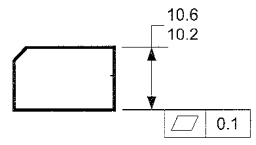
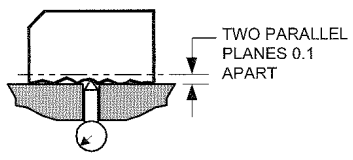
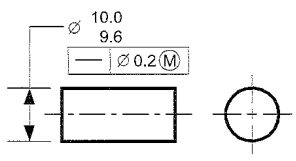
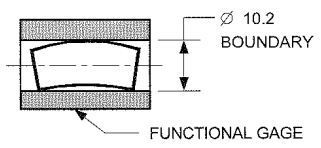
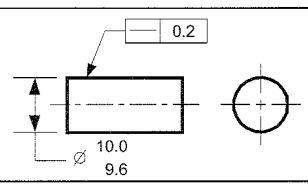
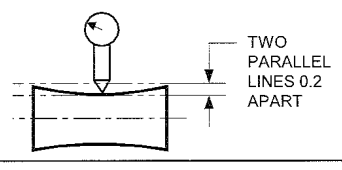
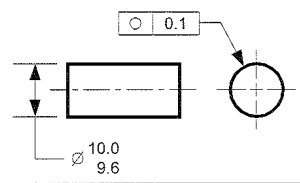
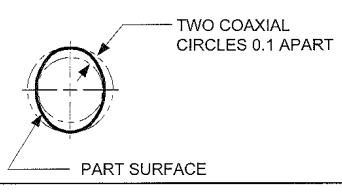
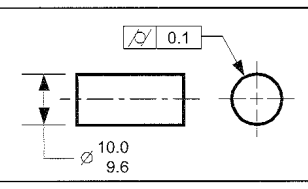
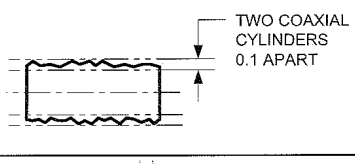
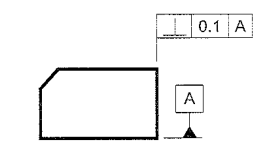
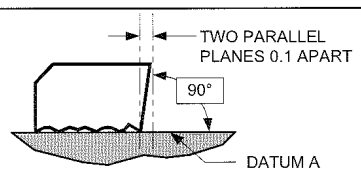
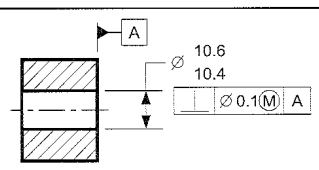
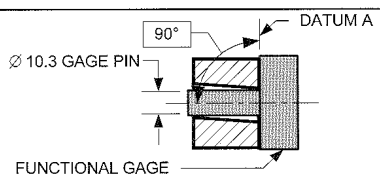
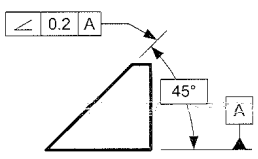
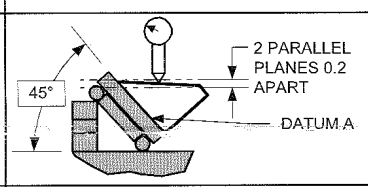
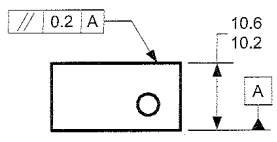
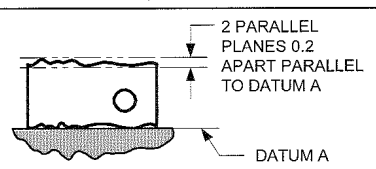
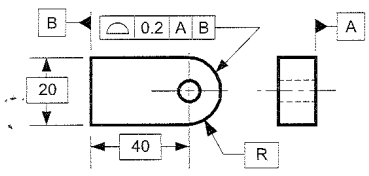
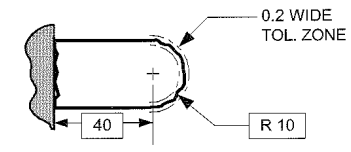
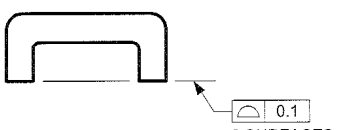
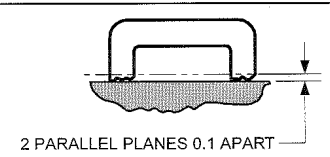
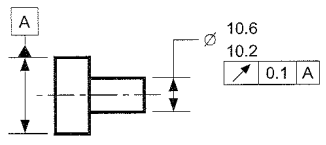
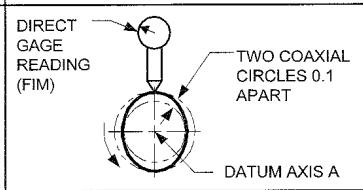
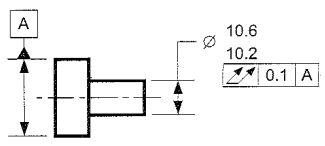
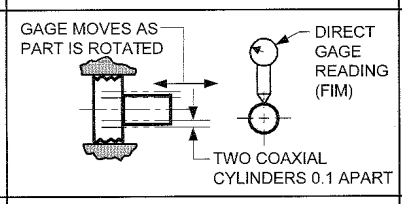
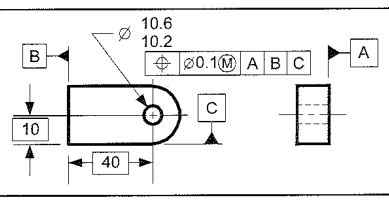
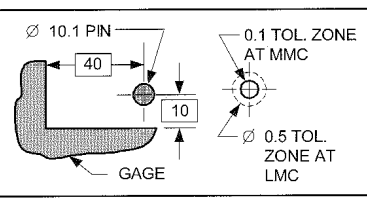
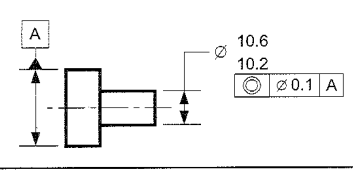
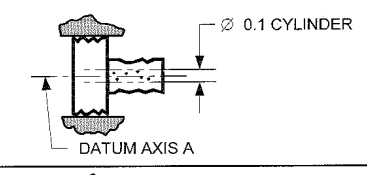
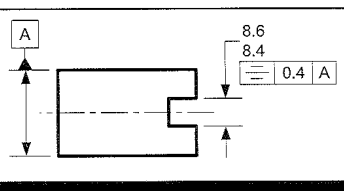
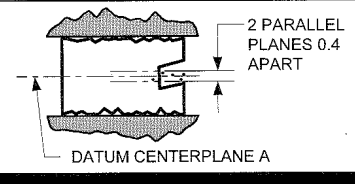
GEOMETRIC TOLERANCING REFERENCE CHART

01/02

PER ASME Y14.5M-1994

GEOMETRIC TOLERANCE	EXAMPLE AS SHOWN ON A DRAWING	MEANING/TOLERANCE ZONE SHAPE	DATUMS USED	MMC OR LMC MODIFIER	BONUS TOL.	OVERALL BOUNDARY	COMMENTS
FLATNESS			No	No Always RFS	No	Not Affected	• Part must also be within size limits • Tolerance value must be less than the size tolerance • Rule #1 applies
STRAIGHTNESS (OF AN AXIS)			No	Ⓜ Shown in example otherwise RFS implied (per Rule #2)	Yes when Ⓜ or Ⓛ is used.	Affected 10.0 + 0.2 = 10.2	• Part must also be within size limits • Tolerance value may be greater than the size limit • Rule #1 overridden
STRAIGHTNESS (OF A SURFACE ELEMENT)			No	No Always RFS	No	Not Affected	• Tolerance value must be less than the size tolerance • Tolerance zone applies to each line element • Rule #1 applies
CIRCULARITY			No	No Always RFS	No	Not Affected	• Tolerance value must be less than the size tolerance • Part must also be within the size limits • Rule #1 applies
CYLINDRICITY			No	No Always RFS	No	Not Affected	• Tolerance value must be less than the size tolerance • Part must also be within the size limits • Rule #1 applies
PERPENDICULARITY (SURFACE TO SURFACE)			Yes	No Always RFS	No	Does not apply	• Also controls flatness of the surface
PERPENDICULARITY (DIAMETER TO SURFACE)			Yes	Ⓜ Shown in example otherwise RFS implied (per Rule #2)	Yes when Ⓜ or Ⓛ is used.	Affected 10.4 - 0.1 = 10.3	• Hole must be within the size limits • Rule #1 applies
ANGULARITY (SURFACE TO SURFACE)			Yes	No Always RFS	No	Not Affected	• Part must also be within size limits • A basic dimension must be used from the tolerance feature to the datum referenced • Also controls the flatness of the surface
PARALLELISM (SURFACE TO SURFACE)			Yes	No Always RFS	No	Not Affected	• Part must also be within size limits • Parallelism tolerance value must be less than the size tolerance • Also controls the flatness of the surface • Rule #1 applies
PROFILE OF A SURFACE			Yes	No Always RFS	No	Does not apply	• Tolerance zone may be bilateral or unilateral • Frequently used in place of coordinate tolerancing • Basic dimensioning must be used to define the true profile
PROFILE OF A SURFACE (COPLANAR)			No	No Always RFS	No	Does not apply	• Controls flatness of the surfaces • Unilateral tolerance zone
CIRCULAR RUNOUT			Yes	No Always RFS	No	Affected 10.6 + 0.1 = 10.7	• Applies to each circular element independently • A composite control that controls circularity and axis offset • The maximum axis offset is 1/2 of the tolerance value • Rule #1 applies
TOTAL RUNOUT			Yes	No Always RFS	No	Affected 10.6 + 0.1 = 10.7	• Applies to entire surface simultaneously • A composite control that controls straightness, taper, and axis offset • The maximum axis offset is 1/2 of the tolerance value • Rule #1 applies
POSITION			Yes	Ⓜ Shown in example otherwise RFS implied (per Rule #2)	Yes when Ⓜ or Ⓛ is used.	Affected 10.2 - 0.1 = 10.1	• Allows round tolerance zones • Ⓜ is used when function of part feature is assembly • Ⓛ is used to control a min. part distance • Basic dimension must be used to establish the true position • Rule #1 applies
CONCENTRICITY			Yes	No Always RFS	No	Affected 10.6 + 0.1 = 10.7	• It is not a direct gage reading • Should only be used in high speed rotating part applications • Midpoints of toleranced dia. must be within toleranced zone • Rule #1 applies
SYMMETRY			Yes	No Always RFS	No	Affected 8.4 - 0.4 = 8.0	• Slot must also be within size limits • Limits the location of the midpoints of two-point measurements • Rule #1 applies

www.etinews.com • (800) 886-0909

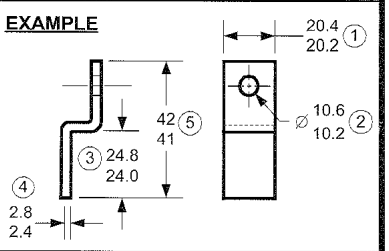
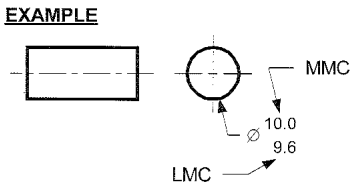
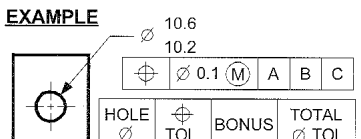
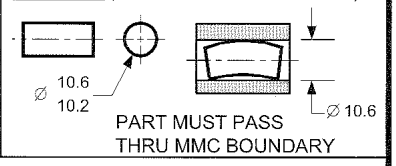
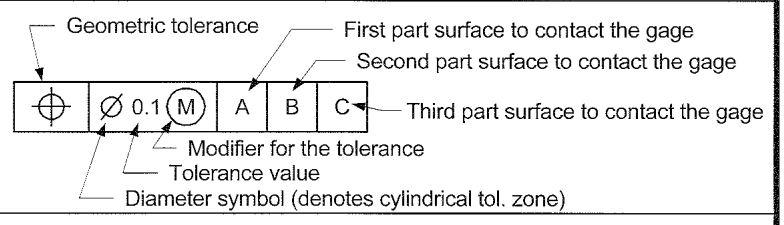
ISBN 0-924520-12-4

GEOMETRIC TOLERANCING REFERENCE CHART

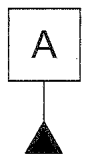
01/02

PER ASME Y14.5M-1994

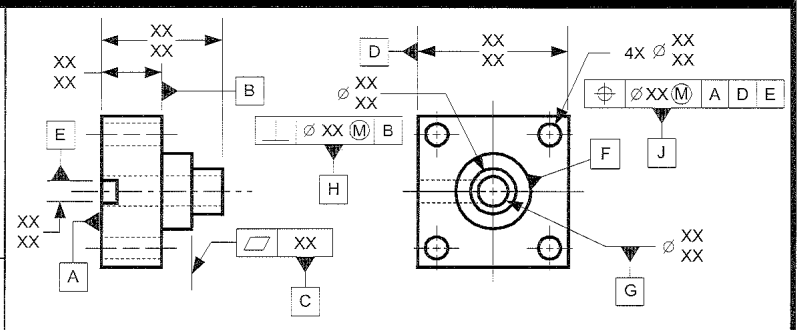
DEFINITIONS

ITEM	MEANING	
FEATURE OF SIZE	One cylindrical or spherical surface, set of two opposed elements, or opposed parallel surfaces, associated with a size dimension. In the example, dimension 1, 2 & 4 are features of size.	**EXAMPLE** 
MMC/LMC	**MAXIMUM MATERIAL CONDITION (MMC)** The condition of a feature of size where the part contains the maximum amount of material. **LEAST MATERIAL CONDITION (LMC)** The condition of a feature of size where the part contains the least amount of material.	**EXAMPLE** 
BONUS TOLERANCE	When an MMC Modifier is shown, the stated tolerance applies only when the feature being controlled is at its MMC size. As the actual size departs from MMC, you may add the amount of departure to the original tolerance. When the MMC modifier is used, the part may be checked with a functional (fixed) gage.	**EXAMPLE**  HOLE Ø / TOL. / BONUS / TOTAL Ø TOL. 10.2 / 0.1 / 0 / 0.1 10.4 / 0.1 / 0.2 / 0.3 10.6 / 0.1 / 0.4 / 0.5
RULE #1	**INDIVIDUAL FEATURE OF SIZE RULE** For each feature of size where only a tol. of size is specified, the feature of size cannot extend beyond a boundary of perfect form at MMC. The LMC of the dimension must also be verified, usually with a two point check.	**EXAMPLE** (EXTERNAL FEATURE OF SIZE) 
RULE #2	**ALL APPLICABLE GEOMETRIC TOLERANCES RULE** Regardless of feature size applies, with respect to the individual tolerance, datum reference, or both, where no modifier is specified.	
FEATURE CONTROL FRAME		
VIRTUAL CONDITION	The worst-case boundary generated by the collective effects of a feature of size specified at MMC or LMC and the geometric tolerances for that material condition.	

DATUM IDENTIFICATION

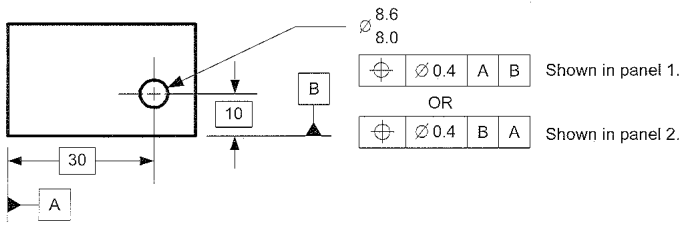


Datum identification symbol



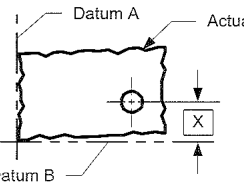
DATUM LETTER	DATUM TYPE	DENOTED BY THE DATUM SYMBOL TOUCHING . . .
A	PLANAR	The edge view of a surface
B		An extension line (must be offset from the dimension lines)
C		A symbol applied to the surface
D	CENTER PLANE	An extension line in line with the dimension
E		An extension line replacing one side of the dimension line and arrowhead
F	AXIS	The surface of a diameter
G		The beginning of a leader line of a diameter dimension
H		A symbol applied to a diameter
J		A symbol applied to a pattern of holes

IMPORTANCE OF DATUM SEQUENCE

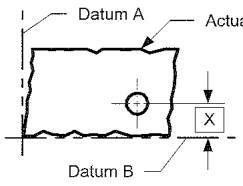


The sequence of the datums referenced affects the measurement obtained.

① A = Primary datum, B = Secondary datum



② B = Primary datum, A = Secondary datum



MODIFIERS AND SYMBOLS

ITEM	MEANING
Ⓜ	**MAXIMUM MATERIAL CONDITION MODIFIER** Means the tolerance applies when the part feature is at maximum material condition.
Ⓛ	**LEAST MATERIAL CONDITION MODIFIER** Means the tolerance applies when the part feature is at least material condition.
No symbol	**REGARDLESS OF FEATURE SIZE (RFS)** RFS means a tolerance applies at whatever size a feature is produced. The symbol for regardless of feature size has been dropped. See Rule #2.
Ⓣ	**TANGENT PLANE MODIFIER** Means that only the tangent plane of the toleranced surface must be within the tolerance zone.
Ⓕ	**FREE STATE MODIFIER** Means that the geometric tolerance applies when the part is in the free state condition.
Ⓟ	**PROJECTED TOLERANCE ZONE MODIFIER** Means the tolerance zone is projected above the part surface.
Ø	**DIAMETER SYMBOL** Replaces the word "diameter" on a drawing. Is used inside a geometric tolerance to denote a cylindrical tolerance zone.
SØ	**SPHERICAL DIAMETER** Replaces the words "spherical diameter" on a drawing.
R	**RADIUS SYMBOL** Means flats and reversals are allowed on the surface of the radius.
SR	**SPHERICAL RADIUS SYMBOL** Replaces the words "spherical radius" on a drawing.
CR	**CONTROLLED RADIUS SYMBOL** Means flats or reversals are not allowed on the surface of the radius.
⌵	**COUNTERSINK SYMBOL** Replaces the word "countersink" on a drawing.
⌴	**COUNTERBORE SYMBOL** Replaces the word "counterbore" on a drawing.
↧	**DEPTH SYMBOL** Replaces the word "depth" on a drawing.
□	**SQUARE SYMBOL** Indicates that a single dimension applies to a square shape.
X	**TIMES SYMBOL** Replaces the words "no. of places" on the drawing.
10	**ARC LENGTH SYMBOL** Indicates that a linear dimension is an arc length measured on a curved surface.
X.X	**BASIC DIMENSION SYMBOL** A rectangle around a dimension indicates it is a theoretical exact dimension. Basic dimensions simply locate tolerance zones and must be supplemented with a geometric tolerance.
	DIMENSION ORIGIN SYMBOL A circle replacing one of the arrowheads of a dimension indicates that a toleranced dimension originates from the end with the circle.
	ALL AROUND SYMBOL The circle at the bend of the leader line indicates the tolerance zone applies all around the outline of the part in the view shown.
A ⟷ B	**BETWEEN SYMBOL** The double-ended arrow indicates that the tolerance zone applies between the points (A&B) indicated.
(0.1)	**REFERENCE DIMENSION SYMBOL** The information inside the parentheses is for reference only. Not to be used for manufacturing or inspection.
ST	**STATISTICAL TOLERANCING SYMBOL** Indicates that a tolerance is based on statistical methods. Features identified with this symbol should be produced with statistical process controls.
A1	**DATUM TARGET SYMBOL** Used to establish datums from partial part surfaces. Datum targets describe the type of gage feature (point, line, or area) used to establish a datum plane.
	DATUM TARGET POINT SYMBOL Denotes a spherical-tipped gage pin should be used at the exact location shown to simulate a point on the datum plane.
	DATUM TARGET LINE SYMBOL Denotes the side of a gage pin should be used at the exact location shown to simulate a portion of the datum plane.
	DATUM TARGET AREA SYMBOL Denotes a flat-tipped gage pin, the exact shape of the area shown, should be used to simulate a portion of the datum plane.

METRIC CONVENTIONS

12	When a metric dimension is a whole number, the decimal point and zero are omitted.
0.2	When a metric dimension is less than one millimeter, a zero proceeds the decimal point.
12.2	When a metric dimension is not a whole number, a decimal point with the portion of a millimeter is specified.

www.etinews.com • (800) 886-0909

ISBN 0-924520-12-4