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, 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 MMB or LMB 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 datum 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 establish a perfect dimension, and then the GD&T takes over, in the form of 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:

Ⓕ = free state   Ⓜ = max. mat'l condition/boundary   Ⓛ = least mat'l condition/boundary   Ⓘ = independency rule

Ⓟ = projected tolerance zone   Ⓤ = unequal tolerance zone   Ⓣ = tangent plane

    www.gdtseminars.com

Changes from the 1994 Y14.5 standard

The standard for dimensioning and tolerancing was revised in 2009. ASME Y14.5-2009 now replaces the 1994 version.
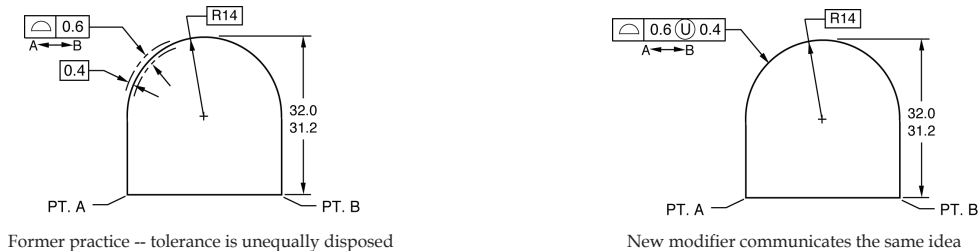
However, each company makes the decision as to when their new designs will begin to use the new standard. The title block or a general note should always be used to clarify which standard is being imposed on a given drawing.

This page highlights the main changes that GD&T users need to be aware of. A more detailed overview can be found in Appendix A of Y14.5.

New modifiers

Ⓘ   This is the "independency" modifier, and it can be used on a feature-of-size dimension to override Rule #1. Recall that Rule #1 requires a given dimension to meet two requirements: one check for the cross-sectional size, and another for the overall envelope size. Both of these measurements must be within the given tolerance range, according to Rule #1.   But with the Ⓘ modifier, only the cross-sectional size is to be measured. It is the equivalent of the fomer note, "Perfect form at MMC not req'd." See paragraph 2.7.3.

Ⓤ   This can be used with the two profile symbols, and it indicates an unequal tolerance zone. A profile zone is presumed to be equally disposed around the perfect profile, but sometimes it is desired to have the zone be all to one side, or have a larger portion on one side than the other. In former versions of Y14.5, it was accomplished by visually showing the tolerance zone on the drawing with one or more phantom lines. The Ⓤ modifier eliminates this reliance on graphical means, and embeds the idea into the feature control frame. Important: the number after the Ⓤ indicates the portion of the full tolerance zone that lies away from the part's material. See paragraph 8.3.1.2.

Former practice -- tolerance is unequally disposed

New modifier communicates the same idea

The "all over" symbol is new; it means that the given tolerance (profile) applies to all surfaces of the part. The "all around" symbol has been around for years (a single circle), but it only applied the tolerance to the perimeter in a given view. See paragraph 8.3.1.6.

⟨CF⟩   CF stands for "continuous feature." This is not directly related to GD&T, but it is a way to denote that two or more holes, for instance, are to be coaxial, essentially creating one feature that is interrupted in the middle. See paragraph 2.7.5.

This is similar to the regular datum target, but the angled lines indicate that it is a "movable datum target." A physical simulator may be moved to control a degree of freedom; the movement is normal to the true profile unless otherwise indicated. See paragraph 4.24.6.
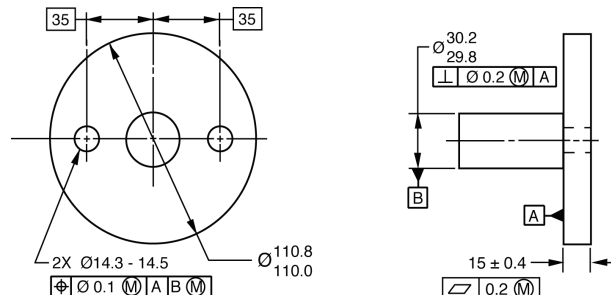
This is called the "translation modifier," and it is similar to the movable datum target. It can be placed after the datum reference letter at the end of a feature control frame, and it means that a physical simulator for the datum feature of size can slide back and forth to find the best fit with that feature. See paragraph 4.11.10.

The new standard allows a datum reference to be called out in a feature control frame with only certain degrees of freedom being constrained. The specific degrees of freedom are given after the datum letter by noting x, y, z, u, v, or w. See paragraphs 4.22 and 4.23.

New definitions

In past editions of Y14.5, the MMC, LMC, and RFS conditions were described when applied to datum references. (This is not the notion of "bonus tolerance," but rather the idea of "datum shift.") While the Ⓜ and Ⓛ modifiers can still be applied to a feature-of-size datum, it is now understood that they invoke the concepts of MMB (maximum material boundary) and LMB (least material boundary), respectively.

As seen below, datum B is referenced with the Ⓜ modifier. Taken literally, the MMC of the datum feature (the pin) would be 30.2. But to have the part assemble properly, the datum simulator really needs to be 30.4, because of the perpendicularity tolerance on the pin. So the new term "maximum material boundary" describes the effect of Ⓜ on the datum reference, and it is understood to include any geometric tolerance on the datum feature. If no modifier is given, the concept is called RMB -- regardless of material boundary -- rather than RFS. See paragraph 7.3.6.

New use for flatness

Flatness usually applies to a surface. The new standard allows flatness to apply to a center plane, if the feature control frame is shown with the size dimension (see the example above). In previous editions of Y14.5, this same concept was invoked with straightness. See paragraph 5.4.2.1.