PRESENT BY

Symbol ALL

52+ GD & T SYMBOLS

Table of Contents

No	Name	Page No
1	All Around Symbol	04
2	All Over Specification	04
3	All Around This Side of Parting Line [ASME Y14.5-2009 Section 3.14.1]	04
4	All Over This Side of Parting Line [ASME Y14.5-2009 Section 3.14.2]	04
5	Angularity	04
6	Arc Length	05
7	Basic Dimension	05
8	Between	05
9	Concentricity	05
10	Conical Taper	05
11	Continuous Feature [ASME Y14.5-2009 Section 2.7.5]	05
12	Controlled Radius	06
13	Counterbore/Spotface	06
14	Countersink	06
15	Cylindricity	06
16	Datum Feature	07
17	Datum Target	07
18	Depth/Deep	07
19	Diameter	07
20	Dimension Origin	07
21	Feature Control Frame	07
22	Flatness	07
23	Free State Variations	08
24	Least Material Condition (LMC)	08
25	Independency Symbol [ASME Y14.5-2009 Section 2.7.3]	08
26	Maximum Material Condition (MMC	08
27	Movable Datum Targets [ASME Y14.5-2009 Section 4.24.6]	08
28	Number of Places	08
29	Parallelism	08
30	Parting Lines [ASME Y14.5-2009 Section 3.14]	09

31	Perpendicularity	09
32	Position Tolerance	09
33	Profile of a Line	09
34	Profile of a Surface	09
35	Projected Tolerance Zone	09
36	Radius	10
37	Reference Dimension	10
38	Roundness	10
39	Runout	10
40	Slope	10
41	Spherical Diameter	10
42	Spherical Radius	11
43	Spotface [ASME Y14.5-2009 Section 1.8.14]	11
44	Square	11
45	Statistical Tolerance	11
46	Straightness	12
47	Symmetry	12
48	Tangent Plane	12
49	Target Point	12
50	Total Runout	12
51	Datum Translation Symbol [ASME Y14.5-2009 Section 3.3.26]	12
52	Unilateral and Unequally Disposed Profile Tolerance [ASME Y14.5-2009 Section 8.3.1.2]	13

NO.	Symbol	Name	Meaning
1		All Around Symbol	Indicating that a tolerance applies to surfaces all around the part.
2		All Over Specification	In addition to a general profile of a surface tolerance there is the option of specifying that the tolerance applies all over on the field of the drawing. It is important to realize that this specification, whether in a general note or on the field of the drawing, applies UNLESS OTHERWISE SPECIFIED.
3		All Around This Side of Parting Line [ASME Y14.5-2009 Section 3.14.1]	To apply a requirement to all features all around one side of a parting line, the graphical symbol for all around this side of parting line is indicated on the leader line.
4		All Over This Side of Parting Line [ASME Y14.5-2009 Section 3.14.2]	To apply a requirement to all features all over one side of a parting line, the graphical symbol for all over this side of parting line is indicated on the leader line.
5		Angularity	Is the condition of a surface, axis, or center plane, which is at a specified angle from a datum plane or axis.

6		Arc Length	Indicating that a dimension is an arc length measured on a curved outline. The symbol is placed above the dimension.
7		Basic Dimension	Used to describe the exact size, profile, orientation or location of a feature. A basic dimension is always associated with a feature control frame or datum target. (Theoretically exact dimension in ISO)
8		Between	To indicate that a profile tolerance applies to several contiguous features, le1ers may designate where the profile tolerance begins and ends. These le1ers are referenced using the between symbol (since 1994) or the word between on drawings made to earlier versions of the Standard.
9		Concentricity	Describes a condition in which two or more features, in any combination, have a common axis.
10		Conical Taper	Is used to indicate taper for conical tapers. This symbol is always shown with the vertical leg to the le5.

11		Continuous Feature [ASME Y14.5-2009 Section 2.7.5]	The note CONTINUOUS FEATURE or the continuous feature symbol is used to identify a group of two or more features of size where there is a requirement that they be treated geometrically as a single feature of size. Although the definition only mentions features of size, there is an example of CF being applied to a pair of planar features.
12		Controlled Radius	Creates a tolerance zone defined by two arcs (the minimum and maximum radii) that are tangent to the adjacent surfaces. Where a controlled radius is 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
13		Counterbore/ Spotface	Is used to indicate a counterbore or a Spotface. The symbol precedes the dimension of the counterbore or Spotface, with no space.
14		Countersink	Is used to indicate a countersink. The symbol precedes the dimensions of the countersink with no space.

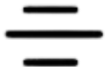
15		Cylindricity	Describes a condition of a surface of revolution in which all points of a surface are equidistant from a common axis.
16		Datum Feature	Is the actual component feature used to establish a datum
17		Datum Target	Is a specified point, line, or area on a part that is used to establish the Datum Reference Plane for manufacturing and inspection operations.
18		Depth/Deep	Is used to indicate that a dimension applies to the depth of a feature. This symbol precedes the depth value with no space in between.
19		Diameter	indicates a circular feature when used on the field of a drawing or indicates that the tolerance is diametrical when used in a feature control frame
20		Dimension Origin	Signifies that the dimension originates from the plane established by the shorter surface and dimensional limits apply to the other surface
21		Feature Control Frame	Is a rectangular box containing the geometric characteristics symbol, and the form, runout or location tolerance. If necessary, datum references and modifiers applicable to the feature or the datums are also contained in the box.

22		Flatness	Is the condition of a surface having all elements in one plane.
23		Free State Variations	Is a term used to describe distortion of a part a5er removal of forces applied during manufacture.
24		Least Material Condition (LMC)	Implies that condition of a part feature of size wherein it contains the least (minimum) amount of material, examples, largest hole size and smallest sha5 size. It is opposite to maximum material condition.
25		Independency Symbol [ASME Y14.5-2009 Section 2.7.3]	The Independency symbol is applied to the size dimension in order to invoke the principle of independency to regular features of size and override Rule #1.
26		Maximum Material Condition (MMC	Is that condition of a part feature wherein it contains the maximum amount of material within the stated limits of size. That is: minimum hole size and maximum sha5 size
27		Movable Datum Targets [ASME Y14.5-2009 Section 4.24.6]	The movable datum target symbol may be used to indicate movement of the datum target datum feature simulator.
28		Number of Places	The X is used along with a value to indicate the number of times a dimension or feature is repeated on the drawing.

29		Parallelism	Is the condition of a surface, line, or axis, which is equidistant at all points from a datum plane or axis.
30		Parting Lines [ASME Y14.5-2009 Section 3.14]	Are depicted on casting/forging/molded part drawings as a phantom line extending beyond the part in applicable views, with the parting line symbol added.
31		Perpendicularity	Is the condition of a surface, axis, or line, which is 90 deg. From a datum plane or a datum axis.
32		Position Tolerance	Defines a zone within which the axis or center plane of a feature is permi1ed to vary from true (theoretically exact) position.
33		Profile of a Line	Is the condition permiting a uniform amount of profile variation, ether unilaterally or bilaterally, along a line element of a feature.
34		Profile of a Surface	Is the condition permiting a uniform amount of profile variation, ether unilaterally or bilaterally, on a surface.

35		Projected Tolerance Zone	Applies to a hole in which a pin, stud, screw, etc., is to be inserted. It controls the perpendicularity of the hole to the extent of the projection from the hole and as it relates to the mating part clearance. The projected tolerance zone extends above the surface of the part to the functional length of the pin, stud, and screw relative to its assembly
36		Radius	Creates a zone defined by two arcs (the minimum and maximum radii). The part surface must lie within this zone.
37		Reference Dimension	A dimension usually without tolerance, used for information purposes only. It does not govern production or inspection operations. (Auxiliary dimension in ISO)
38		Roundness	Describes the condition on a surface of revolution (cylinder, cone, sphere) where all points of the surface intersected by any plane.
39		Runout	Is the composite deviation from the desired form of a part surface of revolution through on full rotation (360 deg) of the part on a datum axis.

40		Slope	Is used to indicate slope for flat tapers. This symbol is always shown with the vertical leg to the le5.
41		Spherical Diameter	Shall precede the tolerance value where the specified tolerance value represents spherical zone. Also, a positional tolerance may be used to control the location of a spherical feature relative to other features of a part. The symbol for spherical diameter precedes the size dimension of the feature and the positional tolerance value, to indicate a spherical tolerance zone.
42		Spherical Radius	Precedes the value of a dimension or tolerance.
43		Spotface [ASME Y14.5-2009 Section 1.8.14]	Counterbore and Spotface previously used the same symbol. A Spotface now looks like the counterbore symbol with the addition of the le1ers SF.
44		Square	Is used to indicate that a single dimension applies to a square shape. The symbol precedes the dimension with no space between.

45		Statistical Tolerance	Is the assigning of tolerances to related components of an assembly on the basis of sound statistics (such as the assembly tolerance is equal to the square root of the sum of the squares of the individual tolerances). By applying statistical tolerancing, tolerances of individual components may be increased or clearances between mating parts may be reduced. The increased tolerance or improved fit may reduce manufacturing cost or improve the product's performance, but shall only be employed where the appropriate statistical process control will be used.
46		Straightness	a condition where an element of a surface or an axis is a straight line.
47		Symmetry	is a condition in which a feature (or features) is symmetrically disposed about the center plane of a datum feature.
48		Tangent Plane	indicating a tangent plane is shown. The symbol is placed in the feature control frame following the stated tolerance.
49		Target Point	indicates where the datum target point is dimensionally located on the direct view of the surface.

50		Total Runout	Is the simultaneous composite control of all elements of a surface at all circular and profile measuring positions as the part is rotated through 360.
51		Datum Translation Symbol [ASME Y14.5-2009 Section 3.3.26]	This symbol indicates that a datum feature simulator is not fixed at its basic location and shall be free to translate.
52		Unilateral and Unequally Disposed Profile Tolerance [ASME Y14.5-2009 Section 8.3.1.2]	To indicate that a profile of a surface tolerance is not symmetrical about the true profile, this symbol is used. The first value in the feature control frame is the total width of the profile tolerance. The value following the symbol is the amount of the tolerance that is in the direction that would allow additional material to be added to the true profile.