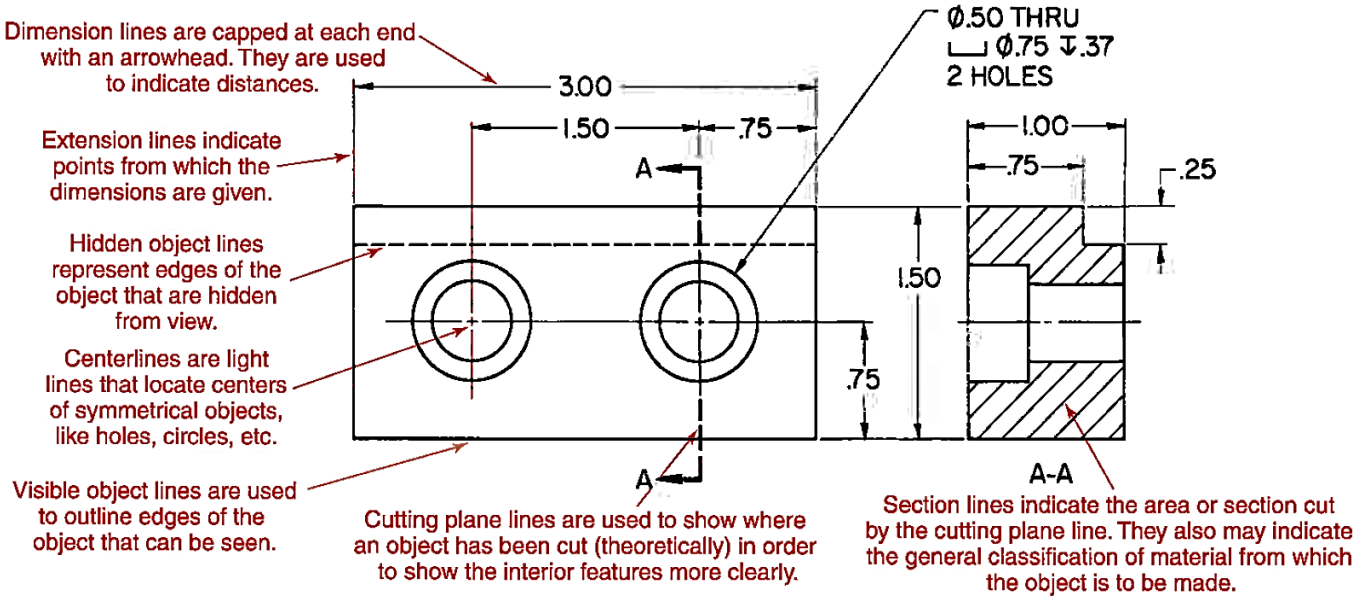
MEE2006: Introduction to Mechanical Design

Instructor: Prof. Keun Ryu (kryu@hanyang.ac.kr; http://turbolab.hanyang.ac.kr)

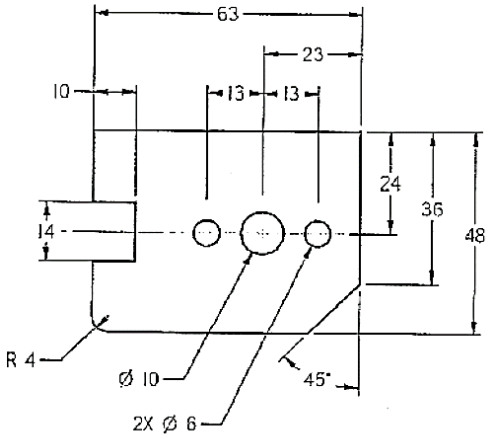
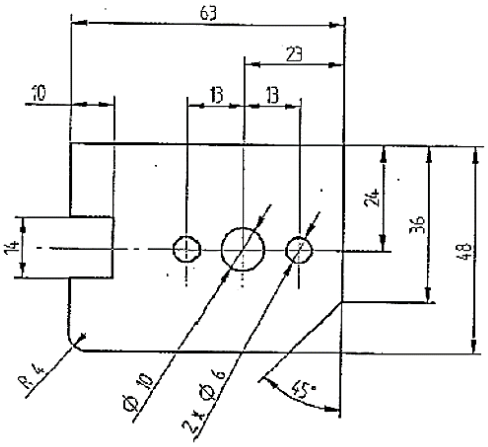
Note 6: Understanding drawings: Basics

Information in engineering drawing: Example



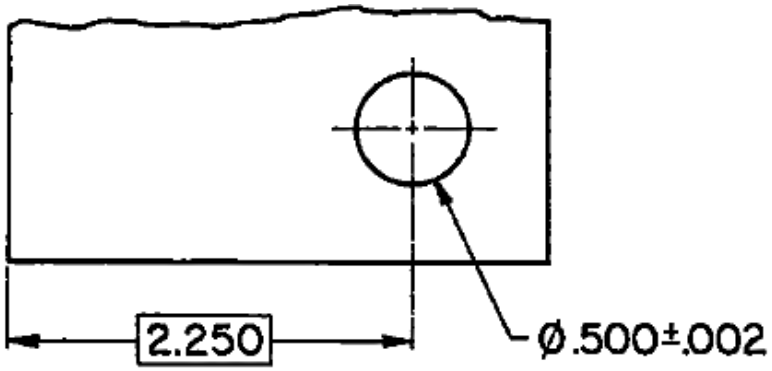
It is important to note that each standard (ASME-American Society of Mechanical Engineers-, ISO-International Organization for Standardization-, ANSI- American National Standards Institute-, etc) change symbols periodically. You must be familiar with both old and new (updated) symbols because either may be used on the drawings. The below is one of examples.

Dimension Value Positioning

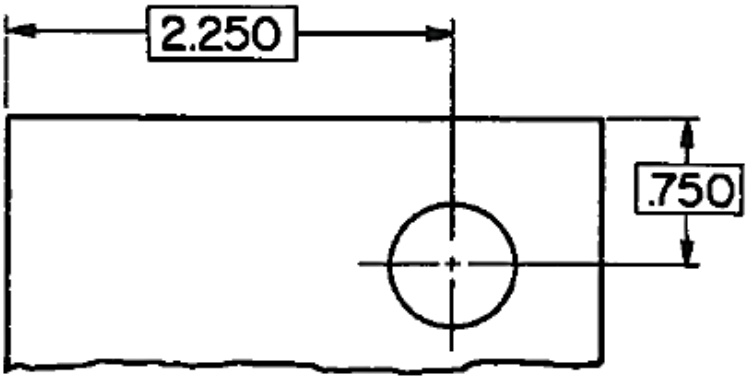
ASME Y14.5M-1994	ISO 129-1:2004
	
All dimensional values are parallel to bottom of drawing except base line dimensioning.	All dimensional values are parallel to and above the dimension line.

Decimal Separator Symbol

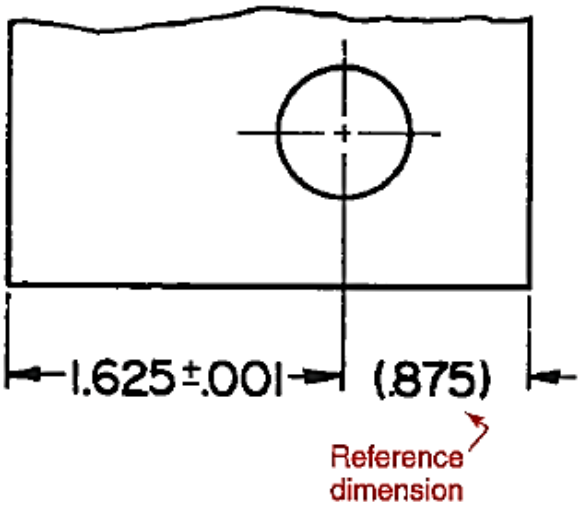
ASME Y14.5M-1994	ISO 31:2000 Amendment 2
2.4	2,4 OR 2.4
A decimal point (.) shall separate the whole number from the decimal fraction.	A comma (,) shall separate the whole number from the decimal fraction; a decimal point (.) may be used on English language drawings and documents.



Basic dimensions

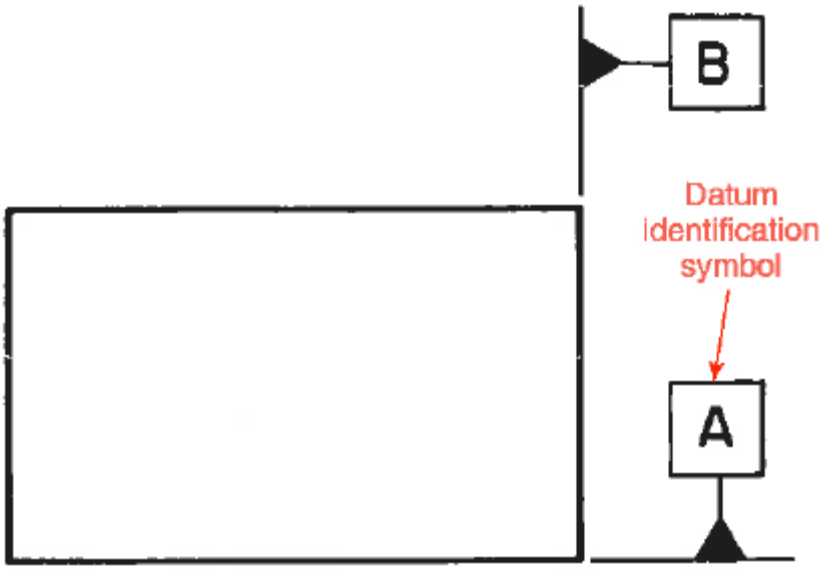


True position



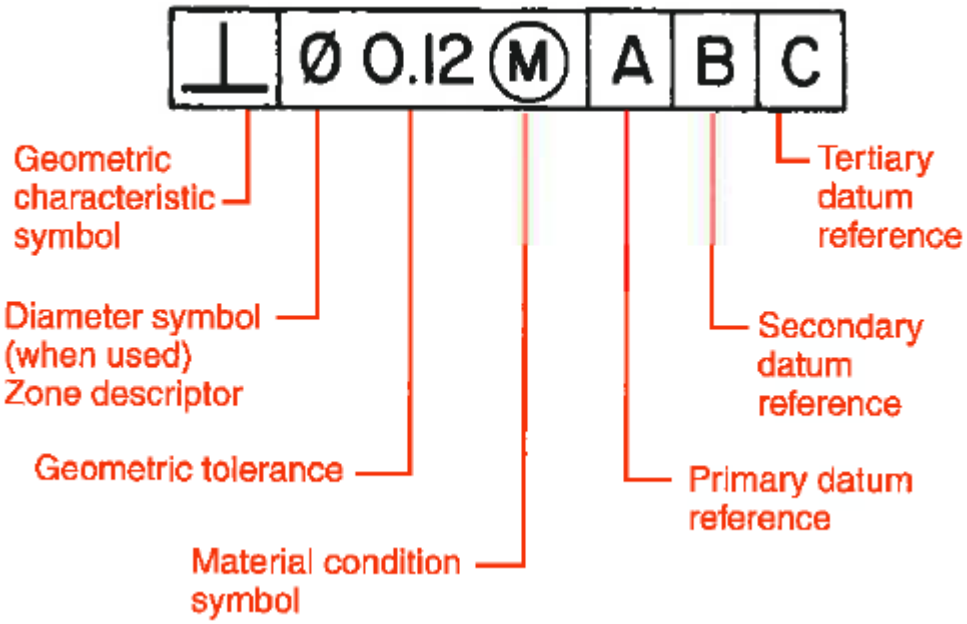
Basic dimensions are usually indicated by being enclosed in a rectangular frame. They are not toleranced. True position is the theoretical exact location of feature. It is established by basic dimensions. Reference dimensions are not used for production or inspection purposes. On a drawing, they are shown enclosed in parentheses.

Datum identification symbol



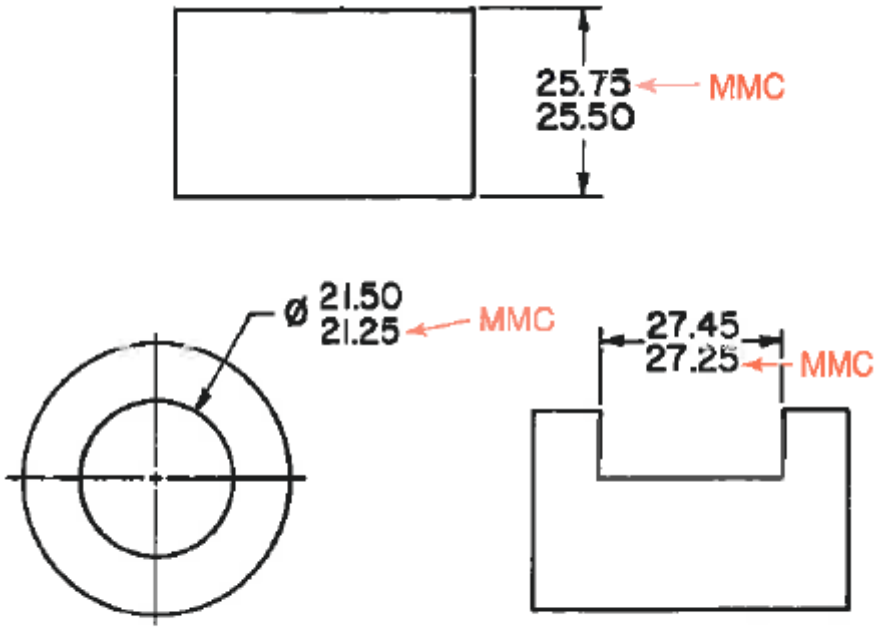
Datums are exact points, axes, or planes from which features of a part are located.

Feature control frame



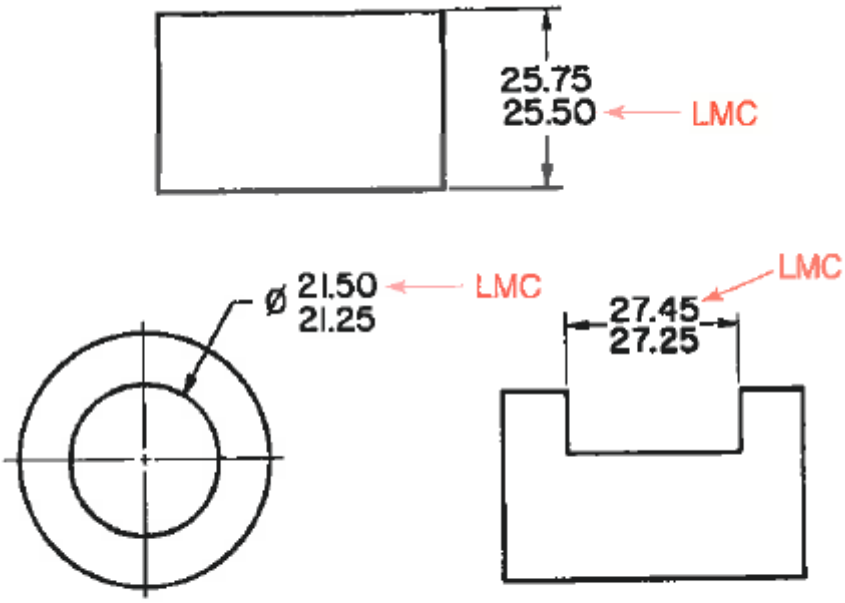
MMC & LMC

Maximum material condition (MMC) refers to a feature-of-size that contains the greatest amount of material, yet remains within its tolerance zone. Some examples of MMC include: Largest pin diameter & Smallest hole size.

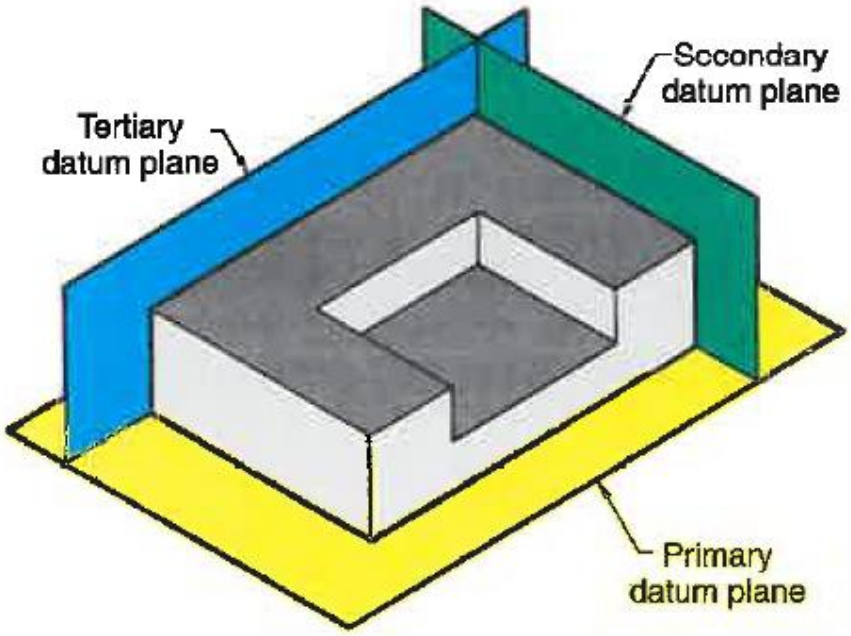


Maximum material condition (MMC) indicates that the size of a feature contains the maximum amount of material within the stated tolerance limits.

In contrast, **least material condition (LMC)** refers to a feature of size containing the least amount of material, yet remains within its tolerance zone: Smallest pin diameter & Largest hole size.

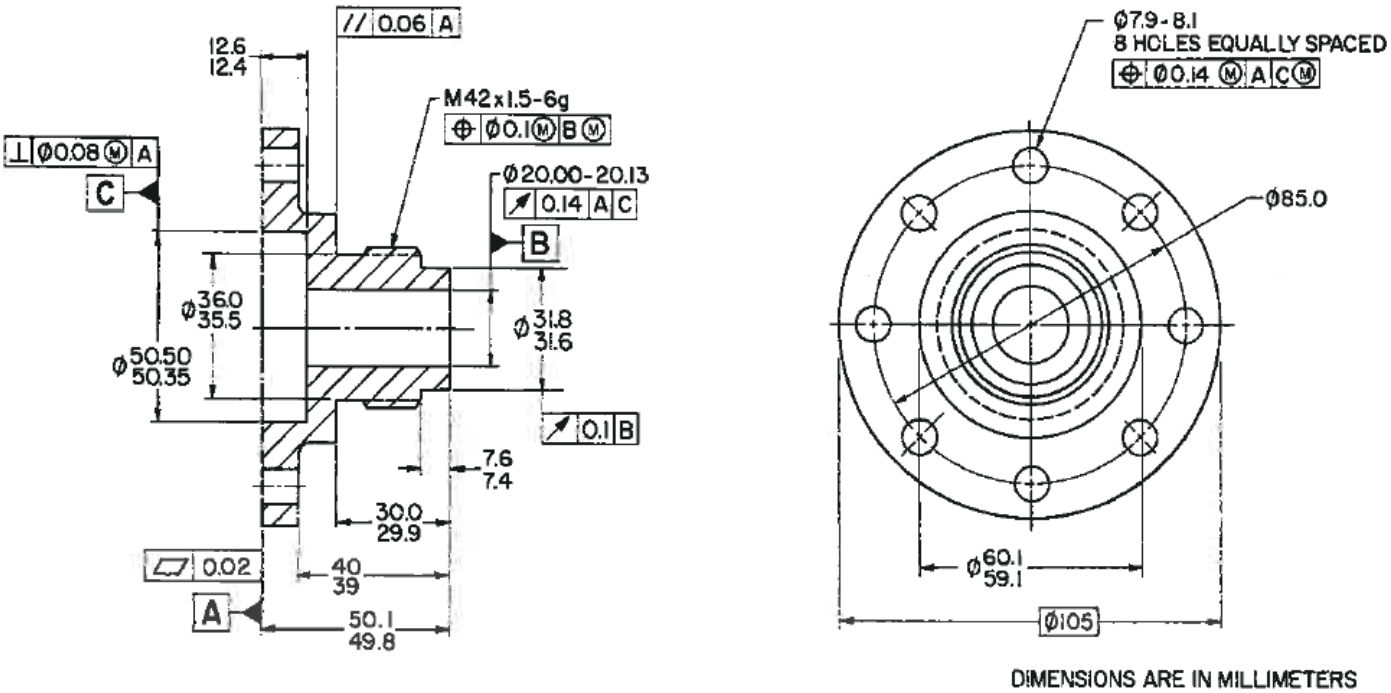


Least material condition (LMC) indicates that the size of a feature contains the least amount of material within the stated limits of size.

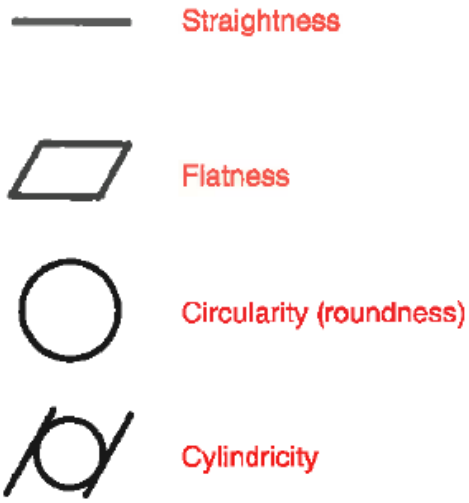


Datum references are perpendicular planes. The first datum referenced is the primary datum, followed by the secondary and tertiary datums.

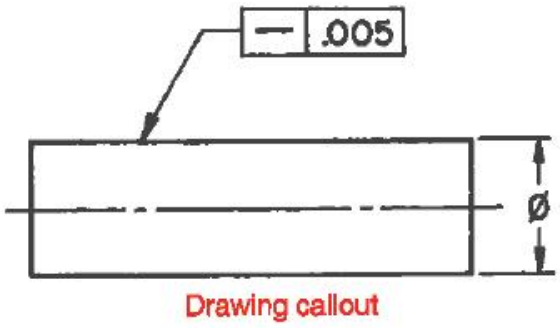
A feature control frame is employed when a location or form tolerance is related to a datum.



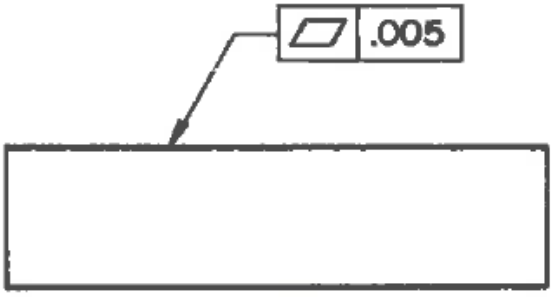
Form geometric tolerances



Form geometric symbols.



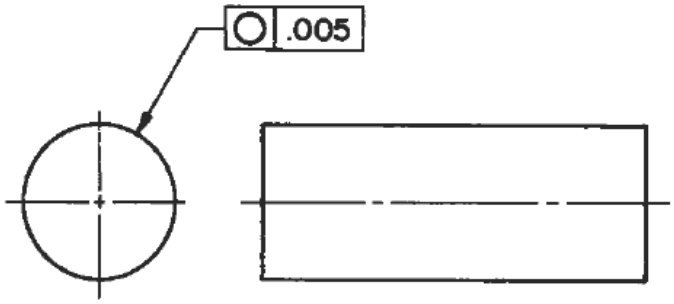
A straightness geometric form tolerance establishes a tolerance zone of uniform width along a straight line. All elements of the surface must lie within this zone.



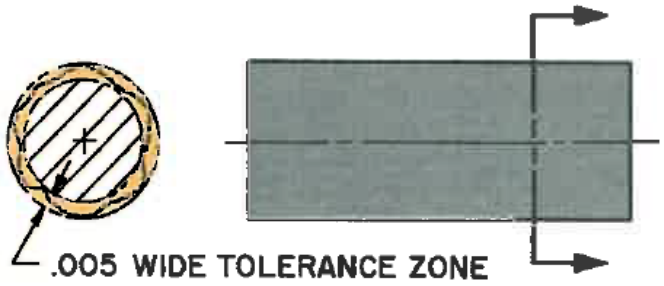
Drawing callout

Interpretation

The flatness geometric form tolerance specifies the two parallel planes within which a surface must lie.

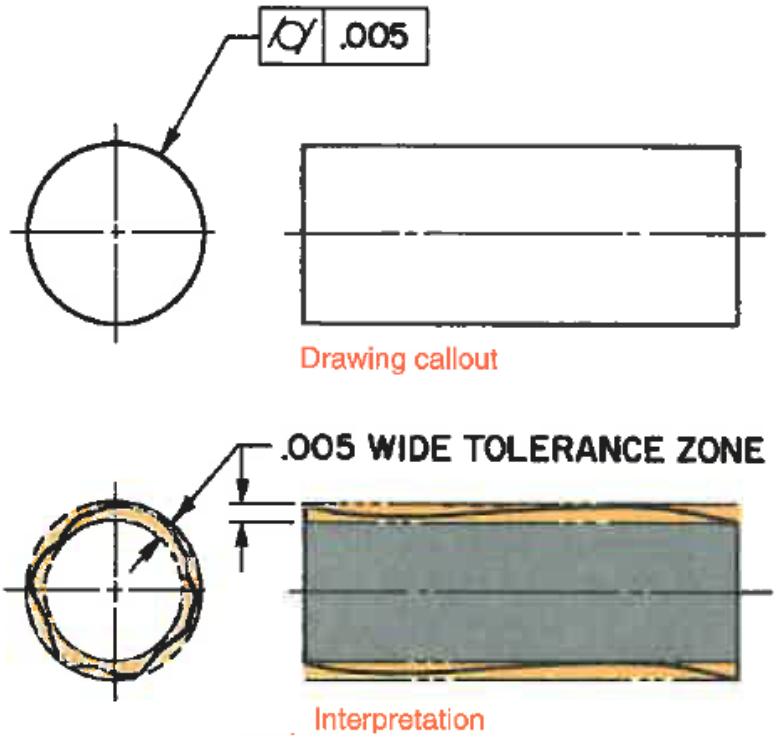


Drawing callout



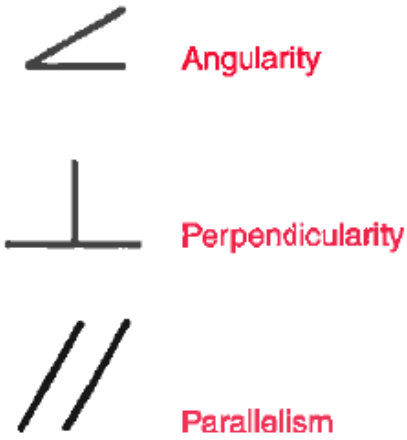
Interpretation

A circularity geometric tolerance specifies a tolerance zone bounded by two concentric circles on a plane perpendicular to the axis of a cylinder or cone, within which each circular element must lie.

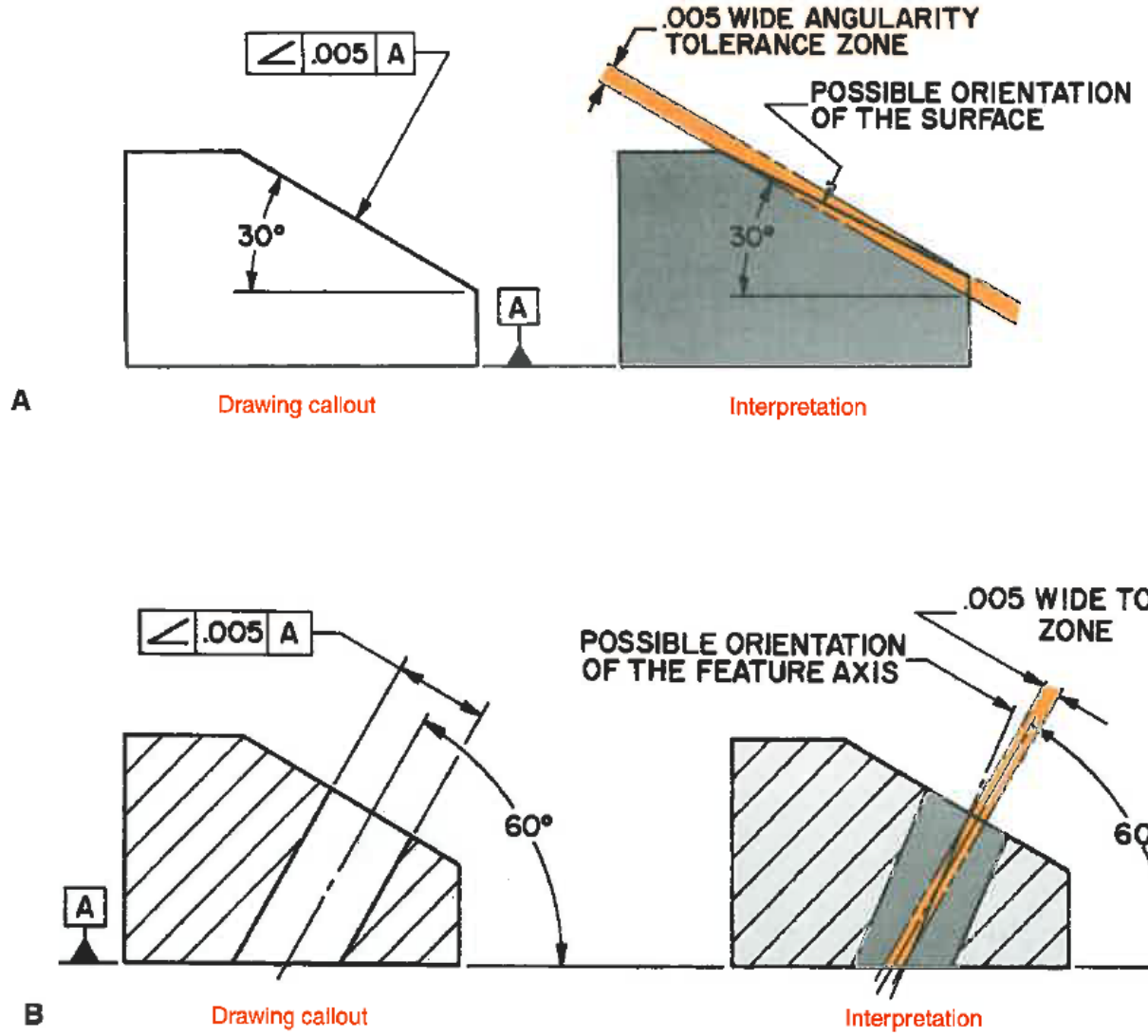


The cylindricity geometric tolerance establishes a tolerance zone that controls the diameter of a cylinder throughout its entire length.

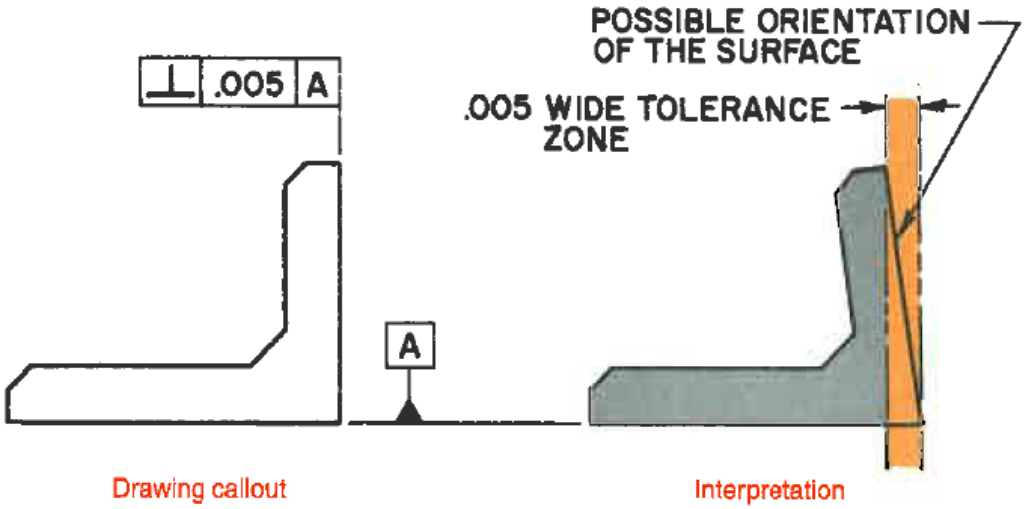
Orientation geometric tolerances



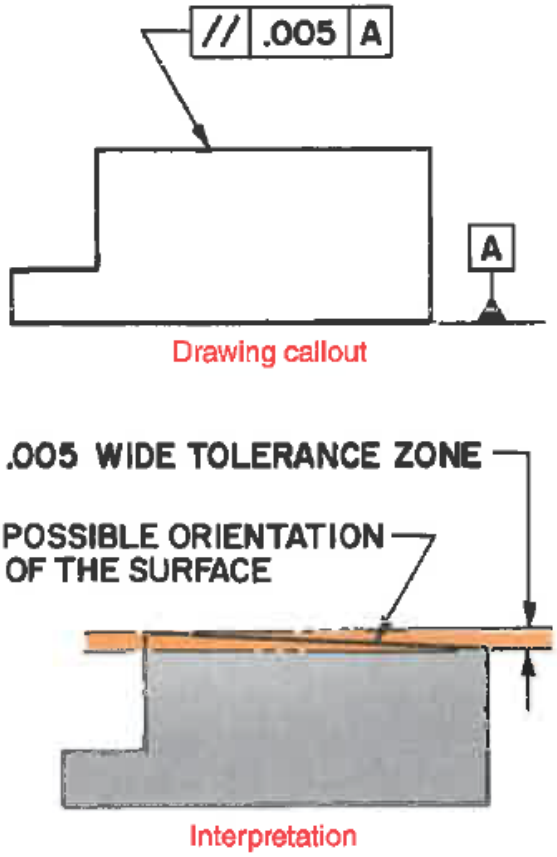
Orientation geometric tolerance symbols.



An angularity geometric tolerance establishes a tolerance zone defined by two parallel lines, planes, or a cylindrical zone at a specified basic angle other than 90°. A—Angularity of a surface. B—Angularity of an axis.

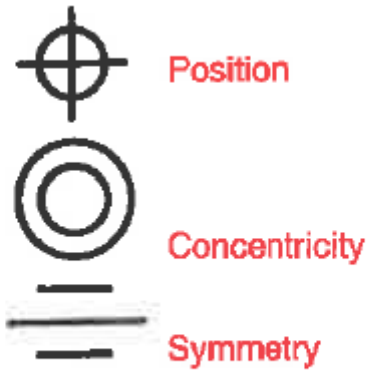


The line, surface, or axis of a considered feature must lie within the perpendicularity geometric tolerance zone.

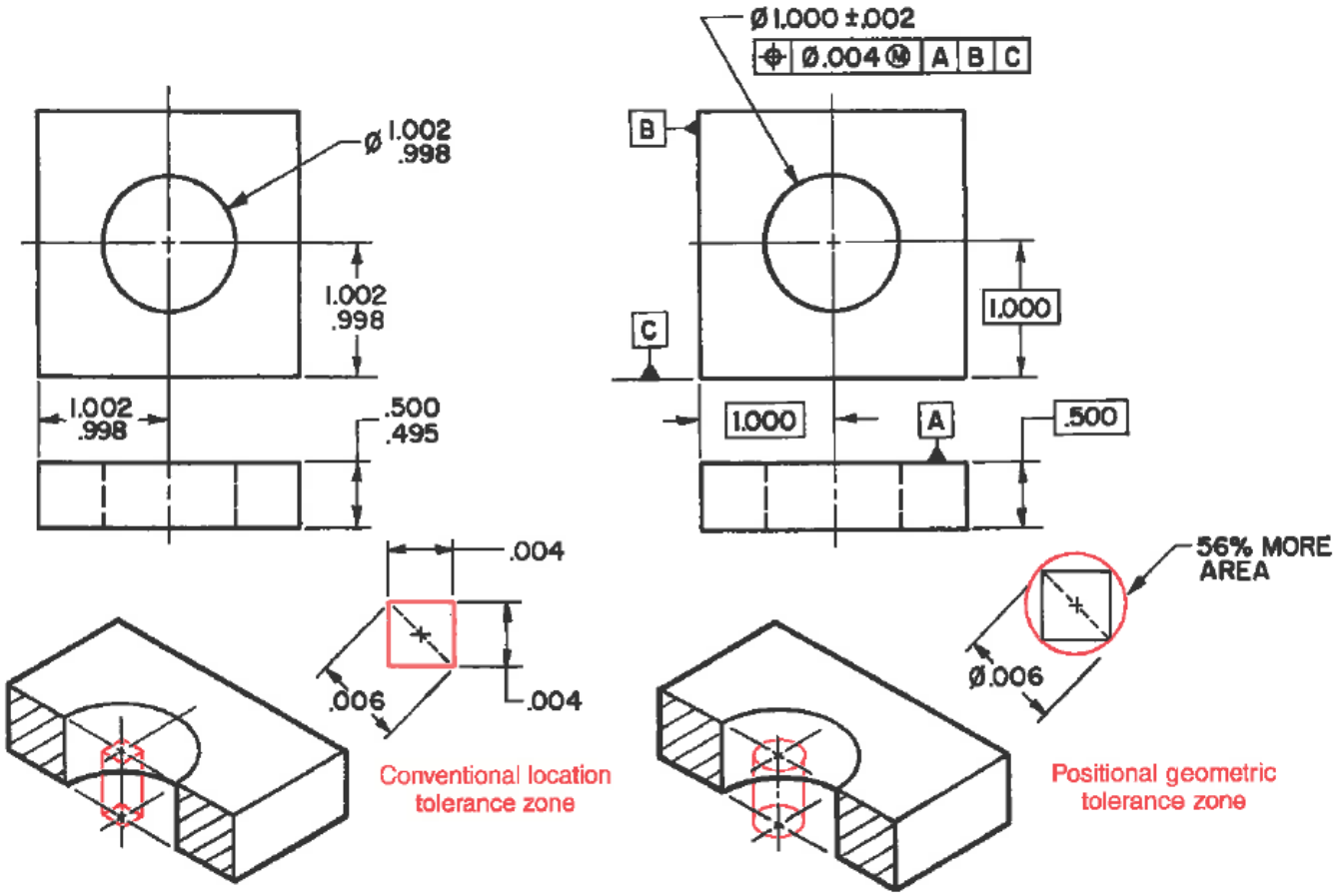


A parallelism geometric tolerance is a tolerance zone defined by two lines parallel to a datum within which the elements of a surface or axis must lie.

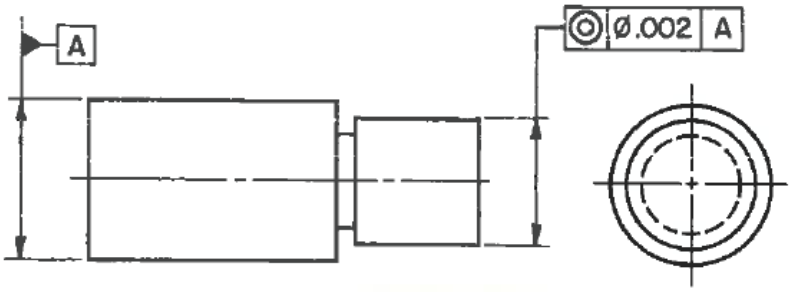
Location of geometric tolerances



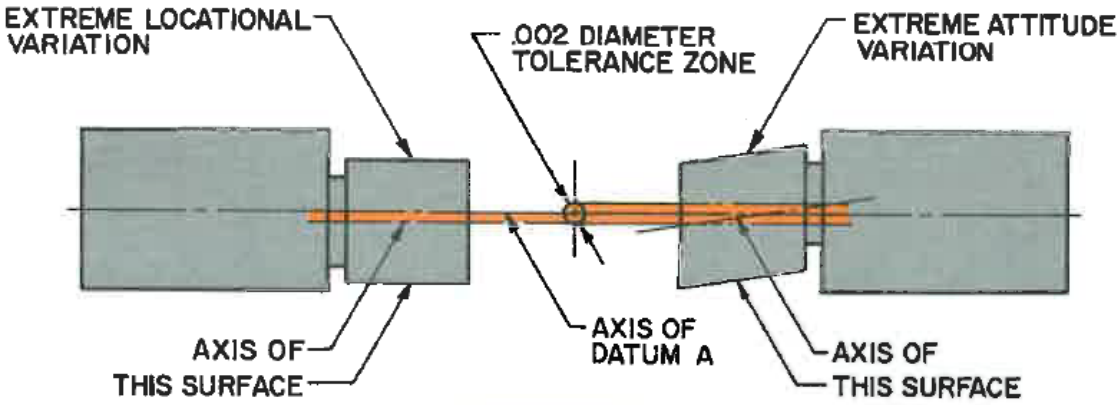
Location or positional tolerance symbols.



A positional geometric tolerance establishes how far a feature may vary from its true position.

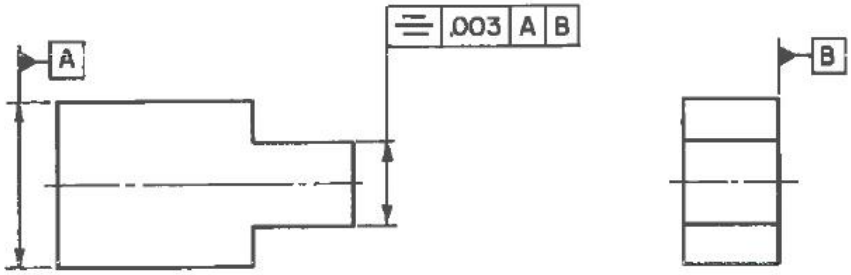


Drawing callout

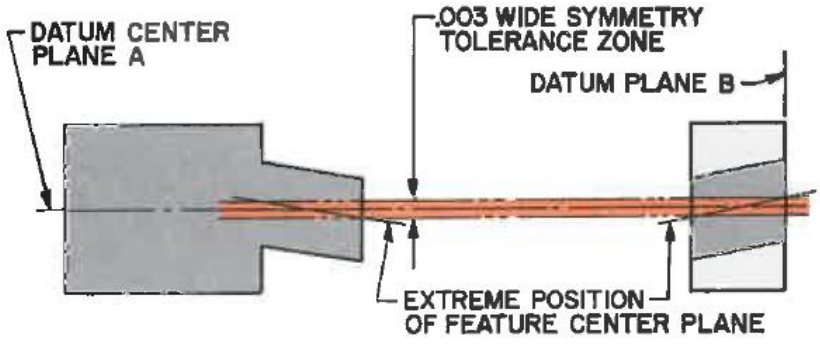


Interpretation

A concentricity geometric tolerance is expressed as a cylindrical tolerance zone. The axis or center point of this zone coincides with a datum axis.



Drawing callout



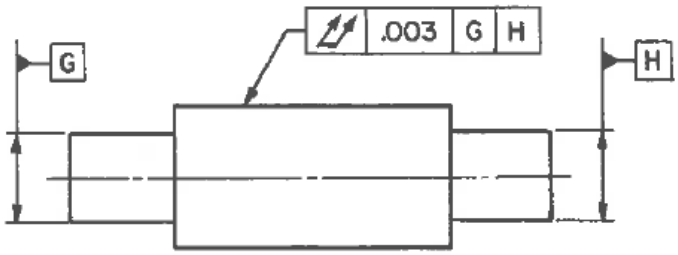
Interpretation

A symmetry geometric tolerance is a zone within which the symmetrical surfaces align with the datum of a center plane or axis.

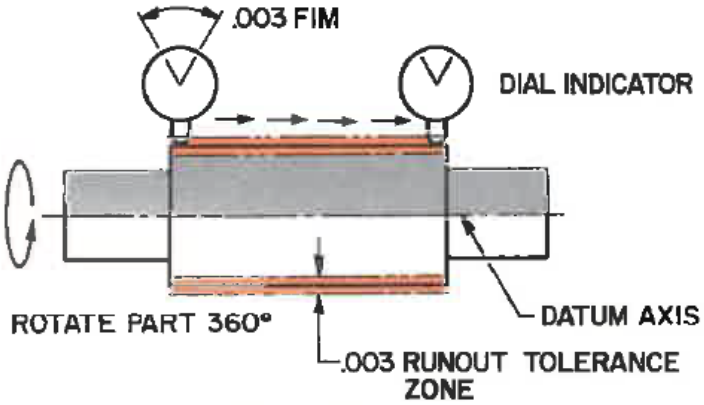
Runout geometric tolerances

- **Total runout** controls circularity, straightness, angularity, and cylindricity of a part when applied to surfaces rotated around a datum axis.
- **Circular runout** is applied to features independently and controls circularity of a single circular cross section.

Runout geometric tolerance symbols. Arrows may be filled or unfilled.

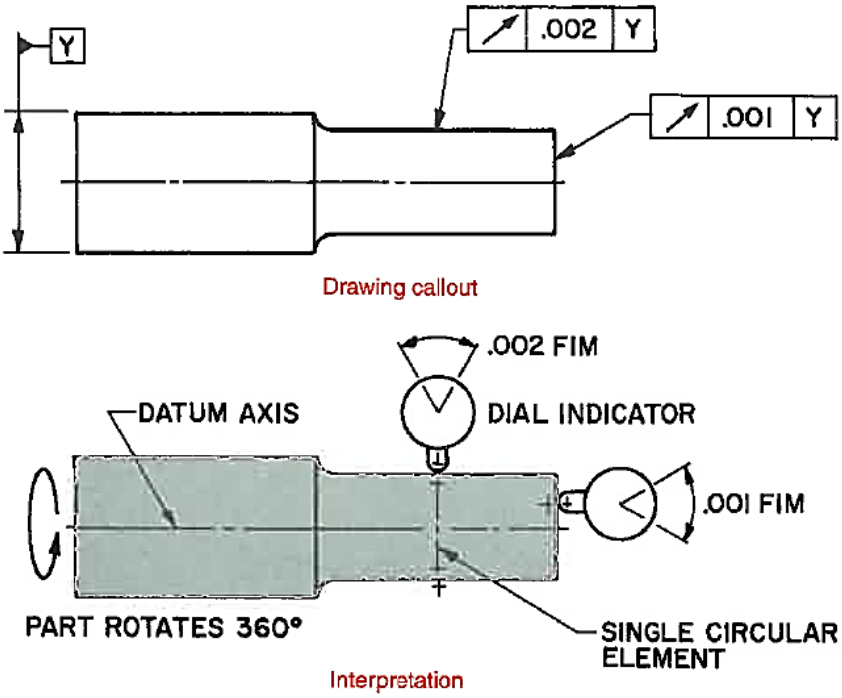


Drawing callout



Interpretation

Total runout controls circularity, straightness, angularity, and cylindricity of a part when applied to surfaces rotated around a datum. The entire surface must lie within the tolerance zone.



Circular runout controls circularity of a single circular cross section.