	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	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	Profile of a Line		Surface	a, d, f	No**	Usually	No
	Profile of a Surface		Surface	b, e, g	No**	Usually	No
Orientation	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	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	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

* Concentricity and symmetry were deleted from ASME Y14.5 in its 2018 edition

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

† Exception: position applied to coaxial holes

†† 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 is ASME Y14.5-2018, from the American Society of Mechanical Engineers. It replaces the 2009 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 perfect dimensions, 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 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:

Ⓜ = max. mat'l condition/boundary
Ⓕ = free state Ⓟ = projected tolerance zone
Ⓛ = least mat'l condition/boundary
Ⓤ = unequal tolerance zone
Ⓘ = independency rule
Ⓣ = tangent plane