GD&T

Geometric Dimensioning & Tolerancing

GD&T

- GD&T is an international language that is used on engineering drawings to accurately describe a part
- This language consists of well defined set of symbols, rules, definition & conventions.
- GD&T is a Design Tool : designer can properly apply geometric tolerance, they must carefully consider the fit and function of each feature of any part.
- Feature tolerance with GD&T reflects the actual relationship between mating parts.
- GD&T encourage a dimensional philosophy called “FUNCTIONAL DIMENSIONING”: functional dimensioning that defines a part based on how it functions in the final product.
- GD&T encourages a process called “SIMULTANEOUS ENGINEERING” where design is a result of input from marketing, manufacturing, inspection, assembly & service.

GD&T Terminology

Feature:

Feature is the general term applied to a physical portion of a machine part, such as a surface, pin, tab, hole, slot …

GD&T Terminology

- **Maximum Material Condition (MMC):** The condition where a size feature contains the maximum amount of material within the stated limits of size. i.e., largest shaft and smallest hole.

  When MMC is used, the Geometric tolerance only applies when the feature of size or datum is at its MMC size.

  Usage: fits, clearances.

- **Least Material Condition (LMC):** The condition where a size feature contains the least amount of material within the stated limits of size. i.e., smallest shaft and largest hole.

  When LMC is used, the Geometric tolerance only applies when the feature of size or datum is at its LMC size.

  Usage: Wall thickness.

GD&T Rules – Rule #1

GD&T Rules – Rule #2

GD&T Bonus Tolerance

Bonus Tolerance

GD&T Bonus Tolerance

The characteristics to which it can be applied are as follows:

- Straightness, parallelism, squareness, angularity, position, concentricity, symmetry

The characteristics to which the maximum material condition concept cannot be applied are as follows:

- Flatness, roundness, cylindricity, profile of a line, profile of a surface, run-out.

GD&T – Virtual Condition

GD&T – Datums

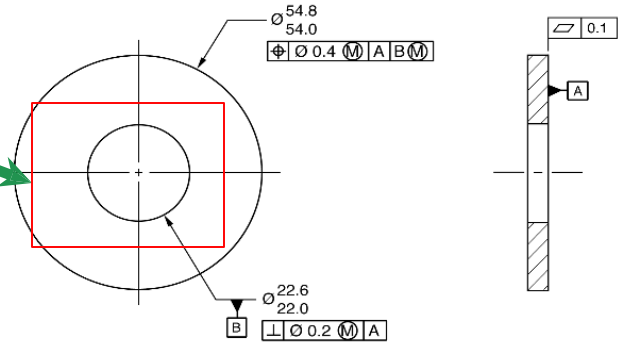
Datum

- It is theoretically exact point, axis, or plane
- A datum is feature that is used as base / origin for dimensions and tolerances
- A datum is considered as an exact base for dimensioning purposes.
- A feature is any physical item on machine part, while a datum is a feature that is used as a base for dimensioning purposes.

Datum Feature

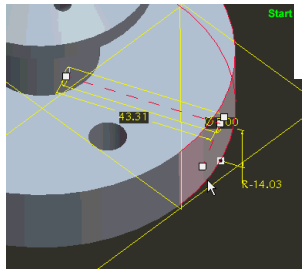
An actual feature of a part that is used to establish a datum is known as

Datum-feature

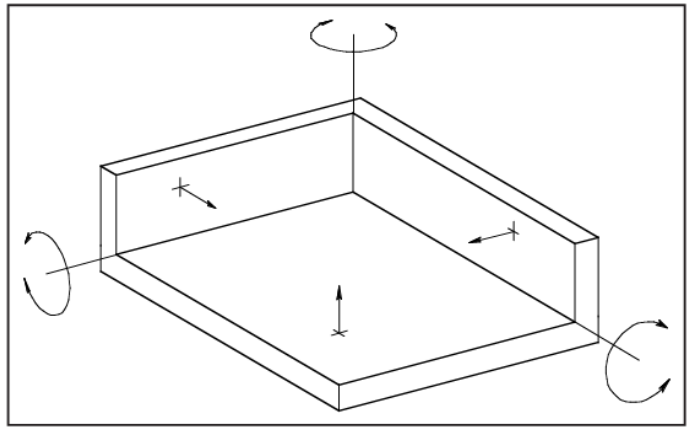


Datum Feature Symbol

A GD&T symbol in a print indicating a part feature that acts as a datum feature and that contacts a datum reference frame simulator.



GD&T – Datums



A part has six degrees of freedom:

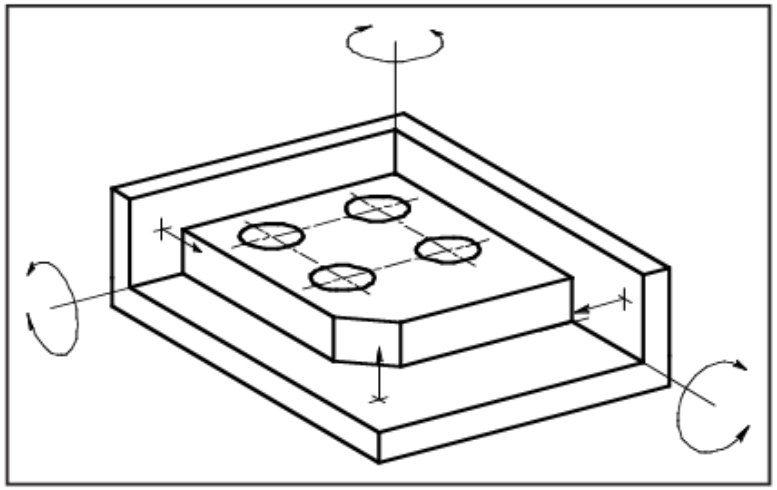
a. **3 Translational**
b. **3 Rotational**

Immobilizing the part using 3 mutually perpendicular planes.

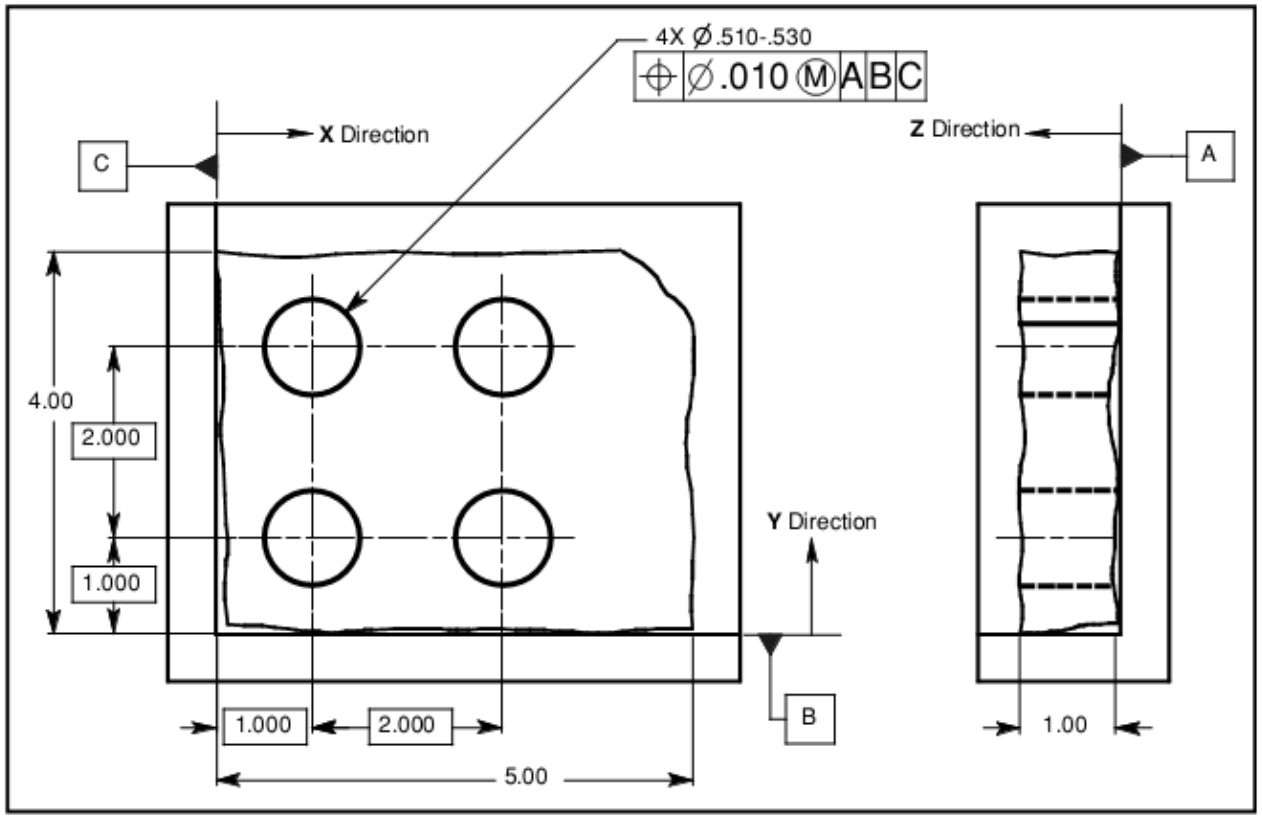
Primary Datum: should have min. 3 pt of contact with part

Secondary Datum: should have min. 2 pt of contact with part.

Tertiary Datum: should contact at min 1 pt with part.



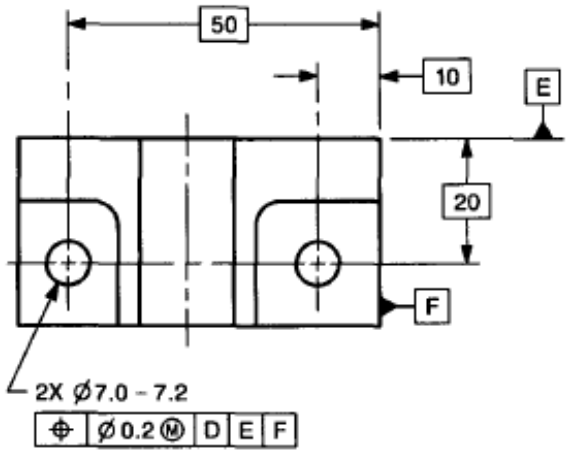
GD&T – Datums



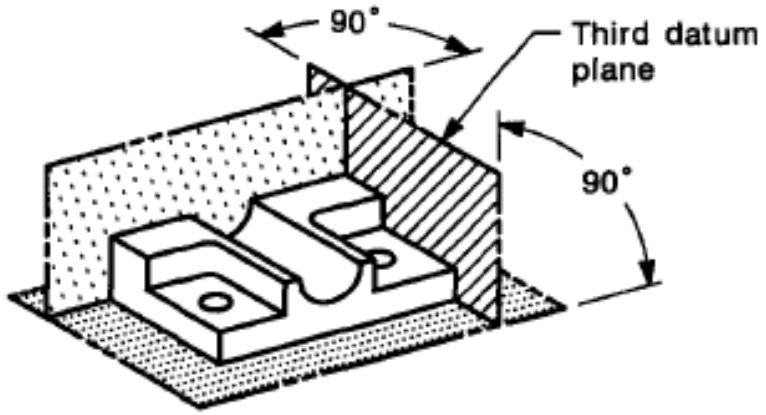
Order of precedence

GD&T – Datums

- An actual feature of a part that is used to establish a datum.
- **@ Virtual Condition:** A size where its axis / center plane is controlled by geometric tolerance. In such cases the datum feature applies at its virtual condition.
- **IDENTIFICATION:** on the drawing by means of a datum feature symbol. This is not applied to center line, centre plane or axes except planes

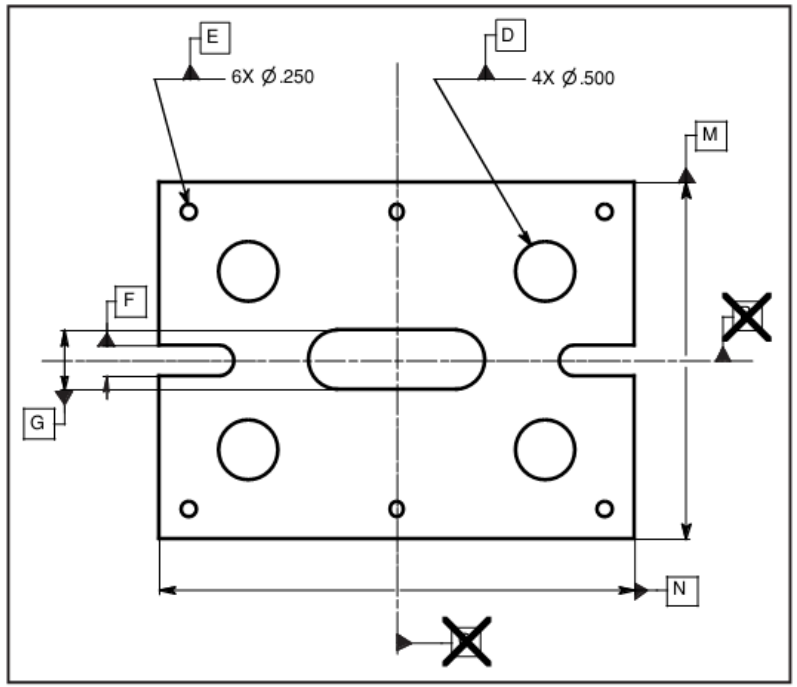


SPECIFYING DATUM FEATURES IN AN ORDER OF PRECEDENCE

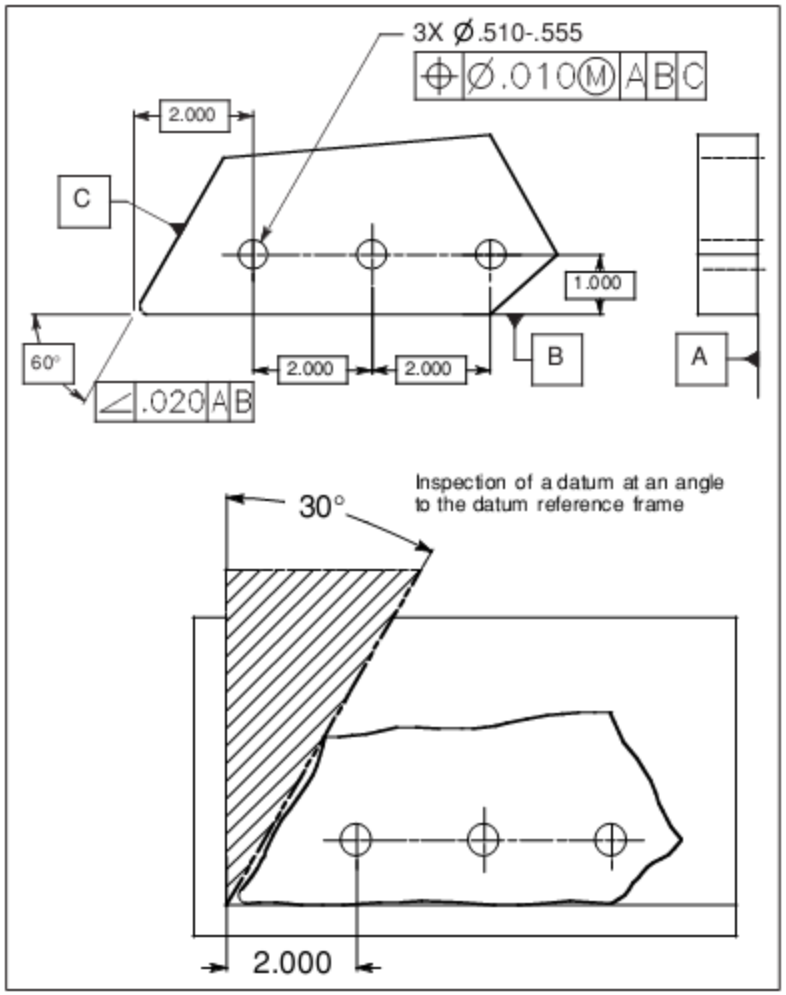


GD&T – Datums

Datum feature symbols must not be applied to centerlines, center planes and axes.



GD&T – Datums



Inclined Datum Features:

Datum features are not required to be perpendicular to each other. Only the datum reference frame is defined as three mutually perpendicular intersecting planes. To inspect this part, a precision 30deg wedge is placed in a datum reference plane.

Geometric Characteristic Symbols

Geometric Characteristics Symbols

There are 14 Geometric Characteristic Symbols

	Type of Tolerance	Characteristic	Symbol
For Individual Symbols	Form	Straightness	⏤
		Flatness	▱
		Circularity	○
		Cylindricity	⌭

Geometric Characteristics Symbols

	Type of Tolerance	Characteristic	Symbol
For Individual or Related Features	Profile	Profile of a Line	⌒
		Profile of a Surface	⌓

Geometric Characteristics Symbols

	Type of Tolerance	Characteristic	Symbol
For Related Features	Orientation	Angularity	
		Perpendicularity	
		Parallelism	
	Location	Position	
		Concentricity	
		Symmetry	
	Runnout	Circular Runnout	
		Total Runnout	

Geometric Characteristics Symbols - Modifiers

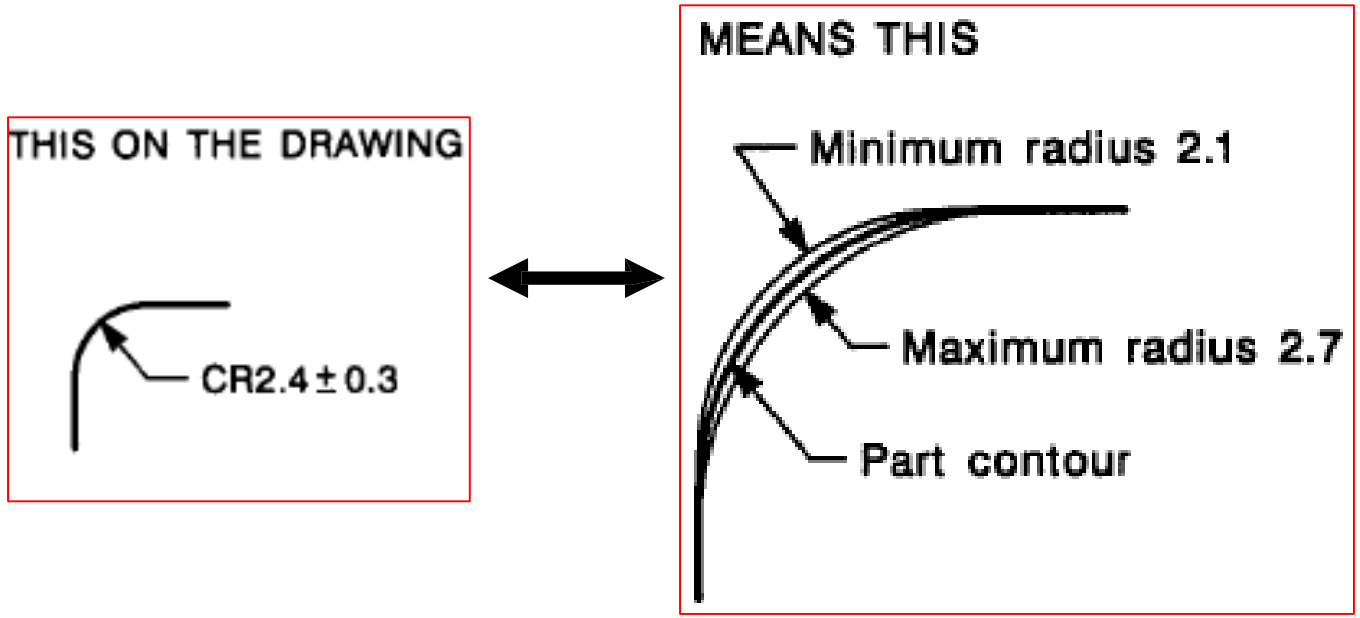
TERM	SYMBOL
AT MAXIMUM MATERIAL CONDITION	Ⓜ
AT LEAST MATERIAL CONDITION	Ⓛ
PROJECTED TOLERANCE ZONE	Ⓟ
FREE STATE	Ⓕ
TANGENT PLANE	Ⓣ
DIAMETER	⌀
SPHERICAL DIAMETER	S⌀
RADIUS	R
SPHERICAL RADIUS	SR
CONTROLLED RADIUS	CR
REFERENCE	( )
ARC LENGTH	⌒
STATISTICAL TOLERANCE	⟨ST⟩
BETWEEN	◄►

Modifying symbols

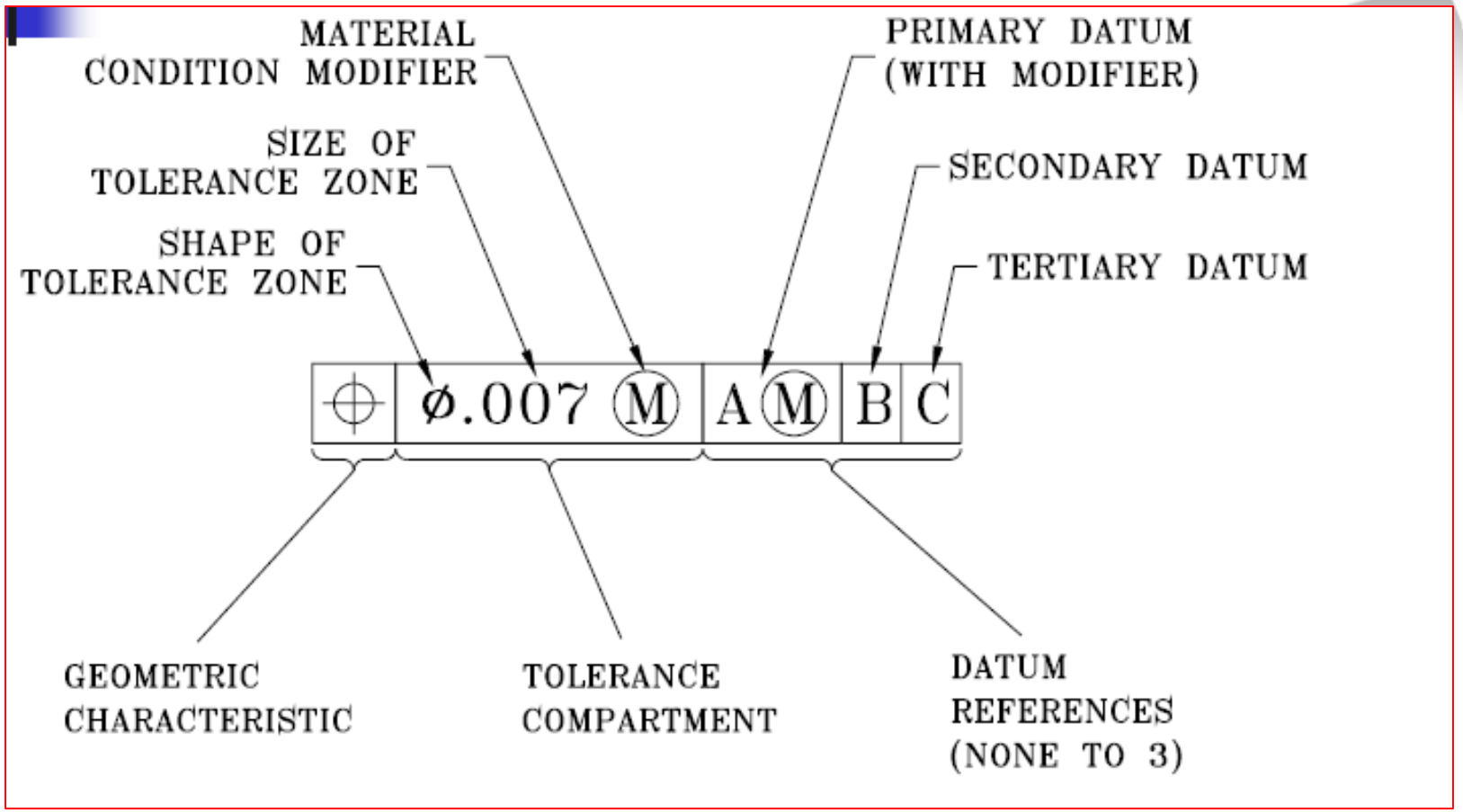
Additional Symbols

Controlled Radius

- A radius crates a tolerance zone defined by two arcs that are tangent to adjacent surface. In addition, at no point on the radius can the curve be greater than the maximum limit, nor smaller than the minimum limit.



Feature Control Frame

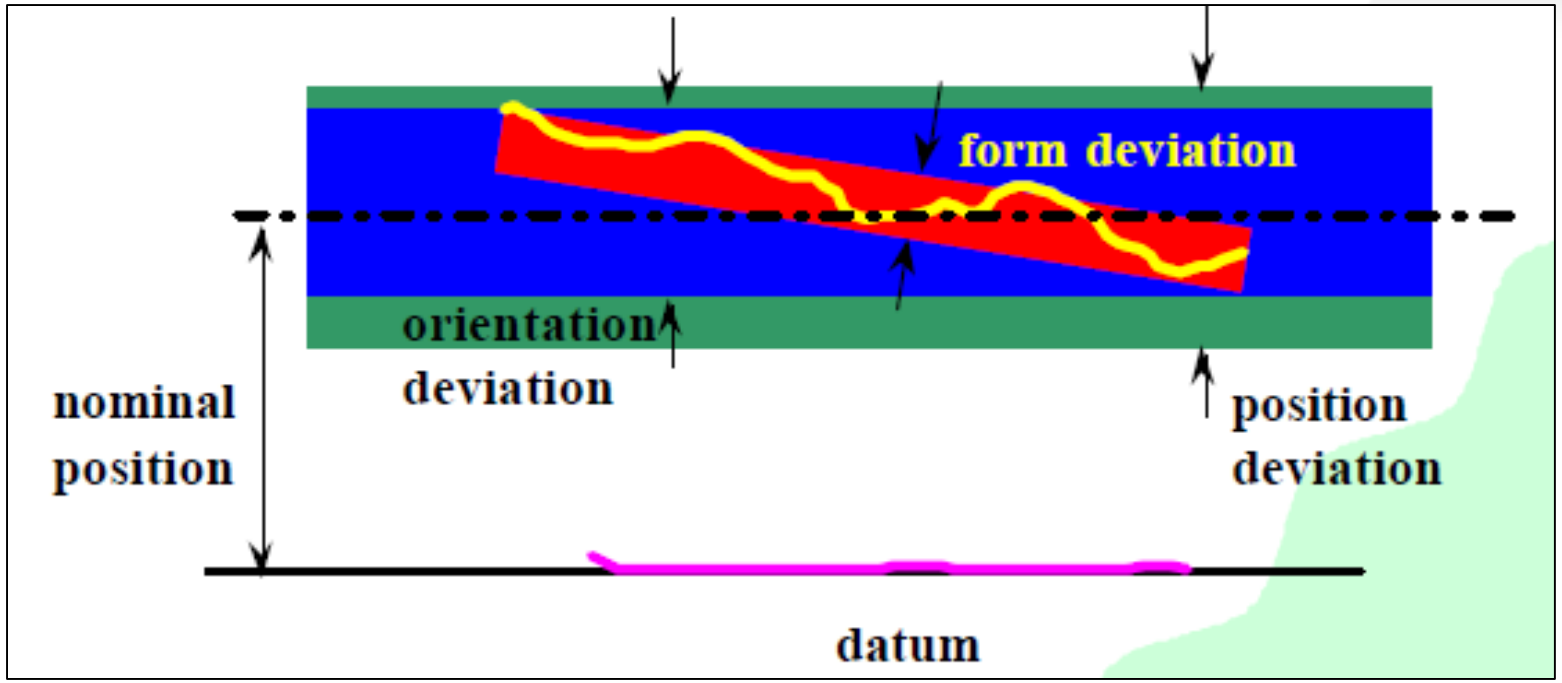


Form Tolerance

Orientation Tolerance

Location Tolerance

Location


form deviation
orientation
deviation
nominal
position
position
deviation
datum

Runout Tolerance

Profile Tolerance

Thank You !