GD&T REFERENCE GUIDE

	NAME	SYMBOL	FOR SURFACE OR F.O.S.?	TOLERANCE ZONE SHAPE (see below)	CAN USE MMC OR LMC MODIFIER?	DATUM REF?	CREATES VIRTUAL CONDITION?
Form controls	Straightness	—	Either	a, b, c	Yes, if a F.O.S.	Never	Yes, if on F.O.S.
	Flatness	▱	Either	b	Yes, if a F.O.S.	Never	Yes, if on F.O.S.
	Circularity	○	Surface	d	No	Never	No
	Cylindricity	⌭	Surface	e	No	Never	No
Profile controls	Profile of a Line	⌒	Surface	a, d, f	No*	Usually	No
	Profile of a Surface	⌓	Surface	b, e, g	No*	Usually	No
Orientation controls	Parallelism	//	Either	a, b, c	Yes, if a F.O.S.	Always	Yes, if on F.O.S.
	Perpendicularity	⊥	Either	a, b, c	Yes, if a F.O.S.	Always	Yes, if on F.O.S.
	Angularity	∠	Either	a, b, c	Yes, if a F.O.S.	Always	Yes, if on F.O.S.
Location controls	Position	⊕	F.O.S.	b, c, h	Yes	Always†	Yes
	Concentricity	◎	F.O.S.	c	No	Always	Yes
	Symmetry	≡	F.O.S.	b	No	Always	Yes
Runout controls	Circular Runout	↗	Surface	d	No	Always	Yes**
	Total Runout	↗↗	Surface	e	No	Always	Yes**

a = 2 parallel lines   b = 2 parallel planes   c = cylinder   d = 2 coaxial circles
e = 2 coaxial cylinders   f = 2 irregular line boundaries
g = 2 irregular surface boundaries   h = sphere

*The datum reference(s) may use MMC or LMC modifiers, if a F.O.S. datum

† Exception: position applied to coaxial holes

F.O.S. stands for "feature of size" (a pin, hole, slot, etc.)

**It is not a true "virtual condition" as defined in the standard, but it does create a "worst-case boundary" that may be important for assembly or function.

Helpful things to remember:

The current standard for GD&T is ASME Y14.5-2009, from the American Society of Mechanical Engineers. It replaces the 1994 version.

The rectangular box that contains a GD&T callout is known as the "feature control frame."

A geometric tolerance shown in a feature control frame is always total, not plus/minus. Depending on how it is used, it may be centered around a fixed location, or it may float within a given size limit.

The datum references (the letters at the end of a feature control frame) are given in a specific order to show the relative importance of each (primary, secondary, and tertiary). They do not have to be in alphabetical order, but rather order of precedence.

The modifier Ⓜ is helpful for clearance fits. It allows the tolerance to increase as the size of the feature varies. It can also be used on datum references if there might be looseness or "play" on those features.

Datum features should be identified on physical items (surface, hole, pin, etc.) not on an imaginary center line. Even if the true datum might be a center, the symbol should still appear on the feature from which the center is derived.

Basic dimensions (boxed dimensions) do not have any direct tolerance. Instead, they are indirectly toleranced from a feature control frame. Basic dimensions are most common in conjunction with position and profile controls.

Concentricity is expensive to inspect. Often, position or runout can be used to achieve the same goal. (Reason: concentricity measures the centers of every cross-section, but position measures the center of an envelope, and runout measures the physical surface.)

One of the most powerful GD&T symbols is profile of a surface. It controls a shape (which is defined by basic dimensions) by building a three-dimensional tolerance zone around it. And depending on how it relates to the datums, it can also control orientation and location.

Modifiers to know: Ⓜ = maximum material condition   Ⓛ = least material condition   Ⓕ = free state
Ⓟ = projected tolerance zone   Ⓤ = unequal tolerance zone   Ⓣ = tangent plane

       www.gdtseminars.com