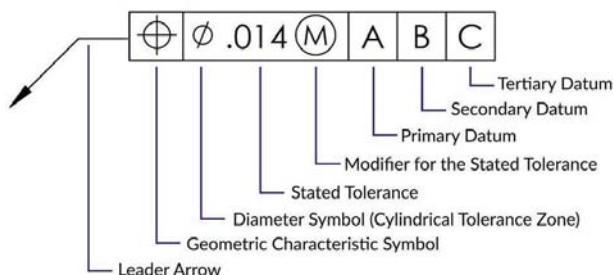


GD&T Engineering Symbols Geometric Dimensioning and Tolerancing

GD&T Symbol	Control Type	Name	Summary Description
—	Form	Straightness	Controls the straightness of a feature in relation to its own perfect form
▭	Form	Flatness	Controls the flatness of a surface in relation to its own perfect form
○	Form	Circularity	Controls the form of a revolved surface in relation to its own perfect form by independent cross sections
⊘	Form	Cylindricity	Like circularity, but applies simultaneously to entire surface
⌒	Profile	Profile of a Surface	Controls size and form of a feature. In addition it controls the location and orientation when a datum reference frame is used.
⌒	Profile	Profile of a Line	Similar to profile of a surface, applies to cross sections of a feature
⊥	Orientation	Perpendicularity	Controls the orientation of a feature which is nominally perpendicular to the primary datum of its datum reference frame
∠	Orientation	Angularity	Controls orientation of a feature at a specific angle in relation to the primary datum of its datum reference frame
//	Orientation	Parallelism	Controls orientation of a feature which is nominally parallel to the primary datum of its datum reference frame
⊕	Location	Position	Controls the location and orientation of a feature in relation to its datum reference frame
◎	Location	Concentricity	Controls concentricity of a surface of revolution to a central datum
≡	Location	Symmetry	Controls the symmetry of two surfaces about a central datum
↗	Runout	Circular runout	Controls circularity and coaxiality of each circular segment of a surface independently about a coaxial datum
↗↗	Runout	Total runout	Controls circularity, straightness, coaxiality, and taper of a cylindrical surface about a coaxial datum

Symbol	ANSI Y14.5 Meaning
Ⓛ	LMC – Least Material Condition
Ⓜ	MMC – Maximum Material Condition
Ⓣ	Tangent Plane
Ⓟ	Projected Tolerance Zone
ⓕ	Free State
∅	Diameter
R	Radius
SR	Spherical Radius
S∅	Spherical Diameter
CR	Controlled Radius
Ⓢ	Statistical Tolerance
77	Basic Dimension
(77)	Reference Dimension
5X	Places
←⊕	Dimension Origin
Ⓛ	Counterbore
∨	Countersink
Ⓣ	Depth
↻	All Around
↔	Between
⊗	Target Point
▽	Conical Taper
∇	Slope
□	Square

Feature Control Frame



Geometric Dimensioning and Tolerance (GD&T) is the symbolic engineering language used by mechanical designers, manufacturers and inspection personnel to communicate and integrates the functional requirements of the part into the tolerances.