

GD&T reference

Reference for Geometric Dimensioning and Tolerancing — the symbols, modifiers, and feature controls that translate design intent into manufacturable parts.

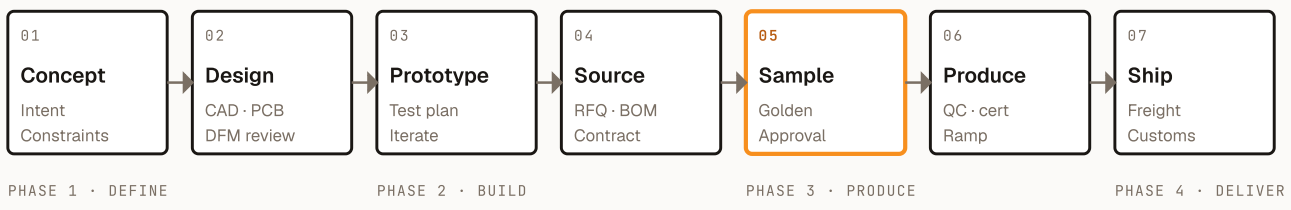
REVISION	ISSUED	OWNER	COMPANION
1.0	May 2026	Ideambox engineering	PDF reference

ABSTRACT

GD&T (Geometric Dimensioning and Tolerancing) is the symbolic language that defines part geometry beyond simple \pm dimensions. Used per ASME Y14.5 (US) and ISO 1101 (international), it specifies form, orientation, location, and runout of features. Without GD&T, "25.0 \pm 0.05 mm" leaves the manufacturer to interpret which surface to measure and which datum to use.

Section 1 covers fundamentals (features, datums, tolerance zones). Section 2 covers the 14 geometric symbols. Section 3 covers modifiers and material conditions. Section 4 covers drawing best practices. Section 5 covers verification and inspection.

HARDWARE PRODUCT DEVELOPMENT – 7-STAGE PIPELINE



GD&T SITS IN PHASE 2 (BUILD) – AT THE POINT WHERE MECHANICAL CAD BECOMES MANUFACTURING DRAWINGS. EACH TOLERANCE IS ALSO A MANUFACTURING COST.

CONTENTS

- | | |
|----------------------|------------------------------|
| 1. GD&T fundamentals | 4. Drawing best practices |
| 2. Geometric symbols | 5. Verification + inspection |
| 3. Modifiers | 6. Common GD&T applications |

1. GD&T fundamentals

GD&T defines geometric requirements with symbols, datum references, and tolerance values.

1.1 Why GD&T over \pm tolerancing

- **Defines the entire feature, not just one dimension.**
- **Establishes a datum reference frame**
every measurement is relative to known features.
- **Enables larger tolerance zones when geometry allows (bonus tolerance via MMC modifier).**
- **Standardised across engineering**
every supplier interprets the same way.
- **Required for safety-critical, automotive, aerospace, and any precision assembly.**

1.2 Key terms

Feature	A physical part of the geometry (hole, surface, slot, axis, plane)
Datum	A theoretically perfect geometric reference (plane, axis, point)
Datum Feature	The physical feature used to establish a datum (the surface the part rests on)
Tolerance Zone	The 3D region within which the feature must lie
Material Condition	MMC, LMC, or RFS modifier on tolerance
Feature Control Frame (FCF)	The full GD&T callout including symbol, tolerance, datums, modifiers

1.3 Datum reference frame

A part has up to 6 degrees of freedom (3 translation, 3 rotation). The datum reference frame fixes the part in 3D space:

- **Primary datum (A)**
Plane that fixes 3 DOF (1 translation, 2 rotations)
- **Secondary datum (B)**
Plane perpendicular to A; fixes 2 more DOF
- **Tertiary datum (C)**
Plane perpendicular to A and B; fixes 1 more DOF

All measurements reference this frame. Per ASME Y14.5: "A measurement made without an explicit datum reference is meaningless."

2. Geometric symbols

14 standard symbols, organized into 5 categories. Each defines a type of geometric requirement.

2.1 The 14 symbols (ASME Y14.5 / ISO 1101)

Form (no datums needed)

SYMBOL	NAME	WHAT IT CONTROLS
—	Straightness	Line/axis is straight within tolerance
○	Roundness (Circularity)	Cross-section is circular
⊘	Cylindricity	Surface is a perfect cylinder (combined straightness + roundness + taper)
▭	Flatness	Surface is planar

Orientation (1 datum)

SYMBOL	NAME	WHAT IT CONTROLS
⊥	Perpendicularity	90° to datum
∥	Parallelism	Parallel to datum
∠	Angularity	At specific angle to datum

Location (1 or more datums)

SYMBOL	NAME	WHAT IT CONTROLS
⊕	Position	Center, axis, or median plane location
◎	Concentricity	Axes coincide (median points coincide)
=	Symmetry	Median plane in correct location

Profile (1 or more datums optional)

SYMBOL	NAME	WHAT IT CONTROLS
⌒	Profile of a Line	2D profile within tolerance
⌒	Profile of a Surface	3D surface within tolerance

Runout (1 datum, often referencing axis)

SYMBOL	NAME	WHAT IT CONTROLS
↗	Circular Runout	Per-cross-section runout
↗↗	Total Runout	Across the entire feature

2.2 Feature Control Frame anatomy

↑ ↑ ↑ ↑ ↑ Symbol Tol value + Primary Secondary Tertiary

 diameter sign datum datum datum + modifier

Reading the frame: "Position of this feature, within a \varnothing 0.10 mm tolerance zone, at Maximum Material Condition (M), relative to datums A (primary), B (secondary), C (tertiary)."

3. Modifiers

Material condition modifiers expand tolerance based on feature size.

3.1 Material conditions

SYMBOL	MODIFIER	WHEN APPLIED
Ⓜ	Maximum Material Condition (MMC)	Tolerance applies at MMC; bonus when feature departs from MMC
Ⓛ	Least Material Condition (LMC)	Tolerance applies at LMC; bonus toward LMC
(none)	Regardless of Feature Size (RFS)	Tolerance applies regardless of size; no bonus
Ⓟ	Projected Tolerance Zone	Tolerance extends above the surface
Ⓠ	Free State	Applies to non-rigid parts when not constrained

3.2 MMC bonus tolerance

At Maximum Material Condition, a feature has the most material (largest external feature, smallest internal feature).

Example: $\varnothing 10.00$ mm hole with tolerance $\pm \varnothing 0.10$ Ⓜ A B

- **MMC = $\varnothing 10.00$ mm (smallest hole) → 0.10 mm position tolerance**
- **Hole at $\varnothing 10.10$ mm (departs from MMC by 0.10) → 0.20 mm position tolerance available**
- **Hole at $\varnothing 10.20$ mm (departs by 0.20) → 0.30 mm position tolerance available**

Bonus = (Actual size – MMC size) for internal features, or (MMC size – Actual size) for external.

3.3 Why MMC matters

- **Functional purpose**
A hole that's larger has more "room" to be off-center while still allowing a bolt to pass through.
- **Cost savings**
A part may be in spec with a larger hole + more position error, vs. a tight hole + perfect position.
- **Easier inspection**
Functional gauges can verify MMC at the gauge surface.

4. Drawing best practices

The drawing is the contract. Every GD&T callout has implications.

4.1 Drawing fundamentals (ASME Y14.5 + ASME Y14.100 series)

- **Drawing format**
Title block, scale, units, projection (third-angle in US, first-angle in EU), revision history.
- **Datum identification**
Clear labels (A, B, C). Place at the feature they reference.
- **Tolerance scheme**
Either \pm tolerances OR GD&T (don't mix on the same dimension).
- **Default tolerance block**
Stated in title block (e.g., ± 0.3 mm linear, $\pm 0.5^\circ$ angular).
- **Critical dimensions called out**
Tight tolerance dimensions explicitly highlighted.
- **Surface finish symbols**
Per ISO 1302 or ASME Y14.36 (Ra values, surface texture).

4.2 Datum selection rules

- **Primary datum**
The most critical alignment surface. Usually largest mating surface or functional reference.
- **Secondary datum**
Perpendicular to primary; second most critical.
- **Tertiary datum**
Perpendicular to both; least critical.
- **Datum referencing order matters**
A | B | C means A is primary, B secondary, C tertiary. The order affects how the part is constrained.

4.3 Tolerance allocation

Tighter tolerances cost more. Allocate budget across the assembly.

APPLICATION	TYPICAL TOLERANCE
Visible mechanical feature (cosmetic)	± 0.3 mm
Mating surface (general)	± 0.1 mm
Bearing or precision mating	± 0.05 mm
Optical alignment	± 0.01 – 0.02 mm
Aerospace / medical critical	± 0.005 mm

4.4 Common GD&T mistakes

- **Datums not specified**
Tolerance is meaningless without a reference frame.
- **MMC on internal feature when LMC needed**
Or vice versa. Functional consideration matters.
- **Tolerance zones smaller than process capability**
Forces 100 % inspection, drives cost.

- **Drawing dimensions inconsistent with model**

CAD model is reference; drawing dimensions should be derived.

- **Inappropriate symbol choice**

E.g., concentricity (rarely used) instead of position (more flexible).

5. Verification + inspection

GD&T values must be measurable. Choose tolerance you can verify.

5.1 Standard inspection equipment + accuracy

EQUIPMENT	CAPABILITY	COST	USE
Digital caliper (0.01 mm)	Linear ± 0.02 mm	\$50–300	General dimension
Micrometer (0.001 mm)	Linear ± 0.002 mm	\$100–500	Critical dimensions
Comparator microscope	Linear ± 0.005 mm + angular	\$2 000–8 000	Optical comparison + measurement
Coordinate Measuring Machine (CMM)	± 0.001 – 0.005 mm	\$30 000–200 000	Full 3D verification
Optical CMM (vision)	± 0.005 mm	\$20 000–80 000	Non-contact, fast
Laser scanner	± 0.05 – 0.2 mm	\$20 000–100 000	Free-form surfaces
Profilometer	± 0.0001 mm vertical	\$5 000–30 000	Surface finish, roughness

5.2 Process capability vs. tolerance

A tolerance must be larger than the process variation, or 100 % of parts will fail inspection.

Process capability index (Cp): tolerance / 6σ . Cp ≥ 1.33 is industry-typical target. Below 1.0, the process cannot reliably hit tolerance.

5.3 Inspection methodology per feature type

FEATURE	METHOD
Hole position (MMC)	Functional gauge OR CMM
Hole position (RFS)	CMM only
Flatness	CMM, gauge block, surface plate
Cylindricity	Roundness gauge OR CMM
Surface profile	CMM, optical CMM, laser scanner
Runout	Dial indicator on rotating axis

5.4 Functional gauge (Go/No-Go)

For MMC-modified position tolerances, a functional gauge can verify in a single check:

- **Go gauge**
Maximum allowed size (e.g., $\varnothing 10.10$ mm) for the hole. Hole must accept the gauge.
- **No-Go gauge**
Minimum allowed size (e.g., $\varnothing 9.90$ mm). Hole must reject the gauge.

Faster than CMM for production inspection but only works for MMC tolerances.

6. Common GD&T applications

Examples of GD&T applied to typical hardware features.

6.1 Mounting hole pattern

A 4-hole bolt pattern needs each hole positioned relative to a datum frame: “ $\text{⌀ } 0.10 \text{ } \textcircled{M} \text{ } A \text{ } B \text{ } C$ ” Position tolerance of $\text{⌀ } 0.10$ mm at MMC (with bonus when holes depart from minimum size), referenced to primary datum A, secondary B, tertiary C.

6.2 Mating surface

A face that must be flat to allow gasket sealing: “ $\text{ } \textcircled{F} \text{ } 0.05 \text{ } \text{ (or } - \text{ } 0.05 \text{)}$ ” Flatness of 0.05 mm. No datum needed (form symbol).

6.3 Cylindrical fit

A shaft that must rotate inside a bearing: “ $\text{ } \textcircled{C} \text{ } 0.025 \text{ } \text{ (or } \textcircled{C} \text{ } 0.025 \text{)}$ ” Cylindricity of 0.025 mm — surface must be a perfect cylinder within this tolerance.

6.4 Hole perpendicular to face

A hole that must be perpendicular to the mounting face: “ $\text{ } \perp \text{ } \text{⌀ } 0.10 \text{ } A$ ” Perpendicularity of $\text{⌀ } 0.10$ mm zone, referenced to datum A.

6.5 Optical surface alignment

A lens mount where two surfaces must be parallel: “ $\text{ } \parallel \text{ } 0.01 \text{ } A$ ” Parallelism of 0.01 mm tolerance zone, referenced to datum A (the optical axis).

6.6 Common assembly pattern

A bolted-together housing where 4 holes pass through 4 inserts:

— **Cover plate**

4 clearance holes at $\text{⌀ } 0.20 \text{ } \textcircled{M} \text{ } A \text{ } B$

— **Housing**

4 tapped holes at $\text{⌀ } 0.10 \text{ } \textcircled{M} \text{ } A \text{ } B$

— **Total stack-up**

Worst-case position error 0.30 mm

The MMC bonus enables the assembly to work as long as the holes don't depart too far from their respective MMC sizes.

FINAL NOTE. GD&T is the language of precision manufacturing. A drawing without GD&T is a drawing without instructions. Every tolerance is also a cost: tighter = more inspection, more rework, slower cycle. Allocate tolerance to where function demands it, and accept looser tolerance elsewhere.