Geometric Dimensioning and Tolerancing

Learning Objectives

After studying this chapter, you will be able to:

- Define the common terms used in geometric dimensioning and tolerancing applications.
- List and describe the different types of tolerances used to control fits for machine parts.
- Identify specific symbols used in geometric dimensioning and tolerancing applications.
- Explain the standard practices for applying tolerance dimensions to drawings.
- Describe how tolerance dimensions are created on CAD drawings.

Technical Terms

Actual size
Allowance
Angularity
Angular surface tolerancing
Annular space
Baseline dimensioning
Basic dimension
Basic hole size
Basic hole size system
Basic shaft size system
Basic size
Bilateral tolerance
Chain dimensioning
Circularity
Circular runout
Clearance fit
Concentricity
Cylindricity
Datum
Datum dimensioning
Datum feature symbol
Datum target
Design size
Feature control frame
Fit
Flatness
Force fits
Form tolerances
Geometric characteristic symbols
Geometric dimensioning and tolerancing
Interchangeable manufacture
Interference fit
Lay
Least material condition (LMC)
Limit dimensioning
Limits
Locational fits
Location tolerances
Maximum material condition (MMC)
Nominal size
Orientation
Orientation tolerances
Parallelism
Perpendicularity
Plus and minus tolerancing
Positional tolerances
Press fits
Profile
Profile tolerances
Projected tolerance zone
Reference dimension
Regardless of feature size (RFS)
Roughness
Roughness height
Roughness width
Roughness-width cutoff
Running and sliding fits
Runout
Runout tolerances
Selective assembly
Shrink fits
Straightness
Surface texture
Symmetry
Tolerance
Tolerancing
Total runout
Transition fit
True position
True positional tolerance
Unilateral tolerance
Waviness
Waviness height
Waviness width

The manufacture of a product usually requires the assembly of a number of different components. These components may all be made by the same company in one location. However, in many cases, several different industries supply the components for assembly. Therefore, it is necessary to control dimensions very closely so that all of the parts will fit properly. This is called ***interchangeable manufacture***. Interchangeability is also essential for replacement parts.

Tolerancing Fundamentals

The control of dimensions is called ***tolerancing***. A toleranced dimension means that the dimension has a range of acceptable sizes that are within a "zone." The size of this zone depends on the function of the part. To achieve an exact size (a non-toleranced dimension) is not only very expensive, but virtually impossible under normal conditions. Therefore, tolerances are set as liberal as possible while still being able to produce a functioning part.

Industrial designers and engineers establish tolerances based on industry standards, practice, and the function of the part. Drafters, too, must understand the application of tolerances to engineering drawings.

Types of Tolerances

Tolerances are used to control the size of the features of a part. Tolerances are also used to control the position and form of parts. ***Positional tolerances*** control the location of features on a part. ***Form tolerances*** control the form or the geometric shape of features on a part. These basic types and the uses of tolerances are presented in this chapter.

Tolerancing Terms

There are standard terms used to effectively communicate information related to tolerancing. A drafter must understand the meaning of these terms. A drafter must also be able to correctly apply these terms to a drawing.

Basic Dimension

A ***basic dimension*** is an exact, untoleranced value used to describe the size, shape, or location of a feature. Basic dimensions are used as a "base." From this base, tolerances or other associated dimensions are established.

Basic dimensions are not directly toleranced. Any permissible variation is contained in the tolerance on the dimension associated with the basic dimension. An example of a tolerance on a hole diameter is shown in **Figure 16-1**.

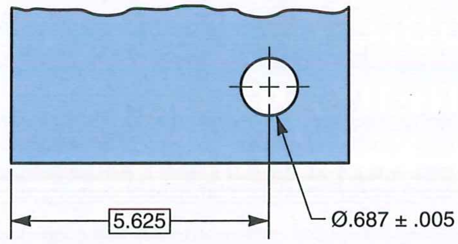

Figure 16-1. Basic dimensions are not toleranced. Instead, the feature that the associated dimension is referring to is toleranced.

Basic dimensions are indicated on the drawing by enclosing the dimension figure in a rectangular frame to indicate that it is a basic dimension. A general note such as "UNTOLERANCED DIMENSIONS LOCATING TRUE POSITION ARE BASIC" can also be used.

Reference Dimension

A ***reference dimension*** is placed on a drawing for the convenience of engineering and manufacturing personnel. A reference dimension is indicated by enclosing the dimension within parentheses, **Figure 16-2**.

Reference dimensions are untoleranced dimensions. They are not required for the manufacturing of a part or in determining the acceptability of the part. Reference dimensions may be rounded off as desired.

Datum

A ***datum*** is an exact plane, line, or point from which other features are located. A datum is usually a plane or point on the part. However, a datum can also be a plane or surface on the machine being used. For example, a datum can be located on the mill table for a part that will be milled.

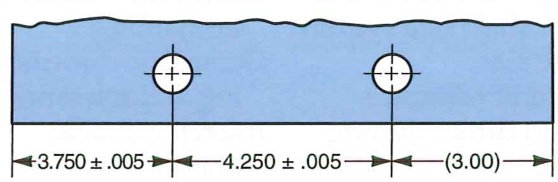

Figure 16-2. A reference dimension is indicated on a drawing by enclosing the dimension in parentheses. Reference dimensions are not toleranced. They are presented on the drawing for the benefit of engineering and manufacturing personnel. Reference dimensions are not used in the manufacture or inspection of the part.

Care should be exercised in the selection of datums on drawings to make sure they are recognizable, accessible, and useful for measuring. Corresponding features on mating parts should be selected as datums to assure ease of assembly. A datum is indicated on a drawing by a ***datum feature symbol***, **Figure 16-3**. This symbol consists of a capital letter enclosed in a square frame connected to a triangle.

A machined part may require more than one datum in its dimensioning. Letters such as "A," "B," and "C" are assigned to each datum. Datums may be considered to be *primary, secondary*, and *tertiary*, depending on the design of the part. The type of datum determines the preference for the order in which datums appear. Dimensions on the part are established in the given sequence. In other words, the primary datum would appear first (on the left) in a feature control frame, and the tertiary datum would appear last (on the right). If a particular feature can be measured in reference to all three datums, the measurement in reference to the primary datum takes precedence over the other two. If the measurement is within tolerance when measured from the secondary and tertiary datums, but not from the primary datum, the feature is not within tolerance.

Nominal Size

The ***nominal size*** is a classification size given to commercial products such as pipe or lumber. It may or may not express the true numerical size of the part or object. For example, a seamless, wrought steel pipe of 3/4″ (.750″) nominal size has an actual inside diameter of 0.824″ and an actual outside diameter of 1.050″, **Figure 16-4A**. In the case of a round rod made of cold-finished, low-carbon steel, the nominal 1″ size is within .002″ of the actual size, **Figure 16-4B**.

Basic Size

The ***basic size*** is the size of a part determined by engineering and design requirements. From this size, allowances and tolerances are applied. For example, the strength and stiffness of a shaft may require 1″ diameter material. This basic 1″ size (with tolerance) is usually applied to the hole size and allowance for a shaft, **Figure 16-5**.

Actual Size

The ***actual size*** is the measured size of a part or object. This measurement is taken from the manufactured part.

Allowance

The ***allowance*** is the intentional difference in the dimensions of mating parts to provide for different classes of fits. This is not the same as a tolerance. Allowance is the minimum clearance space or maximum interference, whichever is intended, between mating parts. In the example shown in **Figure 16-5B**, an allowance of .002″ has been made for clearance (1.000 – .002 = .998).

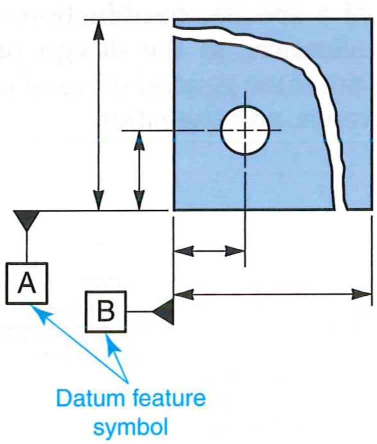

Figure 16-3. A datum is an exact plane, line, or point from which other features are located. Datums are identified by datum feature symbols.

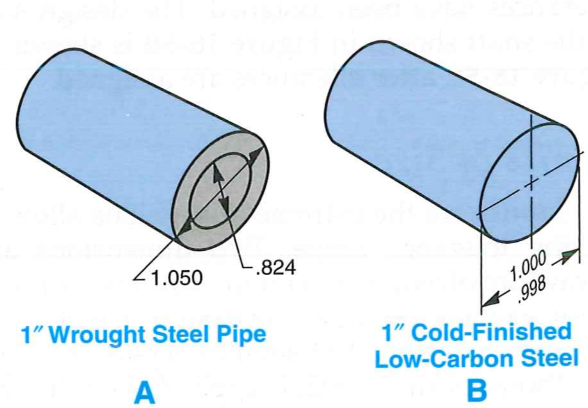

Figure 16-4. The nominal size of a product does not necessarily reflect the actual size.

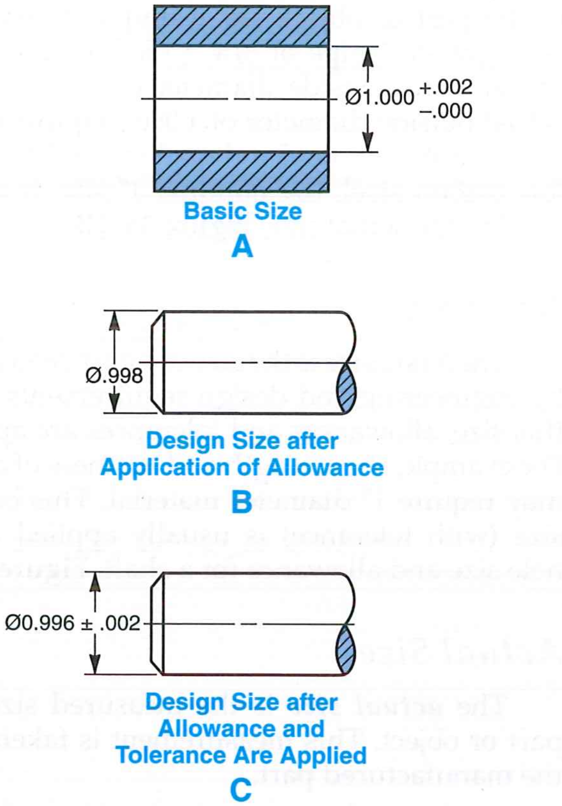

Figure 16-5. The basic size of a part is determined by engineering and design requirements. A—The basic size of a hole for a shaft design. B—The allowance is the maximum variance allowed in the design size. C—The final design size accounts for the allowance and the tolerance for the designed part.

Design Size

The ***design size*** of a part is the size after an allowance for clearance has been applied and tolerances have been assigned. The design size of the shaft shown in **Figure 16-5B** is shown in **Figure 16-5C** after tolerances are assigned.

Limits of Size

Limits are the extreme dimensions allowed by the tolerance range. Two dimensions are always involved, a maximum size and a minimum size. For example, the design size of a feature may be 1.625″. If a tolerance of plus or minus two thousandths (±.002) is applied, then the two limit dimensions are maximum limit 1.627″ and minimum limit 1.623″.

Tolerance

Tolerance is the total amount of variation permitted from the design size of a part. This is not the same as allowance. Tolerances should always be as large as possible while still able to produce a usable part to reduce manufacturing costs. Tolerances can be expressed as limits, **Figure 16-6A**. Tolerances can also be expressed as the design size followed by a plus and minus tolerance, **Figure 16-6B**.

Tolerances can also be given in the title block or in a note, **Figure 16-6C**. If a tolerance is given in a note or the title block, it applies to all dimensions on the drawing, unless otherwise noted.

Unilateral tolerance

A ***unilateral tolerance*** varies in only one direction from the specified dimensions, **Figure 16-7A**. This type of tolerance might be "plus the tolerance" or "minus the tolerance," but never both.

Bilateral tolerance

A ***bilateral tolerance*** varies in both directions from the specified dimension, **Figure 16-7B**. This type of tolerance might vary by the same amount from the given dimension, or it might vary by a different amount in each direction. However, the dimension will vary in both directions.

Fit

Fit is a general term referring to the range of "tightness" or "looseness." Fit results from the application of a specific combination of allowances and tolerances in the design of mating parts. There are three general types of fits: clearance, interference, and transition.

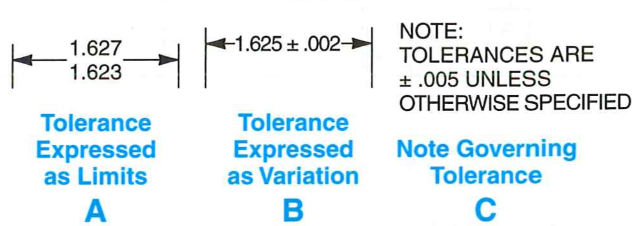

Figure 16-6. Standard methods of indicating a tolerance on a drawing.

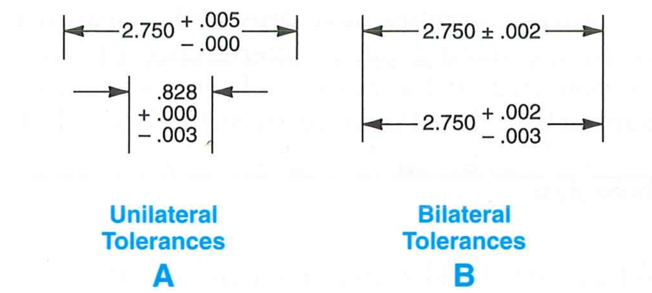

Figure 16-7. Unilateral and bilateral tolerancing. A—Unilateral tolerances vary in one direction only from the specified dimension. B—Bilateral tolerances vary in both directions from the specified dimension.

Clearance fit

A ***clearance fit*** has a positive allowance, or "air space." The limits of size are defined so that a clearance always results when mating parts are assembled, **Figure 16-8A**.

Interference fit

An ***interference fit*** has a negative allowance, or "interference," **Figure 16-8B**. This is often referred to as a *press fit* or a *force fit*.

Transition fit

In a ***transition fit***, the limits of size are defined so that the result may be either a clearance fit or an interference fit. For example, the smallest shaft size allowed by the shaft tolerance will fit within the largest hole size allowed by the hole tolerance and a clearance will result. However, the largest shaft size allowed by the shaft tolerance will interfere with the smallest hole size allowed by the hole tolerance. The two mating parts will have to be "pressed" together, **Figure 16-8C**.

Basic Size Systems

In the design of mating cylindrical parts, it is necessary to assume a basic size for either the hole or shaft. The design sizes of mating parts are then calculated by applying an allowance to this basic size. Manufacturing costs usually determine which mating part becomes the standard size.

If standard tools can be used to produce the holes, the basic hole size system is used. However, if a machine or an assembly requires several different fits on a cold-finished shaft, the basic shaft size is most economical in manufacturing and is the system used. When standard parts (such as ball bearings) are inserted in castings, the basic shaft size system also applies.

Basic hole size system

In the ***basic hole size system***, the basic size of the hole is the design size, and the allowance is applied to the shaft. The ***basic hole size*** is the minimum hole size produced by standard tools, such as reamers and broaches. Allowances and tolerances are specified for this basic or design size to produce the type of fit desired.

An example of a basic hole size is shown in **Figure 16-9A**. In this example, the basic hole size is the minimum size, .500". An allowance of .002" is subtracted from the basic hole size and applied to the shaft for clearance, providing a maximum shaft size of .498". A tolerance of +.002" is then

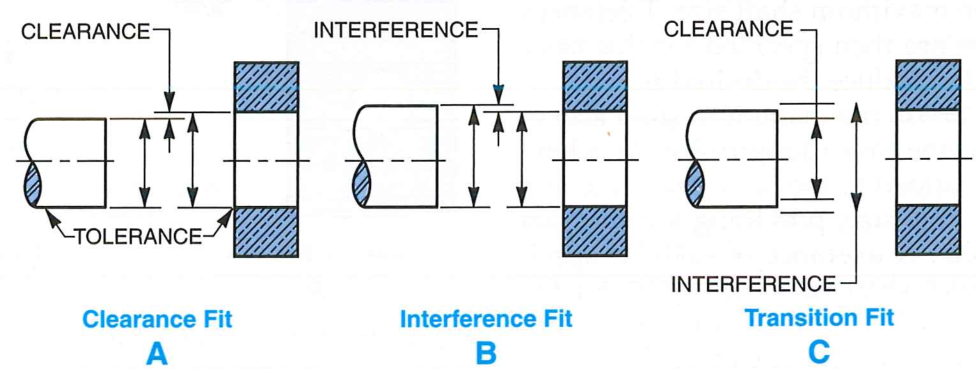

Figure 16-8. Types of fit used in the design of parts. A—A clearance fit is one where a gap, or air space, occurs between two mating parts. B—An interference fit occurs when the part being inserted is larger than the opening that it is being inserted into. C—A transition fit occurs when the dimensions and tolerances are specified so that the mating parts might fit as an interference fit or a clearance fit, depending on how the two fit in the tolerance range.

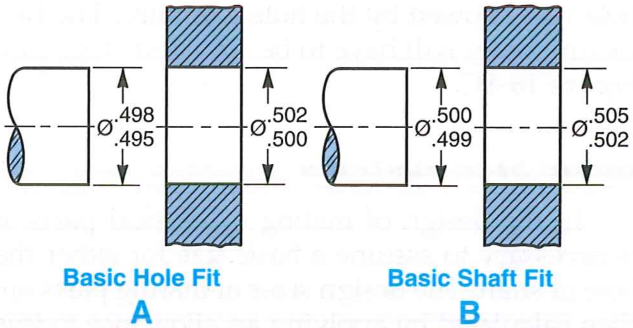

Figure 16-9. Design examples using the basic hole size and basic shaft size systems. A—In the basic hole size system, the size of the hole is the design size. Allowances are applied to the size of the shaft during the design phase. B—In the basic shaft size system, the size of the shaft is the design size and allowances are applied to the hole size during the design phase.

applied to the basic hole size and –.003″ to the shaft size to provide a maximum hole size of .502″ and a minimum shaft size of .495″.

The tightest fit (minimum clearance) is .500″ (smallest hole size) – .498″ (largest shaft size) = .002″. The fit giving maximum clearance is .502″ (largest hole size) – .495″ (smallest shaft size) = .007″.

These examples have provided a clearance fit in mating parts. To obtain an interference fit in the basic hole size system, add the allowance to the basic hole size and assign this value as the largest shaft size.

Basic shaft size system

When the ***basic shaft size system*** is used, the design size of the shaft is the basic size and the allowance is applied to the hole. The basic shaft size is the maximum shaft size. Tolerances and allowances are then specified for this basic or design size to produce the desired fit.

In **Figure 16-9B**, the maximum shaft size of .500″ is taken as the basic (design) size. An allowance of .002″ is added to the basic shaft size and applied to the hole size, providing a minimum hole size of .502″. A tolerance of –.001″ is specified for this basic shaft size and +.003″ for the hole size. This provides a minimum shaft size of .499″ and a maximum hole size of .505″.

The minimum clearance provided is .502″ (smallest hole size) – .500″ (largest shaft size) = .002″. The maximum clearance provided is .505″ (largest hole size) – .499″ (smallest shaft size) = .006″.

These examples have provided a clearance fit in the mating parts. Interference fits may be obtained in the basic shaft size system by subtracting the allowance from the basic shaft size and assigning this value as the minimum hole size.

Maximum Material Condition (MMC)

The ***maximum material condition (MMC)*** is present when the feature contains the maximum amount of material. MMC exists when internal features, such as holes and slots, are at their minimum size, **Figure 16-10A**. MMC also occurs when external features, such as shafts and bosses, are at their maximum size, **Figure 16-10B**.

MMC is applied to the individual tolerance, datum reference, or both. The position or form tolerance increases as the feature departs from MMC by the amount of such departure. In other words, if the feature that controls MMC varies (a standard shaft, for example), the position tolerance of the feature will change (the mating hole, for example).

Least Material Condition (LMC)

Least material condition (LMC) is present when the feature contains the least amount of material within the tolerance range. LMC exists when holes are at maximum size. LMC also occurs when shafts are at minimum size.

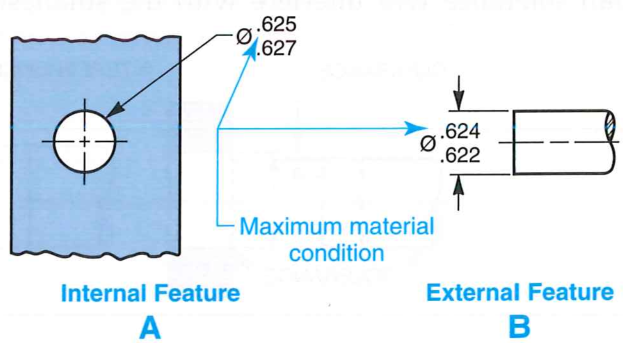

Figure 16-10. The maximum material condition (MMC) occurs when the feature has the most material possible while still staying within the tolerance. A—For an internal feature, MMC occurs when the feature is at the lower limit. B—For an external feature, MMC occurs when the feature is at the upper limit.

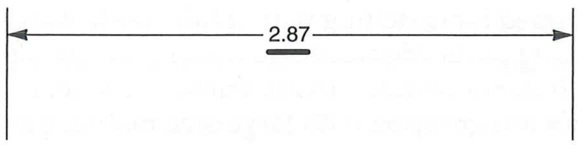

Figure 16-11. A not-to-scale dimension is indicated by a straight, thick line underneath the dimension that is not to scale.

Just as with MMC, if the LMC of a controlling feature varies, the position tolerance of the mating part will change as well. The variance of the position tolerance will be equal to the amount of variance in LMC.

Regardless of Feature Size (RFS)

Regardless of feature size (RFS) means that geometric tolerances or datum references must be met no matter where the feature lies within its size tolerance. Where RFS is applied to a positional or form tolerance, the tolerance must not be exceeded regardless of the actual size of the feature. RFS applies with respect to the individual tolerance, datum reference, or both, where no symbol is specified. RFS is assumed on all dimensions unless otherwise indicated.

Not-to-Scale Dimensions

All drawings (with the exception of diagrammatic and schematic drawings) should be drawn to scale. However, on a drawing revision, the correction of a dimension that is drawn to scale may require an excessive amount of drafting. If the drawing remains clear, the dimension can be changed and underlined with a thick, straight line to indicate a not-to-scale dimension, **Figure 16-11**.

Original drawings should not be issued with out-of-scale dimensions. These dimensions should be kept to an absolute minimum on revisions. Where there is the slightest chance of misinterpretation of an out-of-scale dimension on a revised drawing, the drawing should be redrawn.

Application of Tolerances

When manufacturing items requiring interchangeability of parts, tolerancing of all dimensions is required. Exceptions are basic dimensions, reference dimensions, and single-limit dimensions (dimensions labeled "MAX" or "MIN").

Tolerances are normally expressed in the same number of decimal places as the dimension. Tolerances are applied to dimensions using either limit dimensioning or plus and minus tolerancing. These methods and other standard tolerancing practices are discussed in the following sections.

Limit Dimensioning

In ***limit dimensioning***, only the maximum and minimum dimensions are given, **Figure 16-12**. Use one of the following methods to arrange the limit numerals.

1. For positional dimensions given directly, the maximum (high) limit is always placed above the minimum (low) limit, **Figure 16-12A**. For positional dimensions given in note form, the minimum limit always precedes the maximum limit, **Figure 16-12B**.

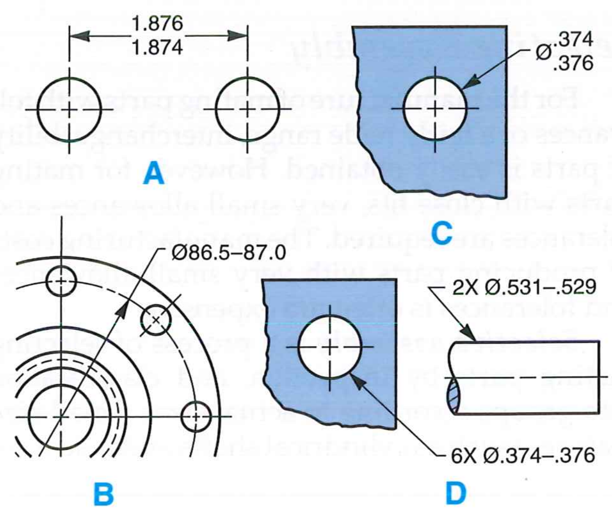

Figure 16-12. Only the maximum and minimum dimensions are given when using limit dimensioning. A—When the limit dimensions are given directly, the maximum dimension always appears above the minimum dimension. B—When the limit dimensions are given as a note, the minimum dimension always comes before the maximum dimension. C—When a size dimension is given directly, the dimension representing MMC is given above the dimension representing LMC. D—For size dimensions given in note form, the dimension representing MMC precedes the dimension representing LMC.

2. For size dimensions given directly, the number representing the maximum material condition (MMC) is placed above the number representing the least material condition, **Figure 16-12C**. For size dimensions given in note form, the MMC number precedes the LMC number, **Figure 16-12D**.

Plus and Minus Tolerancing

In *plus and minus tolerancing*, the tolerances are generally placed to the right of the specific dimension. The tolerances are designated as a number with "stacked" plus and minus signs before it. This expression is the allowed variation of the size or location of the feature. Refer to **Figure 16-6B**.

Calculating Tolerances

The tolerances of a drawing are checked to see that all parts will assemble as specified. Two methods of calculating the tolerance between two features are shown in **Figure 16-13**.

Selective Assembly

For the manufacture of mating parts with tolerances of a fairly wide range, interchangeability of parts is easily obtained. However, for mating parts with close fits, very small allowances and tolerances are required. The manufacturing costs of producing parts with very small allowances and tolerances is often too expensive.

Selective assembly is a process of selecting mating parts by inspection, and classification into groups according to actual sizes. Small-size features (such as cylindrical shafts and holes) are grouped for matching with small-size features of mating parts. Medium-size features are grouped with medium-size mating parts. Large-size features are grouped with large-size mating parts.

The cost of manufacturing is reduced considerably in selective assembly due to less restriction on allowances and tolerances. This method is usually more satisfactory than interchangeable assembly in achieving transition fit mating parts.

Tolerance Accumulation

An accumulation of tolerances occurs when consecutive features on a part are dimensioned and a "buildup" of tolerances results. This can cause an excessive amount of variation between features because the individual dimensions are controlled by more than one tolerance. Depending on the dimensioning system used, different results occur in the accumulation of tolerances. As discussed in Chapter 9, datum dimensioning is more suitable than chain dimensioning for parts requiring greater accuracy in manufacturing (such as mating parts). The effects of these two dimensioning systems on tolerance accumulation are discussed next. See **Figure 16-14**.

Chain dimensioning

Chain dimensioning is the dimensioning of a series of features, such as holes, from point to point, **Figure 16-14A**. When chain dimensions are toleranced, overall variations in the position of features may exceed the tolerances specified, **Figure 16-14B**. The possible variation is equal to the sum of the tolerances on the intermediate dimensions. For example, the variation in

Feature	Normal Method	Graphic Method
HOLE 1.000 $^{+.002}_{-.000}$	1.002 max 1.000 min	1.000 $^{+.002}_{-.000}$
SHAFT .996 $^{+.002}_{-.000}$	.996 min .998 max	.996 $^{+.002}_{-.000}$
Find resultant difference	.006 max .002 min	.004 $^{+.002}_{-.002}$
		+ .006 max − .002 min

Figure 16-13. The normal method and graphic method are two ways of calculating the tolerance between a hole and a shaft. (Sperry Flight Systems Div.)

position between Holes A and B in **Figure 16-14B** ranges from 1.499″ (3.997″ – 2.498″ = 1.499″) to 1.501″ (4.003″ – 2.502″ = 1.501″). This is a difference of .002″ instead of the intended ±.001″.

Chain dimensioning is used on drawings prepared for incremental positioning CNC operations (see Chapter 21). The tolerances are built into the machine and dimensions are given as basic dimensions. Tolerancing of these dimensions would not change the part being machined.

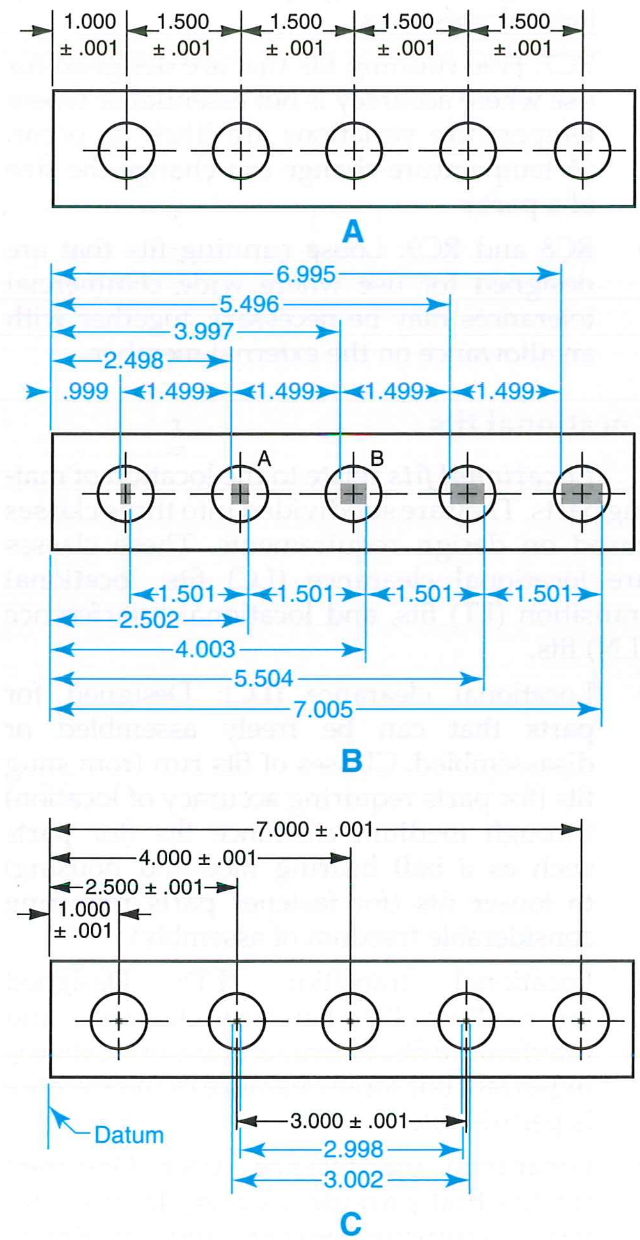

Figure 16-14. Tolerance accumulation in chain dimensioning and datum dimensioning. A—A part dimensioned with chain dimensions. B—In chain dimensioning, error can accumulate even though each dimension stays within tolerance. C—In datum dimensioning, the error is limited by dimensioning every feature of a part from a common point or plane.

Datum dimensioning

In *datum dimensioning*, also called *baseline dimensioning*, features are dimensioned individually from a datum, **Figure 16-14C**. This system of dimensioning avoids accumulation of tolerances from feature to feature. Where the distance between two features must be closely controlled, without the use of an extremely small tolerance, datum dimensioning should be used. Datum dimensioning is also used for absolute positioning CNC operations.

Angular Surface Tolerancing

Angular surface tolerancing uses a combination of linear and angular dimensions, or linear dimensions alone, to dimension and tolerance angular surfaces. A dimension and its tolerance specify a tolerance zone that the surface must lie in, **Figure 16-15A**. The tolerance zone widens as it moves away from the apex of the angle.

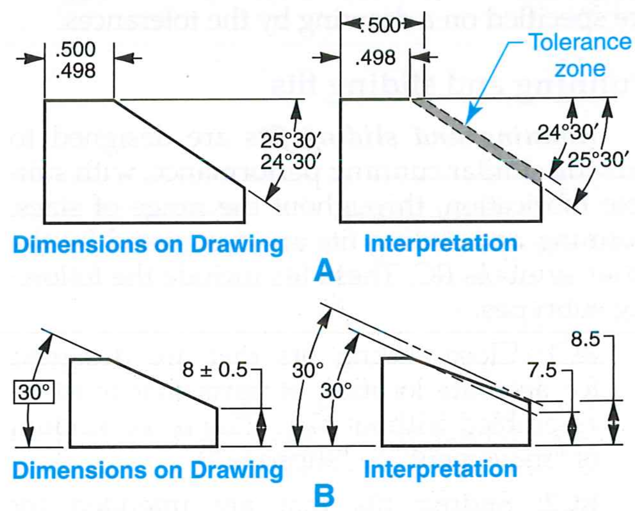

Figure 16-15. Angular surface tolerancing practices. A—An angular surface can be toleranced using a combination of linear and angular dimensions. The tolerance zone in this case will widen as it is moved from the apex of the angle. B—If a tolerance zone with parallel sides is required, a basic angle dimension should be specified.

Where a tolerance zone with parallel boundaries is desired, a basic angle may be specified as in **Figure 16-15B**. By specifying a basic angle, a tolerance zone with parallel sides is indicated. The surface controlled must lie within the tolerance zone. A tolerance zone with parallel sides is useful for parts that have long angular surfaces.

Selection of Fits

Tables listing recommended tolerances for fits and sizes have been developed by the American National Standards Institute (ANSI). The designer or drafter should refer to these tables when it is necessary to select tolerances for a specific size feature and mating part. The use required of a piece of equipment determines the limits of size of mating parts and the selection of type of fit.

Standard Fits

A number of types and classes of fits are given in the Reference Section of this textbook. Any fit of mating parts will usually be required to perform one of three functions: running or sliding fit, locational fit, or force fit. These fits are further divided into classes and assigned letter symbols. Fits are not indicated on a drawing. Fits are specified on a drawing by the tolerances.

Running and sliding fits

Running and sliding fits are designed to provide similar running performance, with suitable lubrication, throughout the range of sizes. Running and sliding fits are designated by the letter symbols RC. These fits include the following subtypes:

- RC1: Close sliding fits that are designed for accurate location of parts that must be assembled without play. (*Play* is the amount of "movement" or "slippage.")
- RC2: Sliding fits that are intended for accurate location but with greater maximum clearance than RC1. Parts move and turn easily, but are not intended to run freely.
- RC3: Precision running fits that are the closest fits that can be expected to run freely at slow speeds and light journal (shaft) pressure.
- RC4: Close running fits that are designed to run freely on accurate machinery with moderate surface speeds and journal pressures. These are designed for use when accurate location and minimum play are needed.
- RC5 and RC6: Medium fits that are designed for higher running speeds and/or heavy journal pressures.
- RC7: Free running fits that are designed for use where accuracy is not essential or where temperature variations are likely to occur. (A temperature change can change the size of a part.)
- RC8 and RC9: Loose running fits that are designed for use where wide commercial tolerances may be necessary, together with an allowance on the external member.

Locational fits

Locational fits relate to the location of mating parts. They are subdivided into three classes based on design requirements. These classes are locational clearance (LC) fits, locational transition (LT) fits, and locational interference (LN) fits.

- Locational clearance (LC): Designed for parts that can be freely assembled or disassembled. Classes of fits run from snug fits (for parts requiring accuracy of location) through medium clearance fits (for parts such as a ball bearing race and housing) to looser fits (for fastener parts requiring considerable freedom of assembly).
- Locational transition (LT): Designed for medium fits, between clearance and interference fits, where accuracy of location is important but some clearance or interference is permissible.
- Locational interference (LN): Designed for fits that provide accurate location for parts requiring rigidity and alignment with no special requirements for bore pressure. These fits are not intended for parts designed to transmit frictional loads from one part to another by virtue of tightness of fit (such conditions are covered by force fits).

Force fits

Force fits or ***shrink fits*** are special interference fits normally characterized by constant bore pressures throughout the range of sizes. The interference varies almost directly with the diameter. These fits are also called ***press fits***. They are designated by the letter symbols FN and include the following subtypes:

- FN1: Light drive fits that require light assembly pressures and produce more or less permanent assemblies. They are used for thin sections or long fits, or in cast iron external members.
- FN2: Medium drive fits that are designed for ordinary steel parts, or for shrink fits on light sections. They usually are the tightest fits that can be used with high-grade cast iron external members.
- FN3: Heavy drive fits that are suitable for heavier steel parts or for shrink fits in medium sections.
- FN4 and FN5: Force fits that are designed for parts that can be highly stressed, or for shrink fits where heavy pressing forces required are impractical.

Geometric Dimensioning and Tolerancing

Geometric dimensioning and tolerancing is a system of dimensioning drawings with emphasis on the actual function and relationship of part features. This system is used where interchangeability is critical. This system does not replace the coordinate dimensioning system. Geometric dimensioning and tolerancing is used in conjunction with coordinate dimensioning. Tolerances applied in the geometric and tolerancing system do not imply tighter tolerances. Rather, the system permits the use of maximum tolerances while maintaining 100% interchangeability.

Geometric dimensioning and tolerancing has become the system used by most industries because of the clarity and preciseness in communicating specifications. Every drafter, designer, and engineer should understand its use.

Geometric Characteristic Symbols

Geometric characteristic symbols for tolerances reduce the number of notes required on a drawing. These symbols are compact, recognized internationally, and designed to reduce misinterpretation.

The standard symbols used for geometric characteristics of part features are shown in **Figure 16-16**. As shown in the table, these symbols are used to convey information in relation to form and positional tolerances. The symbols explain information relating to characteristics such as the form of an object, the profile (or outline) of an object, the orientation of features, the location of features, and the runout of surfaces. Modifying symbols and other dimensioning symbols used in the geometric dimensioning and tolerancing system are shown in **Figure 16-17**. The meanings and applications of these symbols are discussed later in this chapter.

In manual drafting, drawing templates are available for use in drawing the symbols

Geometric Characteristic Symbols

TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL
FORM	Straightness	⏤
	Flatness	⏥
	Circularity (roundness)	○
	Cylindricity	⌭
PROFILE	Profile of a line	⌒
	Profile of a surface	⌓
ORIENTATION	Angularity	∠
	Perpendicularity	⊥
	Parallelism	//
LOCATION	Position	⌖
	Concentricity	◎
	Symmetry	⌯
RUNOUT	Circular runout	↗*
	Total runout	⌰*

* Arrowheads may be filled or not filled.

Figure 16-16. Geometric characteristic symbols used in geometric dimensioning and tolerancing. (American Society of Mechanical Engineers)

Modifying Symbols	
TERM	**SYMBOL**
At maximum material condition	Ⓜ
At least material condition	Ⓛ
Regardless of feature size	NONE
Projected tolerance zone	Ⓟ
Diameter	Ø
Spherical diameter	SØ
Radius	R
Spherical radius	SR
Arc length	⌒105
Between	↔
Datum target	06 A1
Target point	×
Dimension origin	⌖→
All-around	⊖
Conical taper	▷
Slope	◺
Counterbore/spotface	⌴
Countersink	⌵
Depth/deep	↧
Square (shape)	□

Figure 16-17. Modifying symbols used in geometric dimensioning and tolerancing. (American Society of Mechanical Engineers)

for geometric dimensioning and tolerancing. The template in **Figure 16-18** shows geometric characteristic symbols, symbols for specifying surface characteristics, and a complete alphabet. In CAD drafting, some programs provide special dimensioning commands that can be used to insert geometric dimensioning and tolerancing symbols automatically. This is discussed later in this chapter.

Figure 16-18. Templates for use with geometric dimensioning and tolerancing contain standard drawing symbols. (Alvin & Co.)

Datum Feature Symbol

As discussed earlier in this chapter, a datum is identified on a drawing by a datum feature symbol. The symbol consists of a reference letter (any letter except "I," "O," or "Q") enclosed in a square frame connected to a triangle, **Figure 16-19A**. Where more than one datum is used on a drawing, the desired order or precedence of datums is shown from left to right in the feature control frame, **Figure 16-19B**.

Datum Targets

It is not always practical to identify an entire feature as a datum feature. For example, a very large feature might not be practical to use as a datum feature. A ***datum target*** is used when the whole feature is not to be used as a datum feature. The types of targets that may be used are points, lines, and areas of a surface.

Material Condition Symbols

The application of material condition symbols to a drawing is limited to features subject

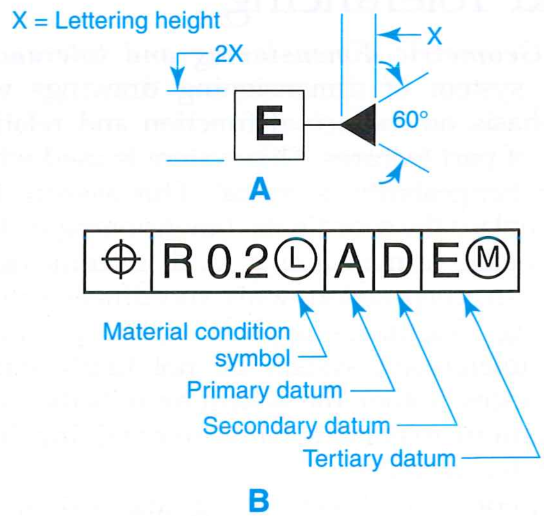

Figure 16-19. A—The elements making up a datum feature symbol. B—Datum reference letters are given in the desired order of precedence in the feature control frame. (American Society of Mechanical Engineers)

to variations in size. The particular feature may be any feature or datum feature whose axis or center is controlled by geometric tolerances. Three different material conditions are used: ***maximum material condition (MMC), least material condition (LMC),*** and ***regardless of feature size (RFS).*** For individual tolerances, MMC or LMC must be specified on the drawing datum reference, as applicable. RFS is always assumed unless MMC or LMC is specified. The only time that RFS is specified is when a previous drafting standard (a standard earlier than ASME Y14.5M-1994) is specified for use.

The symbol Ⓜ is used to designate the maximum material condition. The symbol Ⓛ specifies the least material condition. Regardless of feature size is assumed when no symbol is given. This means that the tolerance must not be exceeded "regardless of the feature size." These symbols may be used only as modifiers in feature control frames. Refer to **Figure 16-19B.** In notes and dimensions, the abbreviations MMC, LMC, and RFS (not the symbols) should be used. Where MMC, RFS, or LMC is specified, the appropriate symbol follows the specified tolerance and applicable datum reference in the feature control frame. Refer to **Figure 16-19B.**

Feature Control Frame

A ***feature control frame*** is the means by which a geometric tolerance is specified for an individual feature, **Figure 16-20.** The frame is divided into compartments containing, in order from the left, the geometric characteristic symbol followed by the tolerance. Where applicable, the tolerance is preceded by the diameter or radius symbol and followed by an appropriate material condition symbol.

If the tolerance is related to a datum (or multiple datums), the datum reference letter (or letters in the case of multiple datum references) follows in the next compartment. Where applicable, the datum reference letter is followed by a material condition symbol. Datum reference letters are entered in the desired order of precedence, from left to right, and need not be in alphabetical order.

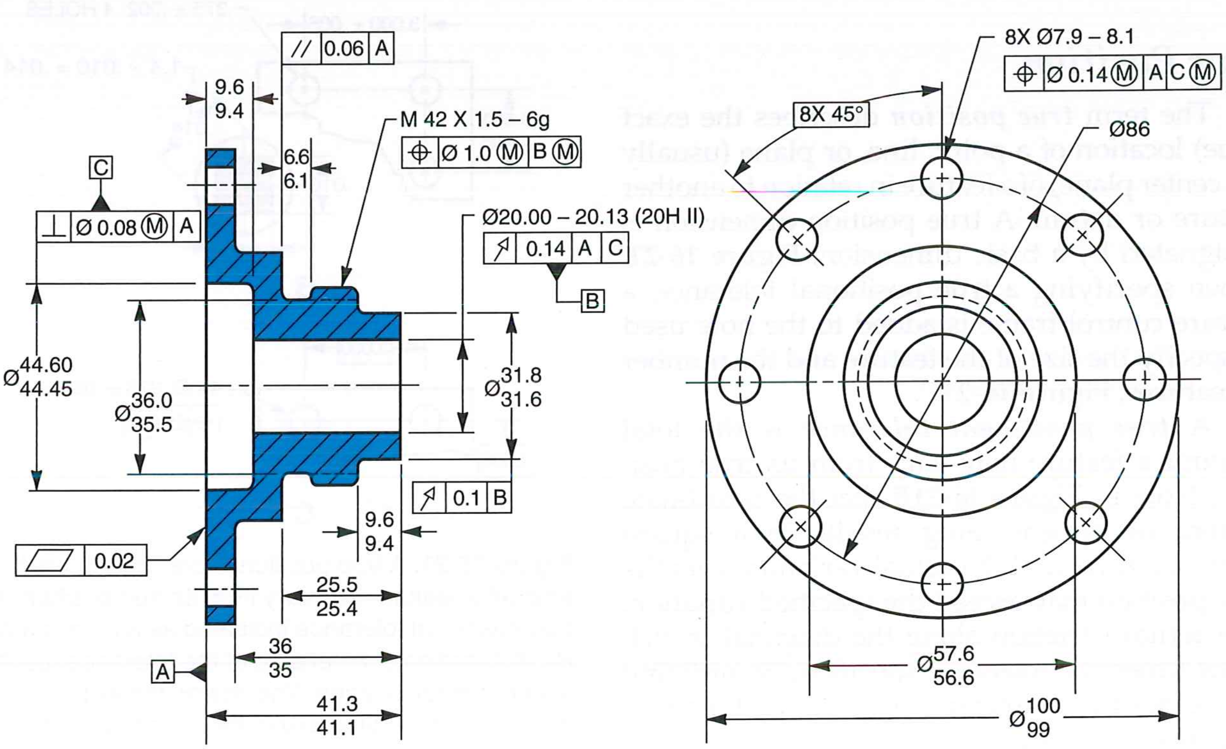

Figure 16-20. Feature control frames specify geometric tolerances for individual features. The frame should be connected to an extension line from the feature, to an extension of the dimension line pertaining to the feature, to a leader running to the feature, or below a leader-directed callout that controls the feature.
(American Society of Mechanical Engineers)

A feature control frame is associated with a feature or features by one of the following methods. Refer to **Figure 16-20**.

1. Attaching a side or end of the frame to an extension line from the feature, provided it is a plane surface.
2. Attaching a side or end of the frame to an extension of the dimension line pertaining to a feature of size.
3. Running a leader from the frame to the feature.
4. Placing the frame below or attached to a leader-directed callout or dimension that controls the feature.

Tolerances of Location

Tolerances assigned to dimensions locating one or more features in relation to other features or datums are known as ***location tolerances*** or ***positional tolerances***. There are three basic types of positional tolerances: true position, concentricity, and symmetry.

True Position

The term ***true position*** describes the exact (true) location of a point, line, or plane (usually the center plane) of a feature in relation to another feature or datum. A true position dimension is designated by a basic dimension, **Figure 16-21**. When specifying a true positional tolerance, a feature control frame is added to the note used to specify the size of the feature and the number of features, **Figure 16-21C**.

A ***true positional tolerance*** is the total amount a feature may vary from its true position. Note in **Figure 16-21B** that the coordinate system of dimensioning results in a square tolerance zone, and the actual variation from the true position may exceed the specified variation. The actual variation along the diagonal is .014, or 1.4 times the tolerance specified. By utilizing a true positional tolerance (where the tolerance zone is a circular zone), the larger tolerance can be specified and interchangeability of parts still maintained, **Figure 16-21C**.

The positional tolerance zone is represented as a circle in one view and is assumed to be a cylindrical zone for the full depth of the hole, **Figure 16-22**. The axis of the hole must be within the tolerance zone.

Projected tolerance zone

A ***projected tolerance zone*** is specified where the variation in perpendicularity of threaded or press-fit holes could cause fasteners such as screws, studs, or pins to interfere with mating parts. The application of a projected tolerance zone to a positional tolerance is shown in **Figure 16-23**. The extent of the projected tolerance zone is indicated in the feature control frame, **Figure 16-23A**. The extent can also be shown as a dimensioned heavy chain line drawn closely to the centerline of the hole, **Figure 16-23B**.

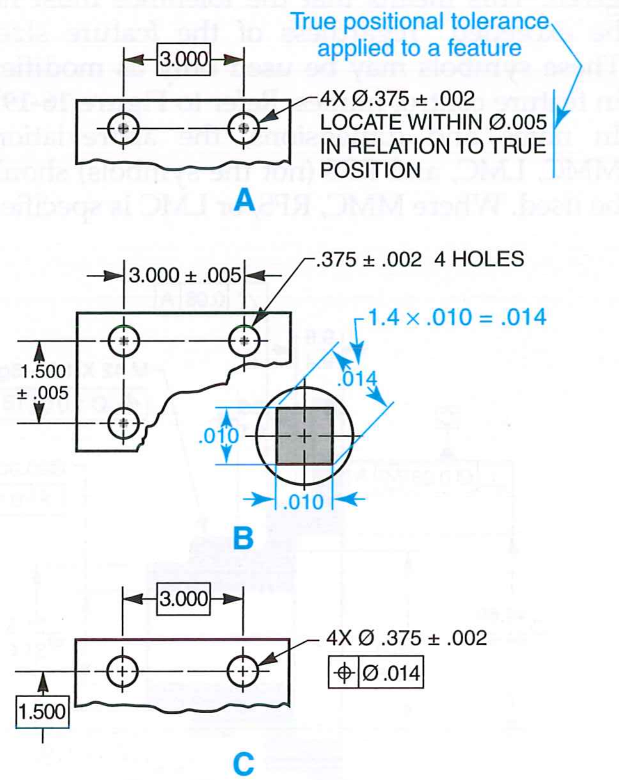

Figure 16-21. A true positional tolerance is the total amount a feature may vary from its true position. A—A true positional tolerance indicated as a note in a callout. B—A coordinate dimension of the tolerance results in a square tolerance zone. The actual variation is equal to the length of the diagonal of the square zone (refer to the detail). This means that the variation from the true position may be outside of the specified tolerance (with the actual variation specified as a circular tolerance zone). C—A feature control frame containing the position symbol indicates the tolerance.

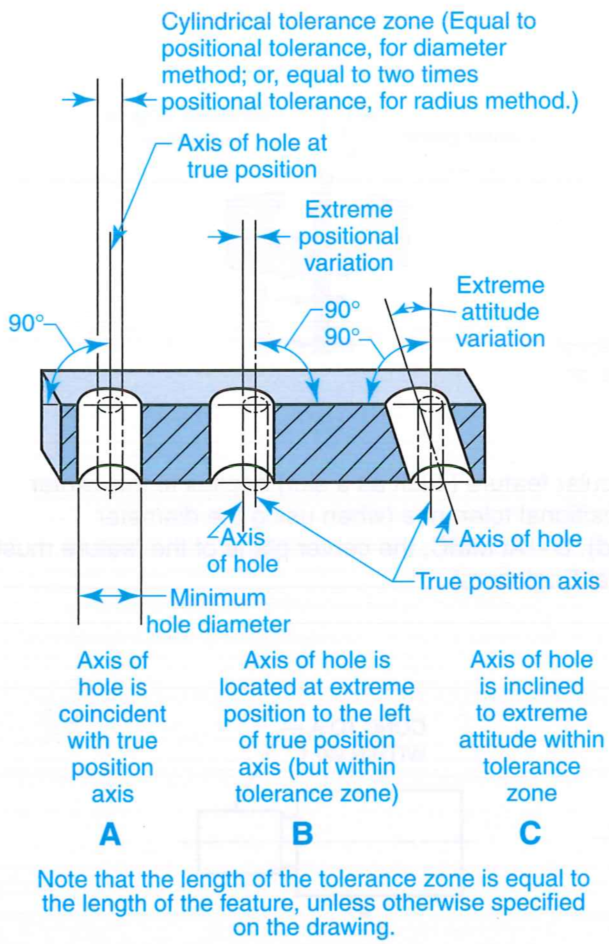

Figure 16-22. The tolerance zone for a hole is assumed to be a cylinder for the full depth of the feature. (American Society of Mechanical Engineers)

True position for noncircular features

Noncircular features such as slots, tabs, and elongated holes may be toleranced for position by using the same basic principles used for circular features. Positional tolerances usually apply only to surfaces related to the center plane of the feature, **Figure 16-24**.

Where the feature is at MMC, its center plane must fall within a tolerance zone having a width equal to the true positional tolerance for the diameter method (or twice the tolerance for the radius method). Note that the tolerance zone also defines the variation limits of the "squareness" of the feature.

Concentricity

Concentricity is the condition of two or more surfaces of revolution having a common axis. A concentricity tolerance callout and interpretation are shown in **Figure 16-25**. In cases where it is difficult to find the axis of a feature (and where control of the axis is not necessary to a part's function), it is recommended that the control be specified as a runout tolerance or a true position tolerance.

Symmetry

Symmetry is a specification for a feature or part having the same contour and size on opposite sides of a central plane or datum feature. A symmetry tolerance may be specified by using a positional tolerance at MMC, **Figure 16-26**. The true position symbol is recommended where a feature is to be located symmetrically about a datum plane and the tolerance is expressed on an MMC or RFS basis, depending upon the design requirements.

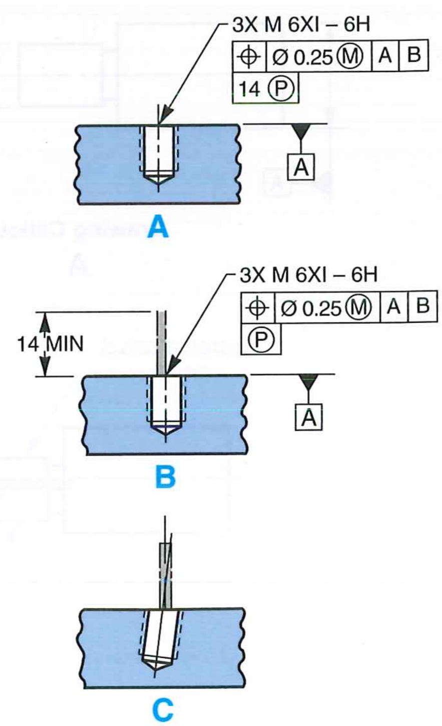

Figure 16-23. A projected tolerance zone indicates the distance above the surface of the part that is critical to a mating feature. A—The projected tolerance zone is indicated using a feature control frame. B—The projected tolerance zone can also be indicated using a combination of a feature control frame and a chain dimension. C—The interpretation of the tolerances specified in A and B. (American Society of Mechanical Engineers)

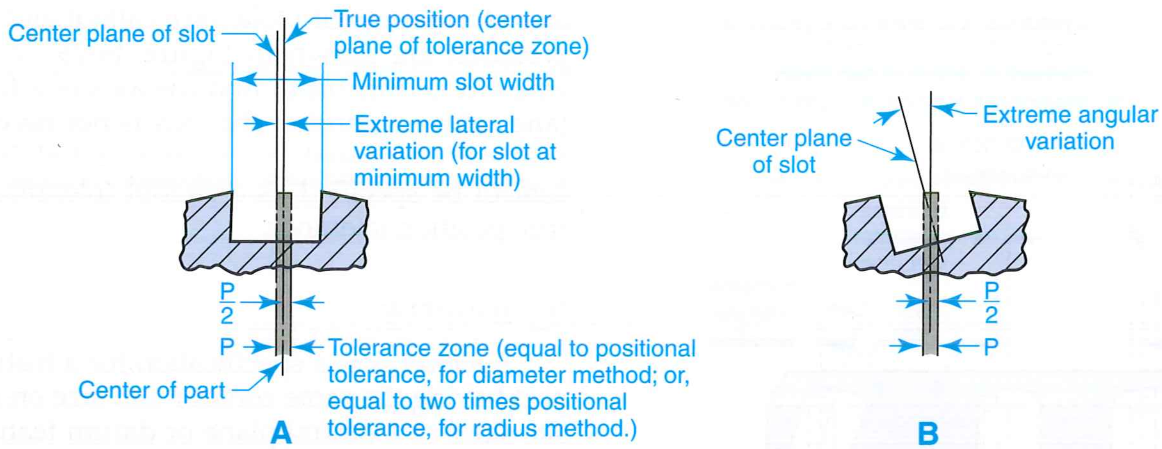

Figure 16-24. The true positional tolerance zone for a noncircular feature (such as a slot) applies to the center plane of the feature. A—The tolerance zone is equal to the positional tolerance (when using the diameter method) or twice the tolerance (when using the radius method). B—At MMC, the center plane of the feature must fall within the tolerance zone. (American Society of Mechanical Engineers)

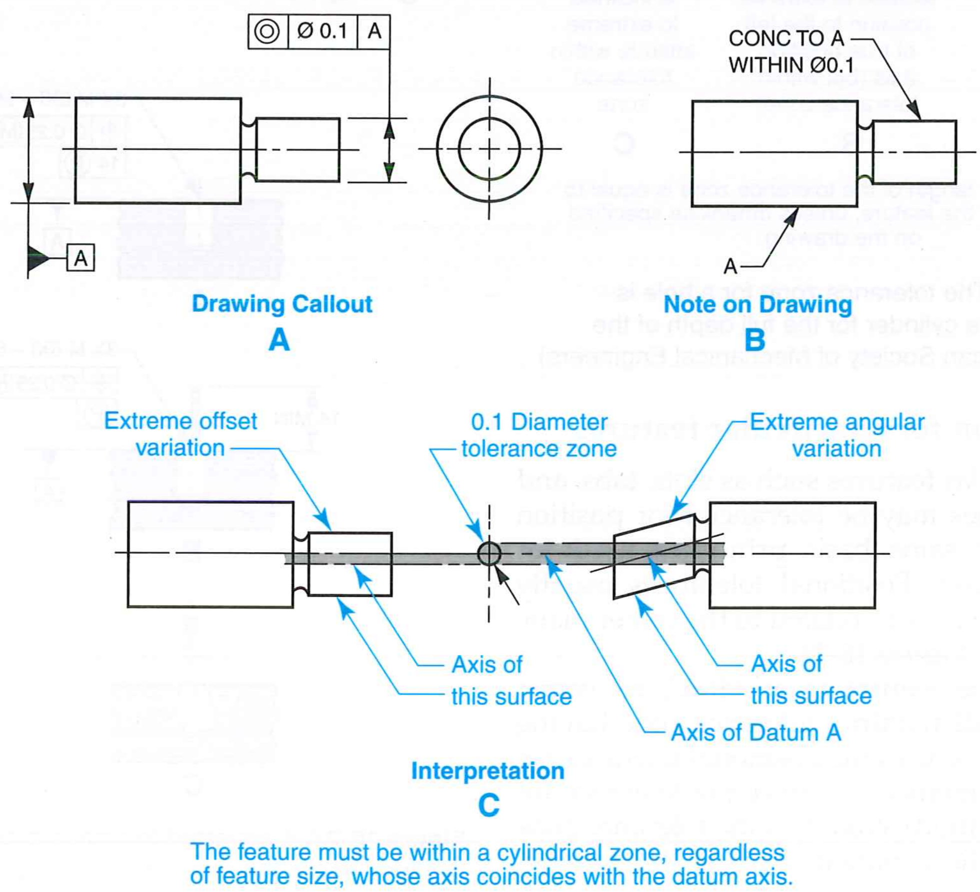

Figure 16-25. Concentricity describes how closely two or more surfaces revolve about a common axis. A—A feature control frame containing the concentricity symbol indicates the tolerance. This is the standard practice. B—A note on the drawing used to indicate concentricity. The note explains the meaning of the drawing callout in A. C—The maximum offset and angular variation that the tolerance allows for the given example. (American Society of Mechanical Engineers)

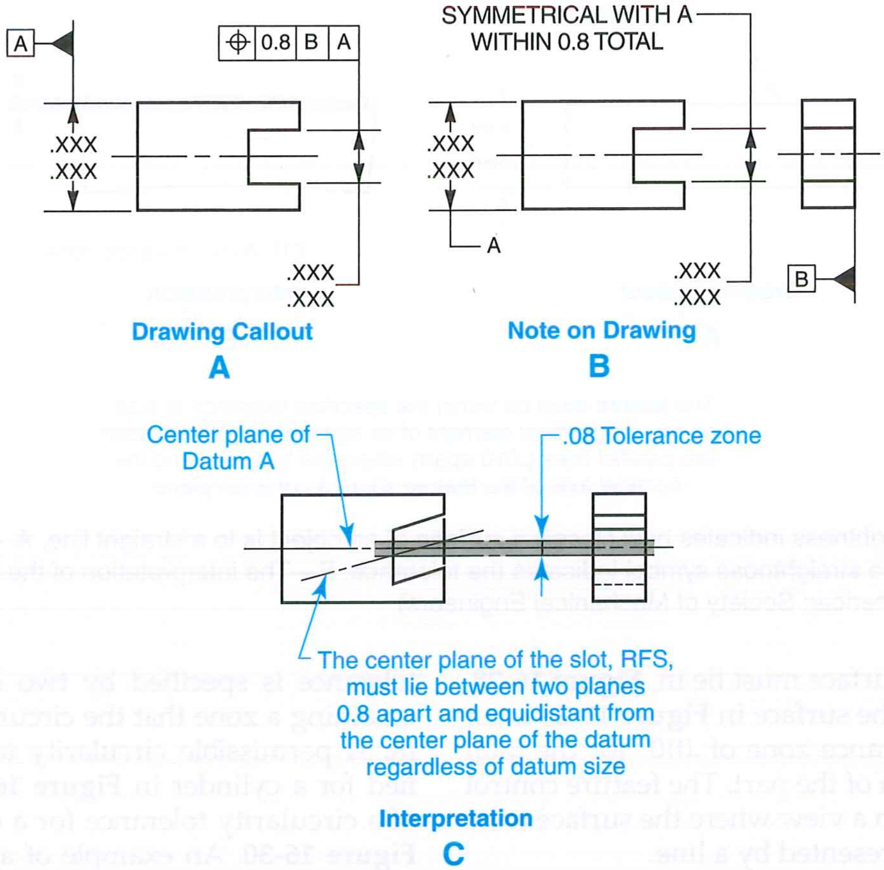

Figure 16-26. Symmetry indicates how closely one side of a feature matches the other side of the feature (about a centerline or a center plane). A—The symmetry may be specified using a positional tolerance. B—A note on the drawing used to indicate symmetry. The note illustrates the meaning of the drawing callout in A. C—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

Tolerances of Form, Profile, Orientation, and Runout

Tolerances can also be used to specify the form, profile, orientation, and runout of features on a part. These tolerances, along with positional tolerances, can be used to fully define every feature of a part, and the parts as a whole.

Form Tolerances

Form tolerances control the forms of geometrical shapes and free-state variations of features. Form tolerances are used to control the conditions of straightness, flatness, circularity (roundness), and cylindricity. A form tolerance specifies a tolerance zone that the particular feature must lie in.

Straightness

Straightness describes how close all elements of an axis or surface are to being a straight line. Straightness is specified by a tolerance zone of uniform width along a straight line within which all elements of the line must lie, **Figure 16-27**. All elements of the feature in **Figure 16-27** must lie within a tolerance zone of .010″ for the total length of the part. Straightness may be applied to control line elements in a single direction or in two directions.

Flatness

Flatness describes how close all elements of a surface are to being in one plane. Flatness is specified by a tolerance zone between two parallel

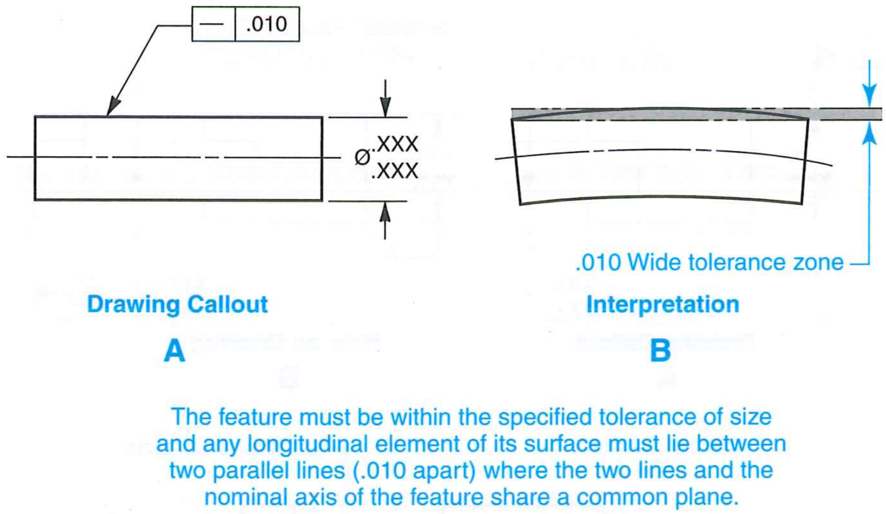

Figure 16-27. Straightness indicates how closely a surface of an object is to a straight line. A—A feature control frame containing the straightness symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

planes that the surface must lie in, **Figure 16-28**. All elements of the surface in **Figure 16-28** must lie within a tolerance zone of .010″ for the total length and width of the part. The feature control frame is placed in a view where the surface to be controlled is represented by a line.

Circularity

Circularity describes the distance from the axis of all the elements of a revolved surface that is intersected by any plane. For a sphere, the intersecting plane passes through a common center. For a cylinder or a cone, the intersection plane is perpendicular to a common axis. A circularity tolerance is specified by two concentric circles confining a zone that the circumference must lie in. A permissible circularity tolerance is specified for a cylinder in **Figure 16-29**. An example of a circularity tolerance for a cone is shown in **Figure 16-30**. An example of a circularity tolerance for a sphere is shown in **Figure 16-31**. Note that the tolerance zone for each of these examples is established by a radius.

Cylindricity

Cylindricity describes the distance of all elements of a surface of revolution from the axis of revolution. If there is perfect cylindricity, all

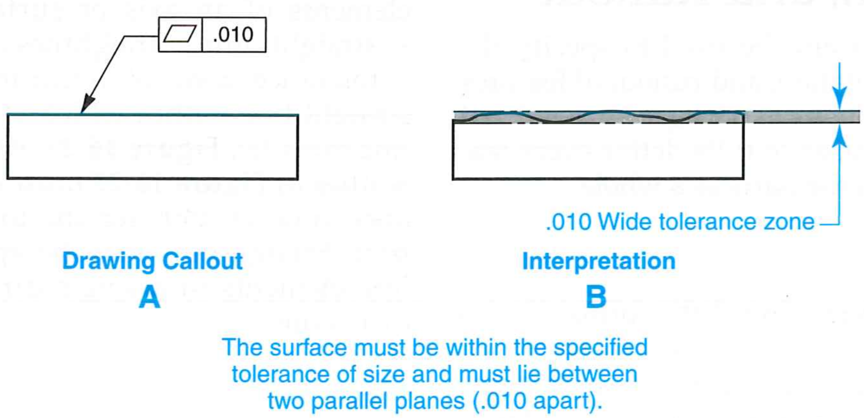

Figure 16-28. Flatness indicates how closely all elements of a surface are to lying in a single plane. A—A feature control frame containing the flatness symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

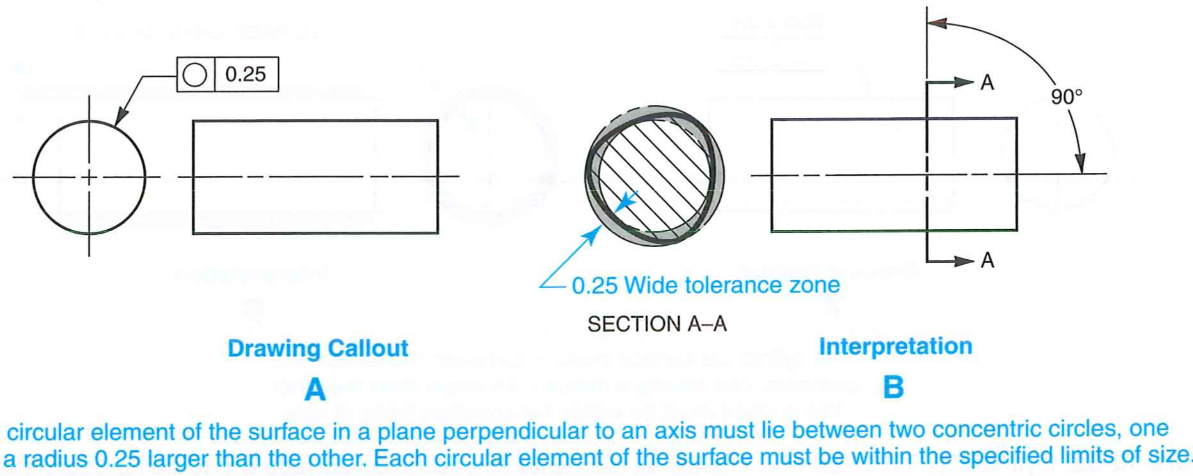

Figure 16-29. Circularity is indicated by a plane cutting a revolved surface perpendicular to the axis of that surface. The circularity tolerance indicates how closely all points of the resulting circle are to being equidistant from the center. Shown is a tolerance specification for the circularity of a cylinder. A—A feature control frame containing the circularity symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

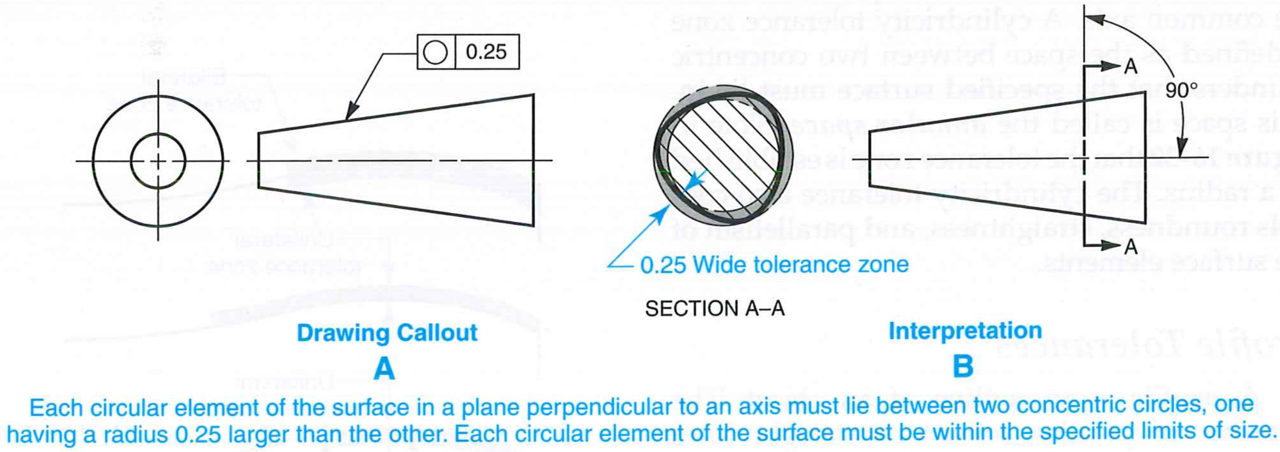

Figure 16-30. A tolerance specification for the circularity of a cone. A—A feature control frame containing the circularity symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

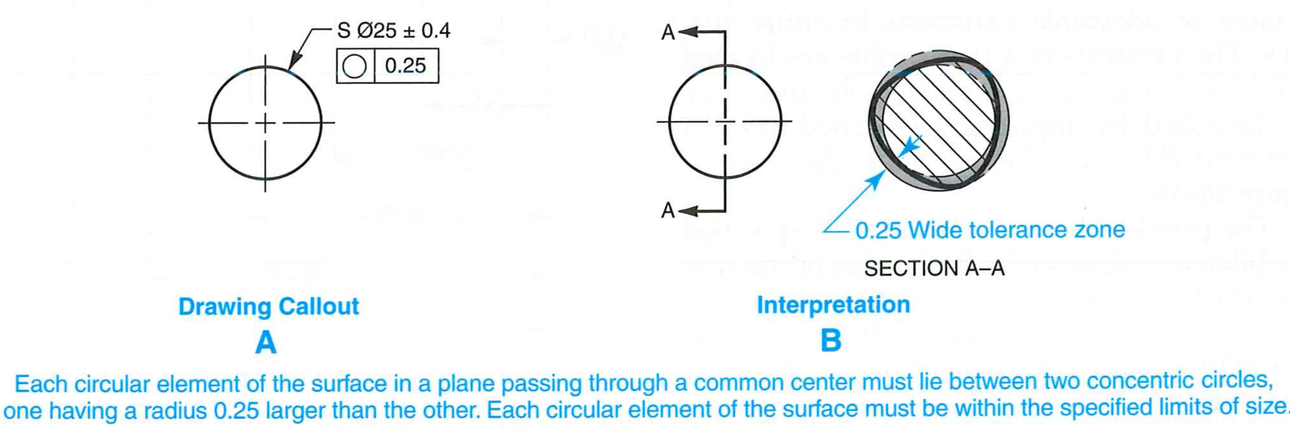

Figure 16-31. A tolerance specification for the circularity of a sphere. A—A feature control frame indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

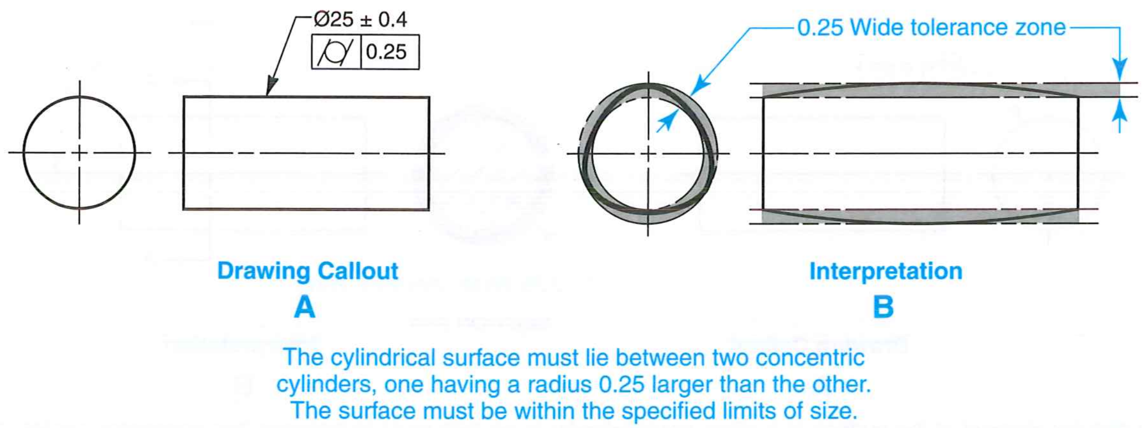

Figure 16-32. Cylindricity indicates how closely all elements of a revolved surface are to being equidistant from a common axis. A—A feature control frame containing the cylindricity symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

elements of the surface are equidistant from the common axis. A cylindricity tolerance zone is defined as the space between two concentric cylinders that the specified surface must lie in. This space is called the ***annular space.*** Note in **Figure 16-32** that the tolerance zone is established by a radius. The cylindricity tolerance also controls roundness, straightness, and parallelism of the surface elements.

Profile Tolerances

A ***profile*** is an outline of an object. The elements of profiles consist of straight lines and curved lines. The curved lines of a profile may be either arcs or irregular curves. ***Profile tolerances*** are used to establish allowable variations in the individual line elements making up surfaces or allowable variations in entire surfaces. The elements of a true profile are located with basic dimensions. A profile tolerance zone is established by applying a specified amount of permissible variation to these dimensions, **Figure 16-33.**

The profile tolerance zone may be specified as a bilateral tolerance (to both sides of the true profile) or a unilateral tolerance (to either side of the true profile). For unilateral tolerances and bilateral tolerances of unequal distribution, the tolerance zone is shown along the profile in a sufficient location by one or two phantom lines parallel to the profile.

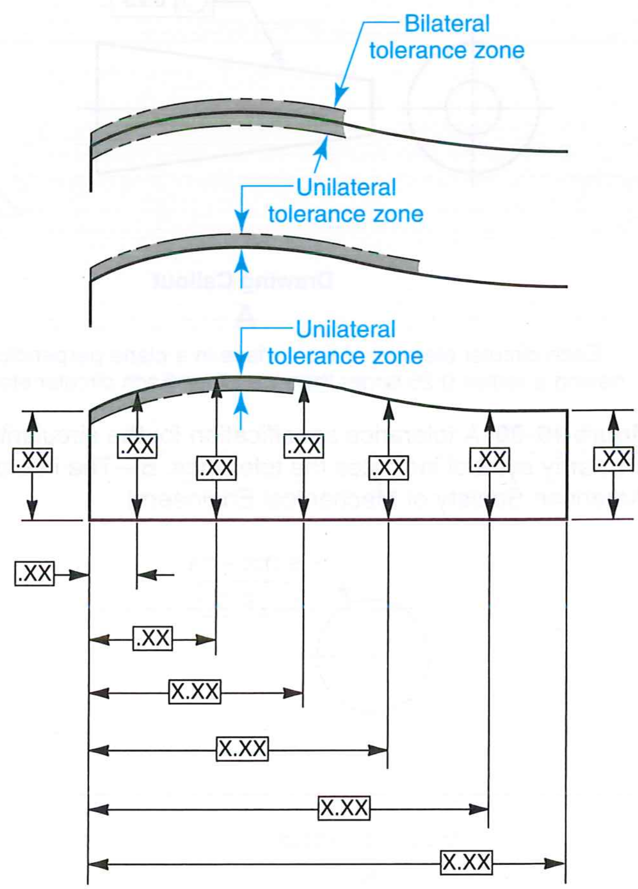

Figure 16-33. Profile tolerance zones indicate how far from the basic dimension a surface can vary. Profile tolerances can be bilateral or unilateral. (American National Standards Institute)

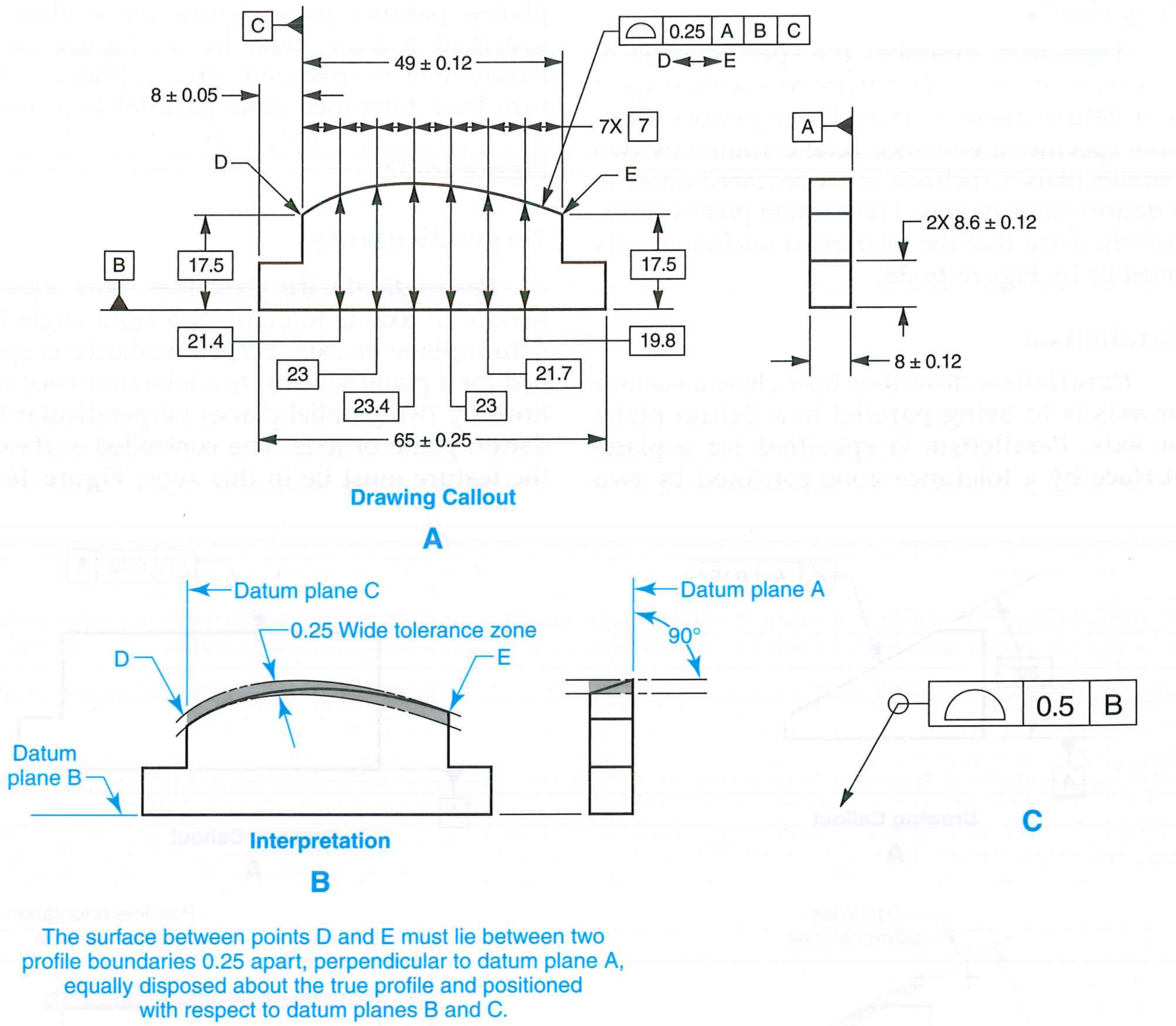

Figure 16-34. A profile tolerance specification for a part. A—A feature control frame containing the profile of a surface symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. C—The all-around profile symbol is used when the profile tolerance applies to surfaces all around the profile. (American Society of Mechanical Engineers)

A dimensional part with a profile tolerance specification is shown in **Figure 16-34**. A profile tolerance for a line is specified in the same manner, except the symbol designating the profile of a line is used. Refer to **Figure 16-16**.

When a profile tolerance applies to surfaces all around the part, a circle is located at the junction of the feature control frame leader, **Figure 16-34C**.

Orientation Tolerances

Orientation describes how a feature of an object "sits" on the object. ***Orientation tolerances*** control the orientation of features in relation to one another. Angularity tolerances, parallelism tolerances, and perpendicularity tolerances all describe the orientation of features.

Angularity

Angularity describes the specific angle of a surface or axis (other than 90°) with respect to a datum plane or axis. An angularity tolerance specifies a tolerance zone confined by two parallel planes, inclined at the required angle to a datum plane or axis. The parallel planes establish the zone that the toleranced surface or axis must lie in, **Figure 16-35**.

Parallelism

Parallelism describes how close a surface or axis is to being parallel to a datum plane or axis. Parallelism is specified for a plane surface by a tolerance zone confined by two planes parallel to a datum plane that the specified feature must lie in, **Figure 16-36**. Parallelism is specified for a cylindrical feature by a tolerance zone parallel to a datum feature axis. The feature must lie in this zone, **Figure 16-37**.

Perpendicularity

Perpendicularity describes how close a surface or axis is to being at a right angle to a datum plane or axis. Perpendicularity is specified for a plane surface by a tolerance zone confined by two parallel planes perpendicular to a datum plane or axis. The controlled surface of the feature must lie in this zone, **Figure 16-38**.

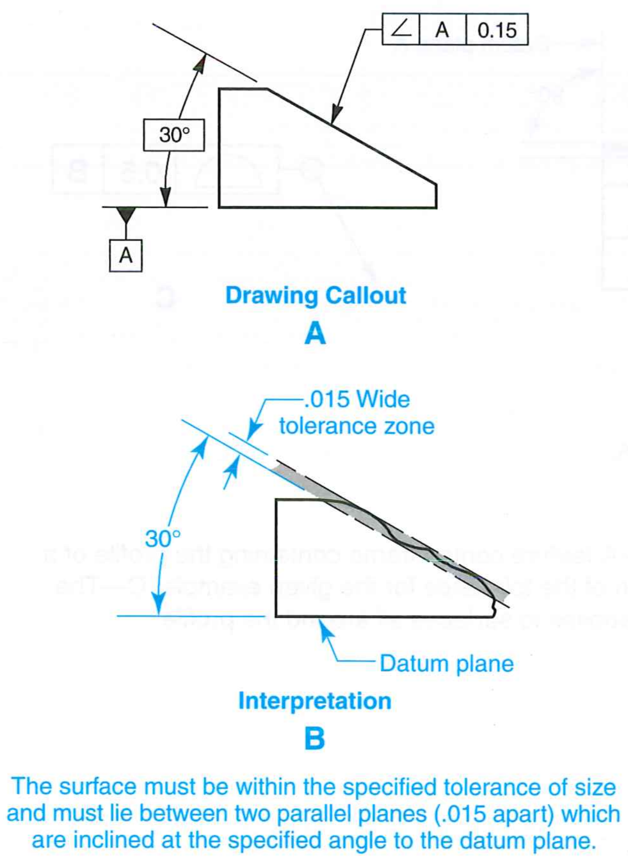

Figure 16-35. Angularity indicates how closely all elements of a surface are to being in a tolerance zone that is inclined at the specified angle. A—A feature control frame containing the angularity symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

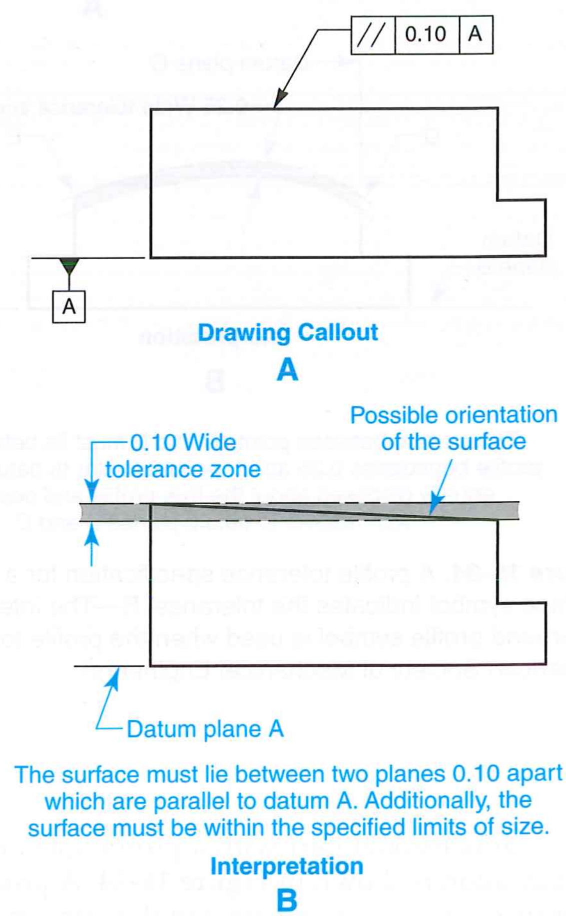

Figure 16-36. Parallelism indicates how closely all elements of a surface are to being equidistant to a given datum surface. A—A feature control frame containing the parallelism symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

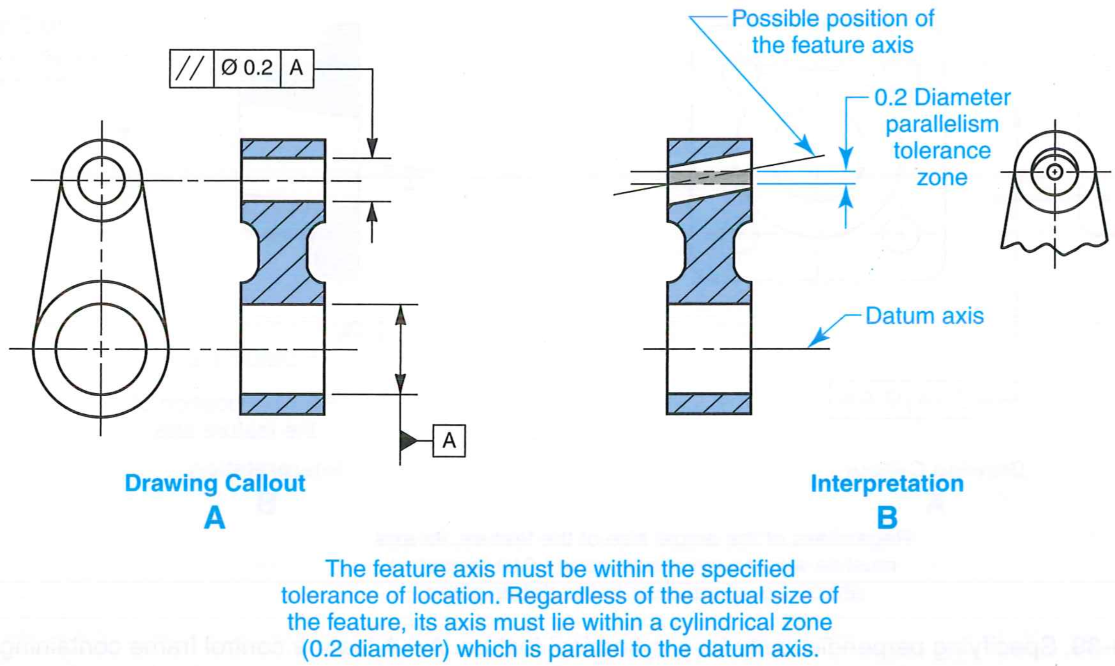

Figure 16-37. Specifying parallelism for a cylindrical feature. A—A feature control frame containing the parallelism symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

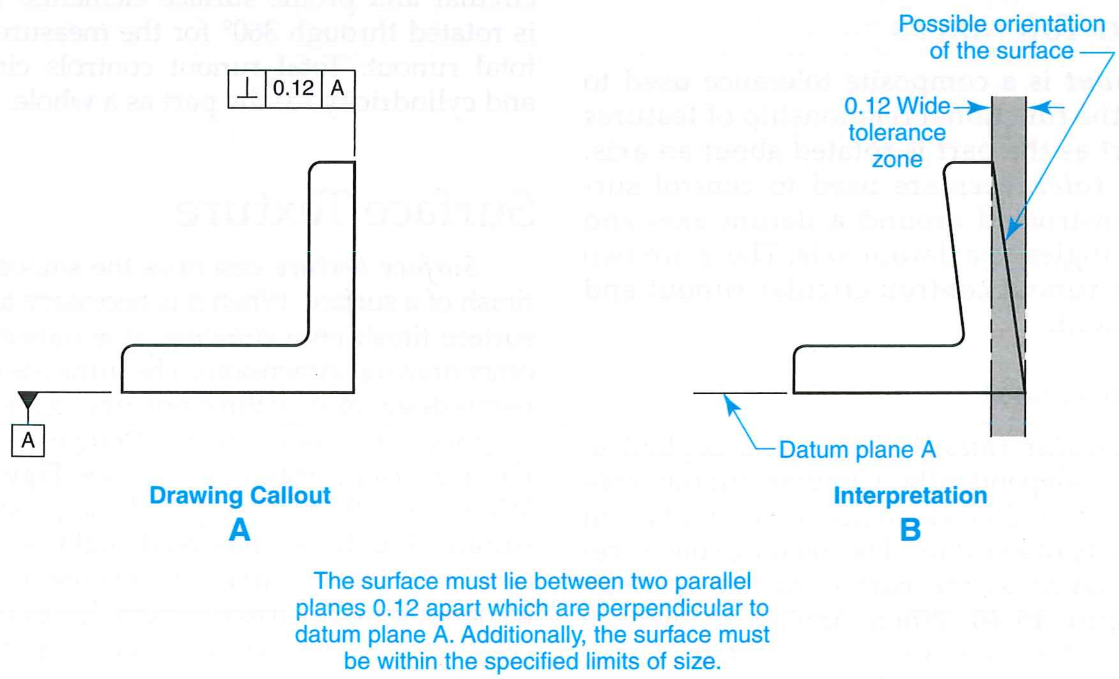

Figure 16-38. Perpendicularity indicates how closely a surface is to being at a right angle to a given datum surface. A—A feature control frame containing the perpendicularity symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

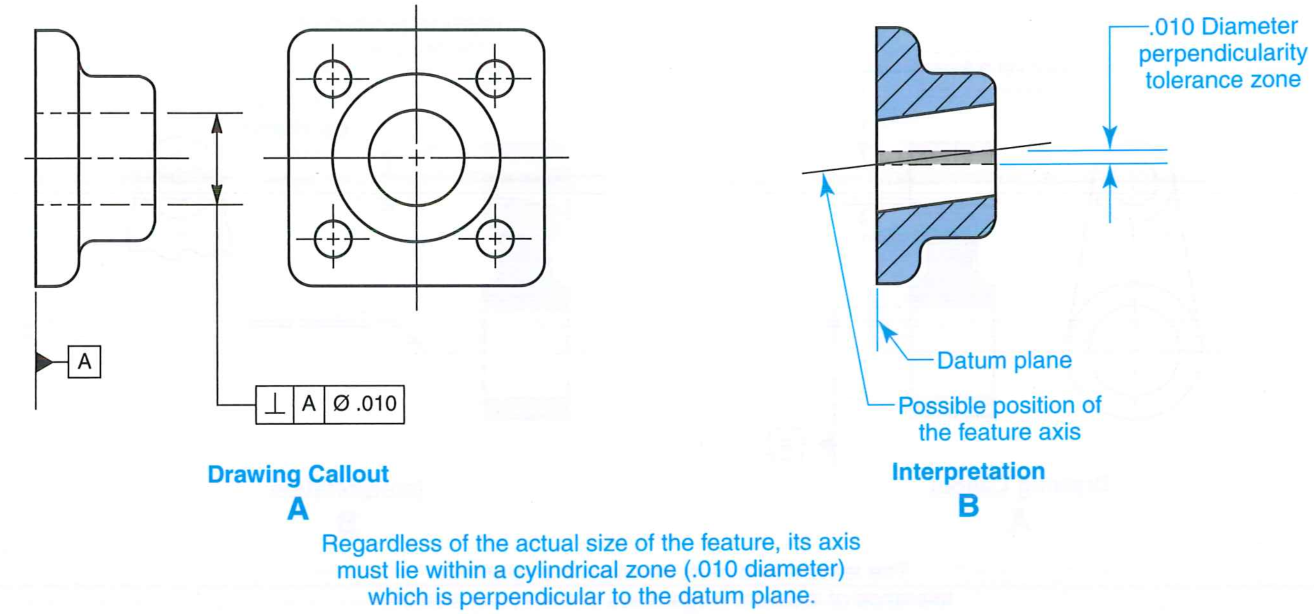

Figure 16-39. Specifying perpendicularity for a cylindrical feature. A—A feature control frame containing the perpendicularity symbol indicates the tolerance. B—The interpretation of the tolerance for the given example. (American Society of Mechanical Engineers)

Specifying perpendicularity for a cylindrical feature is shown in **Figure 16-39**.

Runout Tolerances

Runout is a composite tolerance used to control the functional relationship of features on a part as the part is rotated about an axis. ***Runout tolerances*** are used to control surfaces constructed around a datum axis and at right angles to a datum axis. There are two types of runout control: circular runout and total runout.

Circular runout

A ***circular runout*** tolerance is applied to features independently. Circular runout controls the individual elements of circularity and cylindricity of a surface. The tolerance measurement is taken as the part is rotated through 360°, **Figure 16-40**. Where applied to surfaces constructed at right angles to the datum axis, circular elements of a plane surface (wobble) are controlled. Where the runout tolerance applies to a specific portion of a surface, the extent is shown by a chain line adjacent to the surface profile.

Total runout

A ***total runout*** tolerance is applied to all circular and profile surface elements. The part is rotated through 360° for the measurement of total runout. Total runout controls circularity and cylindricity for the part as a whole.

Surface Texture

Surface texture describes the smoothness or finish of a surface. When it is necessary to specify surface finish on a drawing, it is indicated with other drawing dimensions. The surface texture of a part is designated by three primary characteristics: roughness, waviness, and lay. ***Roughness*** refers to the finer irregularities of a surface, **Figure 16-41**. ***Waviness*** is the widest-spaced component of the surface. Roughness may be thought of as occurring on a "wavy" surface. ***Lay*** is the direction of the predominant surface pattern. For example, the direction may be specified as parallel, perpendicular, or angular to a line representing the surface. When required, these surface characteristics are specified on a drawing using standard symbols. Conventions for applying surface texture symbols are given in the ASME Y14.36M standard.

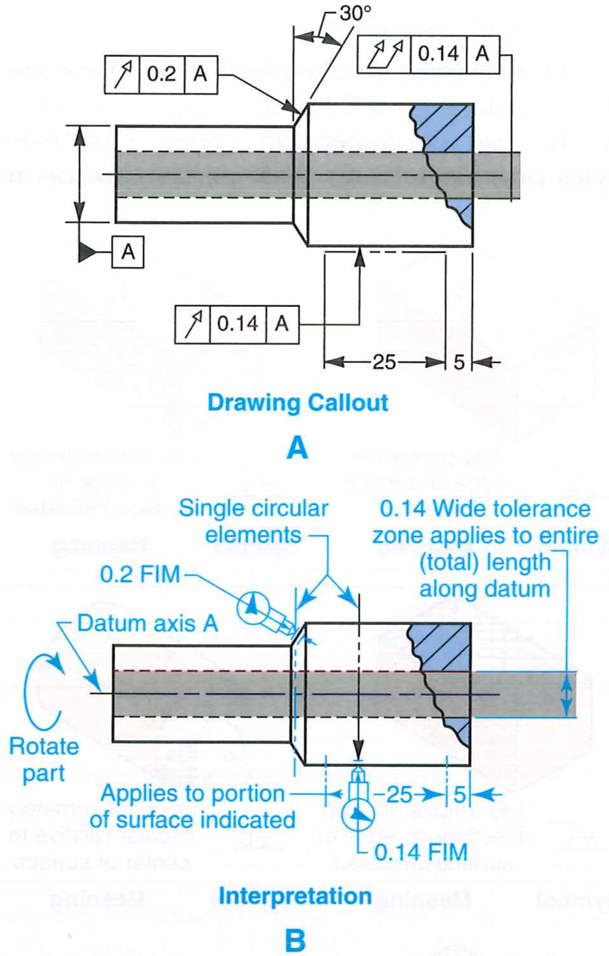

Figure 16-40. Specifying circular runout and total runout tolerances. A—A feature control frame containing the runout symbol indicates the tolerance. B—The interpretation of the tolerances for the given example. When a runout tolerance of 0.2 is given, for example, the full indicator movement (FIM) must not exceed 0.2 as the object is rotated through 360°. (American Society of Mechanical Engineers)

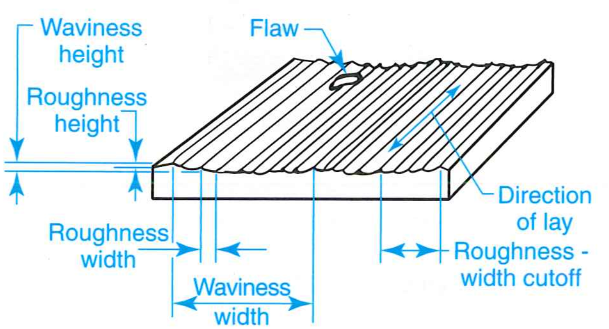

Figure 16-41. Surface texture controls are used to specify classifications of roughness, waviness, and lay.

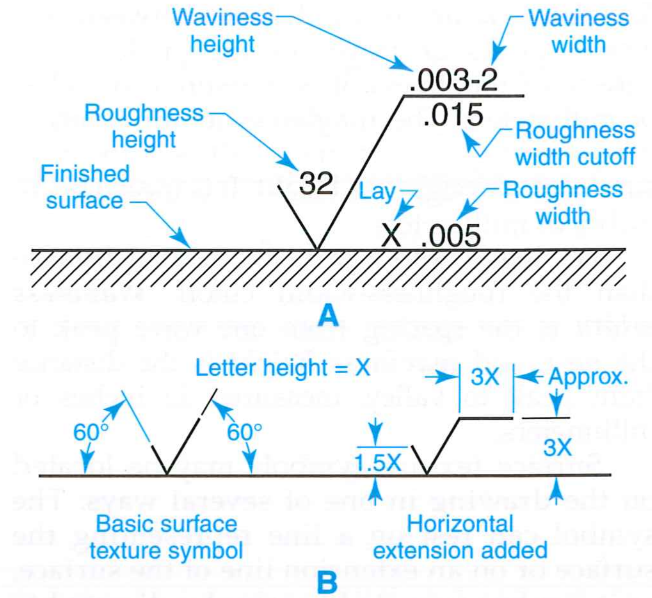

Figure 16-42. Drawing conventions for surface texture symbols. A—Values at specific locations on the symbol designate the surface finish. B—Dimensions for drawing surface texture symbols.

An application of the surface texture symbol and dimensions for creating surface texture symbols are shown in **Figure 16-42.** A basic surface texture symbol is a "V" symbol made up of a pair of inclined lines. The basic symbol is used to designate surface roughness. A horizontal extension bar is added to the symbol where values other than roughness are specified. The symbol is modified when it is necessary to indicate specifications for the removal of material by machining, **Figure 16-43.**

As shown in **Figure 16-41**, surface roughness is measured for height and width. ***Roughness height*** represents the average deviation measured along a nominal centerline. It is designated above the "V" in the surface texture symbol and is measured in microinches or micrometers.

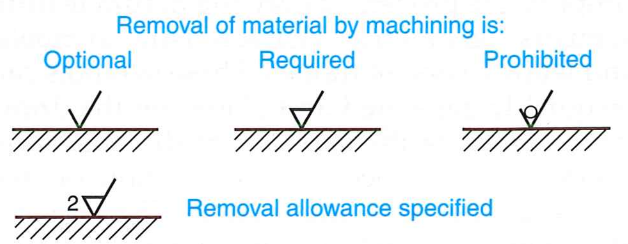

Figure 16-43. Different conventions are used for surface texture symbols when it is necessary to indicate a control for the removal of material by machining.

Roughness width is the distance between successive peaks or ridges of the predominant pattern of roughness. It is measured in inches or millimeters. The ***roughness-width cutoff*** is the greatest spacing of irregularities in the measurement of roughness height. It is measured in inches or millimeters.

Waviness covers a greater horizontal distance than the roughness-width cutoff. ***Waviness width*** is the spacing from one wave peak to the next, and ***waviness height*** is the distance from peak to valley, measured in inches or millimeters.

Surface texture symbols may be located on the drawing in one of several ways. The symbol can rest on a line representing the surface or on an extension line of the surface, or it can be connected to a leader directed to the surface.

Lay symbols are located beneath the horizontal bar on the surface texture symbol. Standard lay symbols for surface controls and their drawing conventions are shown in **Figure 16-44**.

Geometric Dimensioning and Tolerancing on CAD Drawings

Whether drawings are made and dimensioned manually or with a CAD system, the same principles are used when applying geometric dimensioning and tolerancing. One of the most important advantages of using CAD is the ability to place dimensions automatically with dimensioning commands. Some CAD programs provide special dimensioning commands for geometric dimensioning and tolerancing applications. These commands simplify the process of drawing datum feature symbols, geometric characteristic symbols, and feature control frames. These symbols can be quickly generated and placed on the drawing by entering the appropriate dimensioning command and specifying a location on the drawing. The following sections discuss common methods used to dimension CAD drawings using the geometric dimensioning and tolerancing system.

Using Dimension Styles

As discussed in Chapter 9, the appearance of dimensions on a CAD drawing is controlled by the use of dimension styles. Dimension styles provide settings that specify controls for

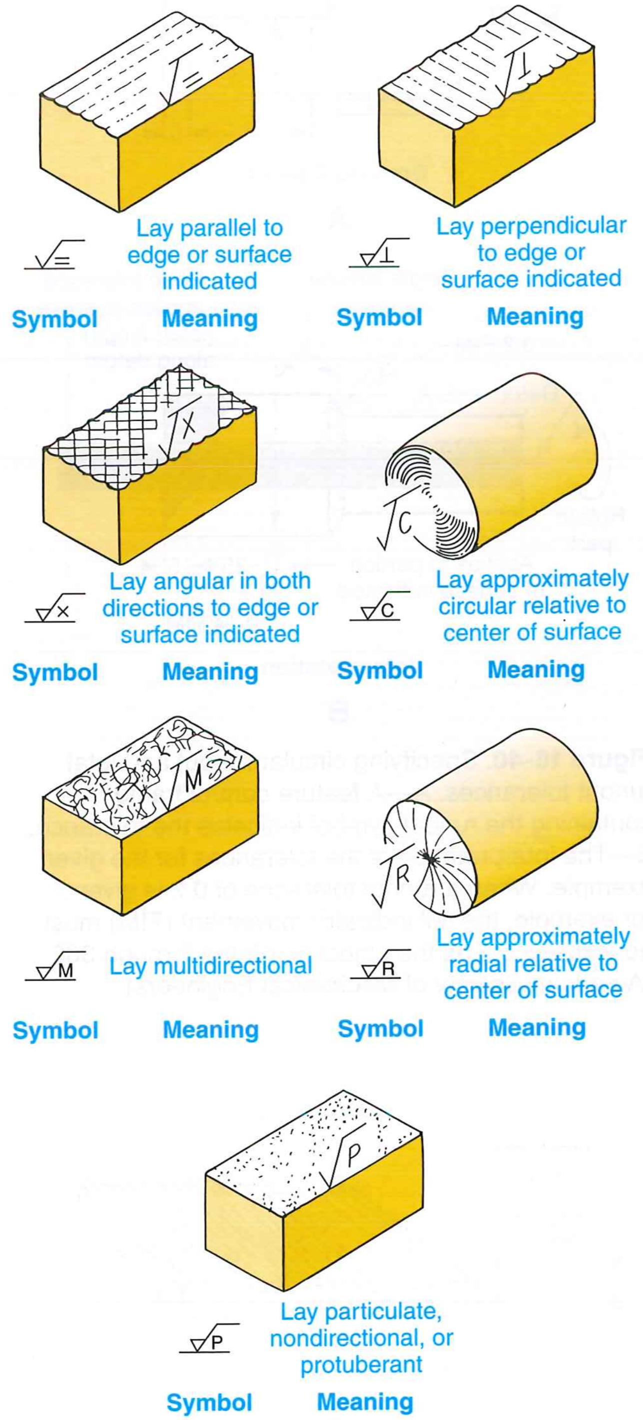

Figure 16-44. Lay symbols and their meaning. (American Society of Mechanical Engineers)

the various dimensioning elements, such as dimension lines and arrowheads. For geometric dimensioning and tolerancing applications, special dimension styles can be created. These styles allow you to draw basic dimensions, limit dimensions, and tolerance dimensions with unilateral or bilateral tolerances. Dimensions drawn in this manner conform to standard drawing conventions. The dimension style settings allow you to control the tolerance values as well as the dimension precision.

Creating Tolerancing Symbols and Feature Control Frames

On CAD drawings, geometric dimensioning and tolerancing symbols can typically be created in one of two ways. Datum feature symbols, geometric characteristic symbols, and feature control frames can be drawn using the **Tolerance** command. This command allows you to create a datum feature symbol or feature control frame without attaching the symbol to a leader arrow. The symbol can then be placed on the drawing as desired. When creating a feature control frame with the **Tolerance** command, a dialog box is used to specify datum identification letters, geometric characteristic symbols, tolerance values, material condition symbols, and other data (such as a projected tolerance zone specification). After entering the necessary values in each compartment, the feature control frame is automatically generated by the program, **Figure 16-45A**.

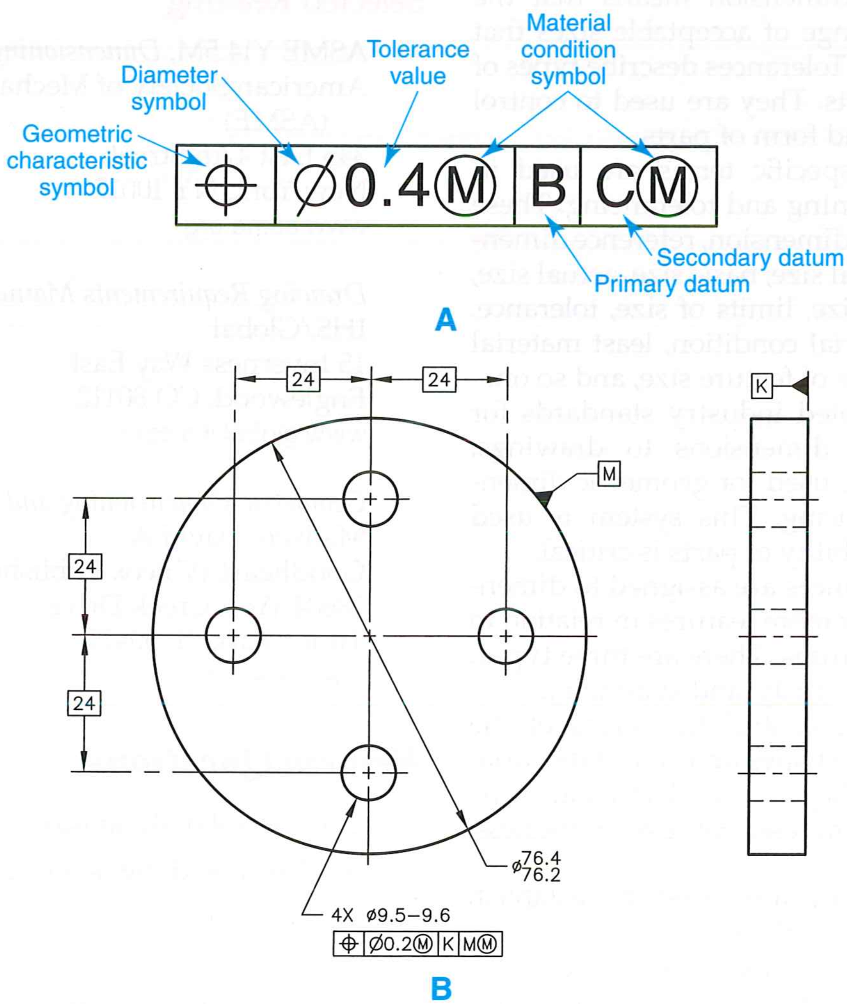

Figure 16-45. Using CAD commands to create geometric dimensioning and tolerancing symbols. A—A feature control frame created with the **Tolerance** command. B—A drawing dimensioned with basic dimensions and geometric dimensioning and tolerancing symbols.

Another way to create geometric dimensioning and tolerancing symbols is to first draw a leader line using the **Leader** command, and then attach a datum feature symbol or feature control frame to the leader. After the leader line is drawn, the command sequence continues and allows you to create the tolerance specification by using a dialog box.

A drawing dimensioned with a feature control frame is shown in **Figure 16-45B**. Notice that basic dimensions are used and the feature control frame appears below a note. In this case, the leader note and feature control frame were created in separate operations.

Chapter Summary

The control of dimensions is called tolerancing. A toleranced dimension means that the dimension has a range of acceptable sizes that are within a "zone." Tolerances describe types of fits for machine parts. They are used to control the size, position, and form of parts.

A number of specific terms are used in geometric dimensioning and tolerancing. These terms include: basic dimension, reference dimension, datum, nominal size, basic size, actual size, allowance, design size, limits of size, tolerance, fit, maximum material condition, least material condition, regardless of feature size, and so on.

There are accepted industry standards for applying tolerance dimensions to drawings. Specific symbols are used for geometric dimensioning and tolerancing. This system is used where interchangeability of parts is critical.

Positional tolerances are assigned to dimensions locating one or more features in relation to other features or datums. There are three types: true position, concentricity, and symmetry.

Form tolerances control the forms of the various geometrical shapes and free-state variations of features. They are used to control the conditions of straightness, flatness, roundness, and cylindricity.

Profile tolerances are used to establish allowable variations in the elements of surfaces or entire surfaces. Orientation tolerances control the orientation of features in relation to one another. Runout tolerances are used to control surfaces constructed around a datum axis and at right angles to a datum axis.

When it is necessary to indicate surface finish on a drawing, specifications are given with other dimensions. Surface finish characteristics include roughness, waviness, and lay. Surface texture symbols are located on the drawing using standard drawing conventions.

On CAD drawings, special dimensioning commands are used to generate geometric dimensioning and tolerancing symbols. As is the case with other drawing applications, CAD dimensioning methods provide a significant advantage to the drafter. Whether drawings are made and dimensioned manually or with a CAD system, the same principles are used when applying geometric dimensioning and tolerancing.

Additional Resources

Selected Reading

ASME Y14.5M, *Dimensioning and Tolerancing*
American Society of Mechanical Engineers (ASME)
345 East 47th Street
New York, NY 10017
www.asme.org

Drawing Requirements Manual
IHS/Global
15 Inverness Way East
Englewood, CO 80112
www.global.ihs.com

Geometric Dimensioning and Tolerancing
Madsen, David A.
Goodheart-Willcox Publisher
18604 West Creek Drive
Tinley Park, IL 60477
www.g-w.com

Review Questions

1. The control of dimensions is called _____.
   A. baseline dimensioning
   B. tolerancing
   C. datum dimensioning
   D. selective assembly
2. _____ tolerances control the location of features on a part.

3. _____ tolerances control the form or the geometric shape of features on a part.
4. What is a *basic dimension*?
5. How are basic dimensions indicated on a drawing?
6. What is a *datum*?
7. A datum is indicated on a drawing by a _____ symbol.
8. The _____ size is a classification size given to commercial products such as pipe or lumber.
   A. actual
   B. basic
   C. design
   D. nominal
9. The _____ size is the size of a part determined by engineering and design requirements.
   A. actual
   B. basic
   C. design
   D. nominal
10. The _____ size is the measured size of a part or object.
    A. actual
    B. basic
    C. design
    D. nominal
11. The _____ is the intentional difference in the dimensions of mating parts to provide for different classes of fits.
12. What are *limits*?
13. _____ is the total amount of variation permitted from the design size of a part.
14. A(n) _____ tolerance varies in only one direction from the specified dimensions.
15. A(n) _____ tolerance varies in both directions from the specified dimension.
16. What are the three general types of fits used to describe mating parts?
17. In the design of mating cylindrical parts, it is necessary to assume a _____ for either the hole or shaft.
18. In the basic _____ size system, the basic size of the hole is the design size, and the allowance is applied to the shaft.
19. When the basic _____ size system is used, the design size of the shaft is the basic size and the allowance is applied to the hole.
20. What condition is present when the feature contains the maximum amount of material?
21. What condition is present when the feature contains the least amount of material within the tolerance range?
22. In _____ dimensioning, only the maximum and minimum dimensions are given to describe tolerances.
23. What is *selective assembly*?
24. In _____ dimensioning, also called baseline dimensioning, features are dimensioned individually from a datum.
25. Running and sliding fits are designated by the letter symbols _____.
26. Force fits or shrink fits are also called _____ fits.
27. A _____ frame is the means by which a geometric tolerance is specified for an individual feature.
28. Name the three basic types of positional tolerances.
29. _____ is the condition of two or more surfaces of revolution having a common axis.
    A. Concentricity
    B. Flatness
    C. Runout
    D. Symmetry
30. _____ describes how close all elements of a surface are to being in one plane.
    A. Concentricity
    B. Flatness
    C. Runout
    D. Symmetry
31. _____ describes the smoothness or finish of a surface.
32. On CAD drawings, what command can be used to draw datum feature symbols, geometric characteristic symbols, and feature control frames?