2023 Digital Manufacturing Challenge powered by SME’s DDM Advisory Team

The webinar titled Fundamentals of Geometric Dimensioning and Tolerancing, Part I will begin shortly

Fundamentals of Geometric Dimensioning and Tolerancing (GD&T)

-Part I-

Host: Carl Dekker
President
MetL-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)

Fundamentals of GD&T
Seminar series overview

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
• 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, \textit{dimensional metrology}, NDE

• B.S. Mechanical Eng., University of Maryland, College Park
• Prior experience: Aerospace, automotive
Why GD&T?

Problems:

(1) Communication between stakeholders
(2) Manufacturing imprecision
(3) Meaningful geometric specification

Solution: Geometric dimensioning and tolerancing (GD&T)
Problem 1: Communication between stakeholders

Design process

- Design concept
  - Customer specs → concepts
- Parametric design
  - Select concepts → optimization → CAD
- Design specifications
  - Eng. drawings
  - Material spec.
  - Quality spec.

Manufacturing

- Manufacturing Process(es)
  - Raw material → finished goods
  - Mfg. eng. → Shop-floor → Quality
  - Did you do it right?

Quality

- Did you do it right?

Functional product

Geometric specifications facilitate communication
Problem 2: Imprecision in manufacturing

No manufacturing method is perfectly precise

- Nothing is ever exactly 1in, 10 mm… etc. in size
- Nothing is ‘perfectly’ flat, round, square… etc.

Therefore, how do engineers specify what size they want something to be, and how do manufacturers achieve that?
Problem 2: Imprecision in manufacturing (example)

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

CAD

* .sldprt, .f360... etc.

FFF additive manufacturing

Dimensional inspection

: 25.000...?

: 24.868

\{ \}

0.307

x16 error
Problem 3: Meaningful geometric specification

Problem:
• Exact dimensions don’t acknowledge imprecision… so let's use tolerances
• Tolerances must be unambiguous and easily interpreted

Solution:
• Designers define tolerance zones which the workpiece must fall in to be ‘in-spec’
Solution: GD&T

Problems:

(1) Communication between stakeholders
(2) Manufacturing imprecision
(3) Meaningful geometric specification

Solution: Geometric dimensioning and tolerancing (GD&T)

Today’s seminar will cover fundamentals like...

• Dimensioning & tolerancing systems (e.g., ASME Y14.5)
• GD&T essentials
  • Datums & datum reference frames
  • Geometric characteristics & features of size
  • Feature control frames & engineering drawing practices

Applications covered in Part II
Dimensioning and tolerancing systems
Dimensioning & tolerancing systems

**Direct dimensioning**, aka plus/minus tolerancing
- Only appropriate to use with features of size
- Difficult to interpret designer intention without the larger context of GD&T
Dimensioning & tolerancing systems

Geometric dimensioning and tolerancing, as per **ASME Y14.5-2018**

- ASME Y14.5 is the bulk of all GD&T concepts
- ASME Y14 committee publishes supporting standards such as Y14.5.1M (GD&T math), Y14.1 (drawing sheet size), Y14.1 (digital product defn.)

Geometric product specification, as per **ISO TC 213 series**, e.g., **ISO 1101:2017**

- ISO technical committee (TC) 213 publishes over 20 standards which are like chapters in the overall concept of GPS

ASME-ISO comparison

- Highly similar symbolic language & associated definitions – if you learn one you will know 90% of the other
- Disagree on:
  - *Exact* implementation of datums, the envelope principle as a default, third- and first-angle projections, drawing style, some symbols, etc.

ASME Y14 landing page
https://www.asme.org/codes-standards/y14-standards

ISO TC 213 landing page
https://www.iso.org/committee/54924.html
Dimensioning & tolerancing systems

Why use GD&T/GPS?

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

Don’t be afraid to go and read it for yourself! It’s long (300+ pg.), but well illustrated and designed for comprehension.
Map of GD&T

1. Datums
   - A

2. Datum ref. frame
   - A|B|C

3. Geometric attributes
   - 3.1 Form
   - 3.2 Orientation
   - 3.3 Location

4. Size
   - Direct dim. aka. (±) tol.
   - Rule #1
     - The envelope principle

5. Geometric characteristics
   - (_ [{}!~^&*#; \ )

Feature control frame
- #:1.00>A|B|C

Rule #1
- The envelope principle

1.00
Datums & datum reference frames
Why do we need datums?

Q: Where is point 1?

Insights on the problem:

• Location is relative!

• By picking a location and orientation for the block within a coordinate system we can answer the question. *Nominally:* (25, 50, 50)

• By placing a workpiece in a coordinate system, we effectively constrain its degrees of freedom

A: Place the workpiece in a coordinate system.
Degrees of freedom

- There are **six degrees of freedom** (DoF) in the motion of a *rigid body*.
- By placing the workpiece in a coordinate system, we effectively constrain all 6 DoF.

<table>
<thead>
<tr>
<th>DoF</th>
<th>Motion</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Translation in X (x)</td>
</tr>
<tr>
<td>2.</td>
<td>Translation in Y (y)</td>
</tr>
<tr>
<td>3.</td>
<td>Translation in Z (z)</td>
</tr>
<tr>
<td>4.</td>
<td>Rotation about X (u)</td>
</tr>
<tr>
<td>5.</td>
<td>Rotation about Y (v)</td>
</tr>
<tr>
<td>6.</td>
<td>Rotation about Z (w)</td>
</tr>
</tbody>
</table>
Why do we need datums?

Q: Where is point 1?
A: Place the workpiece in a coordinate system. ..But which one?...

Solution: Define a *specific* coordinate system relative to features on the workpiece! We call this the **datum reference frame**.
Datums

Datum feature: A nonideal physical reference from which a theoretically exact datum is derived

Datum: A perfect theoretical feature which forms a reference from which a location or orientation is established

Datum simulator: A precision embodiment of the datum feature.

Datum reference frame: A set of datum features which establish a coordinate system
Datum features Planes and cylinders

Many features can serve as datums – planes and cylinders are common.

At maximum, a planar datum controls 2 rot. and 1 trans. DoF.

At maximum, a cylindrical datum controls 2 rot. and 2 trans. DoF.
Datums—Drawing conventions

Many acceptable ways to apply datums to the drawing!

- -A-, -B-, and -C- refer to planar surfaces
- -D- refers to a median plane between two surfaces
- -E- refers to a cylinder
Datum reference frames

Datum reference frames (DRF) constrain degrees of freedom (DoF) based on this precedence.

**Datum ref. frame example**

- **Primary**
- **Secondary**
- **Tertiary**

Shigley, adapted

**Note:**

Lower precedence datums do not influence DoF already controlled by higher precedence datums.
Datum reference frames

Datum precedence matters! This is especially clear considering imperfect datum feature geometry.

1. A-constrains 2 rot. & 1 trans.
2. C-constrains 1 rot. & 1 trans.

Selecting functional datums will be covered in Part II.
Datum reference frames

Consider a plane-cylinder DRF.

Datum ref. frame

Datum axis

Datum planes of datum reference frame

Datum feature

Shigley, adapted

Selecting functional datums will be covered in Part II.
Geometric attributes & geometric characteristics
Map of GD&T

1. Datums
   A

2. Datum ref. frame
   A|B|C

3. Geometric attributes
   3.1 Form
   3.2 Orientation
   3.3 Location

4. Size
   Direct dim. aka. (±) tol.

5. Geometric characteristics
   (\_ [{}!~^&*#; \_ )

Rule #1
The envelope principle

Feature control frame
#;1.00>|A|B|C

1.00

3.4 Basic dim.
Geometric attributes

GD&T concepts categorize geometry to have 4 possible attributes:

- **Size**
  - aka ‘feature-of-size’
  - *(Directly dim.)*

- **Location**

- **Orientation**

- **Form**

Your job as the drafter is to control **geometry**, not just size. GD&T is the tool.
<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></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></td>
<td></td>
</tr>
<tr>
<td>Cylindricity</td>
<td>}</td>
<td></td>
<td>No.</td>
</tr>
<tr>
<td>Profile of a line</td>
<td>!</td>
<td>Profile (location, orientation, size, &amp; form)</td>
<td>Sometimes datum referencing</td>
</tr>
<tr>
<td>Profile of a surface</td>
<td>~</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Angularity</td>
<td>^</td>
<td>Orientation</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Perpendicularity</td>
<td>&amp;</td>
<td>Orientation</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Parallelism</td>
<td>*</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Position</td>
<td>#</td>
<td>Location</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Circular runout</td>
<td>;</td>
<td>Runout (location of a cylinder)</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Total runout</td>
<td>\</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Concentricity</td>
<td>$</td>
<td>Elim. in ASME Y14.5-2018</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Symmetry</td>
<td>%</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
The feature control frame Review

- **Geo. char. symbol**: }
- **Tolerance zone size**: 1.00
- **Cylindricity (form)**: ~
- **Datum references (aka DRF)**: |A|B|
- **Profile (*depends)**: |
- **Datum reference**: *|A|
- **Parallelism (orientation)**: |
- **Tolerance zone shape**: #:
- **Position (Location)**: >|A|B|C|
- **Mat. condition modifier**: *Material condition modifiers not covered in this seminar

*Material condition modifiers not covered in this seminar*
**Form – Straightness**

*Straightness* controls deviation of a surface line element or a feature axis from a perfect linear geometry

- [ ] Datum referencing
- [√] Floating

- Good for: Long, high-aspect features which may need separate size and form control levels
**Flatness** controls deviation of surface from a perfect planar geometry.

- Good for: mating surfaces, faces that must bear lots of load and wear, faces must seal against others, and controlling datum features.

Datum referencing: [ ] Datum [√] Floating

Translation: The entire surface seen as a line in this view, must be flat within 0.05, i.e. all points must lie between two parallel planes separated by 0.05.
Form – Roundness { & Cylindricity

**Roundness** controls deviation of a 2D cross section from perfect circular form.  
**Cylindricity** controls deviation of a surface from perfect cylindrical form

[√] Datum referencing [\sqrt{\text{Floating}} \text{ (shrinks & expands to feature size, too!)}]

- Good for: Boss-on-bore contact (e.g., bushings), bores/bosses that mate with other features (prevents ‘out-of-round’), sliding shaft/bore assembles (prevents binding)

Translation: The entire surface of the cylinder must lie between two concentric cylinders with a radial difference of 0.01.

\[\varnothing 30 \pm 0.1\]
Orientation - Angularity

Parallelism, perpendicularity, and angularity control the deviation a surface, axis, or center plane from 0°, 180°, 90°, or X° relative to a datum reference.[√] Datum referencing

• Good for: controlling how well assemblies mate when put together
• Good for: controlling orientation of a bore/boss to a face, relation of faces, non-primary datums
Orientation - Perpendicularity &

Applications to feature surfaces & axes

Translation: The axis of the hole feature must lie within a cylindrical tolerance zone with 0.05 diameter and perpendicular to datum plane A.
Orientation - Parallelism *

Note: Parallelism is NOT flatness – it has a datum reference
**Position** controls the location of a center point, axis, median plane, or boundary of a feature of size relative to a datum or DRF.

[√] Datum referencing [ ] Floating

Basic dimensions are theoretically exact. They do not have to originate from a datum on the drawing, but it is best practice.
Profile and profile of a surface control the location and/or orientation and/or size of a feature

[√] Datum referencing [√] Floating (It depends!)

- Control a 2D cross section’s or 3D surface’s deviation from their nominal form (no datum reference), orientation and location (with datum references)
- Powerful, but easily abused

Shigley, adapted
Profile! & profile of a surface

Translation: This surface must lie within a tolerance zone defined as the space between two surfaces separated by 0.1 and centered on the ideal surface. The ideal surface is defined with the basic dimensions, while the part is in contact with datum A and datum B.

These surfaces are controlled by the default surface profile of 0.3 (tolerance zones not shown).

0.2 tolerance zone

0.1 tolerance zone

0.05 tolerance zone

What changes if this control frame does not reference the AB-DRF?
Runout & total runout

Runout and total runout control the form, orientation, and location of surfaces relative to a datum axis.

- Use to control radially symmetric features on rotating assemblies
- Controls ‘wobble’ of rotating assemblies, controls balance, prevents binding

\[ \text{Measured with } a \text{ as the “total indicator reading” TIR} \]

---

Shigley, adapted
Many acceptable ways to apply control frames to the drawing!

1. Use a leader pointing to the feature
2. Use an extension line from the feature
3. Associate with a feature of size
## 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></td>
<td></td>
</tr>
<tr>
<td>Circularity</td>
<td>{</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cylindricity</td>
<td>}</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Profile of a line</td>
<td>!</td>
<td>Profile (location, orientation, size, &amp; form)</td>
<td>Sometimes datum referencing</td>
</tr>
<tr>
<td>Profile of a surface</td>
<td>~</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Angularity</td>
<td>^</td>
<td>Orientation</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Perpendicularity</td>
<td>&amp;</td>
<td>Orientation</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Parallelism</td>
<td>*</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Position</td>
<td>#</td>
<td>Location</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Circular runout</td>
<td>;</td>
<td>Runout (location of a cylinder)</td>
<td>Datum referencing</td>
</tr>
<tr>
<td>Total runout</td>
<td>\</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Not too bad, right?
Features of size
Map of GD&T

1. Datums
   - A

2. Datum ref. frame
   - A|B|C

3. Geometric attributes
   - 3.1 Form
   - 3.2 Orientation
   - 3.3 Location

4. Size
   - Direct dim. (aka ± tol.)
   - Rule #1
     - The envelope principle

5. Geometric characteristics
   - [{}!~^&*#; \ ]

3.4 Basic dim.
   - 1.00

Feature control frame
   - #:1.00>|A|B|C
Features of size

**Features of size** have opposing surfaces

- The opposing surfaces may be externally or internally facing
- Features of size may use plus/minus tolerancing
- Not a feature of size...
  - Depth
  - Position

*Rule of thumb*: If you can put caliper jaws around or inside it, it is a feature-of-size.
Direct dimensioning styles

Fig. 2-1  Limit Dimensioning

Fig. 2-2  Plus and Minus Tolerancing

Fig. 2-3  Indicating Symbols for Metric Limits and Fits
Maximum & minimum material conditions

• **Maximum material condition** – The feature condition which creates the maximum amount of material.

• **Least material condition** – The feature condition which creates the minimum amount of material

---

**BOSS (POSITIVE)**

MMC: Ø26.0

LMC: Ø24.0

**Bore (NEGATIVE)**

MMC: Ø24.0

LMC: Ø26.0
Rule #1– The envelope principle

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

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

[Diagram showing MMC and form control examples]
GD&T – Summary

Why use GD&T/GPS?

• **Functional** – related to component functionality

• **Unambiguous** – clearly defined and standardized

• **Inspectable** – specifications relate to inspection methods

Shigley
Resources

Source standards
• ASME Y14.5-2018
• ISO TC 213 (E.g., ISO 1101:2017)

Texts & reference books
• “Shigley’s Mechanical Engineering Design”, 10th Ed. or newer, Chp. 20
• Machinery’s Handbook, 26 Ed. or newer
  • Note that some resource may be slightly out of date
• Professional development through societies or for-profit consulting

ASME Y14 landing page
https://www.asme.org/codes-standards/y14-standards

ISO TC 213 landing page
https://www.iso.org/committee/54924.html

SME Tooling U Intro to GD&T
https://learn.toolingu.com/class/140210
2023 Digital Manufacturing Challenge powered by SME’s DDM Advisory Team

Thank you for your time!

Questions?

Seminar 2 teaser

Part 1: Established the core fundamental concepts of GD&T

Part 2: Apply them! Be sure to attend, Feb. 17th 12pm Eastern!

- GD&T check list for designers & walk-through
- Digital/AM examples of product specification
- DRFs & CAD-Actual measurements

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 $R_a$, $R_t$, ±).
4. Control the relation of datum features to each other (normally $\perp$ and $//$).