GEOMETRIC CHARACTERISTICS AND THEIR ATTRIBUTES

CHARACTERISTIC	SYMBOL	TYPE of FEATURE CONTROLLED	FEATURE CONTROL FRAME PLACEMENT OPTIONS	BOUNDARY / TOL ZONE SHAPE MODIFIER	TOLERANCE MODIFIABLE TO MMC OR LMC	NUMBER OF DATUM REFERENCES ALLOWED	MMC/LMC ALLOWED FOR DATUM REFERENCE (S)	BASIC DIMENSIONS REQD
STRAIGHTNESS / F	—	CYL-SURFACE ELEMENTS	b			NONE		
		CYL-DERIVED MEDIAN LINE	a, d	Ø	√	NONE		
		PLANE-LINE ELEMENTS	b, c			NONE/PRI		
FLATNESS / F	▱	PLANE	b, c			NONE		
		WIDTH-DERIVED MEDIAN PLANE	a, d		√	NONE		
CIRCULARITY / F	○	REVOLUTE, SPHERE	a, b, d			NONE		
PROFILE OF A LINE	⌒	ALL	b			NONE/PRI/SEC/TER	√	REQD
PROFILE OF A SURFACE	⌓	REVOLUTE	b			NONE/PRI/SEC/TER	√	REQD
		OTHER (NON-REVOLUTE)	b			NONE/PRI/SEC/TER	√	REQD
		COPLANARITY OF PLANES	b			NONE		
PERPENDICULARITY / O	⊥	PLANE (INCL LINE ELEMENTS)	b, c			PRI/SEC/TER	√	
		CYLINDER	a, d	Ø	√	PRI/SEC/TER	√	
PARALLELISM / O	//	WIDTH	a, d		√	PRI/SEC/TER	√	
		REVOLUTE-RADIAL ELEMENT	b, c			PRI/SEC/TER	√	
ANGULARITY / O	∠	PLANE (INCL LINE ELEMENTS)	b, c			PRI/SEC/TER	√	REQD
		CYLINDER	a, d	Ø	√	PRI/SEC/TER	√	REQD
		WIDTH	a, d		√	PRI/SEC/TER	√	REQD
		REVOLUTE-RADIAL ELEMENT	b, c			PRI/SEC/TER	√	REQD
POSITION / L	⌖	CYLINDER	a, d	Ø	√	PRI/SEC/TER	√	REQD
		WIDTH	a, d		√	PRI/SEC/TER	√	REQD
		SPHERE	a, d	SØ	√	PRI/SEC/TER	√	REQD
CONCENTRICITY / L	◎	ALL NON-SPHERICAL	a, b, d	Ø		PRI/SEC/TER		
		SPHERE	a, b, d	SØ		PRI/SEC/TER		
SYMMETRY / L	⌯	OPPOSED POINTS	a, d			PRI/SEC/TER		
CIRCULAR RUNOUT	↗	REVOLUTE	a, b, d			PRI/SEC		
TOTAL RUNOUT	⌰	CYLINDER	a, b, d			PRI/SEC		
		PLANE PREP TO AXIS	b, c			PRI/SEC		

FEATURE CONTROL FRAME PLACEMENT OPTIONS (LEGEND)		
(a) Place the frame below or attached to a leader - directed callout or dimension pertaining to the feature.		
(b) Run a leader from the frame to the feature.		
(c) Attach either side or either end of the frame to an extension line from the feature, provided it is a plane surface.		
(a) Place the frame below or attached to a leader - directed callout or dimension pertaining to the feature.		
(d) Attach either side or either end of the frame to an extension of the dimension line pertaining to a feature of size.		
MEANING	SYMBOL	
AT MAXIMUM MATERIAL CONDITION	Ⓜ	The condition where a feature of size contains the maximum amount of material within the stated limits of size. Ex: Minimum Hole Size and Maximum Shaft Size.
AT LEAST MATERIAL CONDITION	Ⓛ	The condition where a feature of size contains the least amount of material within the stated limits of size. Ex: Maximum Hole Size and Minimum Shaft Size.
REGARDLESS of FEATURE SIZE	**R F S**	This is the default condition for all geometric tolerances. No bonus tolerances allowed. Functional gages may not be used.
PROJECTED TOLERANCE ZONE	Ⓟ	When the symbol is shown, it means the stated tolerance zone is extended beyond the surface of the part, not witin the part.
FREE STATE	Ⓕ	This symbol indicates the parts must not be restricted during inspection.
TANGENT PLANE	Ⓣ	The tangent plane modifier means that the form error, flatness and straightness, of the surface is ignored. Therefore, it's necessary to provide some form control.
DIAMETER	⌀	This symbol replaces the word "Diameter". It should be used anywhere there is a diameter on the drawing, and when a tolerance zone is cylindrical.
SPHERICAL DIAMETER	**SØ**	The symbol for spherical diameter precedes the size dimension of the feature and the positional tolerance value, to indicate a spherical tolerance zone.
RADIUS	**R**	Creates a zone defined by two arcs (the minimum and maximum radii). The part surface must lie within this zone.
CONTROLLED RADIUS	**CR**	Where a controlled radius specified, the part contour within the crescent-shaped tolerance zone must be a fair curve without flats or reversals. Additionally, radii taken at all points on the part contour shall neither be smaller than the specified minimum limit nor larger than the maximum limit.
CYLINDRICITY	⌭	Describes a condition of a surface of revolution in which all points of a surface are equidistant from a common axis.
REFERENCE DIMENSION	**( )**	A dimension usually without a tolerance, used for information purposes only. It doesn't govern production or inspection operations.
ARC LENGTH	⌒	Indicating that a dimension is an arc length measured on a curved outline. The symbol is placed above the dimension.
STATISCAL TOLERANCE	⟨ST⟩	A tolerance for a part of an assembly based on the results from a statistical calculation. The desired result is larger tolerances.

BETWEEN	⟷	Indicating the tolerance zone extends to include multiple surfaces.

SYMBOLS, TERMS and RULES

1. What type of geometric tolerance has nodatum features?	FORM CONTROLS
2. Which of the form tolerances is the most common?	FLATNESS
3. What type of geometric tolerances indicates an angular relationship with specific datum features?	ORIENTATION CONTROLS
4. If the datum feature symbol is placed in line with a dimension line or on a feature control frame associated with a feature of size, then the datum feature is what kind of feature?	A FEATURE OF SIZE
5. Where only a tolerance of size is specified, the limits of size of an individual feature prescibe the extent to which variations in its geometric form, as well as size, are allowed.	RULE #1

Relationship Between Material Modifiers and Fits: MMC, LMC and RFS Modifiers and Fit Tolerances

* Bonus Tolerance: When MMC is shown modifying a particular tolerance, the stated tolerance applies only when the feature being controlled is at MMC. The bonus is the difference between the actual size and the MMC size and may be added directly to the orginal tolerance. Actual Location of Hole Axis (X=.008, Y=.006 from true position). The fomulas below used to calculate the bonus tolerance and total positional tolerance: (The Example shown here is the *Calculation of Bonus Tolerance for an Internal Feature*).	Dim & Tol = ⌀ 2.000/2.020 ⌖ ⌀.010Ⓜ A Actual=2.012 MMC=2.000 Bonus=.012 Tol=.010 / Total Pos Tol=.022
Actual Mating Envelope Size - MMC Size = Bonus Tolerance	
Bonus Tolerance + Geometric Tolerance = Total Positional Tolerance	

* Design Intent: Easier Assembly (Implied: Less Location Accuracy)	Worst Case Condition for Assembly:	*Maximum material condition*
MMC: More Clearance, Looser Fit. The design intent of the maximum material modifier being easy and guaranteed assembly, we don't care if the mating pin wiggles around a bit after assembly. So we encourage the hole to be larger, putting a maximum material limit. As the hole gets larger in size (hence moving from maximum material condition toward least material condition), it provides a looser fit within the mating pin. The clearance gained can be used as a bonus tolerance for position tolerance.		Example: Hole ⌀30 +.021/.0 ⌖ ⌀.010Ⓜ A B C Pin: ⌀30 -.007/-.020
* Design Intent: More Location Accuracy (Implied: Less Easy Assembly)	Worst Case Condition for Location Accuracy:	*Least material condition*
LMC: Less Clearance, Tighter Fit. The design intent of the least material modifier being more location accuracy, we don't want the matting pin to wiggles around so much after assembly. So we encourage the hole to be smaller, putting a least material limit. As the hole gets smaller in size, it provides a tighter fit with the mating pin. The tightness gained can be used as a bonus tolerance for position. The use of LMC can pose some risk in the assembly, as it is difficult to design and manufacture the correct tightness limit for the assembly (as opposed to looseness limit in the MMC case). If the hole gets too small, we can end up getting an unwanted interference fit.		Example: Hole ⌀30 +.021/.0 ⌖ ⌀.010Ⓛ A B C Pin: ⌀30 .0/-.013
* Design Intent: High and Robust Location Accuracy	RFS: No Clearance, Interference Fit	*Regardless of Feature Size*
RFS: No Clearance, Interference Fit. The design intent of the Regardless of Feature Size condition is the location accuracy. Once the pin is press fit into the hole, the position of the pin is determined and it cannot wiggle at all. The size of the hole has no relationship with the location of the hole. If the hole size gets larger or smaller, there is no bonus added to position tolerance.		Hole ⌀30 +.021/.0 ⌖ ⌀.010 A B C Pin: ⌀30 .035/+.022

MMC: As the hole get larger, bonus tolerance increases.

LMC: As the hole gets smaller, bonus tolerance increases.

RFS: There is no relationship between the hole size and the bonus tolerance.