

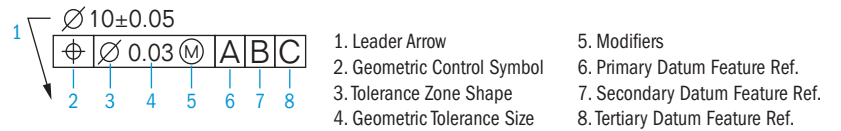
GD&T Symbols Reference Chart

For the best GD&T training, resources, and content visit www.gdandtbasics.com



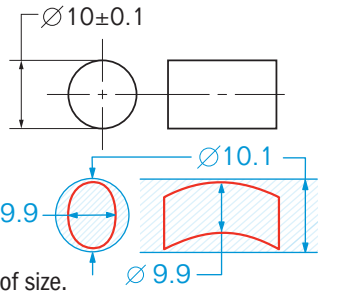
	SYMBOL	ON DRAWING	TOLERANCE ZONE	DATUMS & MODIFIERS
FORM	Straightness (Surface)		Two Parallel Lines 0.03 Apart	A M
	Straightness (Feature of Size)		Ø0.03 Cylindrical Tolerance Zone	A M
	Flatness (Surface)		Two Parallel Planes 0.03 Apart	A M
	Flatness (Feature of Size)		Two Parallel Planes 0.03 Apart	A M
	Circularity		Two Concentric Circles 0.03 Apart	A M
	Cylindricity		Two Concentric Cylinders 0.03 Apart	A M
ORIENTATION	Parallelism (Surface)		Two Parallel Planes 0.03 Apart Datum A	A M
	Parallelism (Feature of Size)		Two Parallel Planes 0.03 Apart Datum A	A M
	Perpendicularity (Surface)		Two Parallel Planes 0.03 Apart Datum A	A M
	Perpendicularity (Feature of Size)		Ø0.03 Cylindrical Tolerance Zone Datum A	A M
	Angularity (Surface)		Two parallel planes 0.03 Apart Datum A	A M
	Angularity (Feature of Size)		Ø0.03 Cylindrical Tolerance Zone Datum A	A M
PROFILE	Profile of Line		Uniform 2D Boundary About True Profile 0.03 Apart Datum A, Datum B	OPTIONAL A M
	Surface Profile		Uniform 3D Boundary About True Profile 0.03 Apart Datum A, Datum B	OPTIONAL A M
LOCATION	Position		Ø0.03 Cylindrical Tolerance Zone Datum A, Datum B	A M
	Concentricity		Measured Diametrically Opposed Points Datum Axis A	A M
	Symmetry		Two Parallel Planes 0.03 Apart Datum A	A M
RUNOUT	Runout		Datum Feature Fixed axially + Rotated	A M
	Total Runout		Datum Feature Fixed axially + Rotated	A M

Feature Control Frame Components



Rule # 1 of GD&T: Envelope Principle

Rule #1 states that the form of a regular feature of size is controlled by its "limits of size." The actual surface of a regular feature cannot extend beyond the envelope prescribed by the feature in perfect form at MMC.



Rule #1 does not apply when:

- The independency symbol is used.
- Straightness/Flatness is applied to a feature of size.
- The part is designated as a "STOCK" size.

Rule # 2: Implied Regardless of Feature Size

Regardless of Feature Size is always implied on geometric tolerances unless otherwise specified by Least Material Condition or Maximum Material Condition modifiers.

RFS - Regardless of Feature Size

Geometric tolerances need to be within tolerance regardless of the size of the feature. This is the default condition on all GD&T unless otherwise specified - See Rule # 2 above.

M Maximum Material Condition Modifier (MMC)

Indicates this specific geometric tolerance is to be assessed at maximum material condition. Bonus geometric tolerance is allowed as the size of the feature deviates away from maximum material condition.

Bonus Tolerance = Actual size deviation from the Max Material Condition

L Least Material Condition Modifier (LMC)

Indicates this specific geometric tolerance is to be assessed at least material condition. Bonus geometric tolerance is allowed as the size of the feature deviates away from least material condition.

Bonus Tolerance = Actual size deviation from the Least Material Condition

Virtual Condition

A static boundary created by the effects of a feature of size's specified MMC or LMC and the geometric tolerance for that material condition.

For internal features like a hole:

Virtual Condition = MMC - Geometric tolerance

For external features like a pin:

Virtual Condition = MMC + Geometric tolerance



Datum Feature Symbol

Indicates a tangible surface or feature of size that is as referenced by a feature control frame.



Datum Target

Reference to datum points, lines or areas needed to create a theoretical datum or datum reference frame for part measurement.



Projected Tolerance

The tolerance zone is extended out from the surface where the feature begins to a specified height. Used commonly with position to represent the virtual condition of a pin stud



Free State

Used to indicate that a specific dimension and its associated tolerance are to be assessed in an unrestrained state. By default, all dimensions are assumed to be evaluated in a "Free State".



Independency

When indicated, Rule #1 is overridden and the part's geometric tolerance is no longer restricted by the limits of size.



Continuous Feature

Used to indicate that a group of two or more features are to be considered as one continuous feature. Rule #1 and any geometric controls will be applied to the considered features as if they are one.



Tangent Plane

A theoretical tangent plane is established on the surface of a part based on the surface high-points. When called, only the virtual tangent plane needs to be within the tolerance zone, instead of all the surface itself.



Basic Dimensions

Dimensions that are listed in a rectangular box that define theoretically exact dimensions. When defining part geometry, GD&T controls will dictate the tolerance for these dimensions as they are untoleranced by themselves.



Between Symbol (For Profile Only)

Specifies the exact limits of the surface or line (usually encompassing multiple features) that the profile tolerance controls. This is indicated by labeling two extremities in a single view.



All Around Symbol (For Profile Only)

Indicates that the geometric tolerance apply to every surface around the part in the specified view.



Unequally Disposed Profile Symbol (For Profile Only)

Indicates the tolerance zone for profile is no longer symmetrical to the true profile. The value the PRECEDES the symbol is the TOTAL tolerance. The value FOLLOWING the symbol indicates how much of the total tolerance is 'shifted' in the direction that adds material to the part.

