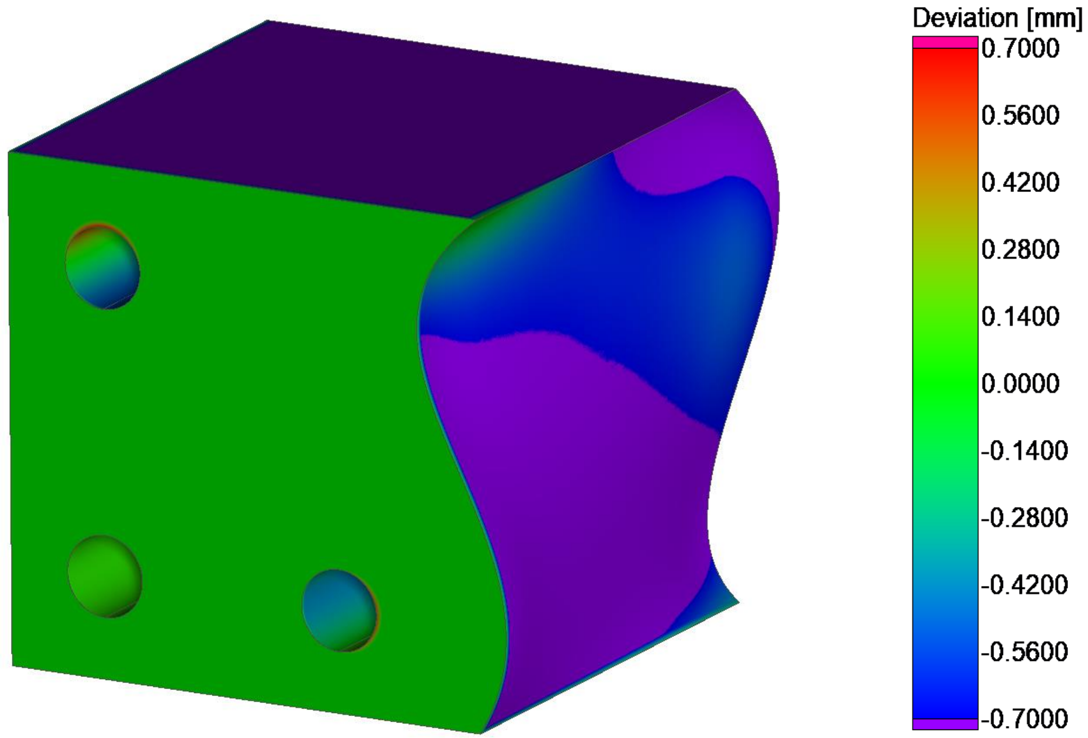
Best Fit Alignment

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

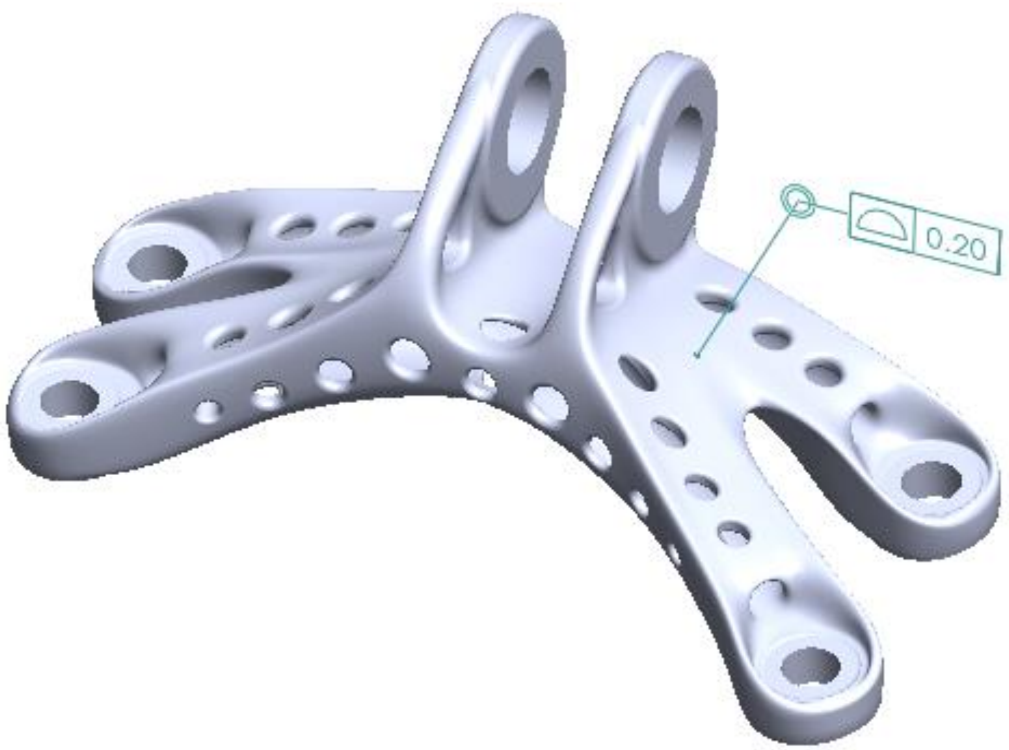


Datum Based Alignment

- Utilizes datums for alignment
- Provides meaningful geometric data

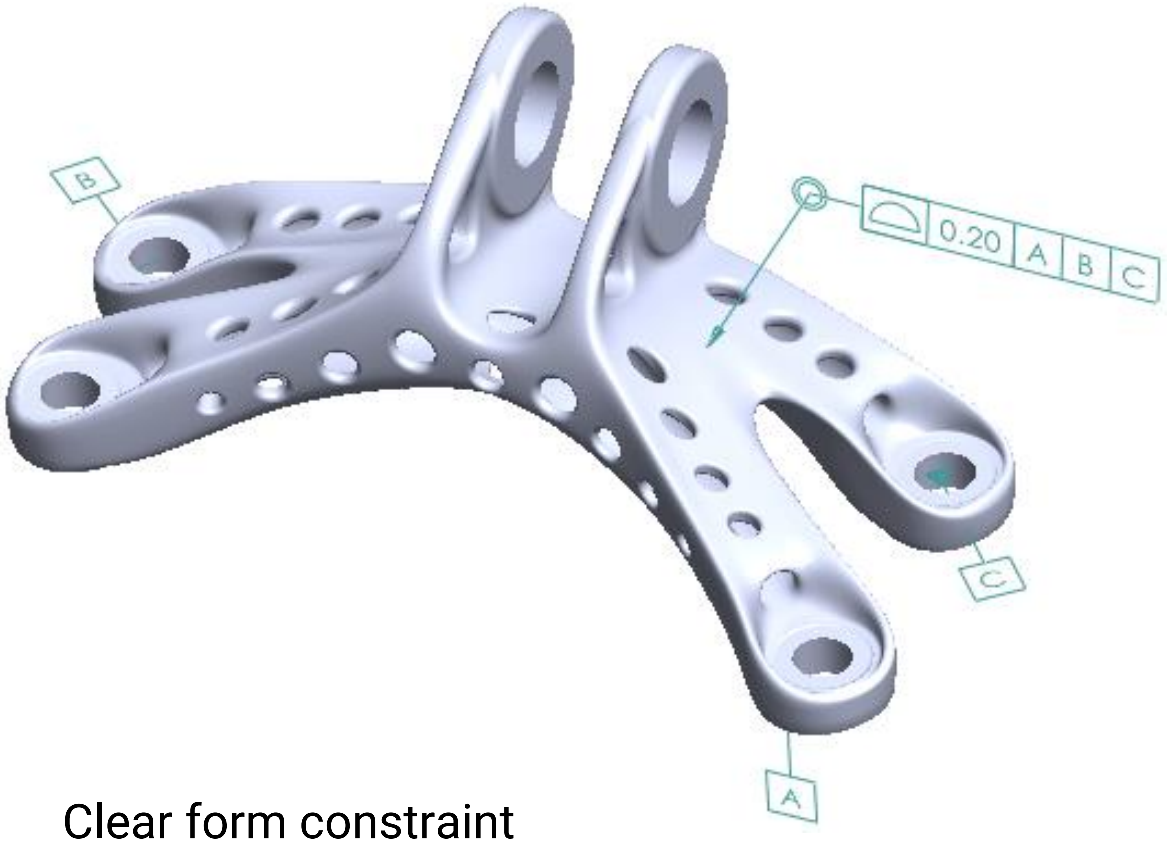


Model-based definition (MBD)



Ambiguous form constraints

- No clear functional requirements

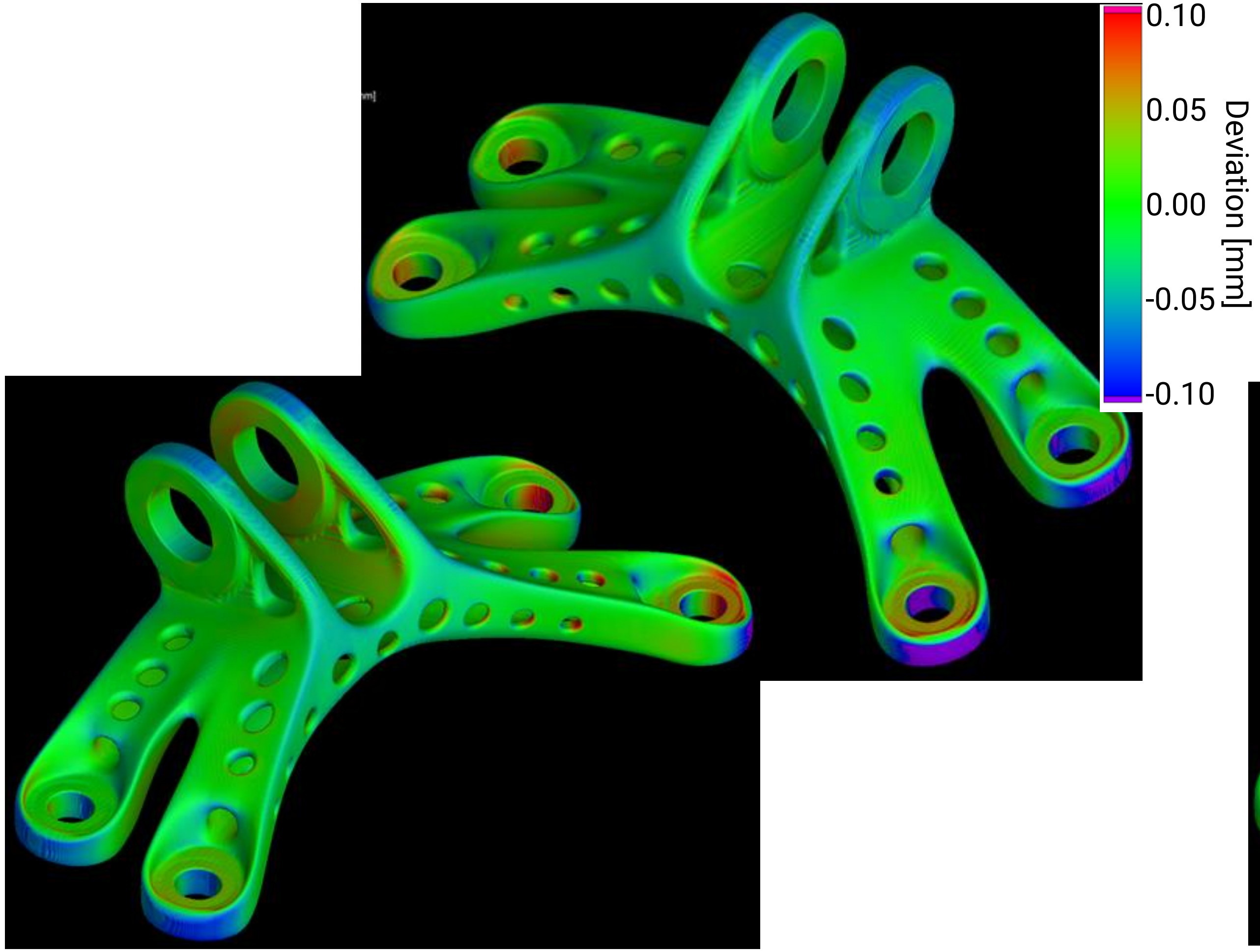


Clear form constraint

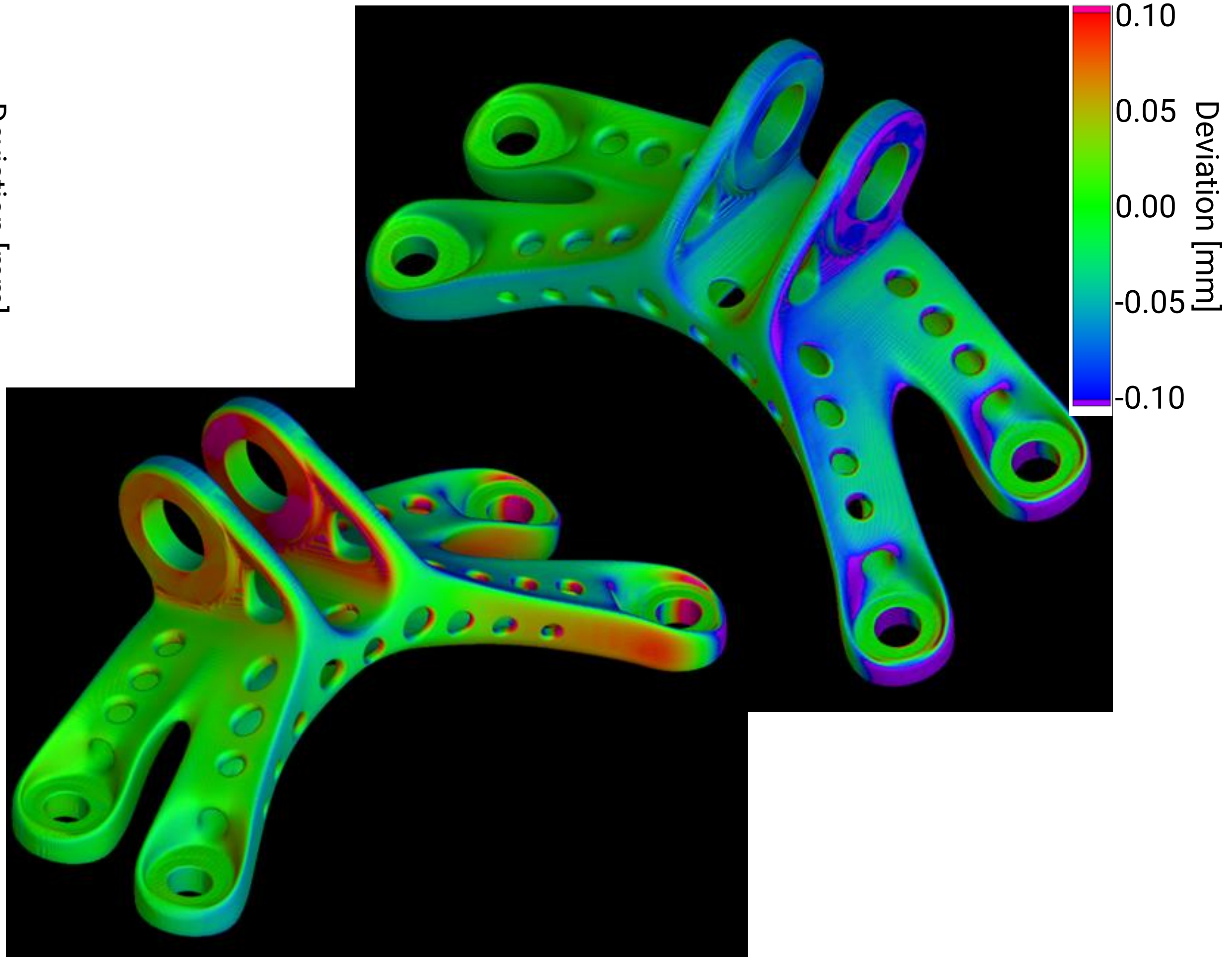
- Datum precedence indicates function

Nominal / Actual Alignment

- Best Fit



- Datum Based

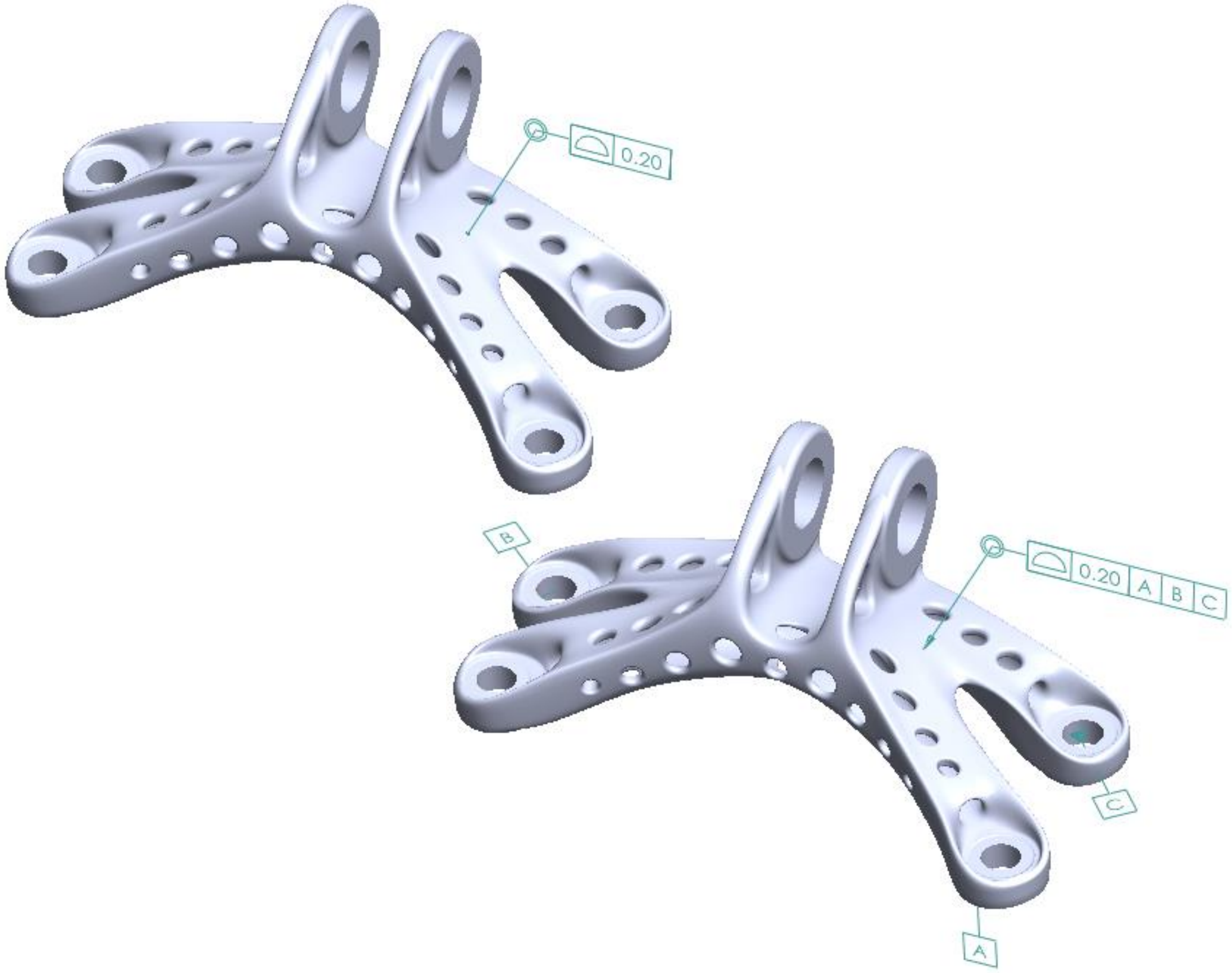


Nominal/Actual can be useful...if defined well!

Dimensioning & tolerancing systems

Why use GD&T?

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

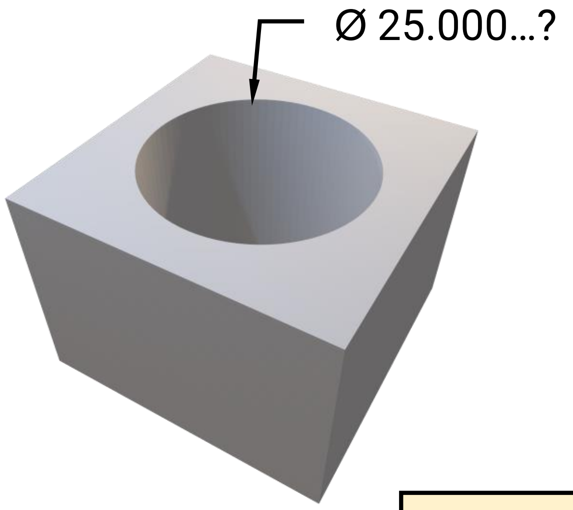


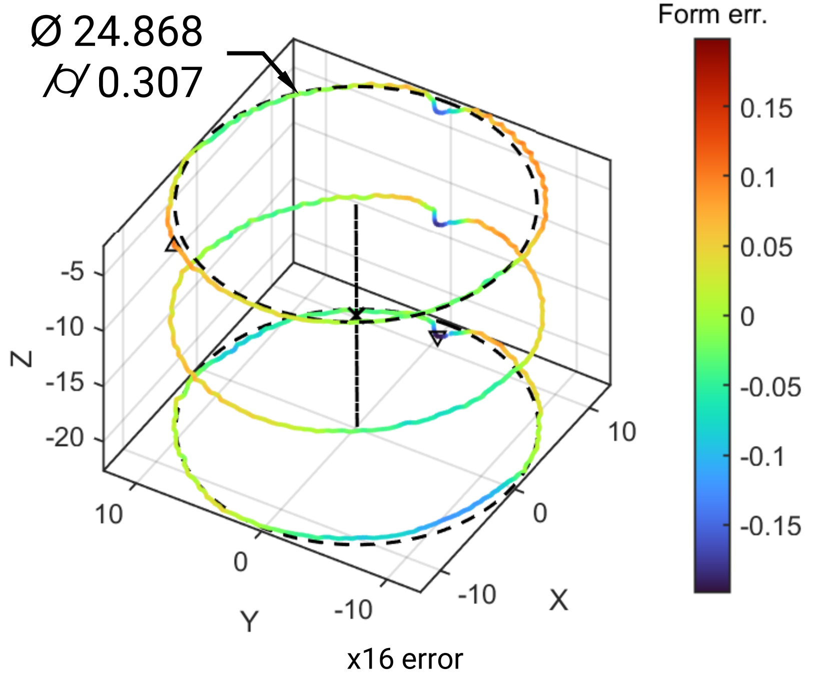
Case study 3 – Imprecision in digital manufacturing

Just because it's digital doesn't mean the manufacturing process is perfect

CAD → FFF* additive manufacturing → Dimensional inspection

.sldprt, .f360… etc. → .STL → .GCODE → (printer)

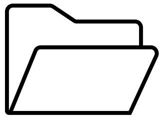




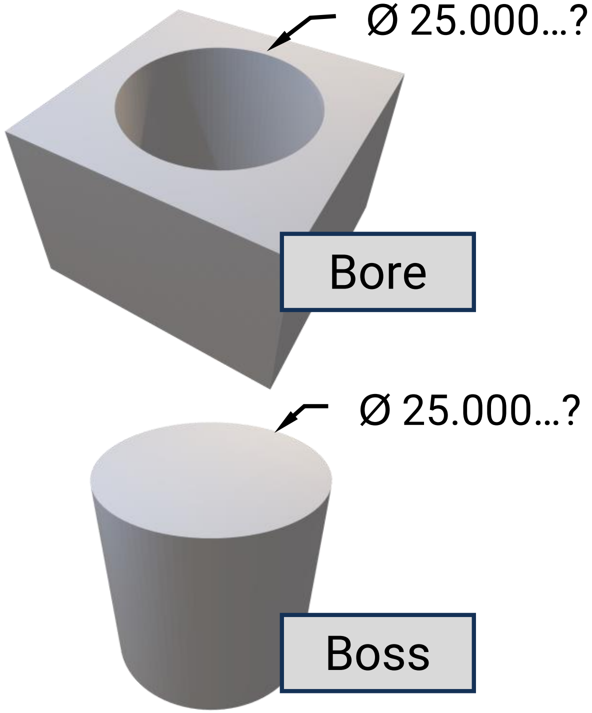
**FFF – Fused filament fabrication*

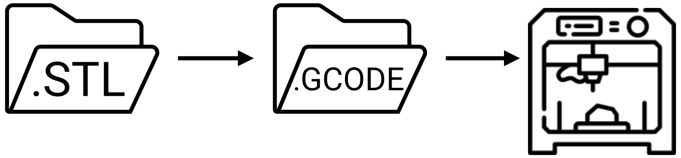
Additive manufacturing example

CAD → FFF additive manufacturing

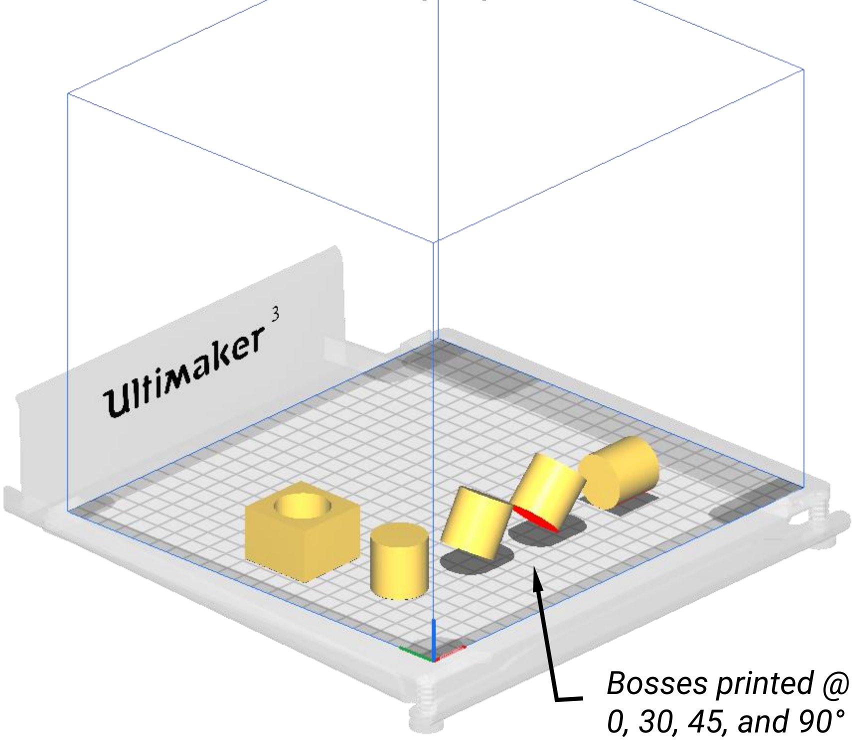


.sldprt, .f360… etc.

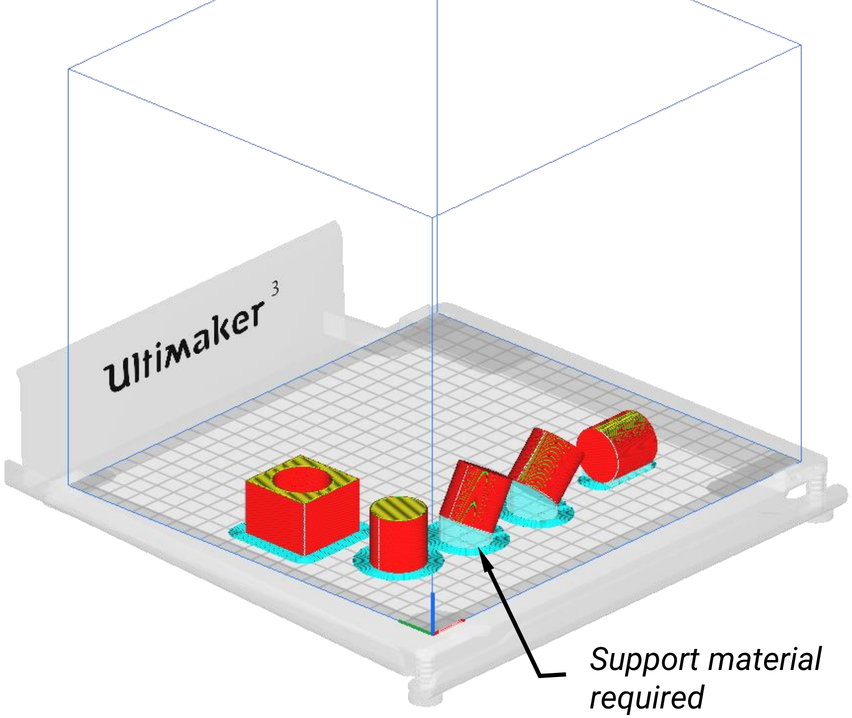




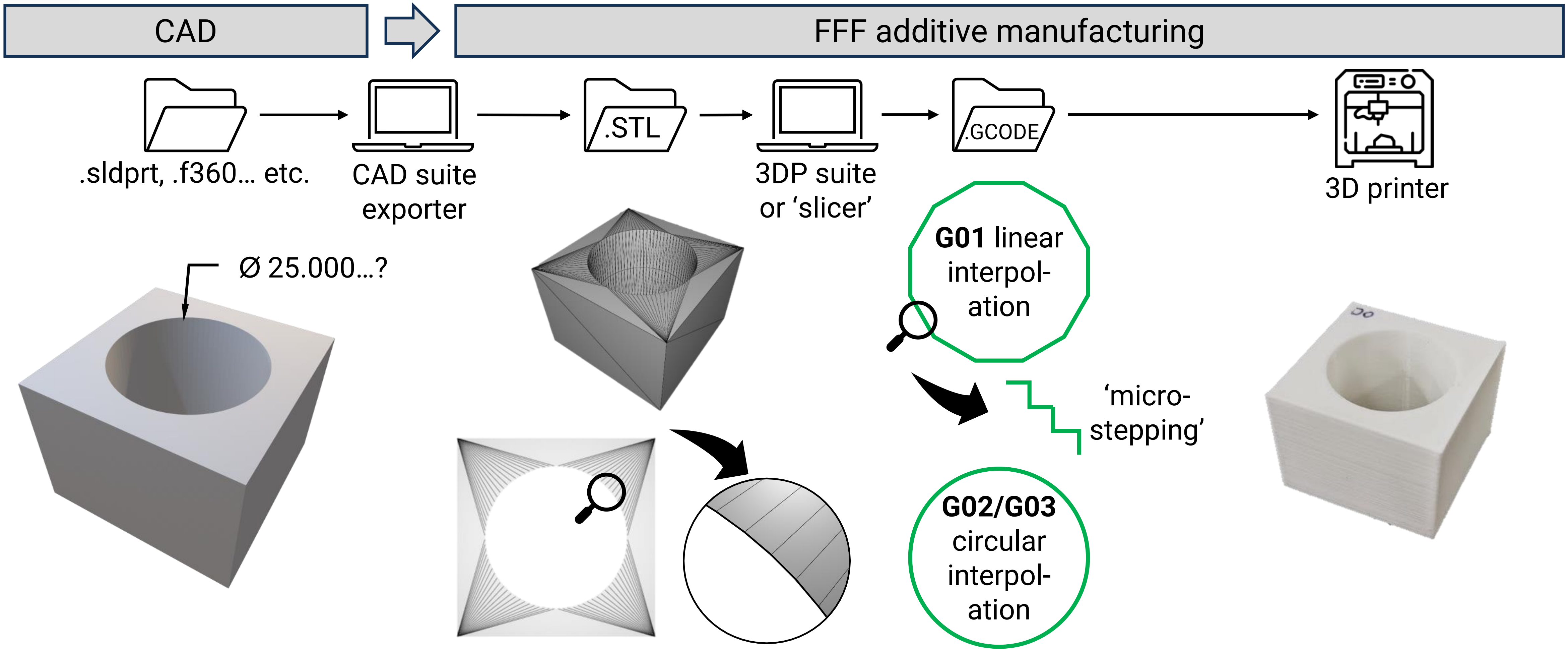
Parts in AM prep software



Parts+support material in AM prep software



Sources of error in digital manufacturing



CAD normally uses 'b-rep' 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…

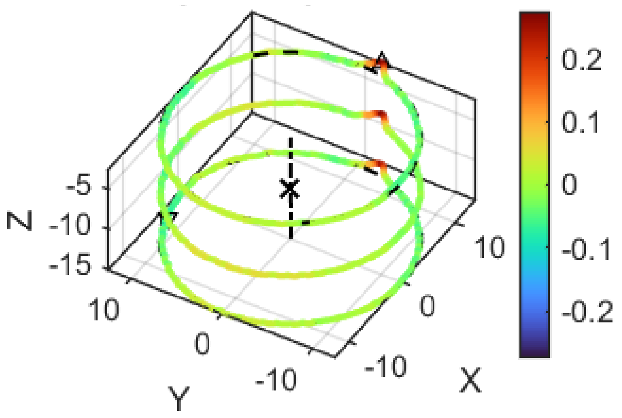
Manufacturing approach determines dimensional error

Designer specification informs manufacturing approach!

Ø24.87, ⌭ 0.31

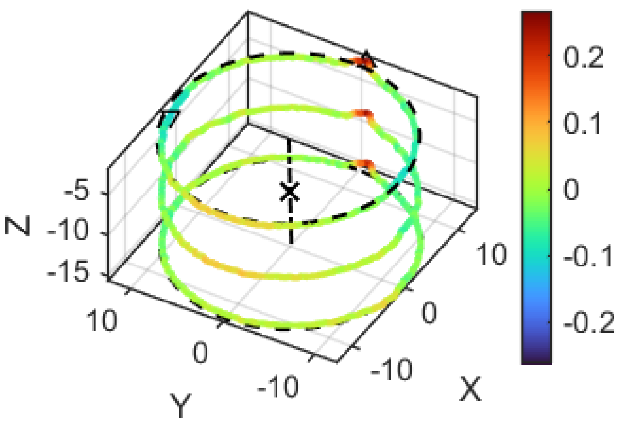
Axis @ 0° to B dir.

Ø24.86, ⌭ 0.36



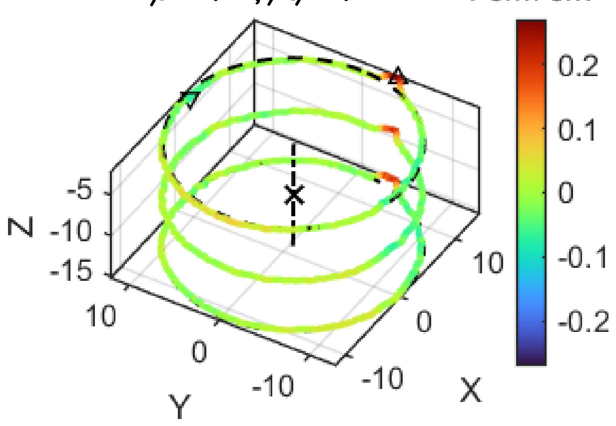
Axis @ 30° to B dir.

Ø24.87, ⌭ 0.38



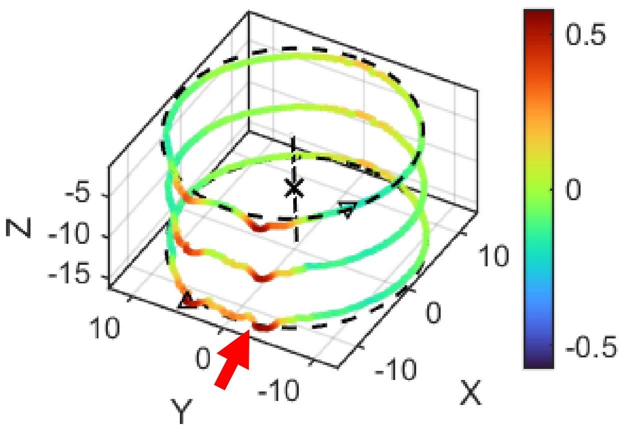
Axis @ 45° to B dir.

Ø24.87, ⌭ 0.35



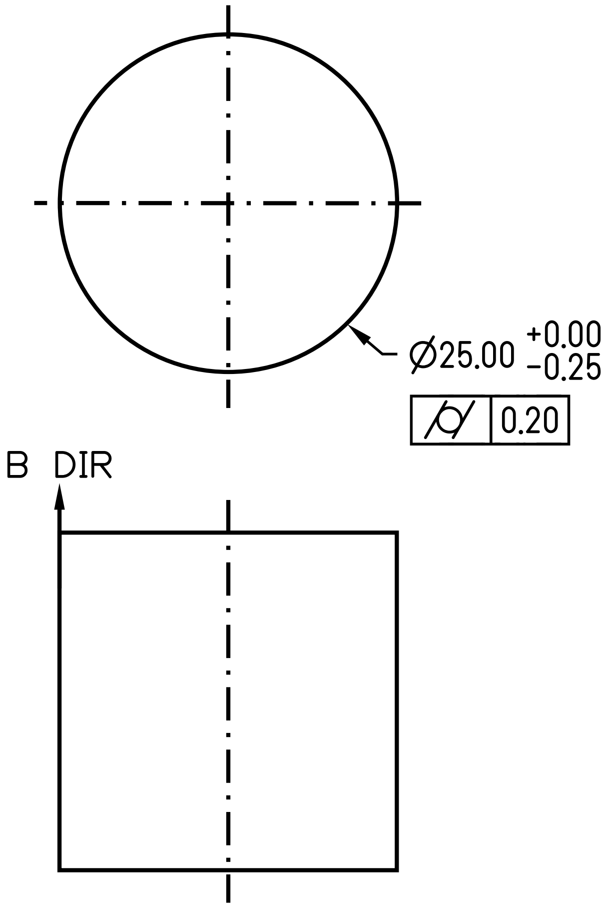
Axis @ 90° to B dir.

Ø25.11, ⌭ 0.79

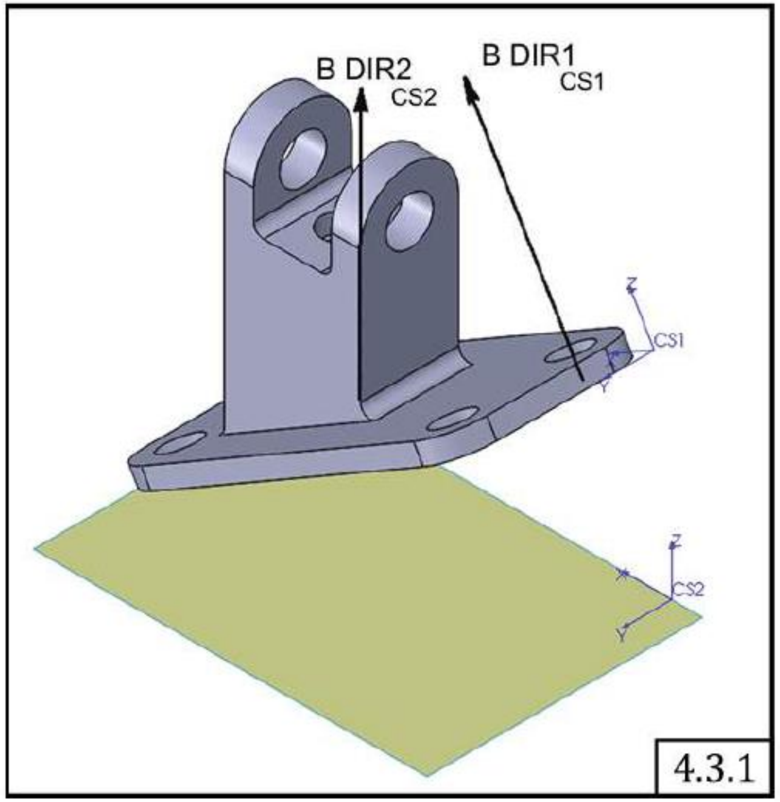


GD&T practices for AM components (a brief look)

This product specification ensures the desired result



Applied to a more complex design



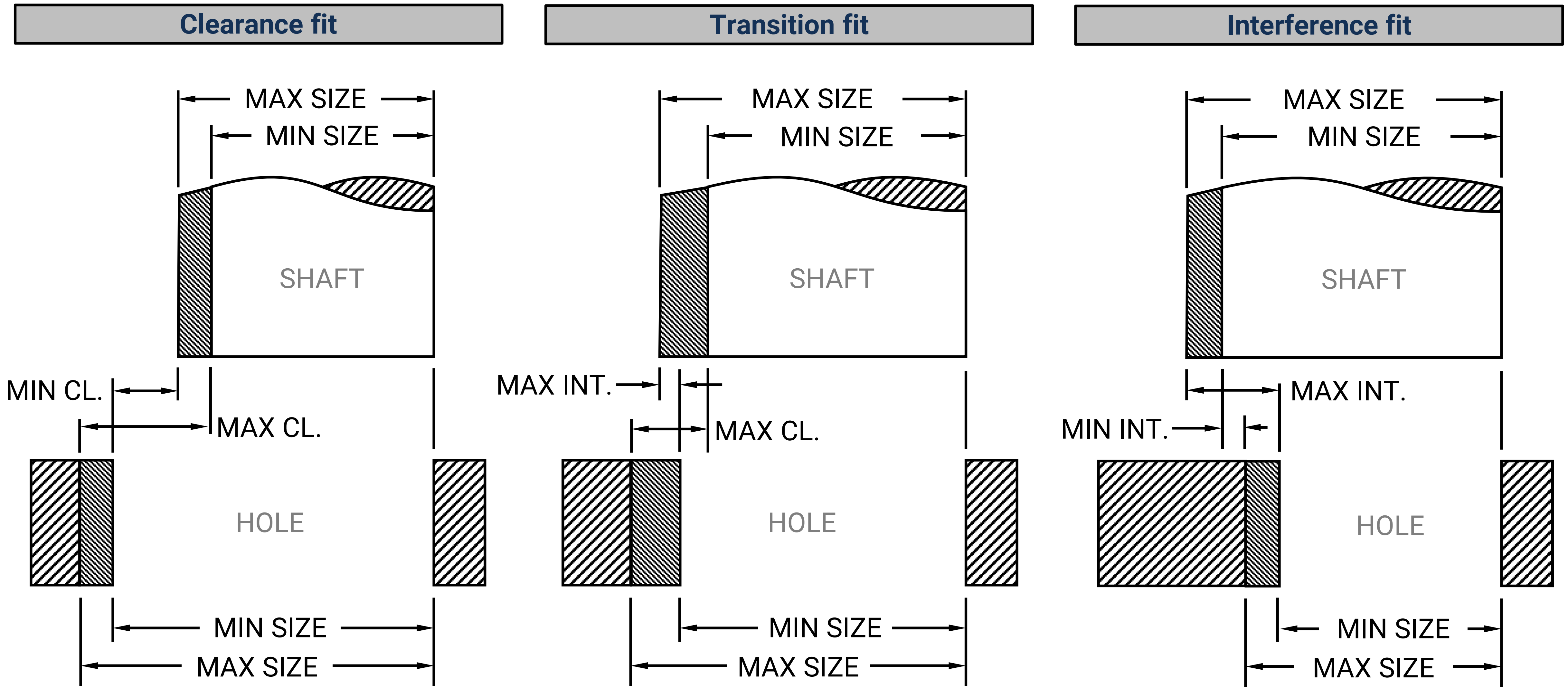
ASME Y14.46-2022

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 B4.2-1994			

References and continuing education

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)

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.

Additional References

- D.A. Maisano, et al. Dimensional measurements in the shipbuilding industry: on-site comparison of a state-of-the-art laser tracker, total station and laser scanner. Prod. Eng. Res. Devel. 17, 625–642 (2023). https://doi.org/10.1007/s11740-022-01170-7
- S. Feng et al., “Calibration of fringe projection profilometry: A comparative review,” Optics and Lasers in Engineering, vol. 143, p. 106622, Aug. 2021, doi: 10.1016/j.optlaseng.2021.106622.
- P. Shah, R. Racasan, and P. Bills, “Comparison of different additive manufacturing methods using computed tomography,” *Case studies in nondestructive testing and evaluation*, vol. 6, pp. 69–78, 2016. https://doi.org/10.1016/j.csndt.2016.05.008