GD&T

Geometric Dimensioning and Tolerancing

WHAT IS GD&T?

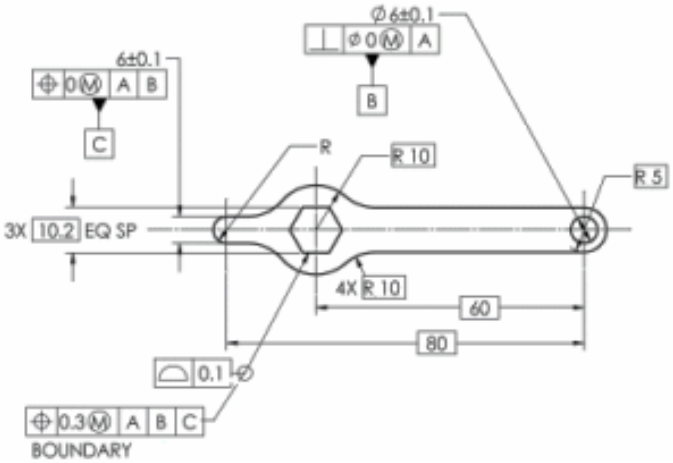
GD&T is a system for defining and communicating engineering tolerances. It uses symbols to describe the nominal geometry (theoretically perfect) of a part and its allowable variation in form and possible size of individual features. It tells the manufacturing company the degree of accuracy and precision needed on each controlled feature of a part.

There are several standards available worldwide that describe the symbols and define the rules used in GD&T. One of them being the ASME Y14.5-2009.

The GD&T allow for accurately defining the requirements for part and assembly geometry and ensuring that the allowable part and assembly defined on the drawing leads to parts that have the desired form, fit and function as intended.

These are some of the fundamental rules:

- All dimensions must have a tolerance since every feature on every manufactured part is subject to variation. Therefore the limits of allowable variation must be specified.
- Dimensions and tolerances should completely define the nominal geometry and allowable variation.



WHY USING GD&T

An engineering drawing is a document that communicates a precise description of a part. It may communicate the geometry of the part, the critical functional relationships, tolerances, material, surface coatings, as well as part documentation such as part number and drawing revision number.

However, without GD&T it could be difficult to communicate functional requirements and the design intent of a part. GD&T is really a method for defining a part's geometry that goes beyond the form description.

COMMONLY USED GD&T TERMS AND SYMBOLS

GD&T symbols are known universally as a method of specifying requirements without using notes or words on the drawing. Most of the symbols used between ASME and ISO are identical but there are some differences. The chart below regroups the most common GD&T symbols and their appearance for both ASME and ISO.

SYMBOL	ASME Y14.5M	ISO
FEATURE CONTROL FRAME	⌖ Ø.030 Ⓜ A B C	⌖ Ø.030 Ⓜ A B C
DIAMETER	Ø	Ø
SPHERICAL DIAMETER	SØ	SØ
AT MAXIMUM MATERIAL CONDITION	Ⓜ	Ⓜ
AT LEAST MATERIAL CONDITION	Ⓛ	Ⓛ
REGARDLESS OF FEATURE SIZE	Ⓢ	NONE
PROJECTED TOLERANCE ZONE	Ⓟ	Ⓟ
FREE STATE	Ⓕ	Ⓕ
TANGENT PLANE	Ⓣ	Ⓣ (proposed)
STATISTICAL TOLERANCE	ST	NONE
RADIUS	R	R
CONTROLLED RADIUS	CR	NONE
SPHERICAL RADIUS	SR	SR
BASIC DIMENSION (theoretically exact dimension in ISO)	50	50
DATUM FEATURE	B	B or
DATUM TARGET	Ø8 A1	Ø8 A1
TARGET POINT	×	×
DIMENSION ORIGIN	⌖	⌖
REFERENCE DIMENSION (auxiliary dimension in ISO)	(50)	(50)
NUMBER OF PLACES	8X	8X
COUNTERBORE/SPOTFACE	⌴	⌴ (proposed)
COUNTERSINK	⌵	⌵ (proposed)
DEPTH/DEEP	↧	↧ (proposed)
SQUARE	□	□
ALL AROUND		NONE
DIMENSION NOT TO SCALE	150	150
ARC LENGTH	150	150
BETWEEN	↔	NONE
SLOPE		
CONICAL TAPER		
ENVELOPE PRINCIPLE	NONE (implied)	Ⓔ

*MAY BE FILLED OR NOT FILLED

Geometric Characteristic Symbols			
Symbol	Description		Type of tolerance
—	Straightness	Form	Individual features
▱	Flatness		
○	Circularity		
⌭	Cylindricity		
⌒	Profile line	Profile	Individual or related features
⌓	Profile surface		
∠	Angularity	Orientation	Related features
⊥	Perpendicularity		
//	Parallelism		
⌖	Position	Location	
◎	Concentricity		
⌯	Symmetry		
*↗	Runout Circular	Runout	
*⌰	Runout Total		

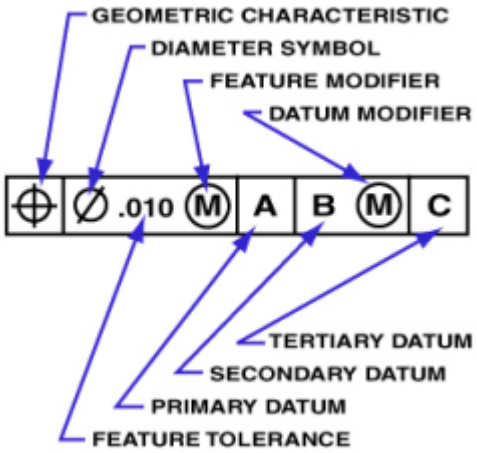
*Either filled or unfilled

No datum or datums needed

Depending on the situation

A datum or datums are required

- Feature Control Frames: This is potentially the most significant symbol in any geometric tolerancing system. It provides the instructions and requirements for the feature to which it is related. Only one requirement is contained in a feature control frame. Multiple features require multiple feature control frames. The first frame contains one of the 14 geometric characteristic, the second contains the total tolerance for the particular feature, the third and subsequent compartments of the feature control frame contain specified datums. It is important to note that the feature control frame controls the surface of a flat feature and the axis or median plane of a feature of size.



- **Material Condition Modifiers:** Often it becomes necessary to refer to a feature in its largest or smallest condition, or it may be necessary to refer to a feature regardless of feature size. These conditions are designated as the maximum material condition (MMC), the least material condition (LMC) and regardless of feature size (RFS). For example, MMC would be used to express the largest pin or the smallest hole. LMC would be used to describe the smallest pin or the largest hole. RFS might be used to show that a geometric tolerance applied to any increment of feature size of any feature within its size tolerance.

HOW TO LEARN MORE ABOUT GD&T

Complete courses

- University of Arizona:
  http://fp.optics.arizona.edu/optomech/files/GDandT%20fundamentals.pdf
- Illinois valley Community College:
  http://www.ivcc.edu/uploadedFiles/_faculty/smith/GDT%20Spring%202010t.pdf
- University of Puerto Rico:
  http://me.uprm.edu/NX%20files/gd___t.pdf

Other courses

- Ohio University:
  - General Tolerance:
    http://www.ohio.edu/people/tc285202/288/General%20Tolerance.pdf
  - Basic GD&T and Datums:
    http://www.ohio.edu/people/tc285202/288/Basic%20GD&T%20-%20Datums.pdf
  - Positional Tolerance:
    http://www.ohio.edu/people/tc285202/288/Positional%20Tolerance.pdf
  - Flatness:
    http://www.ohio.edu/people/tc285202/288/Flatness.pdf
  - Straightness, Circularity, Cylindricity:
    http://www.ohio.edu/people/tc285202/288/Straightness,%20Circularity,%20Cylindricity.pdf
  - Angularity, Parallelism, Perpendicularity:
    http://www.ohio.edu/people/tc285202/288/Angularity,%20Parallelism,%20Perpendicularity.pdf
  - Runout:
    http://www.ohio.edu/people/tc285202/288/Runout.pdf