GD&T
What is GD&T

• ASME Y14.5M-1994
• -The national standard for dimensioning and tolerancing in the United States.
• ASME stands for American Society of Mechanical Engineers.
• The Y14.5 is the standard number.
• "M" is to indicate the standard is metric, and 1994 is the date the standard was officially approved.
What is GD&T

- Geometric Dimensioning and tolerancing (GD&T) is a language used on mechanical engineering drawings composed of symbols that are used to efficiently and accurately communicate geometry requirements for associated features on component and assemblies.

- A method to specify the shape of a piece of hardware on an engineering drawing.
• A set of fourteen symbols used in the language of GD&T. It consists of well-defined symbols, rules, definitions and conventions, used on engineering drawings to accurately describe a part.

• GD&T is a precise mathematical language that can be used to describe the size, form, orientation, and location of part features.

• GD&T is also a design philosophy on how to design and dimension parts.
As drawn

As manufactured –
Without geometric tolerance control
Example of part with no GD&T
Example part with GD&T
Compare!
Manufactured parts
Advantages of GD&T

• Use of this language or tool “can provide economic and technical advantage” stated the ASME.
• Maximizes quality of the products.
• Provides uniformity of specification and interpretation (reducing guesswork and controversy)
• Geometric dimensioning dramatically reduces the need for drawing notes to describe complex geometry requirements on a component or assembly by the use of standard symbology.
Advantages of GD&T

• Ensures the design requirements are carried out.
• GD&T facilitates an efficient means to communicate specific datums on a part. Without the use of a datum system (zero reference) on a part, it is not clear to manufacturing or quality where to manufacture or measure from. Additionally, the use of datums dramatically simplifies the design and specification of parts for use in manufacturing and quality verification steps.
Advantages of GD&T

- Create a part design that focuses on the product function.
- Convert product requirements into dimensional specifications.
- Better define parts without the need for assumptions.
- Document the design for future use.
- Discover problems in the design stage.
- Ensure that parts will assemble.
- Have less "hand fitting" at assembly.
- Ensure that parts are inspected as intended.
- Inspect parts more quickly.
- Reduce scrap or rework.
- Make a replacement that fits into the assembly.
Advantages of GD&T

• Have multiple sources on various parts of an assembly.
• Make valid engineering calculations.
• Have common parts across similar assemblies.
• Design subassemblies in different locations and have them function correctly.
• Do tolerance analysis to study the effect of part tolerances on the assembly.
• Use state of the art software tools to analyze parts in an assembly.
• Use state of the art software tools to inspect the parts.
• Reduce the risk caused by vague specifications.
• Saves money
Geometric Tolerances are divided into five categories

• 1. Form control
• 2. Orientation control
• 3. Location control
• 4. Composite control
• 5. Profile controls
<table>
<thead>
<tr>
<th>Geometric Characteristic</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straightness</td>
<td>![straightness symbol]</td>
</tr>
<tr>
<td>Flatness</td>
<td>![flatness symbol]</td>
</tr>
<tr>
<td>Circularity</td>
<td>![circularity symbol]</td>
</tr>
<tr>
<td>Cylindricity</td>
<td>![cylindricity symbol]</td>
</tr>
</tbody>
</table>
Orientation control

<table>
<thead>
<tr>
<th>Geometric Characteristic</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angularity</td>
<td></td>
</tr>
<tr>
<td>Perpendicularity</td>
<td></td>
</tr>
<tr>
<td>Parallelism</td>
<td></td>
</tr>
</tbody>
</table>
# Location control

<table>
<thead>
<tr>
<th>Geometric Characteristic</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>Position</td>
<td>![Position Symbol]</td>
</tr>
<tr>
<td>Concentricity</td>
<td>![Concentricity Symbol]</td>
</tr>
<tr>
<td>Symmetry</td>
<td>![Symmetry Symbol]</td>
</tr>
</tbody>
</table>
Composite control

Geometric Characteristic

Circular Runout
Total Runout

Symbol
<table>
<thead>
<tr>
<th>Geometric Characteristic</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>Profile of a Line</td>
<td></td>
</tr>
<tr>
<td>Profile of a Surface</td>
<td></td>
</tr>
</tbody>
</table>
Summary of control tolerances

- Orientation is refinement of location.
- Form is refinement of orientation.
Geometric Dimensioning & Tolerancing symbols

**GENERAL TOLERANCE SYMBOLS**

- Ø Diameter
- XX.XX Basic
- M Maximum Material Condition (MMC)
- L Least Material Condition (LMC)
- S Regardless of Feature Size
- Full Indicating Movement (FIM)
- P Projected Tolerance Zone
- □ Dimension Origin
- O All-Around
- R Radius
- ( ) Reference Dimension (REF)
- SØ Spherical Diameter (SD)
- Spherical Radius (SR)
- ( ) Arc Length
- Chain Line
- Conical Taper
- Slope
- Counterbore/Spotface
- Countersink
- Depth/Deep (DP)
- Dimension Not To Scale
- Times/Places

**FORM AND ORIENTATION TOLERANCE SYMBOLS**

- Straightness
- Flatness
- Cylindricity
- Perpendicularity
- Angularity
- Parallelism
- Surface Profile
- Line/Profile
- Total Runout
- Circular Runout
- Unit Control

**TOLERANCE SYMBOLS OF LOCATION**

- Concentricity
- Position
- Symmetry
<table>
<thead>
<tr>
<th>Modifier</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>FREE STATE</td>
</tr>
<tr>
<td>L</td>
<td>LEAST MATERIAL CONDITION</td>
</tr>
<tr>
<td>M</td>
<td>MAXIMUM MATERIAL CONDITION</td>
</tr>
<tr>
<td>P</td>
<td>PROJECTED TOLERANCE ZONE</td>
</tr>
<tr>
<td>S</td>
<td>REGARDLESS OF FEATURE SIZE</td>
</tr>
<tr>
<td>T</td>
<td>TANGENT PLANE</td>
</tr>
<tr>
<td>U</td>
<td>UNILATERAL</td>
</tr>
</tbody>
</table>
Feature control symbols with datum references

Datum - A theoretically exact plane, point or axis from which a dimensional measurement is made.

Datum Feature - A part feature that contacts a datum.

Datum Feature Simulator - The inspection equipment (or gage surfaces) used to establish a datum.

Datum Reference Frame - A set of three mutually perpendicular datum planes.

Datum Shift - The allowable movement, or looseness, between the part datum feature and the gage.

Datum System - A set of symbols and rules that communicate to the drawing user how dimensional measurements are to be made.

Datum Target - A symbol that describes the shape, size, and location of gage elements that are used to establish datum planes or axes.
datum

A theoretically exact point, axis or plane derived from the true geometric counterpart of a specified datum feature.

A datum is the origin from which the location or geometric characteristics of features of a part are established.
Datum vs datum feature
Datum feature symbol

- When the datum is the axis or center plane
Datum feature symbol

When the datum is the axis
Feature Surface and extension line
datum targets

• The datum targets are indicated by a circular frame divided in two compartments by a horizontal line.

• The lower compartment is reserved for a letter and a digit.

• The letter represents the datum feature and the digit datum target number.

• The upper compartment is reserved for additional information, such as dimensions of the target area.
Datum target

- A specified point, line or area on a part used to establish a datum.
Common datum

- A common_datum is a type of Datum that corresponds to a datum that is established from more than one datum feature.
- On technical drawing, a datum that is established from multiple datum features is indicated by placing the identifying letters of the datum features, separated by a dash, within a single compartment in a feature control frame.
- There is no significance to the order of the datum feature identifying letters within a compartment of the feature control frame.
Datum system

• A group of two or more separate datums used as a combined reference for a tolerated feature.

Datum reference frame

Datum reference frames are coordinate systems used to locate and orient part features.
Datum reference frame

• A framework that consists of three mutually perpendicular datum planes, three datum axes (located at the intersection of each pair of datum planes), and a datum point (that is located at the intersection of the three datum planes).

• A Cartesian coordinate system established using the Datums extracted from a set of Datum Features referenced in a Feature Control Frame. Datum Reference Frames serve to orient and locate tolerance zones.
Datum reference frame
Orthogonal Datum Planes

Secondary Datum Plane
2 points of contact
1 d.o.f.

Primary Datum Plane
3 points of contact
3 degrees of freedom

Tertiary Datum Plane
1 point of contact
0 d.o.f.
(i.e. part is fixed)

“3-2-1” Rule
Datum Simulators

High precision surfaces and gages are used to *simulate* theoretically perfect datums, and therefore allow accurate measurement.

Surface plate
Ground flat and highly polished – simulates datum plane
Datum simulators

Gages are used to measure feature variation

V-aligns centers of circular objects, regardless of size

V-blocks are used with cylindrical shapes
Material conditions

- Features of size which includes datum features have size tolerances.
- The size condition or material (amount of metal) condition can vary from maximum metal condition (MMC) to the least metal condition (LMC).
- If the center planes or axes of a feature of size are controlled by geometric tolerances a modifying symbol can be specified in the feature control frame, that applies the tolerance value either MMC or LMC.
- Can also use for a datum that a feature of size.
- If a symbol is not specified – regardless of material condition, use RFS (regardless of feature size).
Regardless of Feature Size (RFS):

• It is applicable if the MMC or the LMC are not specified for individual features of size tolerances or for datum features of size.
• The tolerance is limited to the specified value in the FCF and if applied to a datum feature of size the actual axis or center plane have to be established regardless of the feature size.
• It is always used for run out, concentricity, and symmetry controls.
• It is also used when targets are specified to establish datum axes and center plane because the targets have to contact the datum features to be useful.
• Also it is used to control wall thickness variation between external and internal features. Hard gages are not applicable since there is no additional or bonus tolerance as allowed for MMC and LMC.
Example: RFS

**THIS ON THE DRAWING**

**MEANS THIS**

**ACTUAL LOCAL SIZE**
Ø1.00 TO Ø1.01

**Ø TOLERANCE ZONE**

**ACTUAL MATING ENVELOPE** (AXIS NOT PARALLEL TO DATUM AXIS A)

**DATUM AXIS A** (AXIS OF ACTUAL MATING ENVELOPE)

<table>
<thead>
<tr>
<th>ACTUAL MINIMUM MATING ENVELOPE FEATURE BEING CONTROLLED</th>
<th>Ø TOLERANCE ZONE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ø1.00</td>
<td>.005</td>
</tr>
<tr>
<td>Ø1.01</td>
<td>.005</td>
</tr>
</tbody>
</table>
Maximum Material Condition (MMC)

• The condition in which a feature of size contains the maximum amount of material everywhere within the stated limits of size.
Maximum Material Condition (MMC)

Upper Limit (MMC): 0.497
Lower Limit (LMC): 0.495

Upper Limit (LMC): 0.502
Lower Limit (MMC): 0.498
MMC vs LMC

“Maximum Material Condition” vs “Least Material Condition”
Least Material Condition (MMC)

• The condition in which a feature of size contains the least amount of material everywhere within the stated limits of size
Maximum Material Condition (MMC)

- This is the condition when the actual mating size or envelope size is at the maximum material condition which is maximum size for an external feature such as a cylinder and the minimum size for an internal feature such as a hole. The symbol is “M”
- The added tolerance is the difference between the actual mating envelope size and the MMC size.
- The largest actual mating envelope named virtual condition is equal to the MMC size plus the tolerance specified in the FCF for an external feature and minus for an internal feature.
- The MMC symbol is used to assure that parts will assemble and it allows the use of so called hard gages (go gages) for quick inspections.
- The actual local size has to meet the size tolerance however the actual local size does not affect the geometric characteristic tolerance
Maximum metal condition

THIS ON THE DRAWING

Ø1.00+0.01

Ø0.05 M A

MEANS THIS

Ø TOLERANCE ZONE

ACTUAL LOCAL SIZE
Ø1.00 TO Ø1.01

ACTUAL MATING ENVELOPE (AXIS PARALLEL TO DATUM AXIS A)

DATUM AXIS A (AXIS OF ACTUAL MATING ENVELOPE)

<table>
<thead>
<tr>
<th>ACTUAL MATING ENVELOPE Ø FEATURE BEING CONTROLLED</th>
<th>Ø TOLERANCE ZONE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ø1.01</td>
<td>.005</td>
</tr>
<tr>
<td>Ø1.00</td>
<td>.015</td>
</tr>
</tbody>
</table>
Least Material Condition (LMC)

- This is the opposite of MMC
- This is the condition when the actual minimum mating size or envelope is at the minimum material condition which is minimum size for an external feature such as a cylinder and the maximum size for an internal feature such as a hole. The symbol is “L”
- The smallest actual mating size is equal to the LMC size minus the tolerance specified in the FCF for an external feature and plus for an internal feature.
- The LMC symbol is used to assure a minimum amount of machining stock for features that are to be machined and for assuring a minimum amount of wall thickness between external and internal features.
- Hard gages cannot be used for inspection
- The actual local size has to meet the size tolerance however the local size does not affect the geometric characteristic tolerance
Least metal condition

THIS ON THE DRAWING

MEANS THIS

ACTUAL LOCAL SIZE
Ø1.00 TO Ø1.01

DATUM AXIS A (AXIS OF ACTUAL MATING ENVELOPE)

ACTUAL MINIMUM MATING ENVELOPE
(AXIS PARALLEL TO DATUM AXIS A)

<table>
<thead>
<tr>
<th>ACTUAL MINIMUM MATING ENVELOPE FEATURE BEING CONTROLLED</th>
<th>Ø TOLERANCE ZONE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ø1.00</td>
<td>.005</td>
</tr>
<tr>
<td>Ø1.01</td>
<td>.015</td>
</tr>
</tbody>
</table>
Maximum and minimum material condition – two hole patterns
True-position dimensioning

(a) 4 x \( \phi 12.5 \pm 0.05 \)

\( \phi 0.07 \text{ (M)} \) A B C

\( \phi 0.18 \text{ (M)} \) A

(b) \( \phi 0.18 \) TOLERANCE ZONE

(Hole pattern location)

\( \phi 0.70 \) TOLERANCE ZONE

(interrelationship of holes and a possible location of hole axes)
Each circular element of the surface in any plane perpendicular to a common axis must be within the specified tolerance of size and must lie between two concentric circles – one having a radius 0.25 larger than the other.

Specifying Roundness for a Cylinder or Cone.
Cylindricity

The cylindrical surface must be within the specified tolerance of size and must lie between two concentric cylinders — one having a radius 0.25 larger than the other.

Specifying Cylindricity.
Concentricity

• Concentricity describes a condition in which two or more features (cylinders, cones, spheres, etc.) in any combination have a common axis.
concetricity
Straightness

Each longitudinal element of the surface must be within the specified tolerance size of the perfect form at MMC and lie between two parallel lines (0.02 apart) where the two lines and the nominal axis share a common plane.

Each circular element of the figure must be within the specified tolerance of size. The centerline of the feature must lie within a cylindrical tolerance zone of 0.04 at MMC. The allowed straightness tolerance increases equal to the amount the feature departs from MMC.
Angular

Angular Tolerance Zones.
Flatness

The surface must be within the specified tolerance of size and must lie between two parallel planes 0.25 apart.

Specifying Flatness.
Flatness

- All surface elements of the tolerance feature must lie between two parallel planes 0.2 apart.
Flatness Applications

• To ensure the integrity of mating or mounting surfaces
• To ensure that surfaces seal properly
• Appearance
Flatness – flange mounting

To be used as primary datum feature

A flatness control could be used on this surface to improve measurement accuracy from using this surface as a primary datum feature.
Flatness – sealing surface

- Flatness control - to ensure proper gasket compression
Flatness- sealing surface

On this sealing surface, a flatness control could be used to ensure the surface will contact the O-ring.
Angularity

This on the drawing ....

\[ \angle 0.4 \ A \]

30°

\[ -A- \]

ANGULARITY

.... means this

0.4 wide tolerance zone
Possible attitude of the surface

30°

Datum plane

The surface must be within the specified tolerance of size and must lie between two parallel planes 0.4 apart which are inclined at 30° to the datum plane A.

Specifying Angularity for a Plane Surface.
Profile

THIS ON THE DRAWING......

---

-D-C-

---

8±0.05

---

7±0.12

---

49±0.12

---

0.25

---

A

---

B

---

C

---

BETWEEN X & Y

---

(UNTOLERANCED DIMENSIONS ARE BASIC)

---

49±0.12

---

21.4

---

23

---

23.8

---

23

---

17.5

---

17.5

---

8±0.12

---

12±0.12

---

65±0.25

---

...... MEANS THIS

---

Datum plane C

---

0.25 wide tolerance zone

---

Datum plane A

---

90°

---

Datum plane B

---

The surface between points X & Y must lie between the two profile boundaries 0.25 apart, equally disposed about the true profile, which are perpendicular to datum plane A and positioned with respect to datum planes B & C.

Specifying Profile of a Surface between Points.
Specifying Profile of a Surface All Around.
concentricity

Specifying Concentricity.
perpendicularly
PERPENDICULARITY FOR A PLANE SURFACE

The surface must be within the specified tolerance of size and must lie between two parallel planes 0.12 apart which are perpendicular to the datum plane A.

PERPENDICULARITY FOR A MEDIAN PLANE

The feature center plane must be within the specified tolerance of location and must lie between two parallel planes 0.12 apart, regardless of feature size, which are perpendicular to the datum plane A.

PERPENDICULARITY FOR AN AXIS

The feature axis must be within the specified tolerance of location and must lie between two planes 0.2 apart, regardless of feature size, which are perpendicular to the datum axis A.
Parallelism

This on the drawing......

...... means this

0.12 wide tolerance zone
Possible orientation of the surface

The surface must be within the specified tolerance of size and must lie between two planes 0.12 apart which are parallel to the datum plane A.

Specifying Parallelism for a Plane Surface.
Parallelism

The feature axis must be within the specified tolerance of location. Where the feature is at maximum material condition (10.00), the maximum parallelism tolerance is 0.05 diameter. Where the feature departs from its MMC size, an increase in the parallelism tolerance is allowed which is equal to the amount of such departure.

Specifying Parallelism for an Axis Feature at MMC.
Specifying Parallelism for an Axis Feature RFS.

The feature axis must be within the specified tolerance of location and must lie between two planes 0.12 apart which are parallel to the datum plane, regardless of feature size.

Datum plane A

Possible orientation of the feature axis

0.12 wide tolerance zone
Circular runout

• A dial indicator is often used to verify a runout control
• First, the part is located in a chuck or collet to establish datum axis A.
• A dial indicator is placed on the surface being checked.
• As the part is rotated 360 degree, the dial indicator movement is the run out value of the circular element.
• Several independent dial indicator readings are made at different places along the diameter.
Total runout

- Total run out is used to control the combined variations of circularity, straightness, coaxiality, angularity, taper and profile when applied to surfaces around and at right angles to a datum axis.

- Note that total runout cannot be applied to conical or curved surfaces as can circular runout.
Run-out
Part is rotated and gage is moved along the surface.

Datum axis A

Indicator travels along a helical path along the surface of the diameter.

Chuck or collet.
Application of GD&T
Surface texture symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a) ![Checkmark]</td>
<td>Basic Surface Texture Symbol. Surface may be produced by any method except when the bar or circle, (b) or (d), is specified.</td>
</tr>
<tr>
<td>(b) ![Checkmark]</td>
<td>Material Removal By Machining Is Required. The horizontal bar indicates that material removal by machining is required to produce the surface and that material must be provided for that purpose.</td>
</tr>
<tr>
<td>(c) ![3.5 Checkmark]</td>
<td>Material Removal Allowance. The number indicates the amount of stock to be removed by machining in millimeters (or inches). Tolerances may be added to the basic value shown or in a general note.</td>
</tr>
<tr>
<td>(d) ![Circle]</td>
<td>Material Removal Prohibited. The circle in the vee indicates that the surface must be produced by processes such as casting, forging, hot finishing, cold finishing, die casting, powder metallurgy or injection molding without subsequent removal of material.</td>
</tr>
<tr>
<td>(e) ![Square Root]</td>
<td>Surface Texture Symbol. To be used when any surface characteristics are specified above the horizontal line or to the right of the symbol. Surface may be produced by any method except when the bar or circle, (b) or (d), is specified.</td>
</tr>
</tbody>
</table>

(f) ![Diagram]

*This dimension is adjusted by +1 for each line of values beyond the two lines shown below the horizontal line.*
Application of surface texture symbol

(a) UNLESS OTHERWISE SPECIFIED: ALL SURFACES 3.2

(b) WAVINESS WIDTH
ROUGHNESS WIDTH
ROUGHNESS HEIGHT
WAVINESS HEIGHT
ROUGHNESS WIDTH CUTOFF
# Lay symbols

<table>
<thead>
<tr>
<th>SYM</th>
<th>DESIGNATION</th>
<th>EXAMPLE</th>
<th>SYM</th>
<th>DESIGNATION</th>
<th>EXAMPLE</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Lay parallel to the line representing the surface to which the symbol is applied.</td>
<td><img src="image1.png" alt="Example" /></td>
<td>X</td>
<td>Lay angular in both directions to line representing the surface to which symbol is applied.</td>
<td><img src="image2.png" alt="Example" /></td>
</tr>
<tr>
<td>X</td>
<td>Lay perpendicular to the line representing the surface to which the symbol is applied.</td>
<td><img src="image3.png" alt="Example" /></td>
<td>M</td>
<td>Lay multidirectional</td>
<td><img src="image4.png" alt="Example" /></td>
</tr>
<tr>
<td>C</td>
<td>Lay approximately circular relative to the center of the surface to which the symbol is applied.</td>
<td><img src="image5.png" alt="Example" /></td>
<td>R</td>
<td>Lay approximately radial relative to the center of the surface to which the symbol is applied.</td>
<td><img src="image6.png" alt="Example" /></td>
</tr>
</tbody>
</table>