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)

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

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

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

* Certain commercial entities, equipment, or materials may be identified in this document in order to describe an experimental procedure or concept adequately. Such identification is not intended to imply recommendation or endorsement by the National Institute of Standards and Technology, nor is it intended to imply that the entities, materials, or equipment are necessarily the best available for the purpose.
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.

<table>
<thead>
<tr>
<th>FEATURE TYPE</th>
<th>ON THE DRAWING</th>
<th>DATUM FEATURE</th>
<th>DATUM AND TRUE GEOMETRIC COUNTERPART</th>
<th>DATUM AND CONSTRAINING DEGREES OF FREEDOM</th>
</tr>
</thead>
<tbody>
<tr>
<td>PLANAR (a)</td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
</tr>
<tr>
<td>WIDTH (d)</td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
</tr>
<tr>
<td>SPHERICAL (c)</td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
</tr>
<tr>
<td>CYLINDRICAL (d)</td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
<td><img src="https://via.placeholder.com/150" alt="Diagram" /></td>
</tr>
</tbody>
</table>

*ASME Y14.5:2018, 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.’

*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

4. Size
3.1 Form
3.2 Orientation
3.3 Location

5. Geometric characteristics

1. Datums

2. Datum ref. frame

3.4 Basic dim.

Rule #1

The envelope principle

1.00±0.10

Direct dim.

5. Geometric characteristics

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00

1.00±0.10

Datum

Datums

1.00

3.3 Location

Datum ref. frame

Feature control frame

1.00
## Geometric characteristics

<table>
<thead>
<tr>
<th>Geometric characteristic</th>
<th>Symbol</th>
<th>Geometric attribute</th>
<th>Datum referencing?</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straightness</td>
<td>¶</td>
<td>Form</td>
<td>No</td>
</tr>
<tr>
<td>Flatness</td>
<td>®</td>
<td>Form</td>
<td>No</td>
</tr>
<tr>
<td>Circularity</td>
<td>-</td>
<td>Form</td>
<td>No</td>
</tr>
<tr>
<td>Cylindricity</td>
<td>³</td>
<td>Form</td>
<td>No</td>
</tr>
<tr>
<td>Profile of a line</td>
<td>¹</td>
<td>Profile (location, orientation, size, &amp; form)</td>
<td>Sometimes datum referencing</td>
</tr>
<tr>
<td>Profile of a surface</td>
<td>₀</td>
<td>Profile (location, orientation, size, &amp; form)</td>
<td>Sometimes datum referencing</td>
</tr>
<tr>
<td>Angularity</td>
<td>²</td>
<td>Orientation</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Perpendicularity</td>
<td>¼</td>
<td>Orientation</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Parallelism</td>
<td></td>
<td>Location</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Position</td>
<td>₆</td>
<td>Location</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Circular runout</td>
<td>½</td>
<td>Runout (location of a cylinder)</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Total runout</td>
<td>¾</td>
<td>Runout (location of a cylinder)</td>
<td>Datum referencing</td>
</tr>
</tbody>
</table>
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: Example

1. Understand the functionality of the part. Identify features that control function and assembly.
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 $[,}$, $\pm^*$)

*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 ($\pm$)

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.
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
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.
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.
Digital product definition

Model-based definition (MBD)

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

  - 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

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. {º ~ _ | 1` .

*This is an illustrative example. Drafters should use verbiage appropriate to their company and application.
Dimensional inspection for GD&T
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,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…
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

This on the drawing...

TIR is the highest (most +) reading minus the lowest (most -) reading
Comparators - Perpendicuarity measurement

Note: The entire workpiece surface must be inspected. TIR is directly read as perpendicularity.

This on the drawing...

*Must be calibrated prior

Perpendicularity @ one point

Workpiece

Indicator stand

Surface plate
Comparators: Flatness measurement

Once ‘leveled’ flatness is the TIR
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...

Indicator reading = position error from -B- @ the point inspected

Primary datum contacts @ 3 pts.
Secondary datum contacts @ 2 pts.
Case studies

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

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?

Your Part Drawing!
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?

6X:.257 ±.XXX
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

<table>
<thead>
<tr>
<th>Hole: .252 - .262</th>
<th>Bolt: .2408 - .2489</th>
</tr>
</thead>
<tbody>
<tr>
<td>±.005</td>
<td>±.005</td>
</tr>
</tbody>
</table>

\[ \frac{1}{2} \left( \frac{0.252}{0.005} - \left( \frac{0.2489}{0.005} \right) \right) = -0.0035 \]

Radial Interference!
Case Study: Hole Tolerancing

Tolerance Components

\[ \frac{1}{2} \left( \left( \{0.257 - 0.005\} - 0.005 \right) - (0.2489 + 0.005) \right) = -0.0035 \]

Nominal Diameter | Diameter Tolerance

\[ \frac{1}{2} \left( \left( \{0.257 - 0.001\} - 0.001 \right) - (0.2489 + 0.005) \right) = 0.0006 \]

Smallest Hole Diameter | Largest Bolt Diameter

Hole Position Tolerance | Bolt Position Tolerance

Changing tolerances

Tighter Tolerances = More Cost

Radial Interference! Radial Clearance!
Case Study: Hole Tolerancing

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

Decrease Hole Diameter Tolerance Range

Hole: 0.252 - 0.262

.257 ±.0005

Bolt: 0.2408 - 0.2489

#::005|A|B

Decrease Hole Position Tolerance

.257 ±.0025

Bolt: 0.2408 - 0.2489

#::005|A|B

Increase Hole Diameter

Bolt: 0.2408 - 0.2489

#::005|A|B

.266±.005
Case Study: Hole Tolerancing

Bolt pattern tolerance complete!

\[ \frac{1}{2}[\{0.266 - 0.005\} - 0.005) - (0.2489 + 0.005)] = 0.0011 \]

Radial Clearance!

\[ 6 \times 0.266 \pm 0.005^* \]

\[ \#1 : 0.005 | A | B \]

*0.266 is listed as the minimum close fit clearance hole for a ¼ fastener in ASME B18.2.8
Impacts of geometric specification on cost

*Production time scales with surface finish*

*Cost scales with tolerance*

Kalpakjian & Schmid/American Machinist
Case Study 2 – Complex AM Components

Final Mass: 403gr
Mass reduction: 80.4%

https://grabcad.com/library/spacehugger

SME Digital Manufacturing Challenge | Fundamentals of GD&T Part II | J. Berez, M. Praniewicz 44
High-Density Coordinate Measurement Systems

Structured Light Scanning

Zeiss – GOM Metrology

X-ray Computed Tomography

North Star Imaging

Computed Tomography Scan

3D Model Reconstruction
Pretty pictures, but are they the measurements you want?

Volume Graphics

Zeiss – GOM Metrology
Nominal / Actual Alignment

Nominal

Actual

Alignment

Has a significant effect on comparison results.
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
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

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

![Diagram showing the process from CAD to FFF additive manufacturing to dimensional inspection, with errors depicted in a graph.]
Additive manufacturing example

CAD

.sldprt, .f360... etc.

Bosses printed @ 0, 30, 45, and 90°

Support material required

FFF additive manufacturing

.BSTL → GCODE

25.000...?
Sources of error in digital manufacturing

**CAD**
- .sldprt, .f360... etc.
- CAD suite exporter

**FFF additive manufacturing**
- .STL
- 3DP suite or 'slicer'
- GCODE
- 3D printer

**G01 linear interpolation**
- 'micro-stepping'

**G02/G03 circular interpolation**

**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...**

02/03/2023

SME Digital Manufacturing Challenge | Fundamentals of GD&T Part II | J. Berez, M. Praniewicz

56
Manufacturing approach determines dimensional error

Designer specification informs manufacturing approach!

<table>
<thead>
<tr>
<th>Axis @ 0° to B dir.</th>
<th>Axis @ 30° to B dir.</th>
<th>Axis @ 45° to B dir.</th>
<th>Axis @ 90° to B dir.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ø24.86,³ 0.36</td>
<td>Ø24.87,³ 0.38</td>
<td>Ø24.87,³ 0.35</td>
<td>Ø25.11,³ 0.79</td>
</tr>
</tbody>
</table>
GD&T practices for AM components (a brief look)

This product specification ensures the desired result

Applied to a more complex design

Practices shown are as per ASME Y14.46-2022
Limits and fits: A brief review
Fit types apply to 'external' and 'internal' features, not just shafts and holes.

Clearance fit

- MAX SIZE
- MIN SIZE

- MAX CL.
- MIN CL.

- SHAFT

- HOLE

Transition fit

- MAX SIZE
- MIN SIZE

- MAX CL.
- MAX INT.

- SHAFT

- HOLE

Interference fit

- MAX SIZE
- MIN SIZE

- MIN INT.
- MAX INT.

- SHAFT

- HOLE
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

# Preferred fits

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

*Based on ASME B4.21994*
References and continuing education

Standards

• ASME Y14.52018: 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 Y14.462022 Product Definition for Additive Manufacturing
• Clearance holes for fasteners
  • Machinery's Handbook Tables (*not a standard)
  • ASME B18.2.81999 (R2017)
• Standard limits and fits
  • ASME B4.1 (inch) and 4.2 (metric)

Texts


The slide deck from part I of this seminar series may be found at:
https://doi.org/10.5281/zenodo.7647256
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?