PARTS LIST

No.	Name	Quan.
1	CRANKCASE	1
2	CRANKSHAFT	1
3	CRANKCASE COVER	1
4	CYLINDER	2
5	PISTON	2

A

A1776	NUT	BRASS	6
A1985	BOLT	BRASS	6
B1765	PLATE	ALUMINUM	2
B1767	CYLINDER	CAST IRON	2
Pt. No.	Name	Material	Quan.

BILL OF MATERIALS

B

Figure 3-25. A list of parts and materials is normally included with the drawings for a project. A—A typical, but partial, parts list. B—An example of a partial bill of materials.

on standard-size sheets. This simplifies the stocking, handling, and storage of the completed drawings.

Standard sizes for drawing sheets include the following:

US CONVENTIONAL SHEET SIZES
A size = 8 1/2″ × 11″
B size = 11″ × 17″
C size = 17″ × 22″
D size = 22″ × 34″
E size = 34″ × 44″

SI METRIC SHEET SIZES
A4 size = 210 × 297 mm
A3 size = 297 × 420 mm
A2 size = 420 × 594 mm
A1 size = 594 × 841 mm
A0 size = 841 × 1189 mm

Also, for convenience in filing and locating drawings in storage, each drawing has an identifying number, **Figure 3-12(H).**

3.7 GEOMETRIC DIMENSIONING AND TOLERANCING

Conventional tolerancing is appropriate for many products. However, for accurately machined parts, the amount of variation (tolerances) in form (shape and size) and position (location) may need to be more strictly defined. This definition provides the precision needed to allow for the most economical manufacture of parts. See **Figure 3-26.**

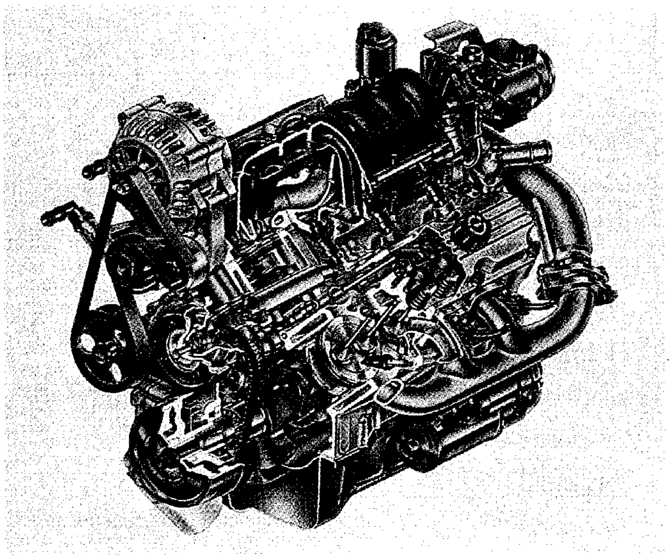


Figure 3-26. A high degree of precision is needed to produce the parts used in this engine. The tolerances allowed for the shape and location of features on the parts must not be exceeded. (Buick Div. of GMC)

Geometric dimensioning and tolerancing is a system that provides additional precision compared to conventional dimensioning. It ensures that parts can be easily interchanged.

Only a brief introduction to geometric dimensioning and tolerancing is included in this text. Detailed information can be found in the publication ASME Y14.5M-1994.

3.7.1 Definitions

Geometric characteristic symbols are employed to provide clarity and precision in communicating design specifications. See **Figure 3-27.** These symbols are standardized by the American Society of Mechanical Engineers (ASME). *Geometric tolerance* is a general term that refers to tolerances which control form, profile, orientation, location, and runout.

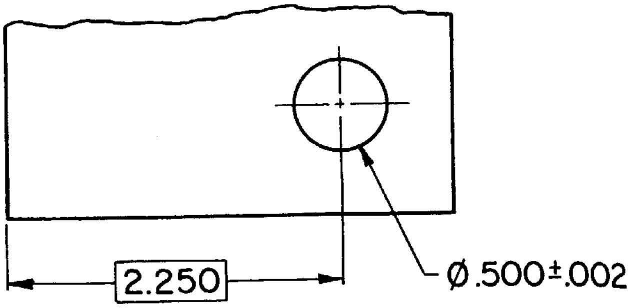
A ***basic dimension*** is a numerical value denoting the exact size, profile, orientation, or location of a feature. The ***true position*** of a feature is its theoretically exact location as established by basic dimensions. A *reference dimension* is a dimension provided for information only. It is not used for production or inspection purposes. See **Figure 3-28.**

Datum is an exact point, axis, or plane. It is the origin from which the location or geometric characteristic of features of a part is established. It is identified by a solid triangle with an identifying letter. See **Figure 3-29.** *Feature* is a general term applied to a physical portion of a part, such as a surface, pin,

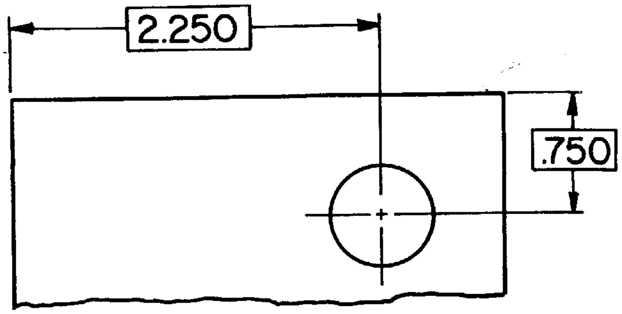
Symbol for:	ASME Y14.5
Straightness	⏤
Flatness	⏥
Circularity	○
Cylindricity	⌭
Profile of a line	⌒
Profile of a surface	⌓
All-around profile	◄⊖—
Angularity	∠
Perpendicularity	⊥
Parallelism	//
Position	⌖
Concentricity/coaxiality	◎
Symmetry	⌯
Circular runout	* ↗
Total runout	* ⌰
At maximum material condition	Ⓜ
At least material condition	Ⓛ
Regardless of feature size	NONE
Projected tolerance zone	Ⓟ
Diameter	Ø
Basic dimension	$\boxed{30}$
Reference dimension	(30)
Datum feature	$\boxed{A}$—◄
Datum target	Ø6/A1
Target point	X
Dimension origin	⌽→
Feature control frame	|⌖|Ø0.5Ⓜ|A|B|C|
Conical taper	⌲
Slope	⌳
Counterbore/spotface	⌴
Countersink	⌵
Depth/deep	↧
Square (shape)	□
Dimension not to scale	<u>15</u>
Number of times/places	8X
Arc length	$\overset{\frown}{105}$
Radius	R
Spherical radius	SR
Spherical diameter	SØ

* May be filled

Figure 3-27. Symbols used to specify positional and form tolerances in geometric dimensioning. (American National Standards Institute)



Basic dimensions



True position

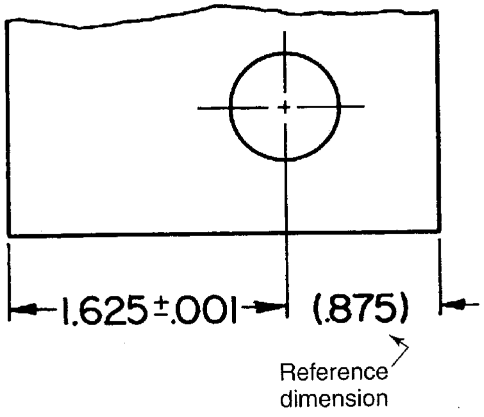


Figure 3-28. *Basic dimensions are usually indicated by being enclosed in a rectangular frame. They are not toleranced. True position is the theoretical exact location of feature. It is established by basic dimensions. Reference dimensions are not used for production or inspection purposes. On a drawing, they are shown enclosed in parentheses.*

hole, or slot. A ***datum feature*** is the actual feature of a part used to establish a datum. See **Figure 3-30.**

Maximum material condition (MMC) is the condition in which the size of a feature contains the maximum amount of material within the stated limits of size. Examples include a minimum hole diameter and maximum shaft diameter, both of which result in the greatest possible amount of material

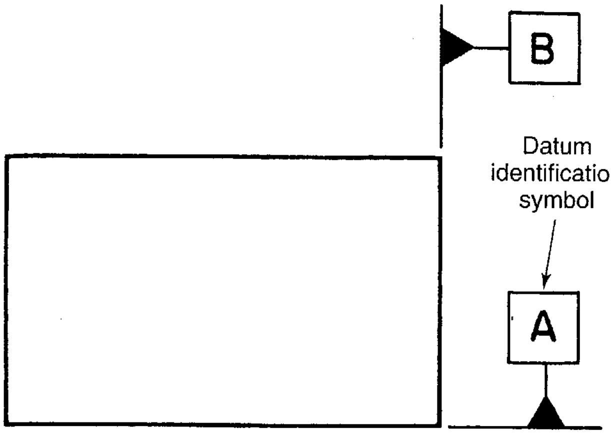


Figure 3-29. Datums are exact points, axes, or planes from which features of a part are located.

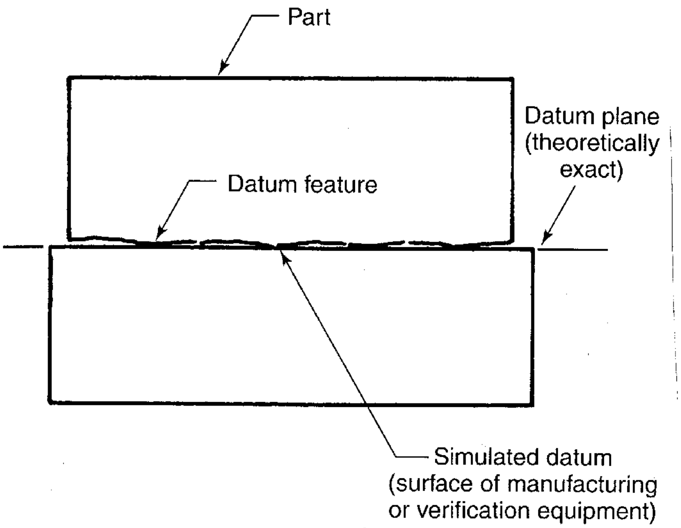


Figure 3-30. A datum feature is a physical feature on a part used to establish a datum.

being used. See **Figure 3-31.** MMC is indicated by an *M* within a circle.

Least material condition (LMC) is the condition in which the size of a feature contains the least amount of material within the stated tolerance limits. Examples include a maximum hole diameter and a minimum shaft diameter. See **Figure 3-32.** LMC is indicated by an *L* within a circle.

Regardless of feature size (RFS) specifies that the size of a feature tolerance must not be exceeded. RFS is assumed for all geometric tolerances unless otherwise specified.

The maximum and minimum sizes of a feature are called the ***limits of size.*** See **Figure 3-33.** The measured size of a part after it is manufactured is the ***actual size.***

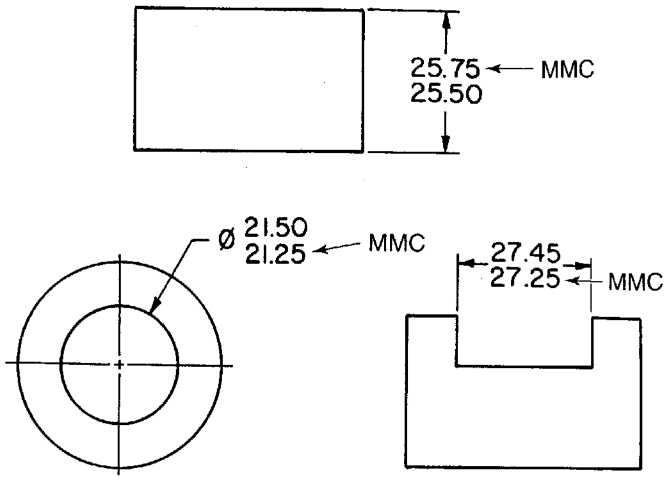


Figure 3-31. Maximum material condition (MMC) indicates that the size of a feature contains the maximum amount of material within the stated tolerance limits.

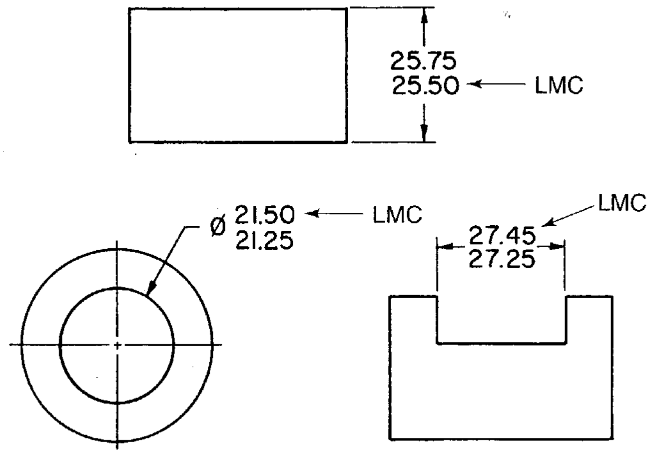


Figure 3-32. Least material condition (LMC) indicates that the size of a feature contains the least amount of material within the stated limits of size.

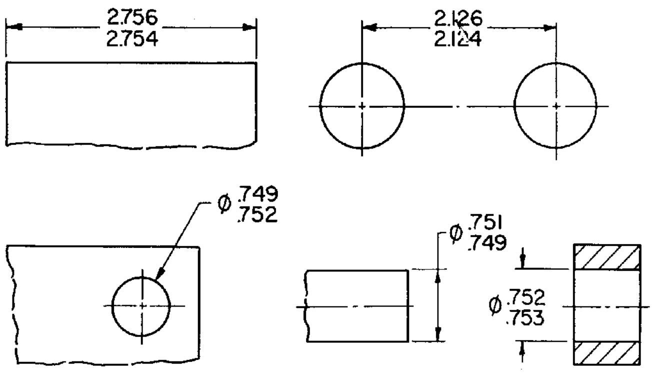


Figure 3-33. Limits of size are the maximum and minimum sizes of a feature.

3.7.2 Application of Geometric Dimensioning and Tolerancing

Datum identification symbol. A datum identifying symbol, **Figure 3-34,** consists of a square frame that contains the *datum reference letter.* All letters but *I, O,* and *Q* may be used. A rectangular frame with the datum reference letter preceded and followed by a dash may be found on older drawings.

Feature control frame. A feature control frame is used when a location or form tolerance is related to a datum. It contains the geometric symbol, allowable tolerance, and the datum reference letter(s). It is connected to an extension line of the feature, a leader running to the feature, or below a leader-directed note of the feature, **Figure 3-35.**

Datum references indicated on the right end of the feature control frame are read from left to right. The letters signify datum preference. They establish three mutually perpendicular planes, **Figure 3-36.**

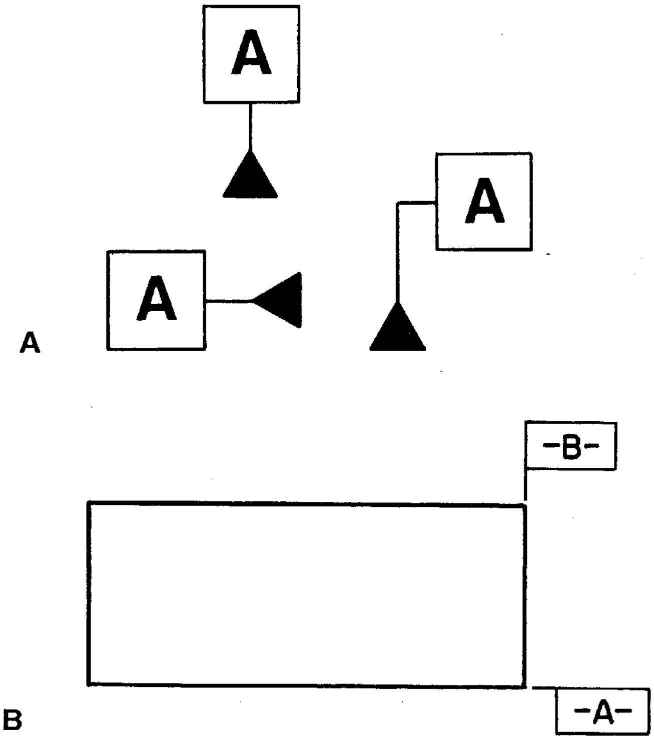
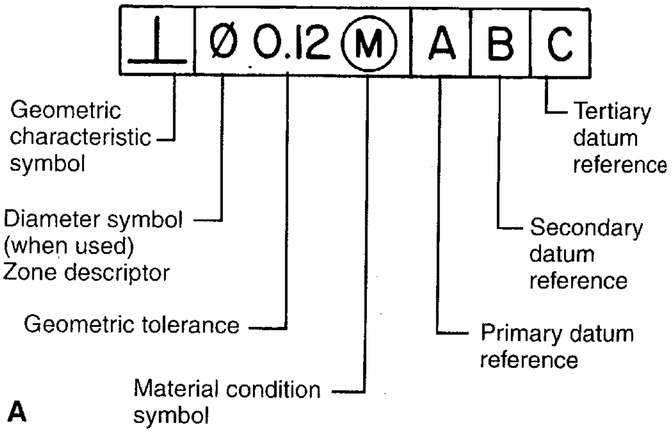


Figure 3-34. Datum points and surfaces are identified by a datum identification symbol. A—Datum identification symbols used on new drawings. B—This type of datum symbol is not used currently, but is still found on old drawings.



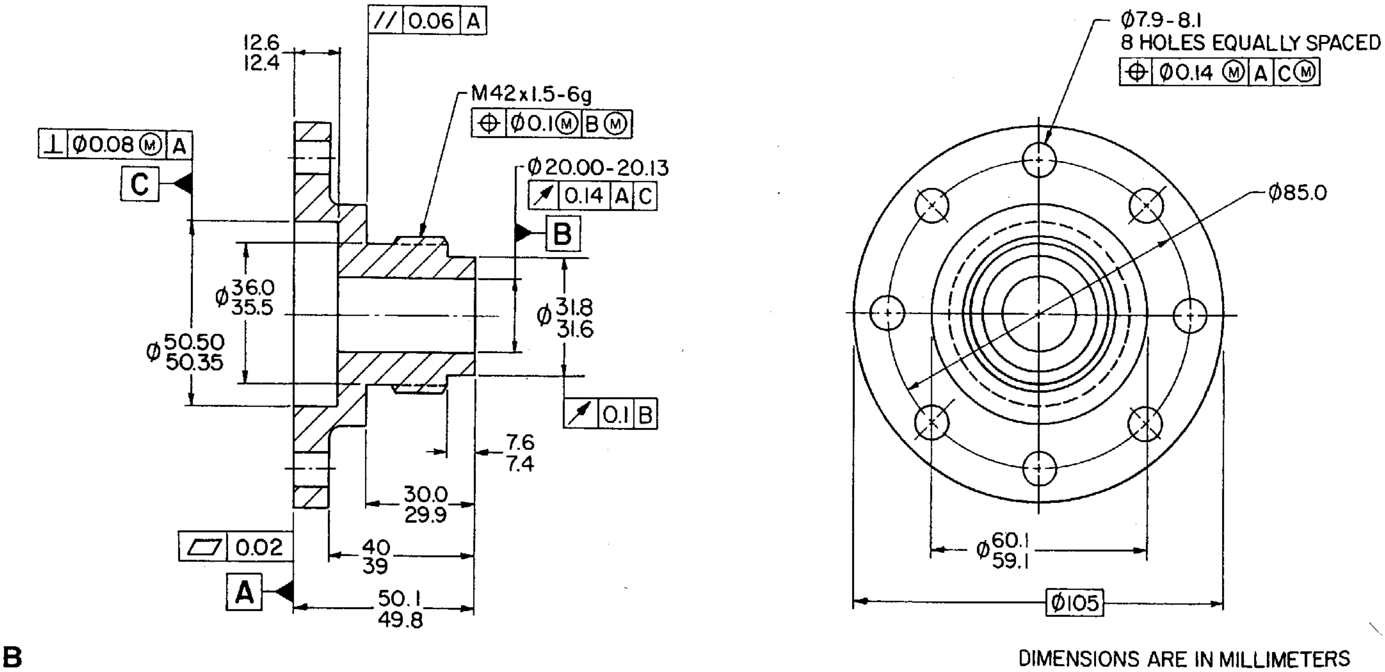


Figure 3-35. A feature control frame is employed when a location or form tolerance is related to a datum. A—Components of a feature control frame. B—Feature control frames are used to specify tolerances on this drawing.

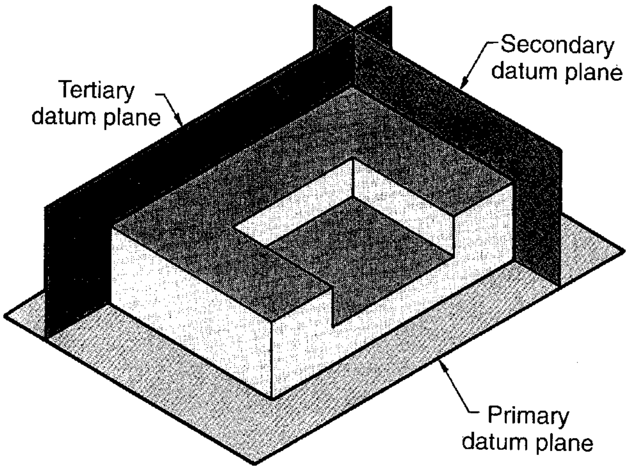


Figure 3-36. Datum references are perpendicular planes. The first datum referenced is the primary datum, followed by the secondary and tertiary datums.

3.7.3 Form Geometric Tolerances

Form geometric tolerances control flatness, straightness, circularity (roundness), and cylindricity. They are indicated by the symbols shown in **Figure 3-37.** Form tolerances control only the variation permitted on a single feature and are used when form variation is less than that permitted by size tolerance.

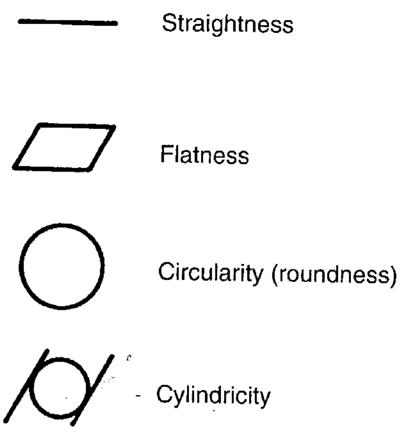


Figure 3-37. Form geometric symbols.

Flatness is a measure of the variation of a surface perpendicular to its plane. The flatness geometric tolerance specifies the two parallel planes within which all points of a surface must lie, **Figure 3-38.**

Straightness describes how closely the surface of an object is to a line. A straightness geometric tolerance establishes a tolerance zone of uniform width along a line. All elements of the surface must lie within this zone, **Figure 3-39.**

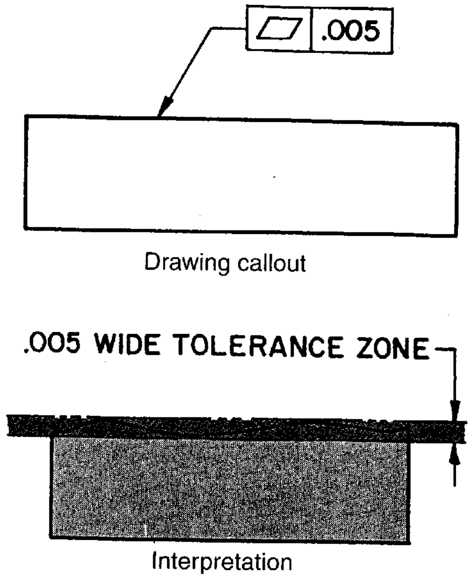


Figure 3-38. The flatness geometric form tolerance specifies the two parallel planes within which a surface must lie.

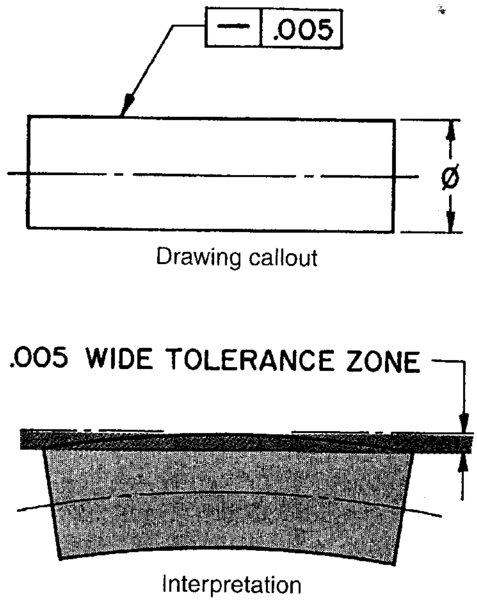
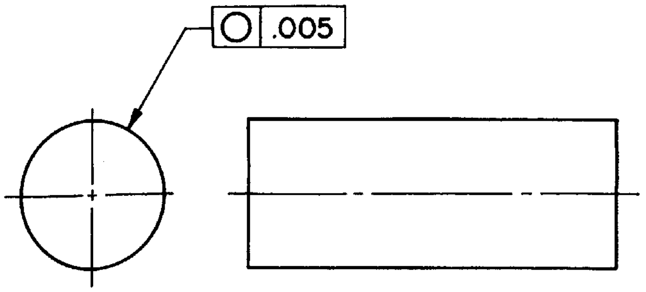
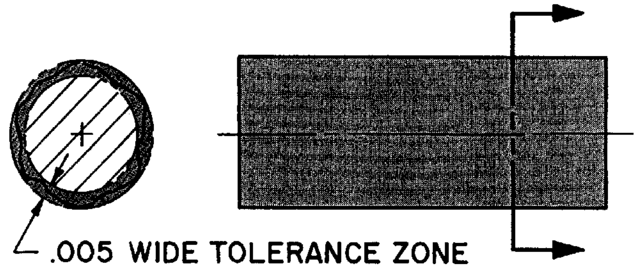


Figure 3-39. A straightness geometric form tolerance establishes a tolerance zone of uniform width along a straight line. All elements of the surface must lie within this zone.

Circularity is characterized by any given cross section taken perpendicular to the axis of a cylinder or a cone, or through the common center of a sphere. A circularity (roundness) geometric tolerance specifies a tolerance zone bounded by two concentric circles, indicated on a plane perpendicular to the axis of a cylinder or a cone, within which each circular element must lie. It is a single cross-sectional tolerance. See **Figure 3-40.**



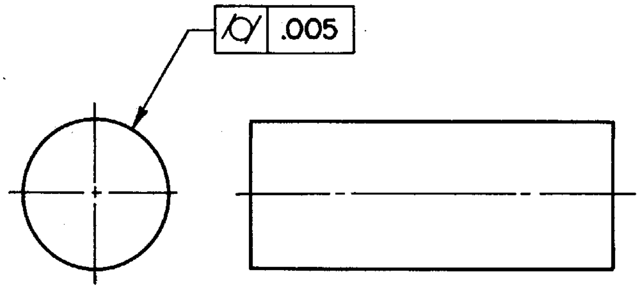
Drawing callout



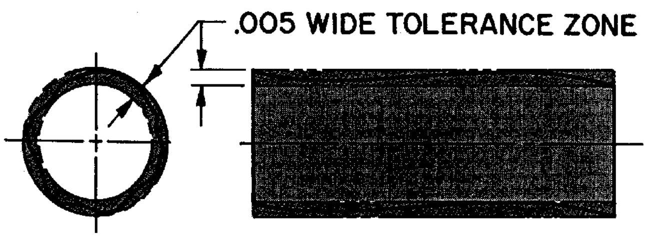
Interpretation

Figure 3-40. *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.*

Cylindricity represents a surface in which all points are an equal distance from a common center. The cylindricity geometric tolerance establishes a tolerance zone that controls the diameter of a cylinder throughout its entire length. It consists of two concentric cylinders within which the actual surface must lie. This tolerance covers both the circular and longitudinal elements. See **Figure 3-41.**



Drawing callout



Interpretation

Figure 3-41. *The cylindricity geometric tolerance establishes a tolerance zone that controls the diameter of a cylinder throughout its entire length.*

3.7.4 Profile Geometric Tolerances

A *profile geometric tolerance* controls the outline or contour of an object and can be represented by an external view or by a cross section through the object. It is a boundary along the true profile in which elements of the surface must be contained. The symbols used to indicate profile tolerances are shown in **Figure 3-42.**

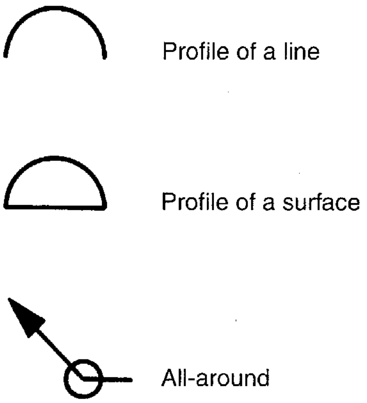


Figure 3-42. *Profile geometric tolerance symbols. When a tolerance is specified for all sides of an object, the "all-around" symbol is used.*

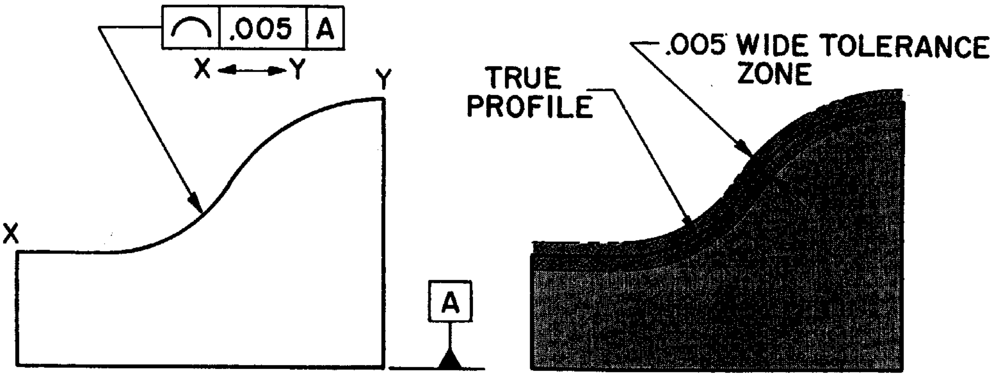
A *profile line geometric tolerance* is a two-dimensional (cross-sectional) tolerance zone extending along the length of the element. It is located using basic dimensions, **Figure 3-43.**

The *profile surface geometric tolerance* is three-dimensional and extends along the length and width of the surface. For proper orientation of the profile, a datum reference is usually required, **Figure 3-44.**

3.7.5 Orientation Geometric Tolerances

Orientation geometric tolerances control the degree of parallelism, perpendicularity, or angularity of a feature with respect to one or more datums. There are three orientation tolerances, **Figure 3-45.**

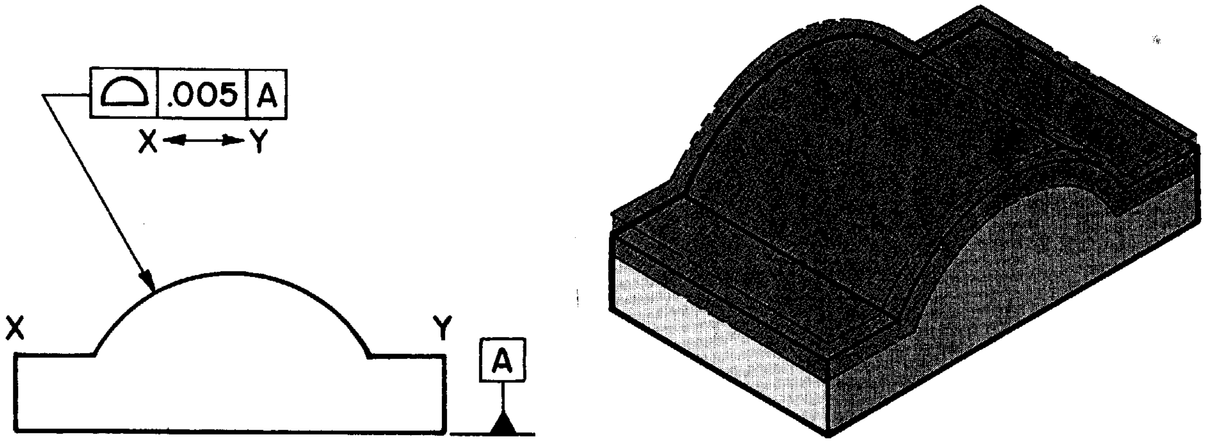
Angularity is concerned with the position of a surface or axis at a specified angle to a datum plane or axis. The specified angle must be other than 90°. An *angularity geometric tolerance* establishes a tolerance zone defined by two parallel lines, planes, or a cylindrical zone at a specified basic angle other than 90°. The line elements, surface, or axis of the considered feature must lie within this zone, **Figure 3-46.**



Drawing callout

Interpretation

Figure 3-43. *A profile line geometric tolerance is a two-dimensional tolerance zone extending along the length of the considered element.*



Drawing callout

Interpretation

Figure 3-44. *The profile surface geometric tolerance is three-dimensional and extends along the length and width of the surface.*

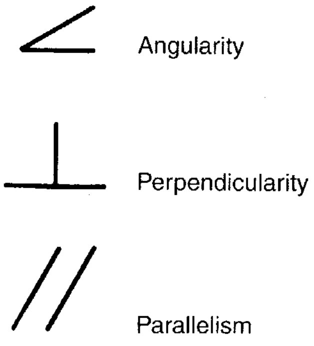


Figure 3-45. *Orientation geometric tolerance symbols.*

A *perpendicularity geometric tolerance* specifies a tolerance zone at right angles to a given datum or axis. It is described by two parallel lines, planes, or a cylindrical tolerance zone. The line, surface, or axis of the considered feature must lie within this zone, **Figure 3-47.**

Parallelism describes how close all elements of a line or surface are to being parallel (equidistant) to a given datum plane or axis. 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, **Figure 3-48.**

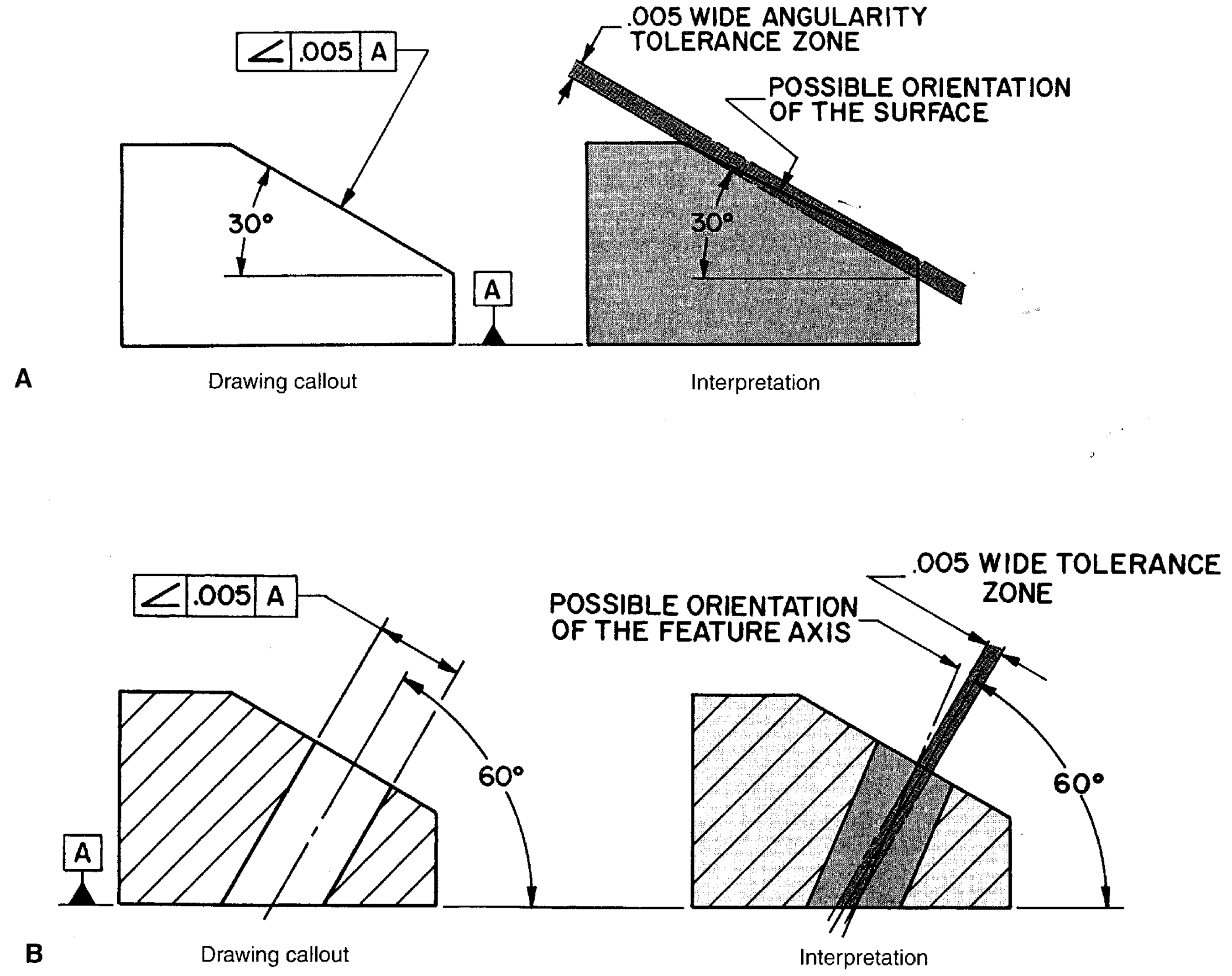


Figure 3-46. *An angularity geometric tolerance establishes a tolerance zone defined by two parallel lines, planes, or a cylindrical zone at a specified basic angle other than 90°. A—Angularity of a surface. B—Angularity of an axis.*

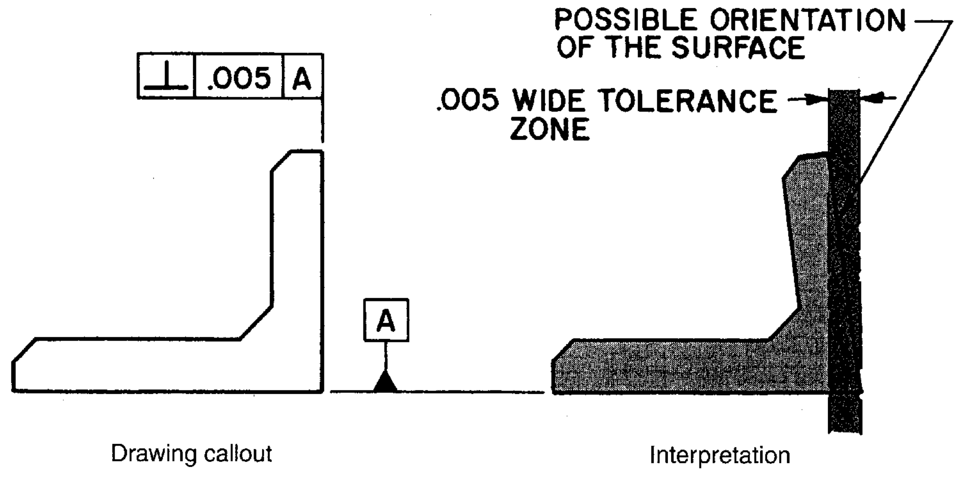
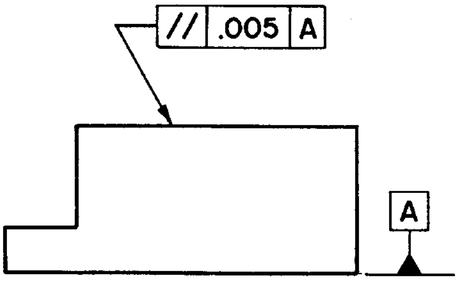
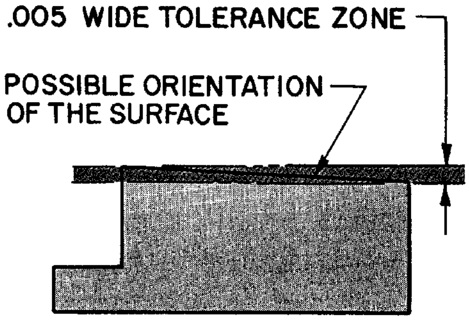


Figure 3-47. *The line, surface, or axis of a considered feature must lie within the perpendicularity geometric tolerance zone.*



Drawing callout



Interpretation

Figure 3-48. 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.

3.7.6 Location Geometric Tolerances

Location geometric tolerances are employed to establish the location of features and datums. They define the zone within which the center, axis, or center plane of a feature may vary from a true (theoretically exact) position. Location tolerances are also known as *positional tolerances* and include position, concentricity, and symmetry. See **Figure 3-49.**

Basic dimensions establish the true position of a feature from specified datums and related features. A *positional geometric tolerance* establishes how far a feature may vary from its true position, **Figure 3-50.**

Concentricity defines the relationship between the axes of two or more of an object's cylindrical features. A *concentricity geometric tolerance* is expressed

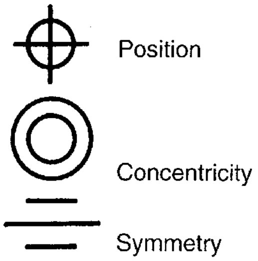


Figure 3-49. Location or positional tolerance symbols.

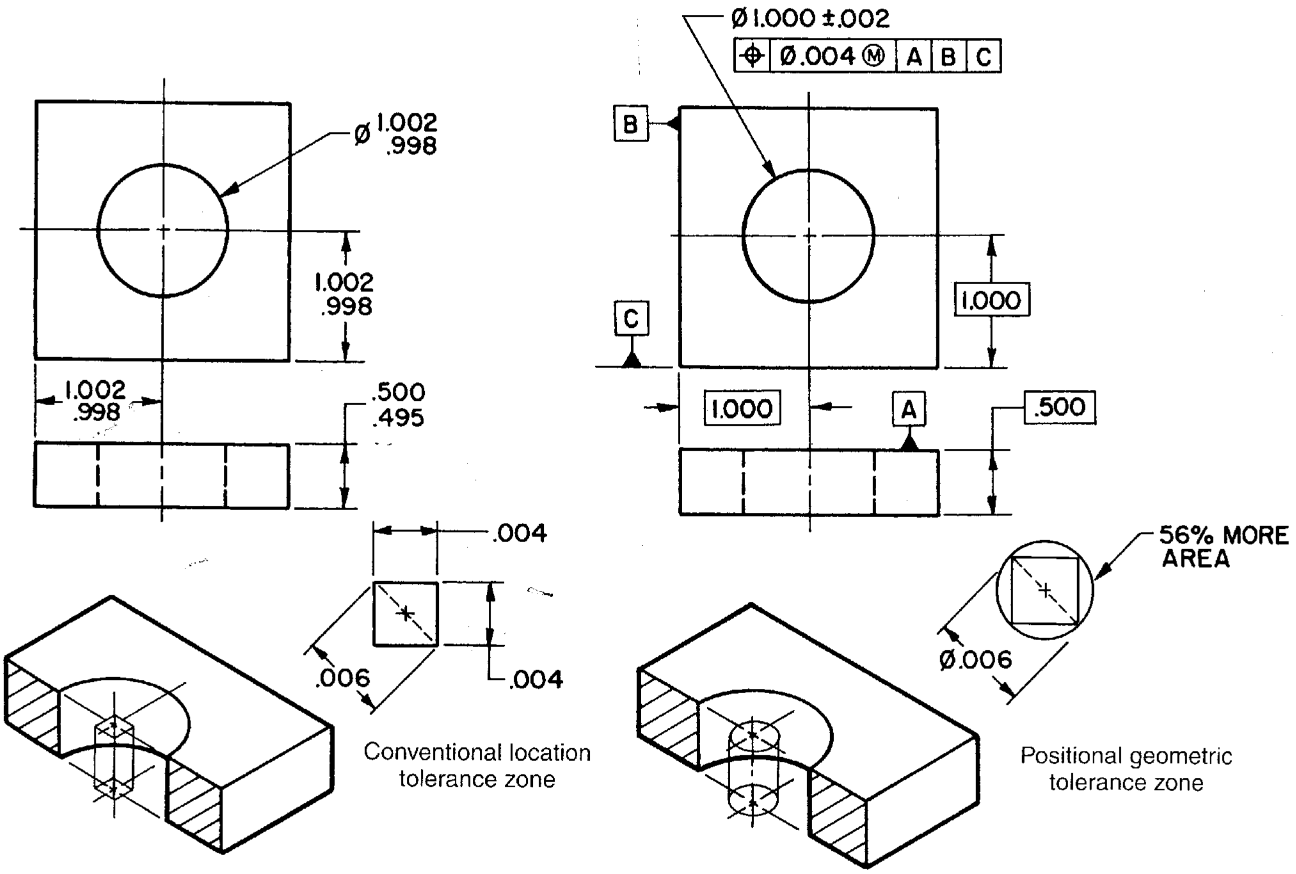


Figure 3-50. A positional geometric tolerance establishes how far a feature may vary from its true position.

as a cylindrical tolerance zone. The axis or center point of this zone coincides with a datum axis, **Figure 3-51.**

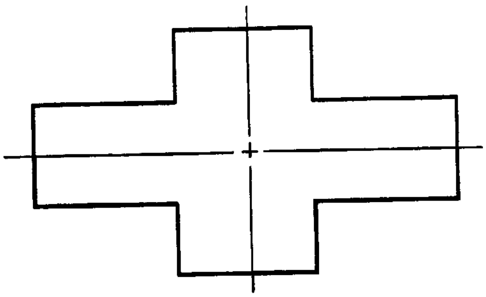
Since this tolerance is sometimes difficult and time-consuming to verify, runout or positional geometric tolerances are often used instead.

Symmetry indicates equal or balanced proportions on either side of a central plane or datum, **Figure 3-52.** A *symmetry geometric tolerance* is a zone within which the symmetrical surfaces align with the datum of a center plane or axis, **Figure 3-53.**

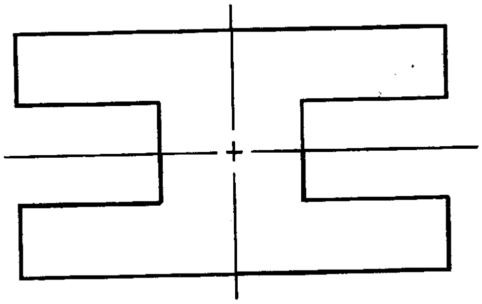
3.7.7 Runout Geometric Tolerances

There are two types of ***runout geometric tolerances***—total runout and circular runout. These tolerances are indicated by the symbols shown in **Figure 3-54.** Runout tolerances are used to control runout of surfaces around or perpendicular to a datum axis.

Total runout controls circularity, straightness, angularity, and cylindricity of a part when applied to surfaces rotated around a datum axis, **Figure 3-55.** The entire surface must lie within the tolerance zone.

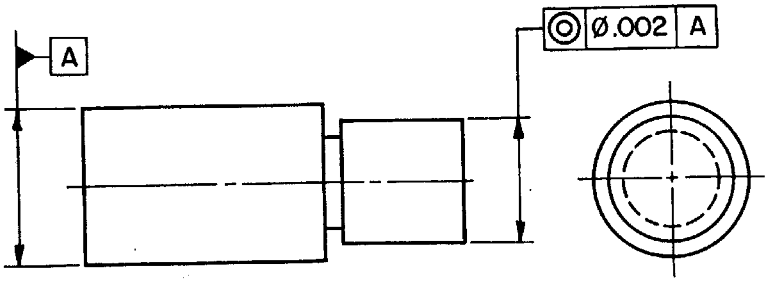


Symmetrical double tab

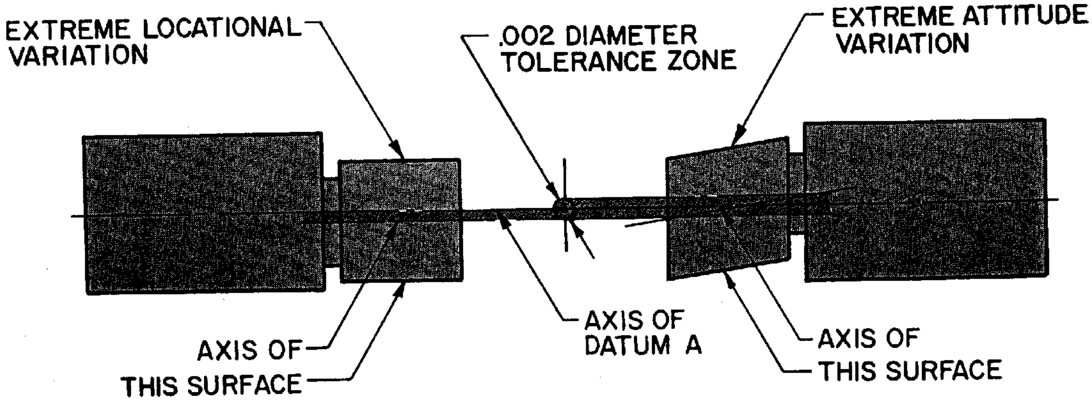


Symmetrical double slot

Figure 3-52. Symmetry indicates equal or balanced proportions on either side of a central plane.

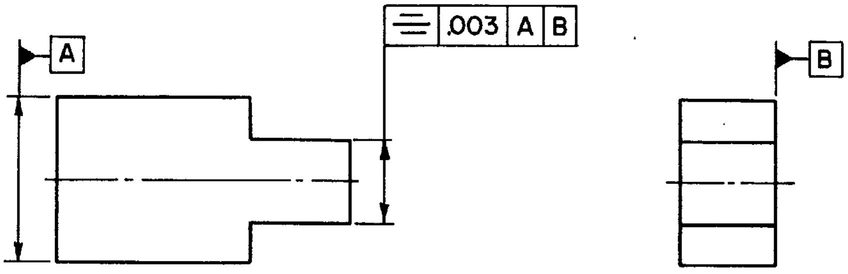


Drawing callout

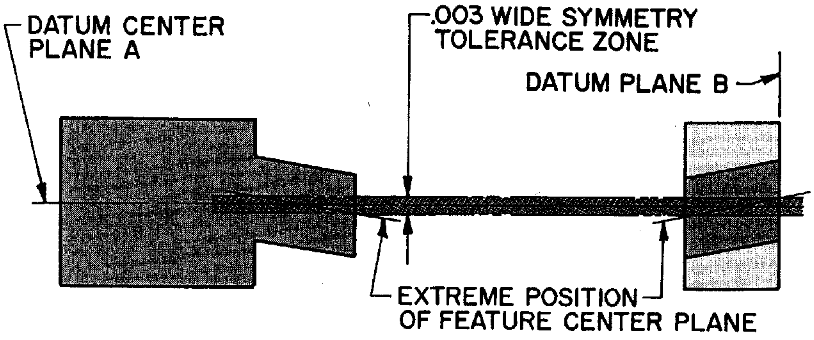


Interpretation

Figure 3-51. 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



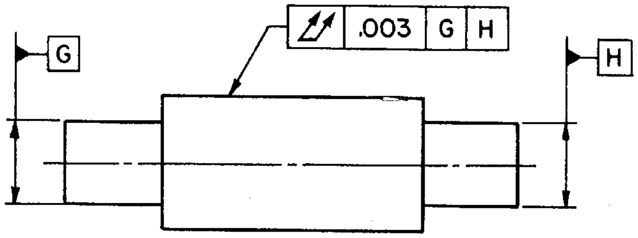
Interpretation

Figure 3-53. *A symmetry geometric tolerance is a zone within which the symmetrical surfaces align with the datum of a center plane or axis.*

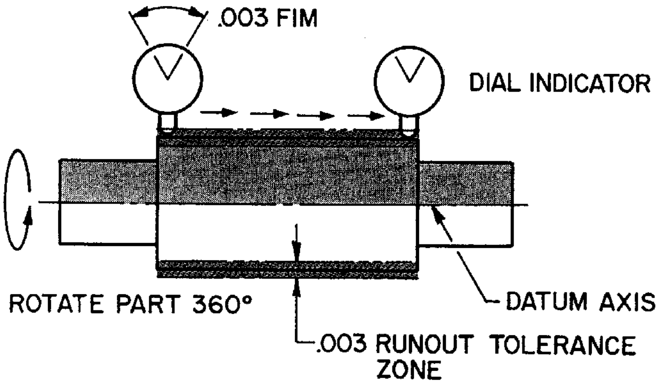
Circular runout

Total runout

Figure 3-54. *Runout geometric tolerance symbols. Arrows may be filled or unfilled.*



Drawing callout



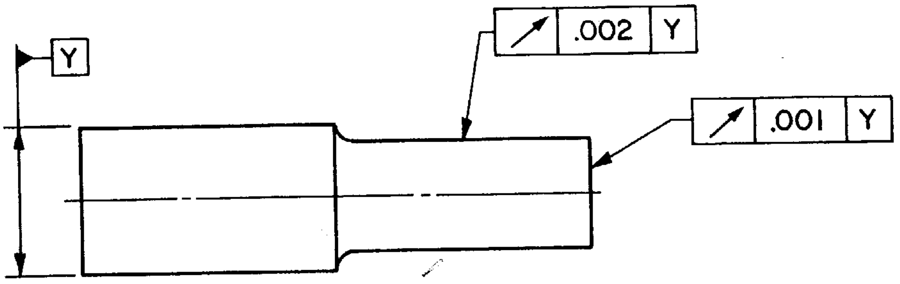
Interpretation

Figure 3-55. *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.*

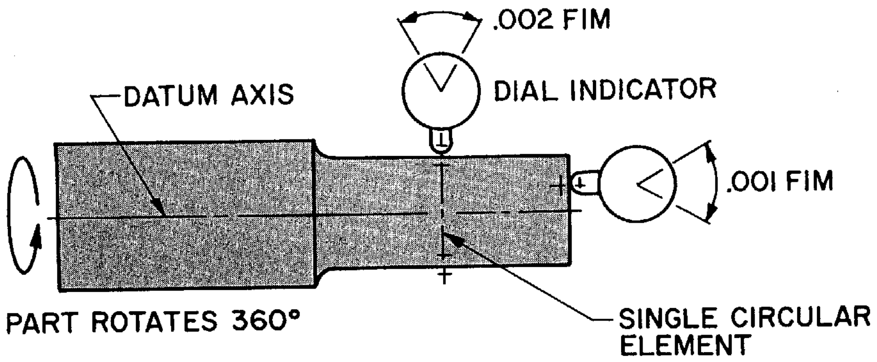
Circular runout is applied to features independently and controls circularity of a single circular cross section, **Figure 3-56.** The tolerance is measured by the *full indicator movement (FIM)* of a dial indicator when it is placed at several positions as the part is rotated.

3.7.8 Summary

Geometric dimensioning and tolerancing is far more involved than described on the preceding pages. As you progress in machining technology, you should consider purchasing a text on the subject, studying a copy of ASME Y14.5M–1994, or enrolling in a class on geometric dimensioning and tolerancing.



Drawing callout



Interpretation

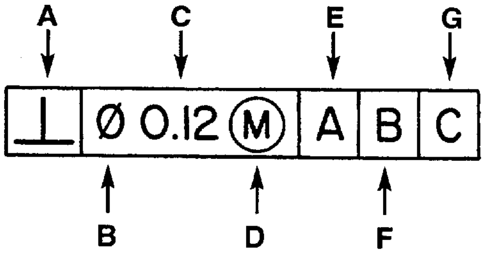
Figure 3-56. *Circular runout controls circularity of a single circular cross section.*

TEST YOUR KNOWLEDGE

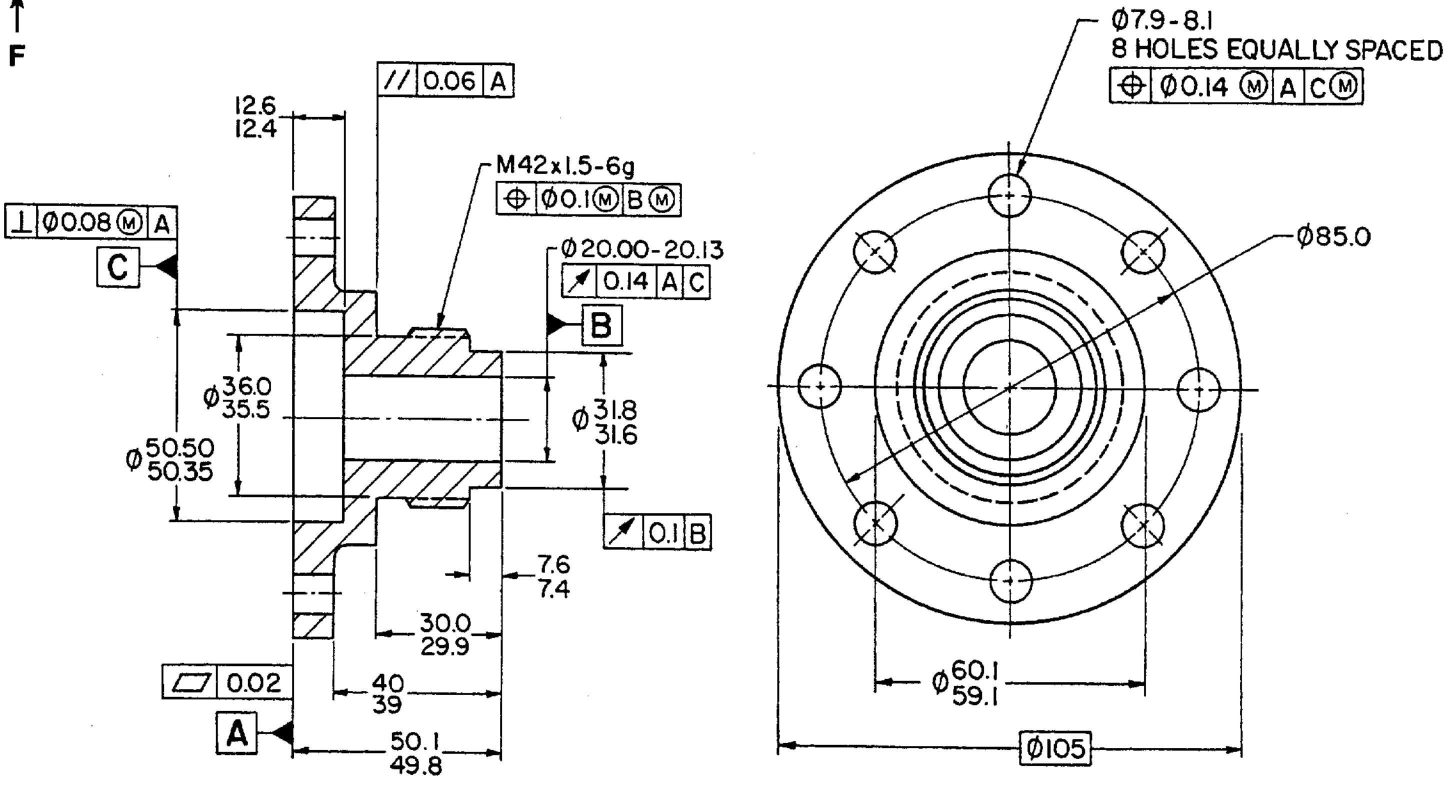
Please do not write in the text. Write your answers on a separate sheet of paper.

1. Drawings are used to:
   a. Show, in multiview, what an object looks like before it is made.
   b. Standardize parts.
   c. Show what to make and the sizes to make it.
   d. All of the above.
   e. None of the above.
2. The symbols, lines, and figures that make up a drawing are frequently called the _____.
3. A microinch is _____ of an inch.
4. A micrometer is _____ of a meter.
5. How can surface roughness of a machined part be checked against specifications on the drawing? How can it be measured electronically?
6. When tolerances are plus and minus, it is called a _____ tolerance.
7. When tolerances are only plus or only minus, it is called a _____ tolerance.
8. Tolerances are:
   a. The different materials that can be used.
   b. Allowances in either oversize or undersize that a part can be made and still be acceptable.
   c. Dimensions.
   d. All of the above.
   e. None of the above.

Geometric Dimensioning and Tolerancing



- **A** Geometric characteristic symbol
- **B** Diameter symbol (when used) Zone descriptor
- **C** Geometric tolerance
- **D** Material condition symbol
- **E** Primary datum reference
- **F** Secondary datum reference
- **G** Tertiary datum reference



DIMENSIONS ARE IN MILLIMETERS