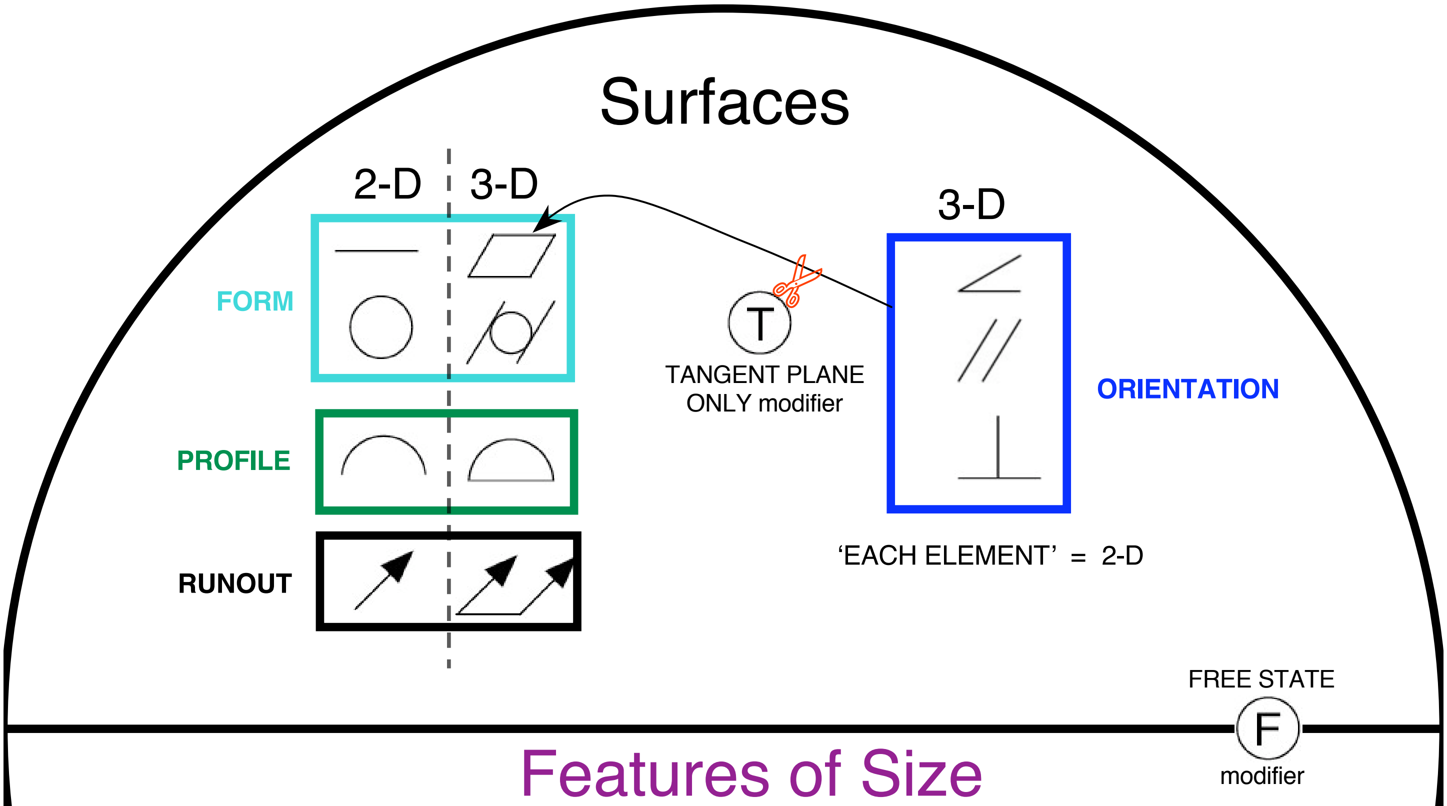
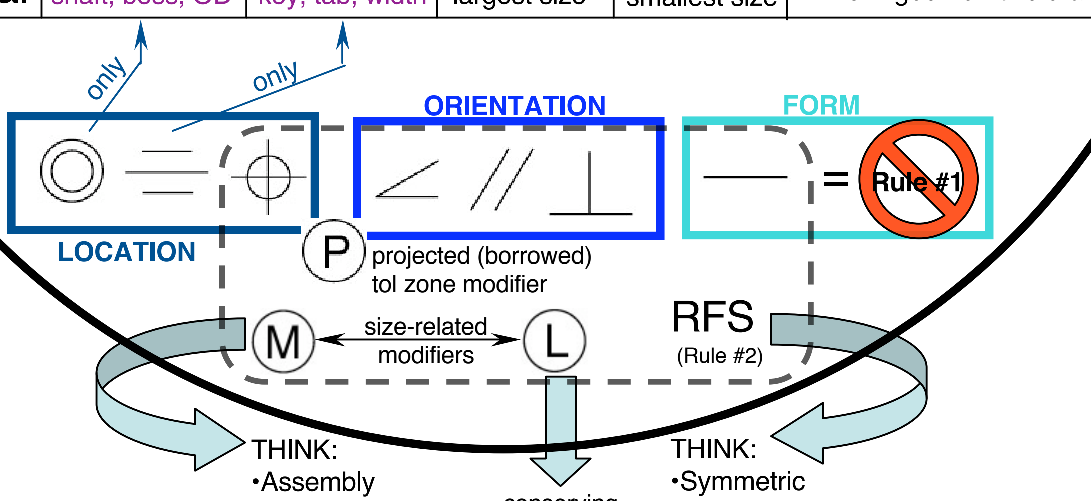
Map of the GD&T World

Based on ASME Y14.5M-1994



	round ∅	square	MMC Ⓜ	LMC Ⓛ	Virtual Condition
internal	hole, ID	keyway, slot	smallest size	largest size	MMC – geometric tolerance
external	shaft, boss, OD	key, tab, width	largest size	smallest size	MMC + geometric tolerance



THINK:
- Assembly
- Attribute gage
- Boundary
- Bonus
- Shift
- Virtual Condition

conserving wall thickness

THINK:
- Symmetric
- Variable gage
- Tolerance zone diameter or planes
- No bonus
- No shift
- Varying AME

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?
Form controls	Straightness	⏤	Either	a, b, c	Yes, if a F.O.S.	Never
	Flatness	⏥	Surface	b	No	Never
	Circularity	○	Surface	d	No	Never
	Cylindricity	⌭	Surface	e	No	Never
Profile controls	Profile of a Line	⌒	Surface	f	No*	Usually
	Profile of a Surface	⌓	Surface	b, e, g	No*	Usually
Orientation controls	Parallelism	//	Either	a, b, c	Yes, if a F.O.S.	Always
	Perpendicularity	⊥	Either	a, b, c	Yes, if a F.O.S.	Always
	Angularity	∠	Either	a, b, c	Yes, if a F.O.S.	Always
Location controls	Position	⌖	F.O.S.	b, c, h	Yes	Always†
	Concentricity	◎	F.O.S.	c	No	Always
	Symmetry	⌯	F.O.S.	b	No	Always
Runout controls	Circular Runout	↗	Surface	d	No	Always
	Total Runout	⌰	Surface	e	No	Always

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 if a F.O.S. datum

† Exception: position applied to coaxial holes

Helpful things to remember:

The accepted standard for GD&T is ASME Y14.5M-1994, published by the American Society of Mechanical Engineers.

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.

Training provided through

STOLTER LLC

www.gd-t.com