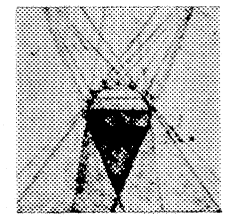
Chapter 16



GEOMETRIC DIMENSIONING & TOLERANCING (ANSI Y14.5 1994)

LEARNING OBJECTIVES

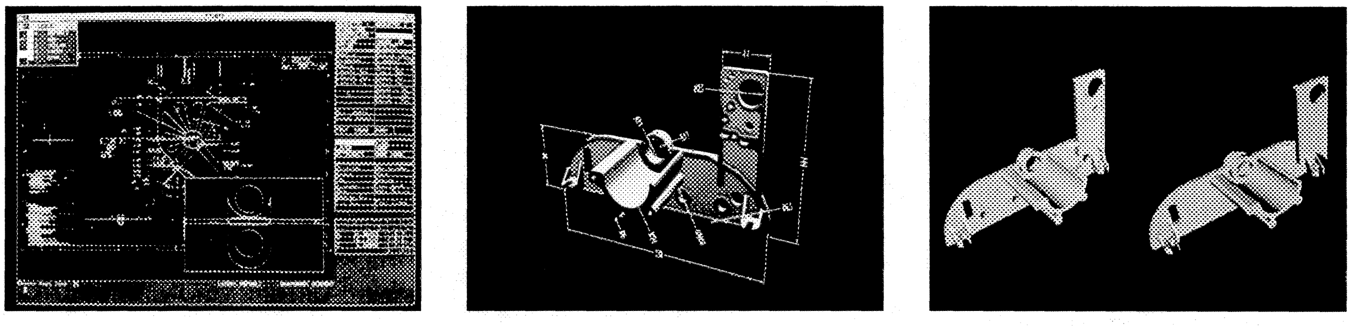
Upon completion of this chapter you will be able to:

1. Differentiate between precision and accuracy, and understand tolerancing terms and techniques.
2. Recognize ISO and ANSI interpretations of angle of projection and limits of size.
3. Understand the use of general and geometric tolerancing rules, symbology, and modifiers.
4. Identify feature control frames.
5. Understand datums and datum systems.
6. Interpret form, profile, orientation, location, and runout tolerances.
7. Apply fixed, floating fastener, and system tolerance formulas.
8. Know how to use guidelines for dimensioning and tolerancing on a CAD system.
9. Understand standardized limits and fits.

16.1 INTRODUCTION

Features of manufactured parts vary in size, form, orientation, or location. Such variation is expected, and, as long as it is understood and controlled, the part will perform as designed. You may have tried to assemble some consumer product and found that holes did not line up between parts or that a hole for a bolt was not drilled perpendicular to the surface. Assembly was no doubt frustrating and the resulting product may not have performed up to expectations without modifications. *Geometric dimensioning and tolerancing (GD&T) is a symbolic system of tolerancing to control the size, form, profile, orientation, location, and runout of a part according to geometry.* Cost-effective designs provide the largest allowable **tolerances** consistent with the function and interchangeability requirements of the design. Statistical process control (SPC) often requires the effective use of GD&T to control process variations and improve product quality.

Even though technical drawings have been employed to communicate engineering information for over 6000 years, the concept of tolerancing, or holding variations within limits, has been around for only about 100 years. At one time, part variations were controlled by the worker rather than by engineering. It was not until the evolution of interchangeable manufacture that the goal of *exact* size gave way to holding parts within *limits.* The *Taylor concept,* introduced in 1905 and still in use today, introduced methods of limit-gaging for holes and shafts. Increased production rates during World War II from larger factories that tapped a wider variety of suppliers created a high rate of scrap that sometimes hampered wartime requirements. Inadequacies in technical drawings for conveying this information became apparent. In 1945, the Gladman papers were published in Great Britain, and these issues of inadequacies in drawings were discussed at the first American, British, Canadian Conference on the Unification of Engineering Standards.



(a) A CAD drawing that uses geometric tolerancing (b) Solid model showing part sizes (c) Solid model family of parts

FIGURE 16.1 Geometric Tolerancing on a CAD System

Unfortunately, geometric tolerancing was put into practice only partially in the 1950s and 1960s. In 1972, the International Organization for Standardization (ISO) established a separate subcommittee to develop dimensioning and tolerancing standards. During the 1970s and 1980s, geometric dimensioning and tolerancing was employed extensively in industry and by the military. ANSI and ISO developed standards to ensure universal interpretation of tolerance requirements on drawings. The drawing in Figure 16.1 shows GD&T applied to a particular part. As product cycle times decrease and demands for quality grow in the increasingly competitive global marketplace of the 1990s, GD&T will play a more and more important role in meeting those demands.

16.1.1 Terms Used in Geometric Dimensioning and Tolerancing

The following terms are used throughout the chapter.

Actual size The measured size.

Basic dimension The theoretically exact size, profile, orientation, or location of a feature or datum target. It is the basis from which permissible variations are established by tolerances on other dimensions, in notes, or in feature control frames.

Basic size The size to which limits or deviations are assigned. This is the same for both members of a fit.

Clearance fit The relationship between assembled parts when clearance occurs under all tolerance conditions.

Datum The origin from which the location or geometric characteristics of features of a part are established.

Datum feature A geometric feature of a part that is used to establish a datum.

Datum target A specified point, line, or area on a part used to establish a datum.

Deviation The difference between the actual size and the corresponding basic size.

Interference fit The relationship between assembled parts when interference occurs under all tolerance conditions.

Lower deviation The difference between the minimum limit of size and the corresponding basic size.

Upper deviation The difference between the maximum limit of size and the corresponding basic size.

Feature A physical portion of a part, such as a surface, a hole, or a slot.

Feature of size A cylindrical or spherical surface, or a set of two parallel surfaces, each of which is associated with a size dimension.

Least material condition (LMC) The condition in which a feature of size contains the least amount of material within stated limits of size, for example, the maximum hole diameter or the minimum shaft diameter.

Limits of size The specified maximum and minimum sizes.

Maximum material condition (MMC) The condition in which a feature of size contains the maximum amount of material within the stated limits of size, for example, the minimum hole diameter or the maximum shaft diameter.

Regardless of feature size (RFS) The geometric tolerance or datum reference applies at any increment of size of the feature within its size tolerance.

Tolerance The total amount by which a specific dimension is permitted to vary; the difference between the maximum and minimum limits.

Tolerance, bilateral A tolerance in which variation is permitted in both directions from the specified dimension.

Tolerance, geometric A tolerance used to control form, profile, orientation, location, or runout.

Tolerance, unilateral A tolerance in which variation is permitted in one direction from the specified dimension.

Tolerance zone An area representing the tolerance and its position in relation to the basic size.

Transition fit The relationship between assembled parts when either a clearance fit or an interference fit results.

True position The theoretically exact location of a feature established by basic dimensions.

Virtual condition The boundary generated by the collective effects of the specified MMC limit of size of a feature and any applicable geometric tolerances.

16.2 STANDARDS AND SPECIFICATIONS

ANSI and ISO standards exist to ensure the universal interpretation of tolerance requirements. However, some companies tailor these standards to meet their particular product requirements. Also, there is not complete agreement between ANSI and ISO standards at this time. To avoid misinterpretation, a note such as "Interpret Drawing in Accordance with ANSI Y14.5M-1982" should appear on the drawing. The note should state the standard, the revision, and the revision date.

Recall that ANSI-standard drawings in the United States use third-angle projection [Fig. 16.2(a)] and that ISO drawings involve first-angle projection [Fig. 16.2(b)]. Although view placement is different, the views that result are the same. However, limits of size are defined differently for the two standards. This book concentrates on ANSI-standard tolerancing techniques, although this chapter also describes ISO techniques.

16.3 SYMBOLOGY

Geometric dimensioning and tolerancing is a **symbolic system** for controlling economically the function, interchangeability, size, form, profile, orientation, position, and runout of features or parts and for establishing datums and other necessary tolerancing practices. This section describes the symbols for specifying geometric characteristics and other dimensional requirements on engineering drawings.

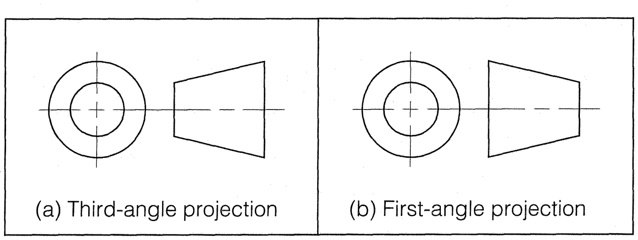


FIGURE 16.2 ANSI and ISO Orthographic Projection Symbols

Symbols should be of sufficient clarity to meet the legibility and reproducibility requirements of ANSI Y14.5M.

Symbols are always preferred to notes because they take less space, overcome language barriers, and are less subject to interpretation. These are the very reasons that GDT uses a symbolic language to communicate specifications. Most individual symbols not only represent an entire standardized engineering concept, but are variously combined in a **feature control frame** to form complete engineering, production, and inspection quality specifications. The form and proportion of geometric tolerancing symbols are shown in Figure 16.3. The geometric characteristic symbols and the modifying symbols are further categorized in Figure 16.4.

Situations may arise where the desired geometric requirement cannot be conveyed completely by symbology. In such cases, a note can describe the requirement, either separately or supplementing a geometric tolerance.

16.3.1 Geometric Characteristic Symbols

Following are the symbols denoting geometric characteristics.

Basic dimension symbols A basic dimension is identified by enclosing the dimension in a rectangle. See Figures 16.3 and 16.4.

Datum feature symbol This consists of a frame containing the datum-identifying letter preceded and followed by a dash.

Letters of the alphabet All letters except I, O, and Q can be used to identify datums. Each datum feature requiring identification is assigned a different letter. When datum features requiring identification on a drawing exceed single alpha lettering, the double alpha series is employed—AA through AZ, BA through BZ, etc.

Datum target symbol This is a circle divided horizontally into two halves. The lower half contains a letter identifying the associated datum, followed by the target number, assigned sequentially starting with 1, for each datum. If the datum target is an area, the area size may be entered in the upper half of the symbol; otherwise, the upper half is left blank. A radial line attached to the symbol is directed to a target point (indicated by an "X"), target line, or target area.

Material condition symbol The symbols for maximum material condition and least material condition are shown in Figures 16.3 and 16.4. If no material condition symbol is present, then regardless of feature size is assumed.

Projected tolerance zone symbol See Figures 16.3 and 16.4.

Diameter and radius symbols The symbols for diameter, spherical diameter, radius, and spherical radius are shown in Figure 16.3. These symbols precede the value of a dimension or tolerance given as a diameter or radius.

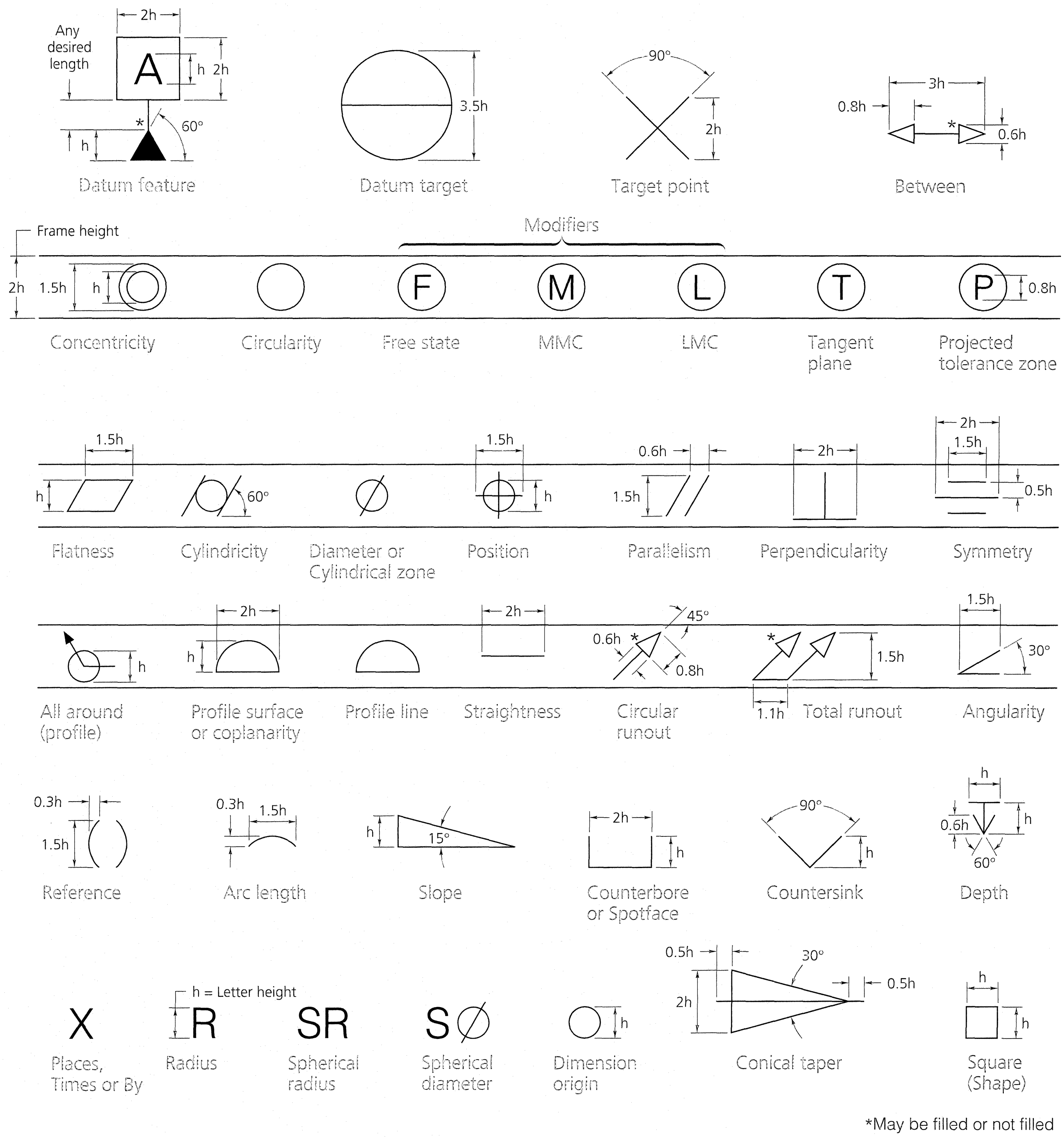


FIGURE 16.3 Geometric Tolerancing Symbols (Sizes)—*Continues*

Reference symbol A reference dimension or reference data is identified by enclosing the dimension or data within parentheses.

Arc length symbol The symbol that indicates that a linear dimension is an arc length measured on a curved outline is shown in Figures 16.3 and 16.4. This symbol is placed above the dimension.

Counterbore or spotface symbol This symbol precedes the dimension of the counterbore or spotface. See Figure 16.3.

Countersink symbol This symbol precedes the dimensions of the countersink. See Figure 16.3.

Depth symbol The symbol for indicating that a dimension applies to the depth of a feature precedes that dimension (Fig. 16.3).

Symbol for:	ASME Y14.5M	ISO
Straightness	—	—
Flatness	⏥	⏥
Circularity	○	○
Cylindricity	⌭	⌭
Profile of a line	⌒	⌒
Profile of a surface	⌓	⌓
All around		(proposed)
Angularity	∠	∠
Perpendicularity	⊥	⊥
Parallelism	//	//
Position	⌖	⌖
Concentricity (concentricity and coaxiality in ISO)	◎	◎
Symmetry	⌯	⌯
Circular runout	↗	↗
Total runout	⌰	⌰
At maximum material condition	Ⓜ	Ⓜ
At least material condition	Ⓛ	Ⓛ
Regardless of feature size	None	None
Projected tolerance zone	Ⓟ	Ⓟ
Tangent plane	Ⓣ	Ⓣ (proposed)
Free state	Ⓕ	Ⓕ
Diameter	⌀	⌀
Basic dimension (theoretically exact dimension in ISO)	$\boxed{50}$	$\boxed{50}$
Reference dimension (auxiliary dimension in ISO)	(50)	(50)
Datum feature	$\boxed{A}$	or $\boxed{A}$

FIGURE 16.3 Geometric Tolerancing Symbols (Sizes)—*Continued*

Symbol for:	ASME Y14.5M	ISO
Dimension origin	⌖→	⌖→
Feature control frame	⌖ | Ø 0.5 Ⓜ | A | B | C	⌖ | Ø 0.5 Ⓜ | A | B | C
Conical taper	⌲	⌲
Slope	⌳	⌳
Counterbore/spotface	⌴	⌴ (proposed)
Countersink	⌵	⌵ (proposed)
Depth/deep	↧	↧ (proposed)
Square	□	□
Dimension not to scale	<u>15</u>	<u>15</u>
Number of places	8X	8X
Arc length	$\overset{\frown}{105}$	$\overset{\frown}{105}$
Radius	R	R
Spherical radius	SR	SR
Spherical diameter	SØ	SØ
Controlled radius	CR	None
Between	*◄—►	None
Statistical tolerance	⟨ST⟩	None
Datum target	(Ø6 / A1) or (• / A1) Ø6	(Ø6 / A1) or (• / A1) Ø6
Target point	×	×

*May be filled or not filled

FIGURE 16.3 Geometric Tolerancing Symbols (Sizes)—*Continued*

Square symbol The symbol that indicates that a single dimension applies to a square shape precedes that dimension (Fig. 16.4).

Dimension origin symbol This symbol, a small circle placed at the origin, indicates that a toleranced dimension between two features originates from one of those features.

Taper and slope symbols Symbols specifying taper and slope for conical and flat tapers are shown in Figure 16.3.

16.3.2 Modifiers

Modifiers stipulate whether a tolerance is to apply regardless of size or only at a specific size (see Fig. 16.5). If no modifier is present, then regardless of feature size is assumed. The following rules for modifiers are based on the size of features and the geometry involved.

- Modifiers may be used only for features and/or datums that have a size tolerance. The MMC modifier may be

Feature	Tolerance Type	Symbol	Characteristic
Individual (single)	Form (shape)	—	Straightness
		⏥	Flatness
		○	Circularity
		⌭	Cylindricity
Individual or related	Profile (contour)	⌒	Profile of a line
		⌓	Profile of a surface
Related	Orientation (attitude)	∠	Angularity
		⊥	Perpendicularity
		//	Parallelism
	Location	⌖	Position
		◎	Concentricity
	Runout	↗	Circular runout
		⌰	Total runout
Modifying symbols		(M)	Maximum material condition-MMC
		(L)	Least material condition
Additional symbols		(P)	Projected tolerance zone
		∅	Diameter (face of dwg.)
		S∅	Spherical diameter
		R	Radius
		SR	Spherical radius
		( )	Reference
		⌒	Arc length

FIGURE 16.4 **Categories of Geometric Tolerancing Symbols**

used in conjunction with the straightness of a feature axis based on the cross-sectional size, flatness (by special note on features of size), datums of size with profile tolerances, and all datums and features of size or orientation and position tolerances.

- The RFS symbol is no longer used in the United States. In all countries, unless otherwise specified, all tolerances automatically apply RFS.
- Position, except in the case of a single-plane surface, requires a modifier for all features and datums (Fig. 16.6).
- Circularity, cylindricity, runout, concentricity, straightness

Modifying Symbols		
Symbol	Abbreviation	Meaning
Ⓜ	MMC	Maximum material condition
Ⓛ	LMC	Least material condition

FIGURE 16.5 Modifying Symbols

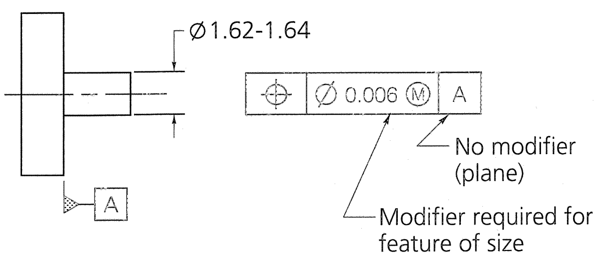


FIGURE 16.6 Single-Plane Surface, No Modifier

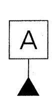
of element lines, and profile of a feature may not use the MMC modifier. The exception is for datums of size used in conjunction with profile. A special note is required to incorporate the MMC modifier in conjunction with flatness:

(PERFECT FORM AT MMC NOT REQUIRED)

16.4 FEATURE CONTROL FRAME

Geometric tolerances are placed in a **feature control frame,** which contains a geometric characteristic symbol, the tolerance, modifiers, and datums (Fig. 16.7). The feature control frame consists of at least the first two compartments shown in Figure 16.8, but may contain three or more compartments. The *first compartment* contains the geometric characteristic symbol (one of the thirteen from Fig. 16.3). The *second compartment* may contain a zone shape symbol, such as the diameter symbol indicating the diameter of a cylindrical zone; the tolerance, in inches or millimeters; and a modifier. The *third compartment* usually contains datums. This compartment may have **separators** to order the datums.

Feature control frames are not repeated or referenced on a technical drawing. **Datum identification symbols,**



may be repeated where it is essential to ensure the correct meaning.

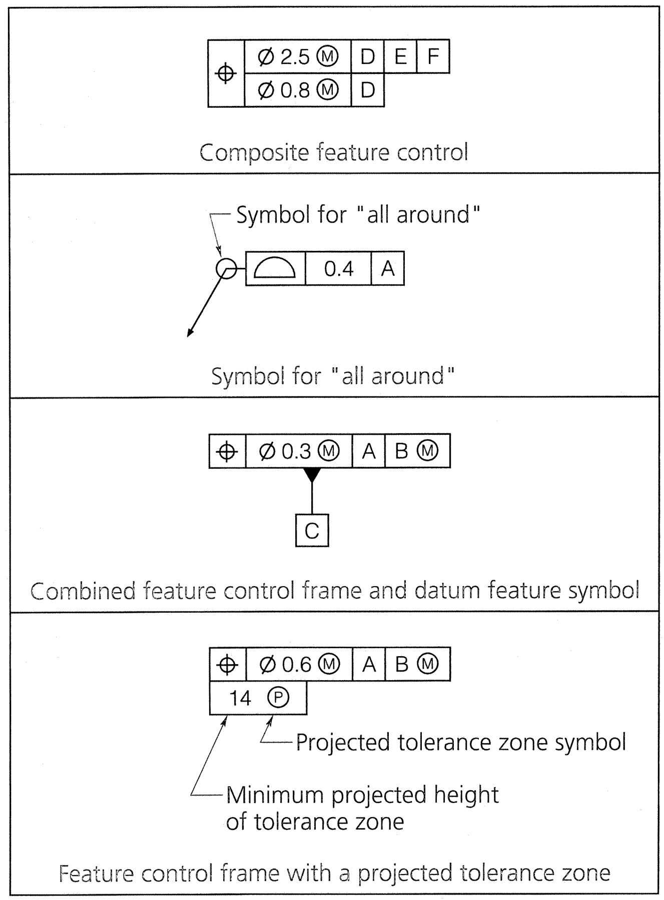


FIGURE 16.7 Feature Control Frames

16.4.1 Maximum Material Condition (MMC)

In the **maximum material condition (MMC),** a feature or datum feature is at the tolerance limit, meaning the part will contain the most material (*weigh the most*). For an external feature, such as a pin or shaft, MMC is the maximum limit (Fig. 16.9). For an internal feature, such as a hole, MMC is the minimum limit (Fig. 16.10). Remember that a part weighs the most when the hole in it is the smallest size in the range. Figure 16.11 shows the MMC and the least material condition (LMC) in both an external and an internal feature. The tightest fit between the two results when both features are at MMC; the loosest fit results when both features are at LMC.

If the MMC modifying symbol appears in a feature control frame (Fig. 16.12), the specified tolerance applies only at MMC. In Figure 16.12, the perpendicularity tolerance is .004 when the feature is at MMC (.512). As the feature deviates from MMC, additional perpendicularity tolerance equal to the deviation is allowed. This is called the *bonus tolerance* (Fig. 16.13). In other words, as the male diameter decreases, the increase in perpendicularity results in the same fit to the mating part. The modifying symbol for MMC, specified in the feature control frame for the feature, datum, or both, works the same way for all geometric tolerances.

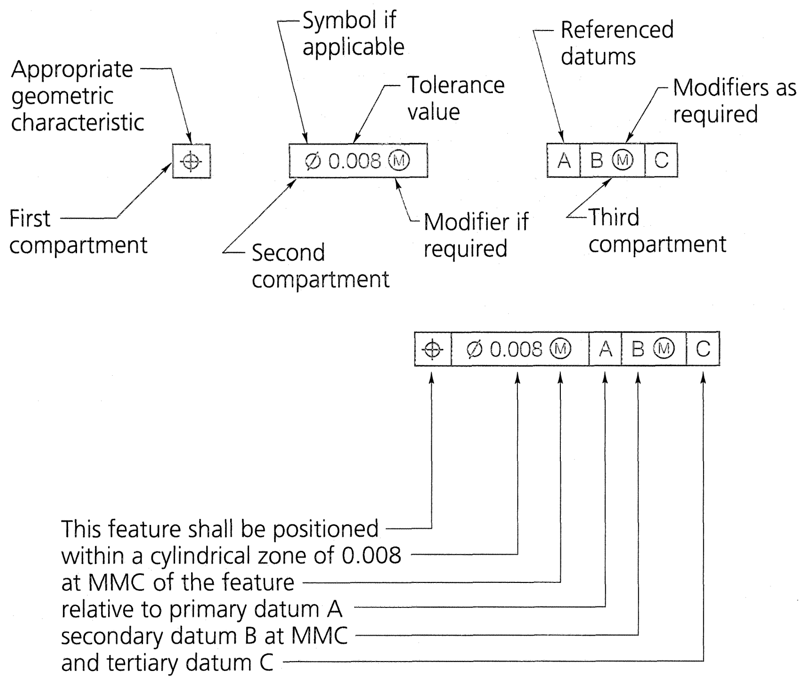


FIGURE 16.8 Typical Configuration of a Feature Control Frame

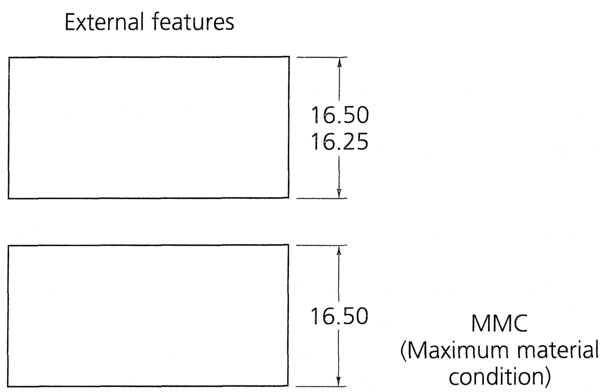


FIGURE 16.9 MMC of an External Feature

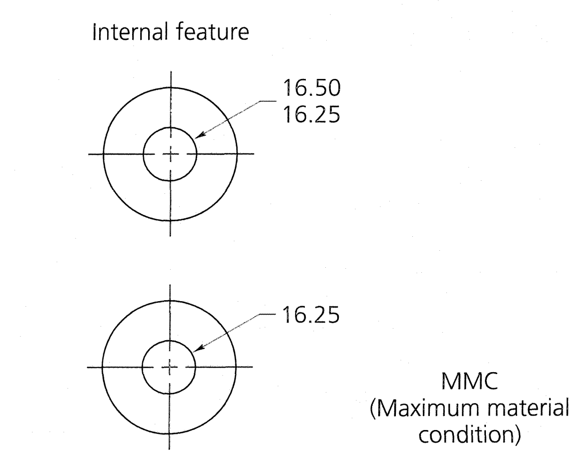


FIGURE 16.10 MMC of an Internal Feature

16.4.2 Least Material Condition (LMC)

In the **least material condition (LMC),** a feature or datum feature is at the tolerance limit, meaning the part will contain the least material (*weigh the least*). For an external feature, such as a pin, LMC is the minimum limit (Fig. 16.14). For an Internal feature, such as a hole, LMC is the maximum limit (Fig. 16.15). The modifying symbol for LMC, specified in the feature control frame for the feature, datum, or both, works the same way for all geometric tolerances. Table 16.1 shows the result of using Figure 16.11 as though LMC, rather than MMC, were specified in the feature control frame. LMC is generally employed where minimum bearing areas, minimum wall thickness, or alignment of parts is the main concern, not fit.

16.4.3 Regardless of Feature Size (RFS)

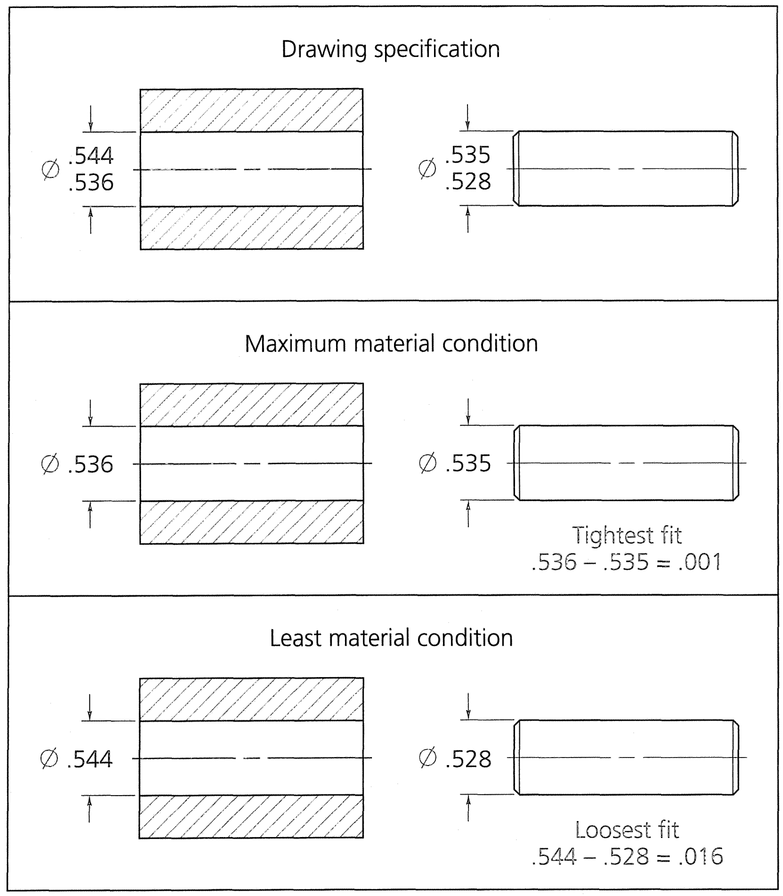
The newest ANSI standard no longer uses the RFS modifier symbol. In the ISO standard, RFS applies to every geometric tolerance unless MMC or LMC is specified. For positional tolerance, MMC or LMC must be specified for all features and datums of size.

If no modifier is placed after the feature tolerance in the second compartment, the tolerance must be met at all sizes.

16.4.4 Virtual Condition

The **virtual condition** (Fig. 16.16) is the condition resulting from the worst-case effect of the size and geometric tolerance applied to the feature. The free assembly of components is

FIGURE 16.11 MMC and LMC Limits for External and Internal Features



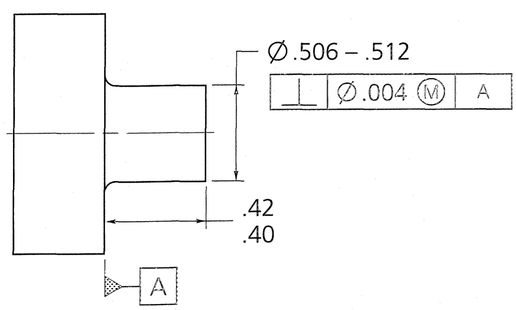
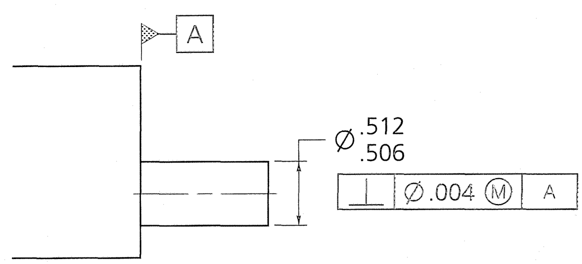


FIGURE 16.12 Perpendicularity When the Feature Is at MMC



MMC size	Actual size	Tolerance allowed	Virtual (fit) condition
.512	.512	.004	.516
.512	.511	.005	.516
.512	.510	.006	.516
.512	.509	.007	.516
.512	.508	.008	.516
.512	.507	.009	.516
.512	.506	.010	.516

FIGURE 16.13 Bonus Tolerance Addition to Geometric Tolerance at MMC

dependent on the combined effect of the actual sizes of the part features and the errors of form, orientation, location, or runout—for example, the axis is out of straight, the size of a shaft is virtually increased, or the size of a hole is virtually decreased. The formulas for determining the virtual condition are as follows.

External Feature:

Virtual condition = MMC size + geometric tolerance

Internal Feature:

Virtual condition = MMC size − geometric tolerance

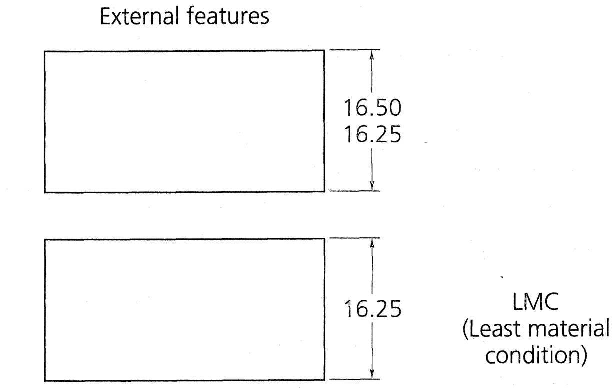


FIGURE 16.14 LMC of an External Feature

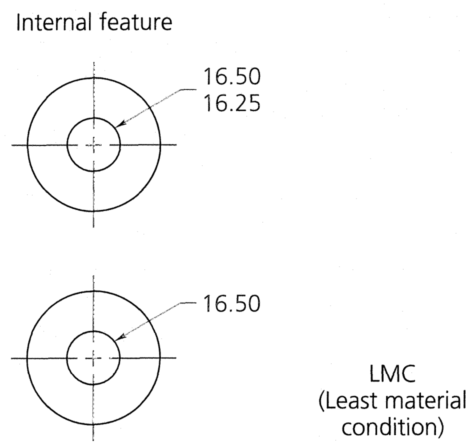


FIGURE 16.15 LMC of an Internal Feature

TABLE 16.1 Bonus Tolerance Addition to Geometric Tolerance at LMC

LMC Size	Actual Size	Tolerance Allowed	Minimum Bearing Area
.506	.506	.004	.502
.506	.507	.005	.502
.506	.508	.006	.502
.506	.509	.007	.502
.506	.510	.008	.502
.506	.511	.009	.502
.506	.512	.010	.502

16.4.5 Angular Surfaces

If an **angular surface** is defined by the combination of a linear dimension and an angle, the surface must lie within a tolerance zone represented by two nonparallel planes (Fig. 16.17). The tolerance zone will be wider as the distance from the apex of the angle increases.

16.5 DATUMS AND DATUM SYSTEMS

Datums are theoretically exact geometric references derived from the datum feature. Figure 16.18 shows the primary datum plane established on a surface by three area contact positions. Datums are not assumed to exist on the part itself, but are simulated by the more precisely made manufacturing or inspection equipment or a computerized mathematical model. A datum plane, for example, could be *simulated* from the datum feature by a surface plate (Fig. 16.19).

Datums are points, lines, and planes. Datums provide repeatable part and feature orientation for manufacturing and inspection consistent with the expected mating characteristics or orientation at assembly. Datums should be established from "hard" features on the part, such as one or two specific diameter(s) on a shaft (Fig. 16.20).

16.5.1 Applicability of Datums

A **datum** is a theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a part are established. A **datum target** is a specified point, line, or area on a part used to establish a datum. Tolerances, as they relate to datums, are described according to the feature they locate.

16.5.2 Part and Feature Direction and Orientation

If a drawing contains two or mor<e features, it is incomplete if one or more datums are not specified. Without datums, reliable engineering interchangeability is difficult or impossible; setup criteria for manufacturing and inspection is then arbitrary. Without datums, the design is compromised and the manufactured part or assembly may not function as intended.

16.5.3 Datum Reference Frame

Locations and measurements are taken relative to three mutually perpendicular planes, collectively called a **datum reference frame** (Fig. 16.21). In inspection, a surface plate and two angle plates perpendicular to it can simulate the datum reference frame. In manufacturing, the bed of the machine and clamps or other devices, along with the direction of machine movement, provide location relative to three mutually perpendicular planes.

16.5.4 Datum Features

Datum features are selected to ensure the orientation of the part and its associated features for interchangeability and to ensure functional relationships. If a functional datum feature is undesirable from a manufacturing or inspection standpoint, a nonfunctional feature with a precise toleranced relationship to the functional feature may be used, provided all design requirements are met.

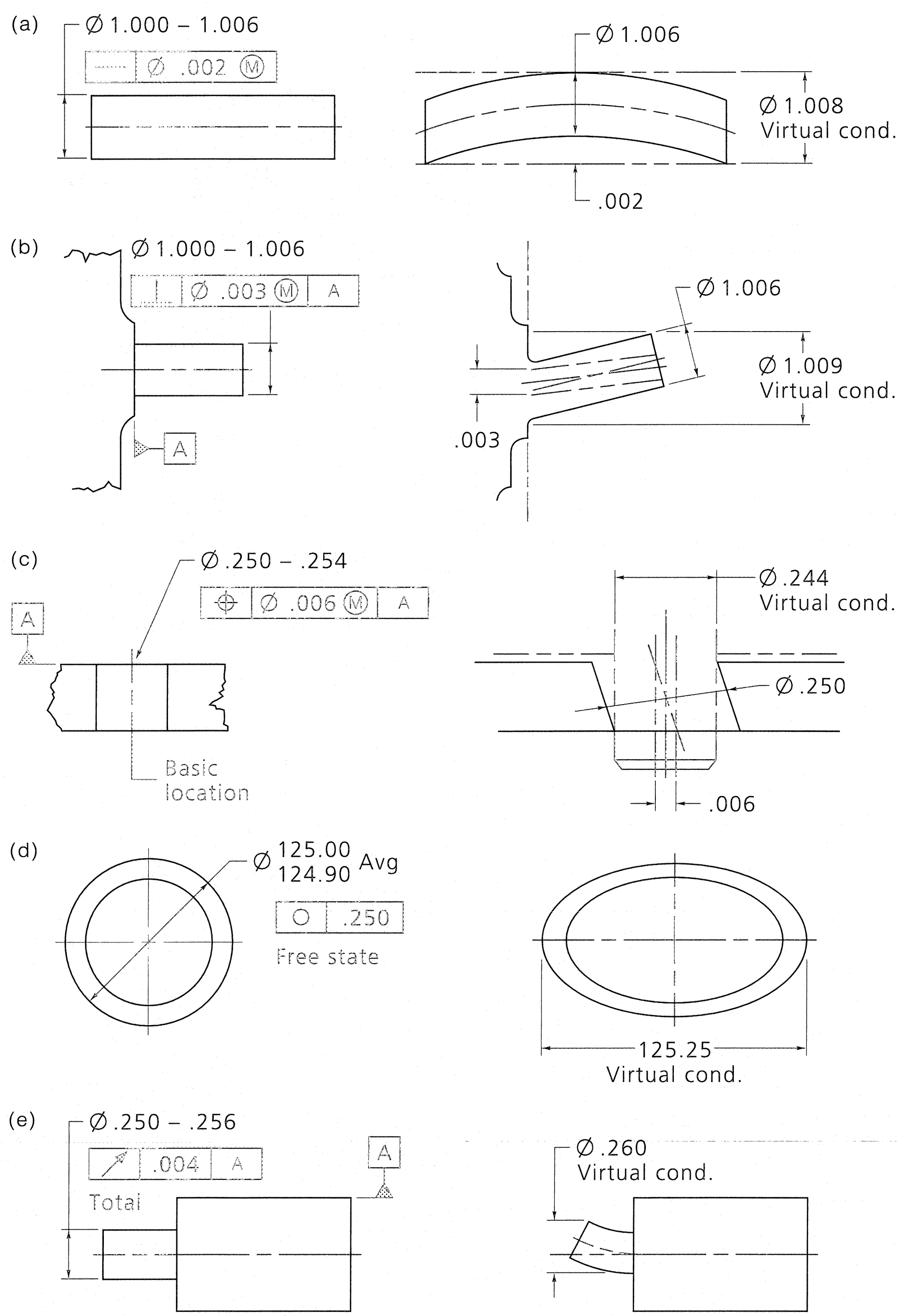


FIGURE 16.16 Examples of Virtual Condition

16.5.5 Datum Precedence

The sequence of datums specified in the feature control frame determines the order in which the datum features contact the datum reference frames (Fig. 16.22):

1. The part primary datum feature is aligned with the primary datum.
2. While in full contact with the primary datum, the secondary datum feature is aligned with the secondary datum.
3. While in full contact with the primary datum and aligned to the secondary datum, the tertiary datum feature is pushed into contact with the tertiary datum.

The **primary datum** is established by full contact with of a minimum of three noncollinear points on the part (recall that three noncollinear points define a plane). The **secondary datum** is perpendicular to the primary datum and is established by contacting a minimum of two points on the part (two points establish a line). The **tertiary datum** is perpendicular to the primary and the secondary datums

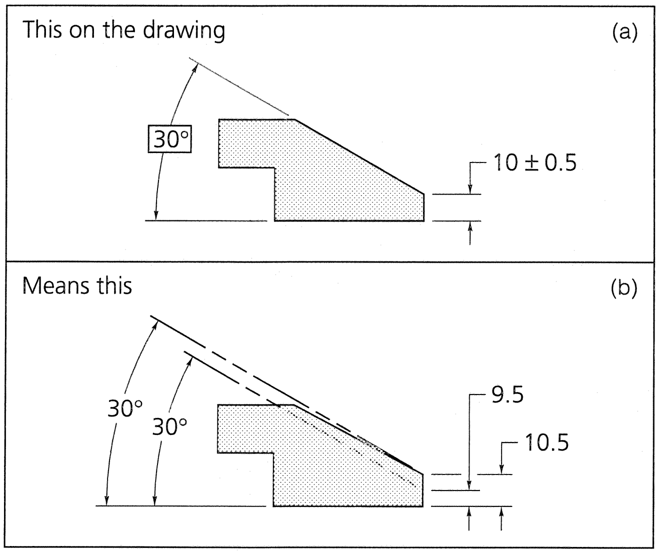


FIGURE 16.17 Tolerancing an Angular Surface Using Linear and Angular Dimensioning

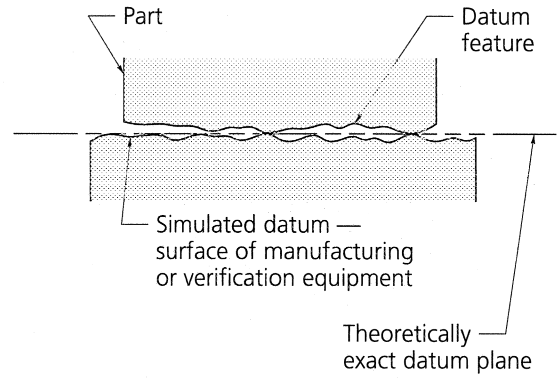


FIGURE 16.19 Theoretical and Simulated Datum and Datum Plane

and, therefore, needs only one point of contact on the part to establish it. In Figure 16.22, notice that the three directions of measurement, **X**, **Y**, and **Z**, are established on the part, as are their origin datums. Precise and repeatable measurements may now be made as the part is oriented and locked in position. Datums are specified on the drawing to ensure the intended datum reference frame.

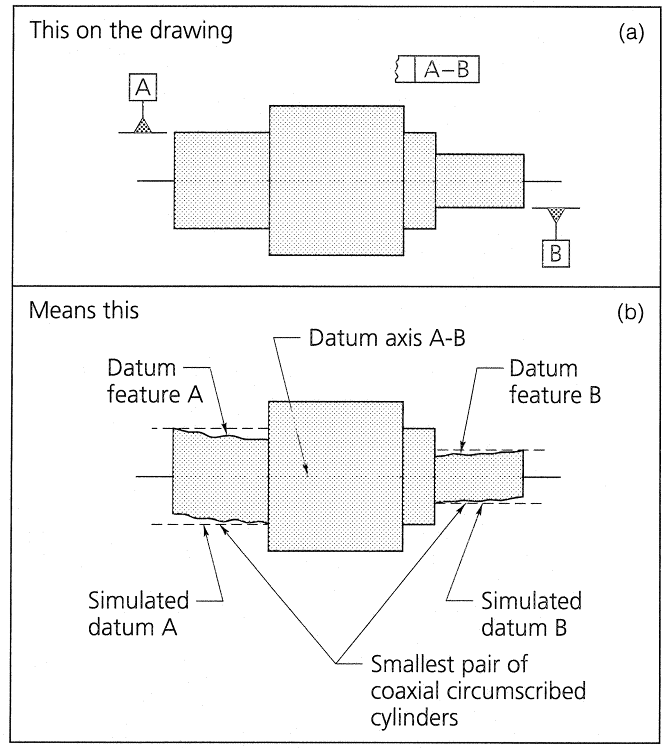


FIGURE 16.20 Coaxial Datum Features

16.5.6 Datum Targets

Datum targets are specific points (Fig. 16.23), lines (Fig. 16.24), or areas (Fig. 16.25) that are used when an entire surface may not be suitable as a datum feature. For example, the rough surfaces of castings and forgings are difficult to use. If a limited portion of a feature is not a point, line, or

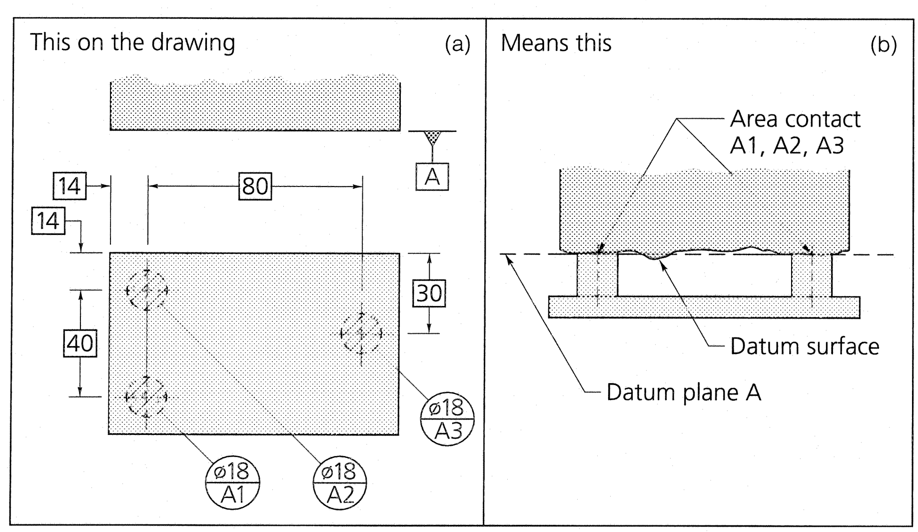


FIGURE 16.18 The Primary Datum Established by Three Area Contact Positions

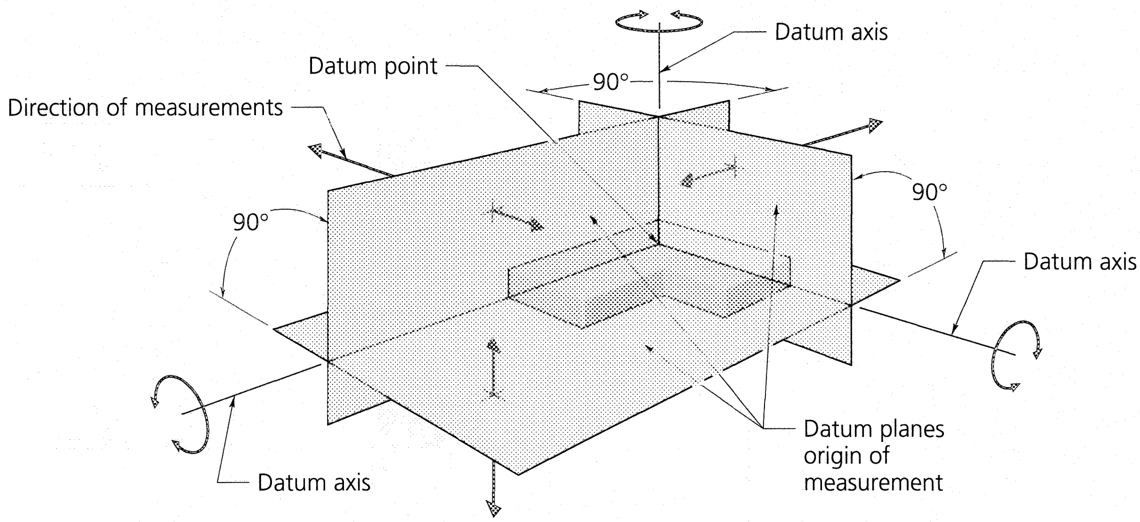


FIGURE 16.21 Datum Reference Frame

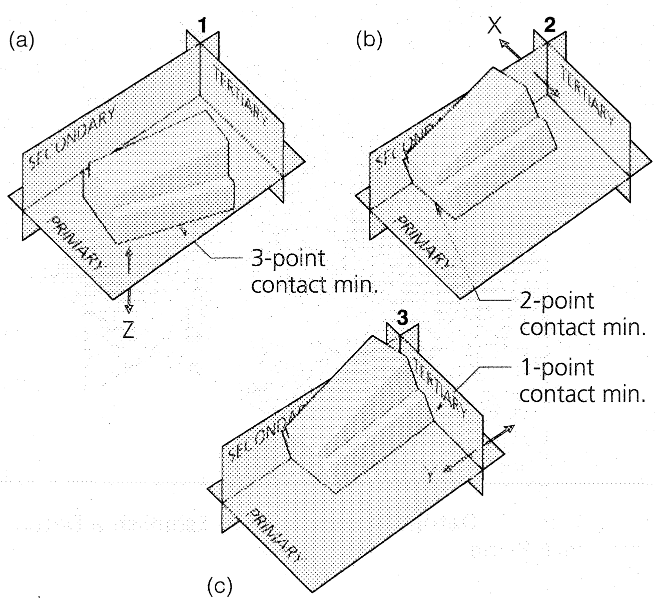


FIGURE 16.22 Datum Reference Frame—Datum Precedence

local flat area (a portion of a cylindrical surface, for example), then partial datums (Fig. 16.26) may be employed instead of targets.

In Figure 16.27 datum targets **A1**, **A2**, and **A3** establish the primary datum; datum targets **B1** and **B2** establish a secondary plane perpendicular to the primary plane; and **C1** establishes the tertiary plane perpendicular to the primary and secondary planes. The datum target identification symbol is shown in Figure 16.28.

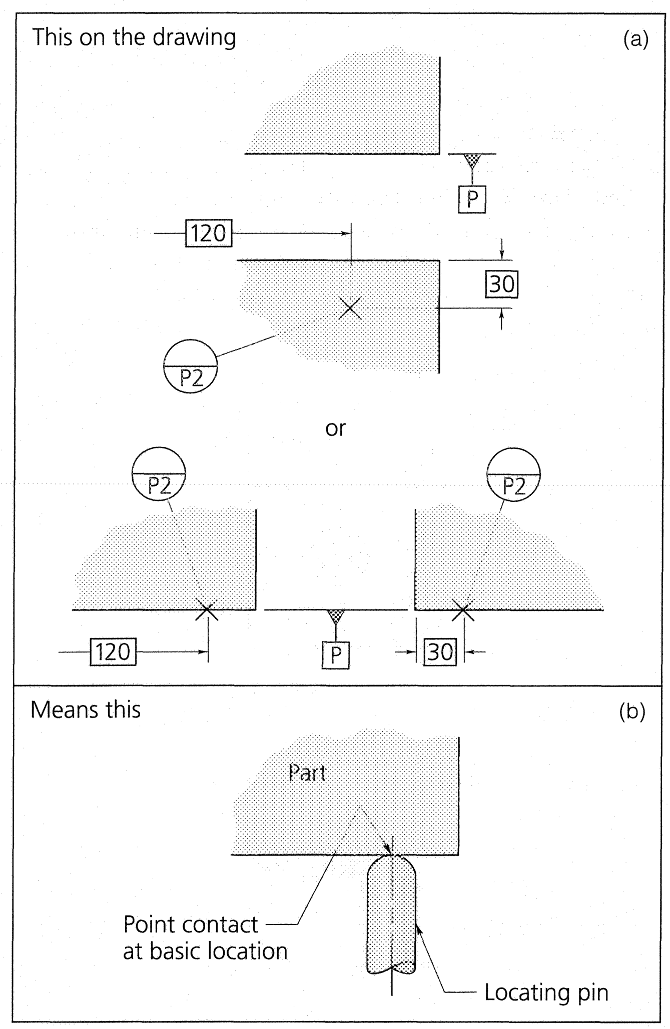


FIGURE 16.23 Datum Target Point

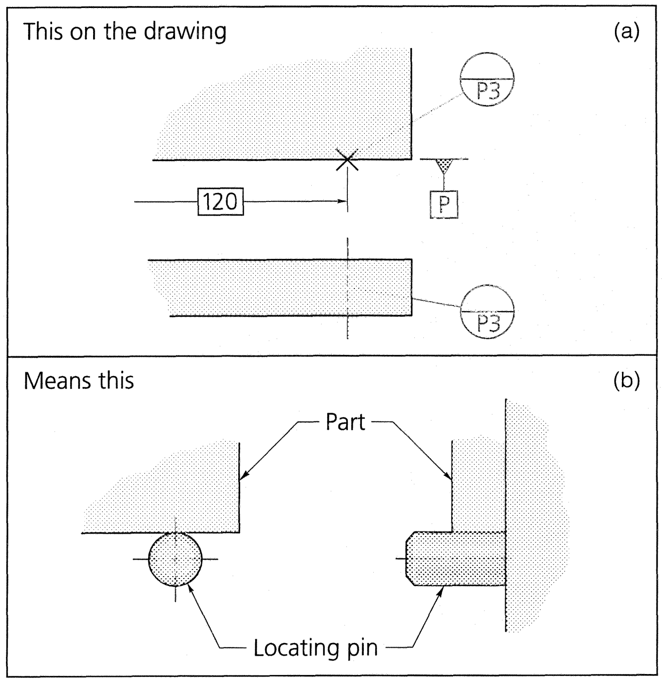


FIGURE 16.24 Datum Target Line

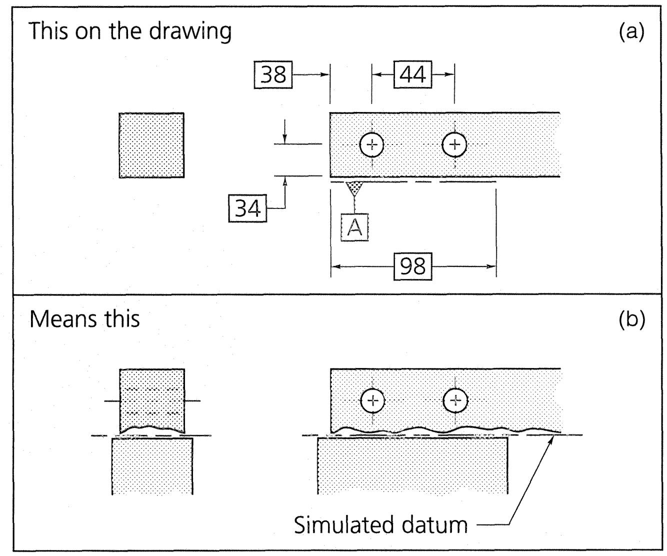


FIGURE 16.26 Partial Datums

16.5.7 Datum Target Depiction

Datum target points are depicted by a dense 90° "cross" (**X**) at 45° to the centerline (Fig. 16.29), at twice the letter height. The leader line from the datum target symbol does not terminate in an arrowhead. A solid leader line indicates that the target is on the near side; a dashed leader line indicates that the target is on the far side. The three mutually

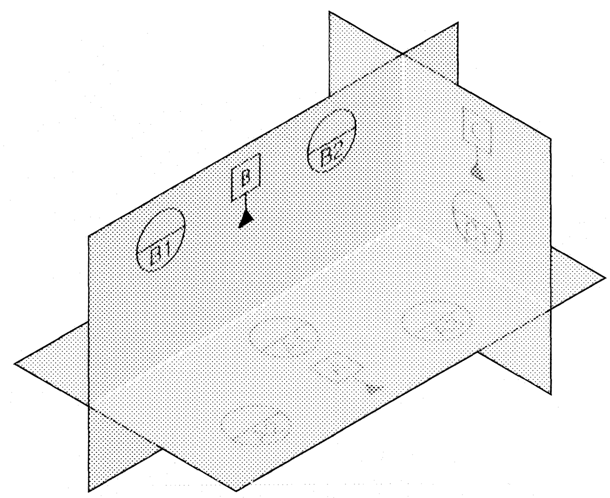


FIGURE 16.27 Datum Targets Used to Establish a Datum Reference Plane

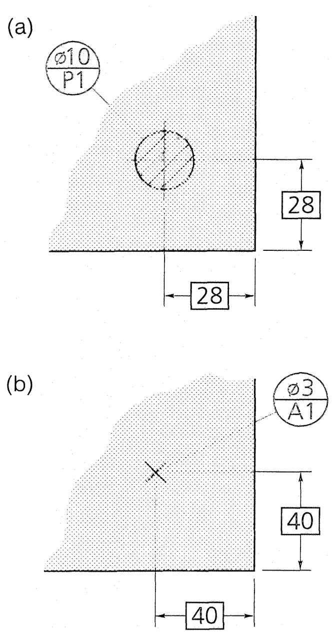


FIGURE 16.25 Datum Target Area

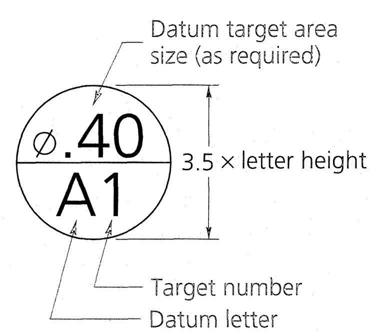


FIGURE 16.28 Datum Target Identification Symbol

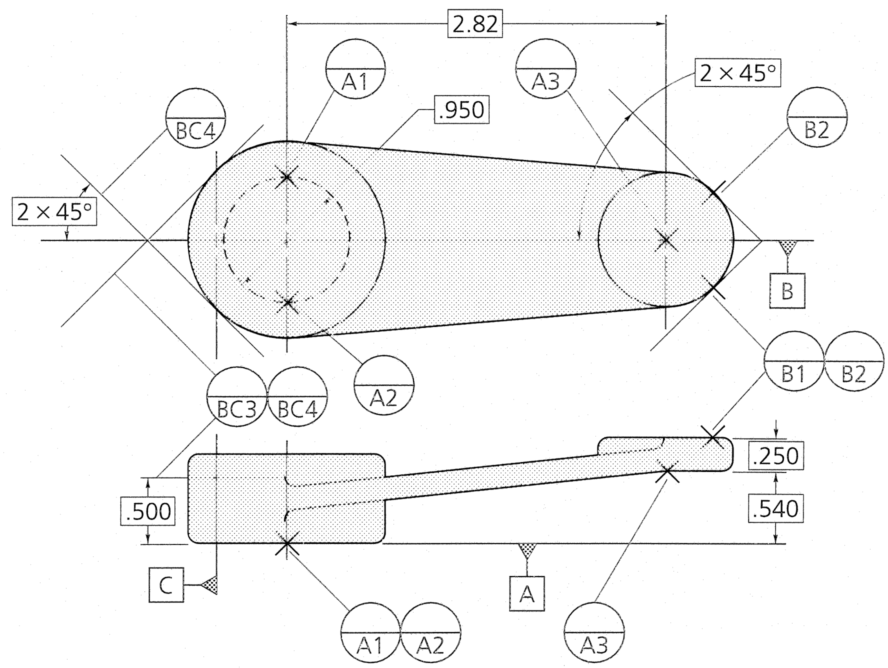


FIGURE 16.29 Datum Targets Showing "Step" and "Equalizing Dimensions"

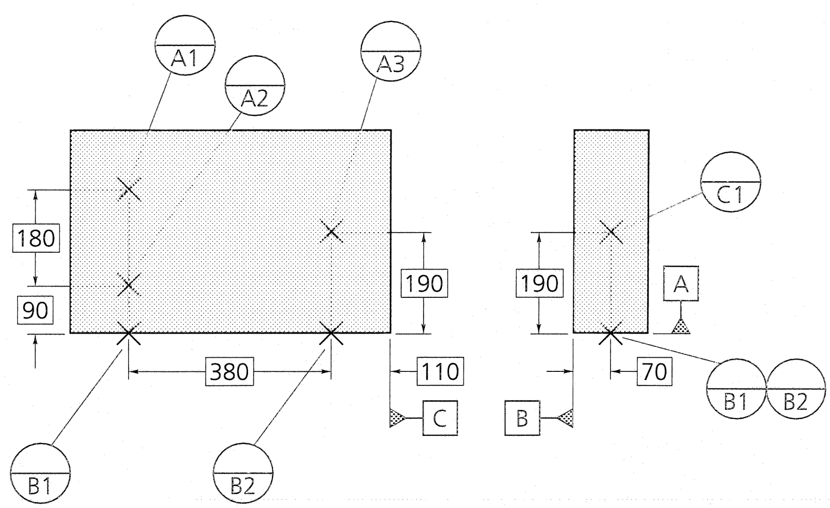


FIGURE 16.30 Dimensioning Datum Targets

perpendicular planes from which to measure **X**, **Y**, and **Z** distances are locked in place and repeatable for each individual part. Datum targets may also be located on the drawing by dimensions (Fig. 16.30).

16.6 GEOMETRIC TOLERANCE OF FORM, PROFILE, ORIENTATION, LOCATION, AND RUNOUT

Geometric tolerances of form, profile, orientation, location, and runout are described in this section.

16.6.1 Form Tolerances

Form tolerances are applicable to individual features or elements of single features. Such tolerances do not use datums because they are related to a perfect counterpart of themselves. These "pure form" tolerances are: straightness, flatness, circularity, and cylindricity.

16.6.2 Straightness of Element Lines

The **straightness** tolerance specifies variation from a straight line. Each element line on the surface must be straight within the specified straightness tolerance. For element control, the leader from the feature control frame must be directed to the outline of the part where the element to be controlled appears as a straight line [Fig. 16.31(b)]. For a

Focus On . . .

TOLERANCING AND ITS ROLE IN INDUSTRY

Mass production of interchangeable parts played an important role in the Industrial Revolution. Much of the technology that we enjoy today also relies on interchangeable components. Automobiles and computer circuits are good examples of mass production and the importance of size control.

While it is impossible to make any part exactly the same size as another part, it is possible to keep component dimensions to a specific range of sizes. Geometrical relationships can also be specified. These dimension restrictions are specified with *tolerances*. Component function determines the degree of tolerance. This process ensures that parts made in one location are interchangeable with parts made in another location.

For example, Eagle Engine Manufacturing produces V-8 engines for top fuel dragsters. The Eagle engine can produce 3000 hp and is designed to allow for different configurations. Cylinders are interchangeable, and the head accommodates one to three spark plugs per cylinder. This means the engine can be configured for a dragster or a tractor. Specific parts for the engine were designed on a CAD system with tolerance capabilities to sixteen decimal places. The design was easily modified to fit another configuration with its tolerance specifications.

Producing components to specific tolerances makes it possible to mass-produce goods and modify existing components to fit different needs. This system gives the manufacturer flexibility, allowing the part to change quickly with market trends and technological advances. This kind of flexibility is essential in the competitive world of today and tomorrow.

rectangular part, the view in which the leader is shown determines the direction of the indicator movement zone. In this case, each element on the surface is to be straight within the specified tolerance, and the feature must meet the size tolerance.

16.6.3 Straightness of an Axis

If straightness of an axis is specified, the leader from the feature control frame must be directed to the size dimension [Fig. 16.31(c)], and a diameter symbol (∅) must precede the tolerance in the feature control frame. An exception is made if the zone is not cylindrical.

16.6.4 Flatness

Flatness means that a surface has all elements in one plane. Flatness must be within the size tolerance, but has no orientation requirement. Therefore, it may be tilted in the size zone. A **flatness tolerance** specifies a tolerance zone defined by two parallel planes within which the surface must lie [Fig. 16.31(d)]. When a flatness tolerance is specified, the feature control frame is attached to a leader directed to the surface or to an extension line of the surface. It is placed in a view where the surface elements to be controlled are represented by a line. If the considered surface is associated with a size dimension, the flatness must be less than the size tolerance.

16.6.5 Circularity

A **circularity tolerance** specifies a tolerance zone bounded by two concentric circles within which each circular element of the surface must lie, and applies independently at any plane described in the list in Figure 16.31(e). The circularity tolerance must be less than the size tolerance, except for those parts subject to three-state variation.

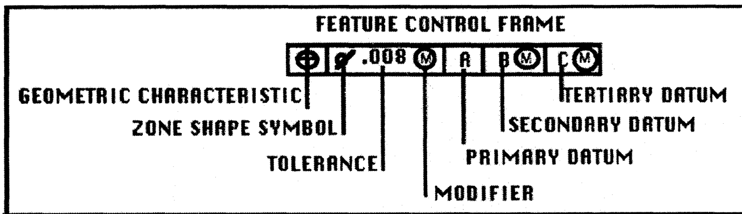
16.6.6 Cylindricity

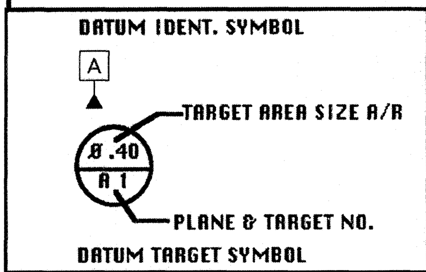
Cylindricity is a surface of revolution in which all points of the surface are equidistant from a common axis. A **cylindricity tolerance** specifies a tolerance zone bounded by two concentric cylinders within which the surface must lie [Fig. 16.31(f)]. In the case of cylindricity, unlike that of circularity, the tolerance applies simultaneously to the entire surface. The leader from the feature control frame may be directed to either view. The cylindricity tolerance must be less than the size tolerance.

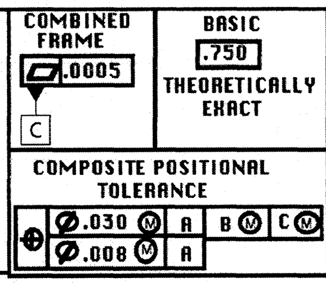
16.7 PROFILE TOLERANCES

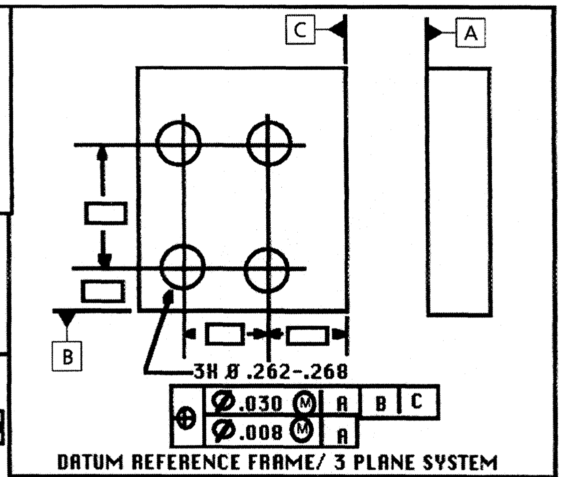
A **profile** is an outline of a 2D part in a given plane. Profiles are formed by projecting a 3D figure onto a plane or by taking cross sections through the figure. The elements of a profile are straight lines, arcs, and other curved lines. If the drawing specifies individual tolerances for the elements or points of a profile, these elements or points must be verified individually. With profile tolerancing, the true profile may be defined by basic radii, basic angular dimensions, basic coordinate dimensions, formulas, or undimensioned drawings.

The **profile tolerance** specifies a uniform boundary along the true profile within which the elements of the surface must lie. It is used to control form or combinations of size,









EXCLUSIONS TO RULE #1 PERFECT FORM AT MMC
1. STOCK SPECIFICATION
2. *FLATNESS NOTE
3. *EXCLUSION BY NOTE
4. FREE STATE VARIATION
5. STRAIGHTNESS-AXIS

* Ⓣ PERFECT FORM AT MMC NOT REQ'D

CONVERSION NUMBERED SCREW TO A DIAMETER
MAX SCREW O.D.= .013 X SCREW NO. + .060

CONVERSION CUSTOMARY TO METRIC & BACK
INCHES X 25.4 = MILLIMETERS (EXACTLY)
MILLIMETERS X .039 370 078 7 = INCHES

POSITIONAL TOLERANCE FORMULAS
H = MMC ⌀ HOLE SIZE
T = MMC ⌀ POSITIONAL TOLERANCE
F = FASTENER ⌀ VIRTUAL CONDITION

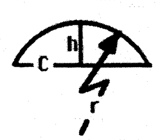
FLOATING FASTENER SYSTEM EQUAL TOLERANCE DISTRIBUTION
1. T = H - F
2. H = F + T

FLOATING FASTENER SYSTEM UNEQUAL TOLERANCE DISTRIBUTION
T1 = MMC ⌀ POSITIONAL TOLERANCE part # 1
T2 = MMC ⌀ POSITIONAL TOLERANCE part # 2
H1 = MMC ⌀ HOLE SIZE-part #1
H2 = MMC ⌀ HOLE SIZE-part #2
3. T1 = (H1 + H2) - (2F + T2)
4. H1 = (T1 + T2) - (2F - H2)

FIXED FASTENER SYSTEM EQUAL TOLERANCE DISTRIBUTION
5. T = (H - F)/2
6. H = F + 2T

FIXED FASTENER SYSTEM UNEQUAL TOLERANCE DISTRIBUTION
7. T1 = H1 - (T2 + F)
8. H1 = T1 + T2 + F

STRAIGHTNESS ON A UNIT BASIS



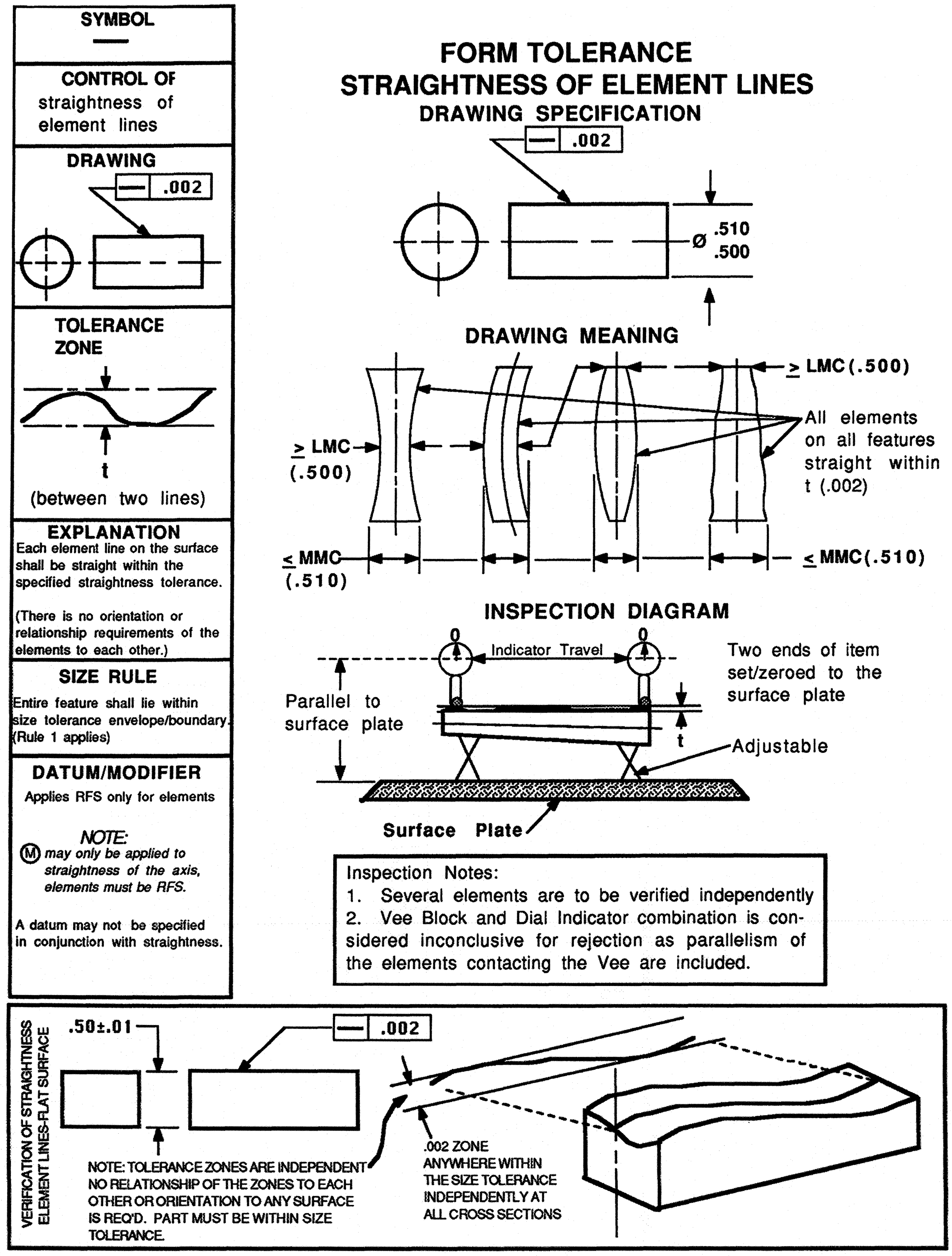
$$r = \frac{C^2}{8h} + \frac{h}{2}$$

$$h = r - \sqrt{r^2 - \frac{C^2}{4}}$$

FEATURE	TOLERANCE TYPE	SYMBOL	CHARACTERISTIC	DATUM REQ'D	MODIFIERS
INDIVIDUAL (SINGLE)	FORM (SHAPE)	⏤	STRAIGHTNESS	Not Allowed-Related to a perfect counterpart	Ⓜ ON AXIS
		⏥	FLATNESS		Ⓜ WHERE RULE 1 N/A
		○	CIRCULARITY		NONE
		⌭	CYLINDRICITY		
INDIVIDUAL OR RELATED	PROFILE (CONTOUR)	⌒	PROFILE OF A LINE	ALLOWABLE	ALLOWED ONLY ON DATUM
		⌓	PROFILE OF A SURFACE		
RELATED	ORIENTATION (ATTITUDE)	∠	ANGULARITY	REQUIRED	Ⓜ FOR FEATURES OR DATUMS OF SIZE
		⊥	PERPENDICULARITY		
		//	PARALLELISM		
	LOCATION	⌖	POSITION	PREFERRED	Ⓜ or Ⓛ REQUIRED
		◎	CONCENTRICITY	REQUIRED	NONE
	RUNOUT	↗	CIRCULAR RUNOUT	REQUIRED	
		⌰	TOTAL RUNOUT	REQUIRED	
MODIFYING SYMBOLS		Ⓜ	MAXIMUM MATERIAL CONDITION		MMC FOR FIT
		Ⓛ	LEAST MATERIAL CONDITION		LMC-ALIGNMENT
ADDITIONAL SYMBOLS		Ⓟ	PROJECTED TOLERANCE ZONE		TAPS/PRESS FITS
		⌀	DIAMETER (FACE OF DWG.)		OR CYLIND. ZONE
		S⌀	SPHERICAL DIAMETER		TOL. ON POINT
		R	RADIUS		DIMENSIONING
		SR	SPHERICAL RADIUS		DIMENSIONING
		( )	REFERENCE		MINIMIZE USAGE
		⌒	ARC LENGTH		ABOVE DIMENSION

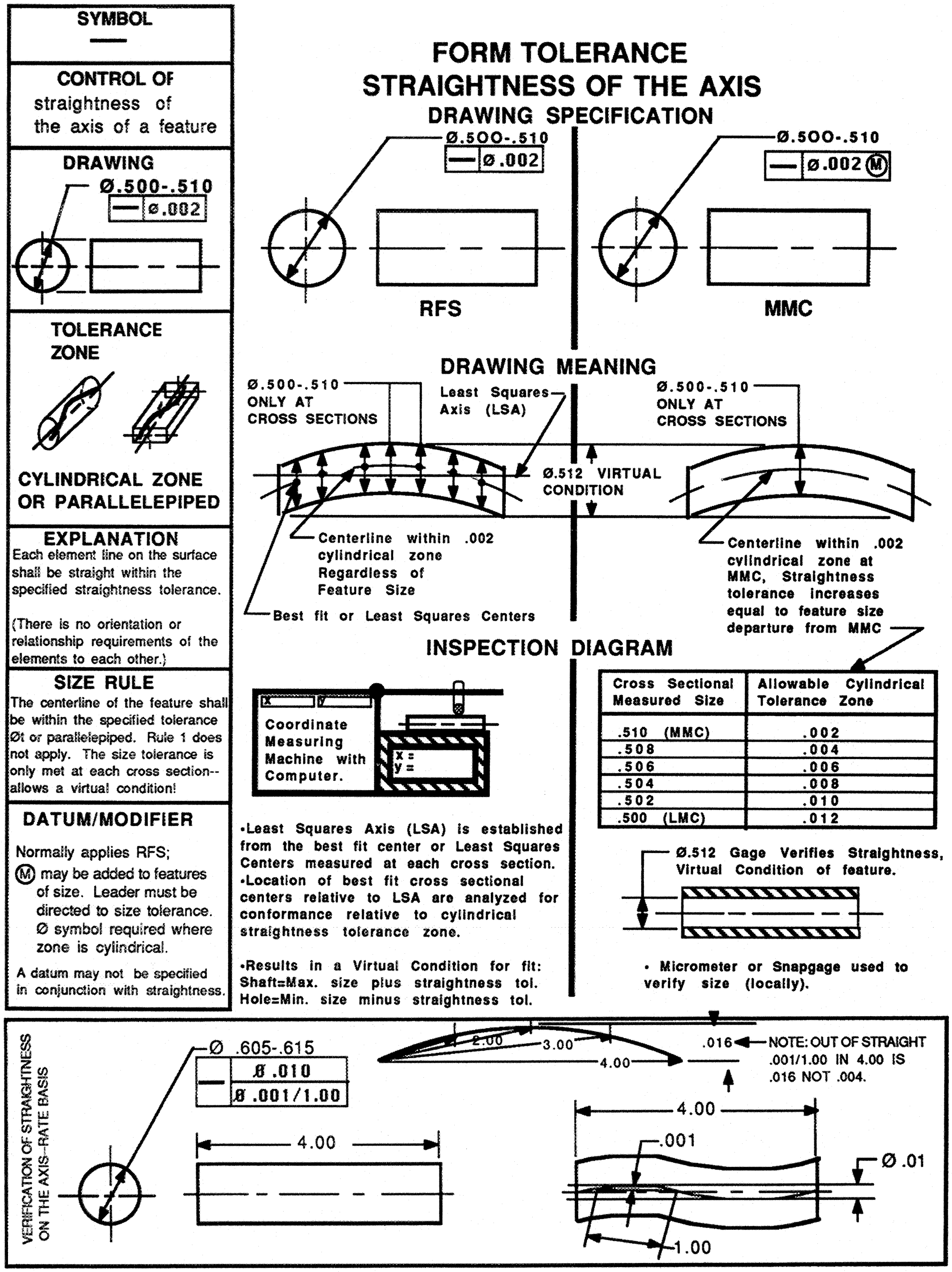
(a) Cover

FIGURE 16.31 Summary Fact Data Sheets



(b) Straightness of element lines

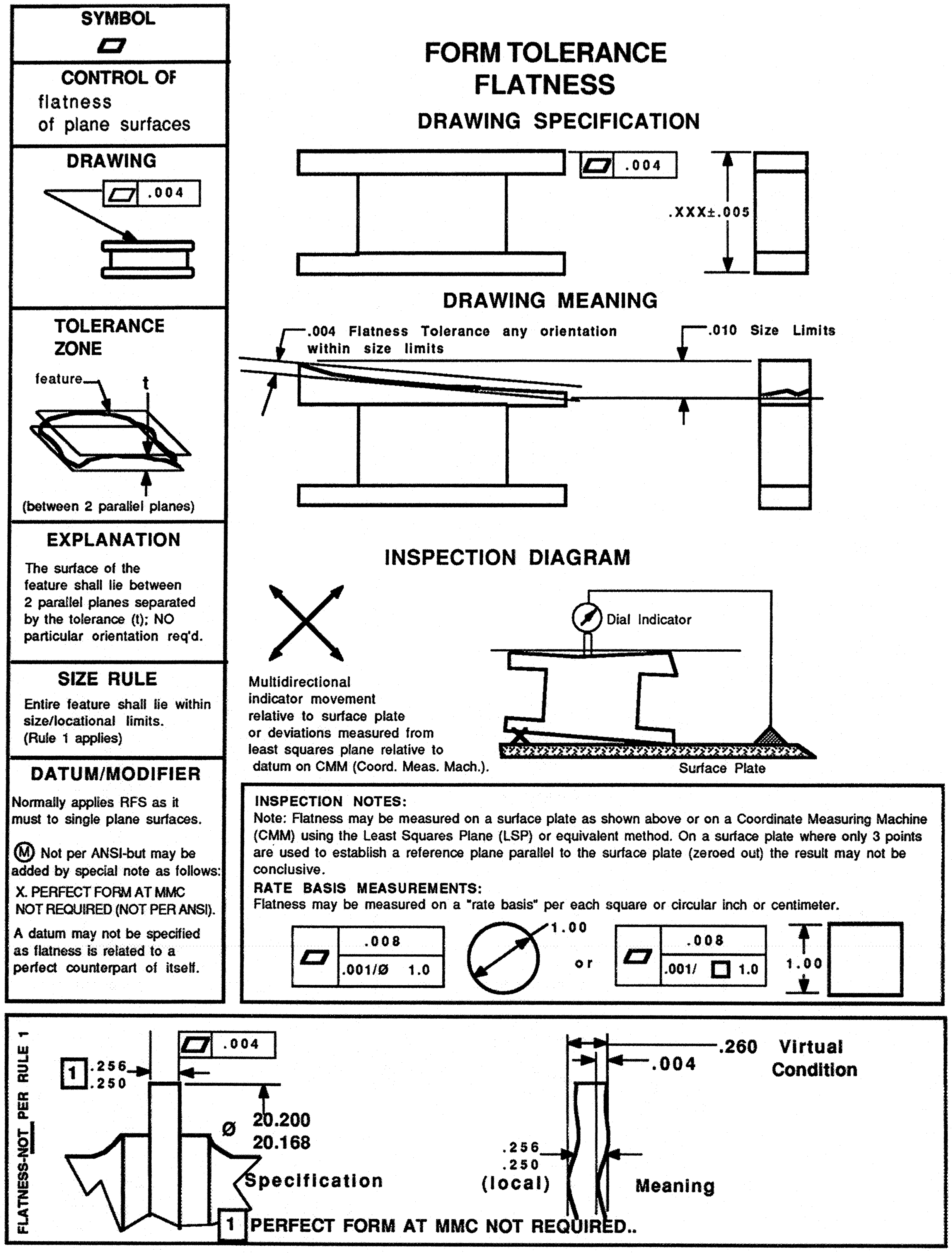
FIGURE 16.31 Summary Fact Data Sheets—*Continued*



Cross Sectional Measured Size	Allowable Cylindrical Tolerance Zone
.510 (MMC)	.002
.508	.004
.506	.006
.504	.008
.502	.010
.500 (LMC)	.012

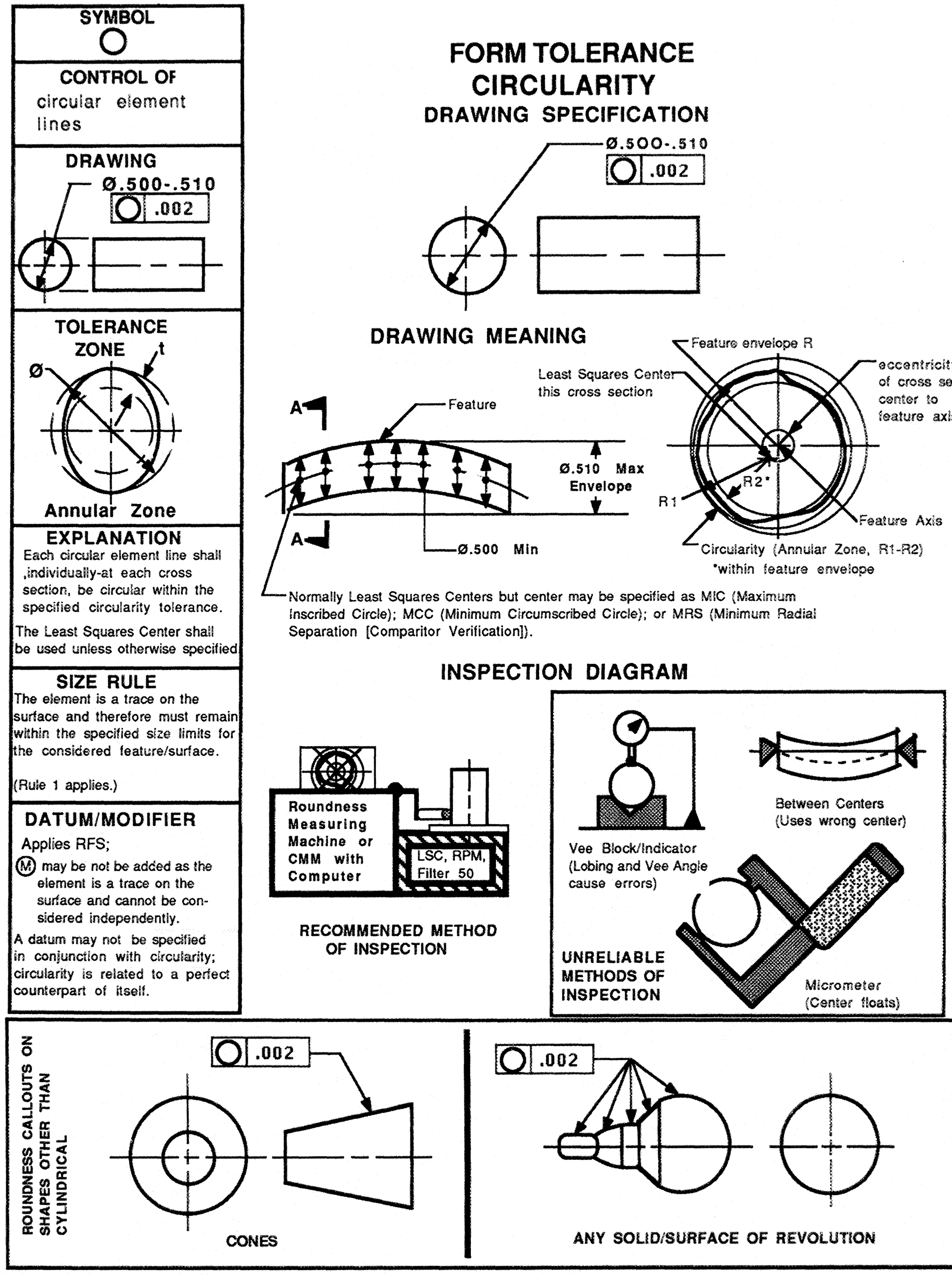
(c) Straightness of the axis

FIGURE 16.31 Summary Fact Data Sheets—*Continues*



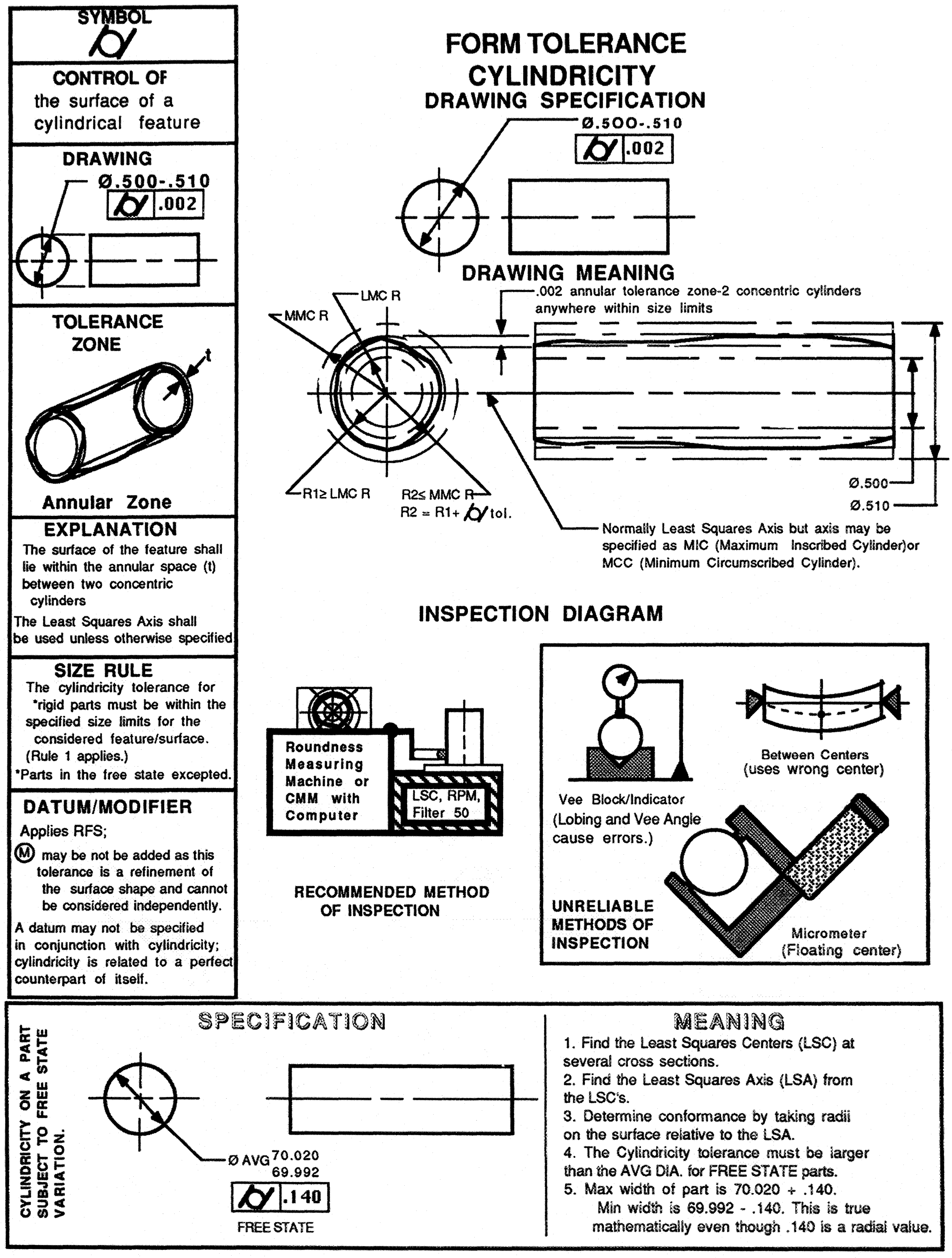
(d) Flatness

FIGURE 16.31 Summary Fact Data Sheets—*Continued*



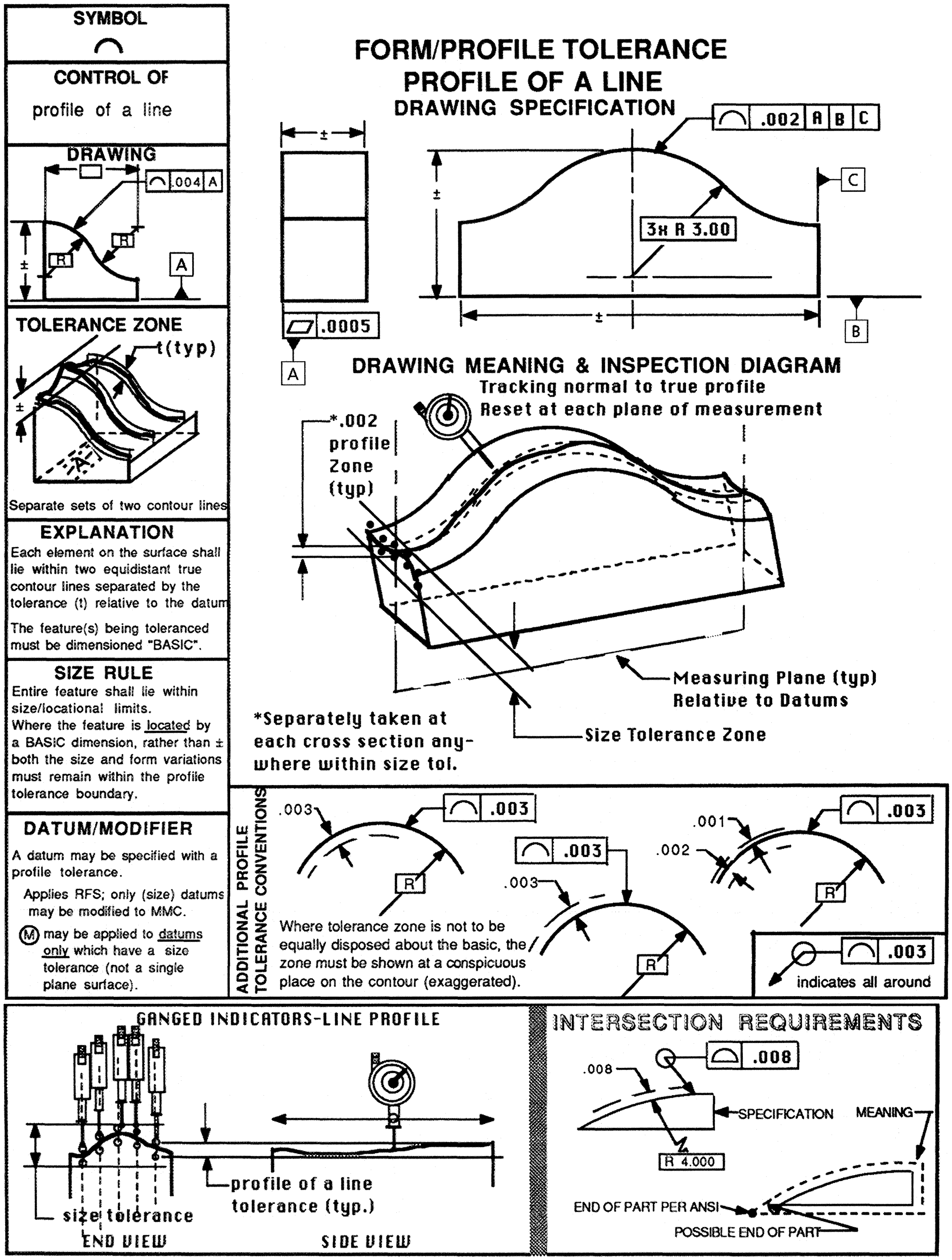
(e) Circularity

FIGURE 16.31 Summary Fact Data Sheets—*Continues*



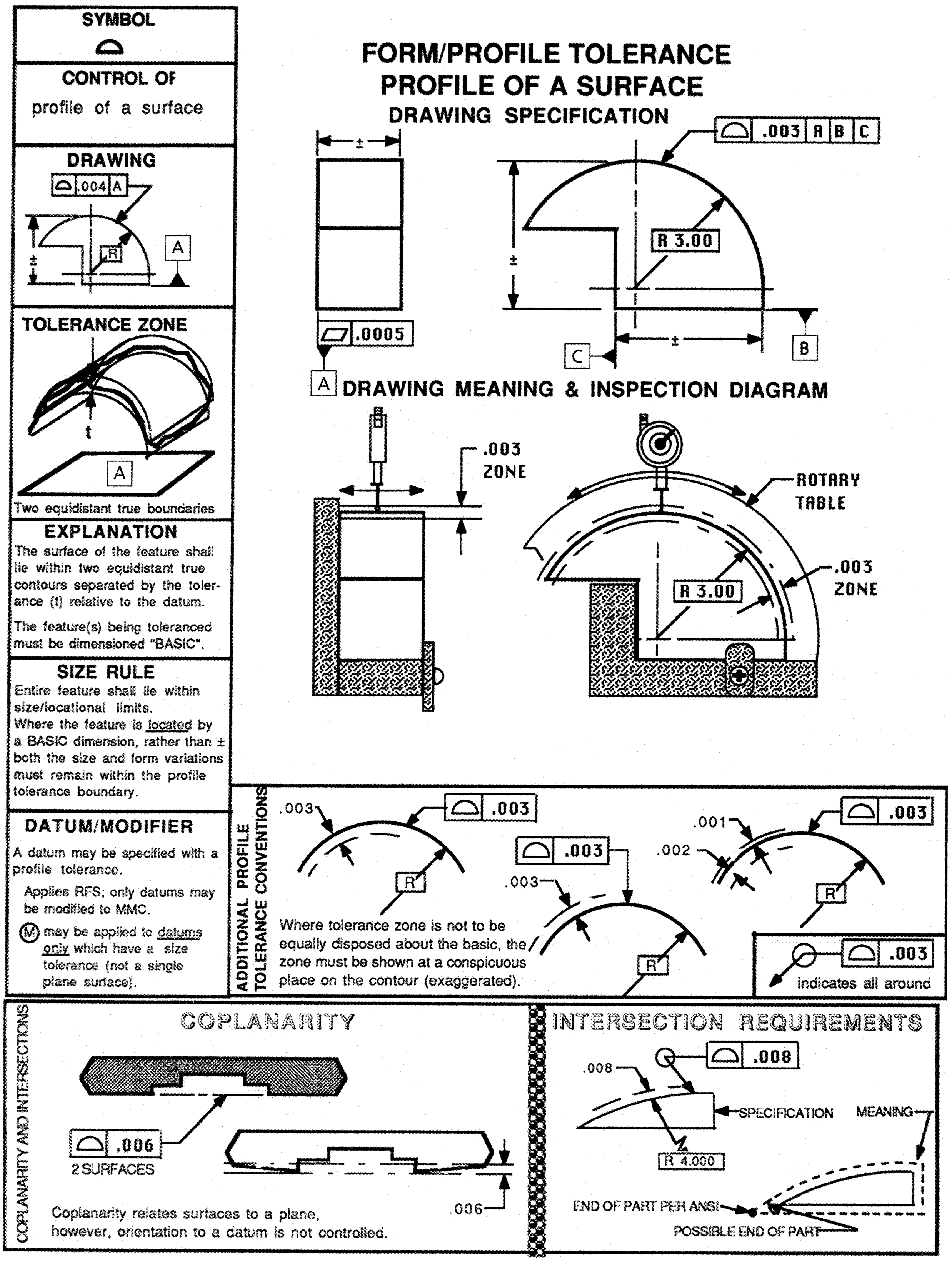
(f) Cylindricity

FIGURE 16.31 Summary Fact Data Sheets—*Continued*



(g) Profile of a line

FIGURE 16.31 Summary Fact Data Sheets—*Continues*



(h) Profile of a surface

FIGURE 16.31 Summary Fact Data Sheets—*Continues*

ORIENTATION TOLERANCE ANGULARITY

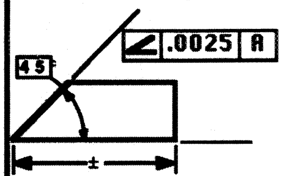
SYMBOL

∠

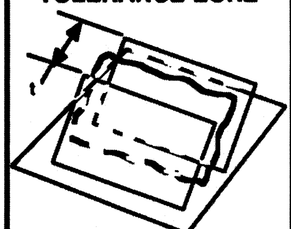
CONTROL OF

angularity of plane surfaces

DRAWING



TOLERANCE ZONE



two parallel planes

EXPLANATION

The surface of the feature shall lie between 2 parallel planes separated by the tolerance (t) which are at the basic angle to the datum.

SIZE RULE

The end of the part is measured to the end of the physical feature; not the zone intersection.

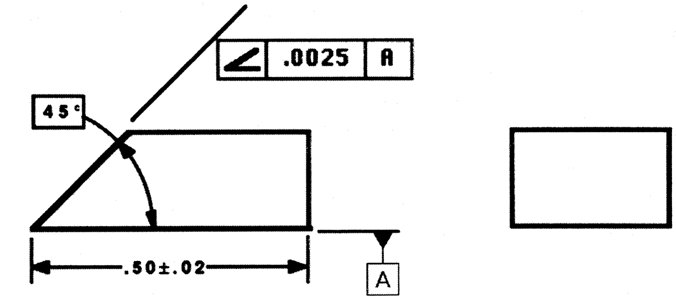
DATUM/MODIFIER

Normally applies RFS as it must to single plane surfaces.

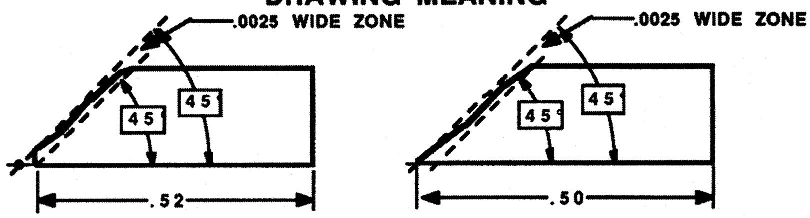
Ⓜ may be applied to datums or features which have a size tolerance (not a single plane surface).

A datum must be specified with angularity even though it may be obvious.

DRAWING SPECIFICATION

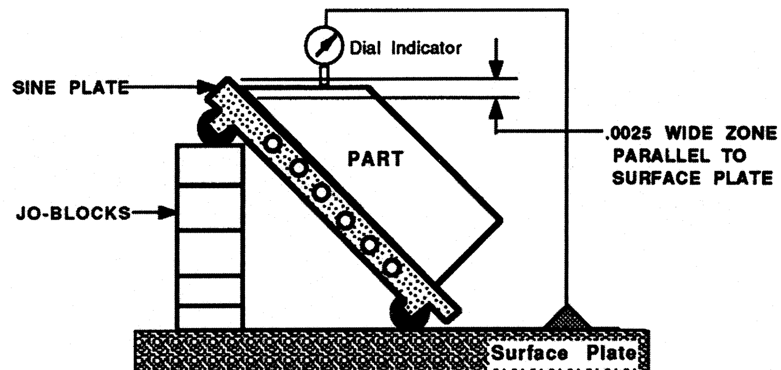


DRAWING MEANING



NOTE: LENGTH IS MEASURED INDEPENDENT OF THE ANGLE.

INSPECTION DIAGRAM

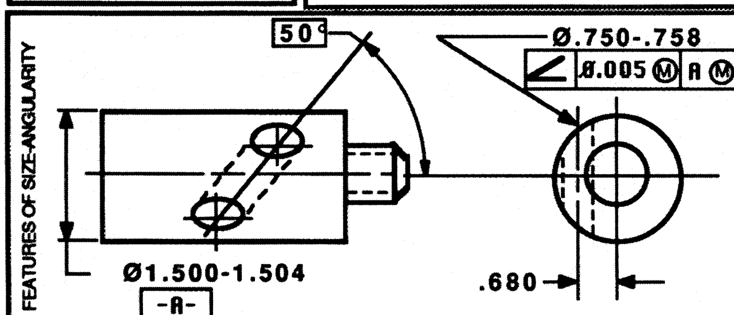


INSPECTION AT MMC

Where features or datums (or both) of size are specified angular at MMC the angularity tolerance increases as the feature or datum (or both) depart from MMC by a like amount. (e.g. A male feature .502 ±.002 with an actual size of .502; and a male datum of .754±.004 with an actual size of .754 which are

∠ .005Ⓜ AⓂ may be .002+.004+.005 out of angularity (.011).

FEATURES OF SIZE-ANGULARITY



Examples of feature/datum sizes vs. allowable Angularity

Actual Datum Diameter	Actual Feature Diameter	Allowable Angularity
1.504 (MMC)	.750 (MMC)	.005
1.503	.751	.007
1.502	.752	.009
1.501	.753	.011
1.500	.754	.013
1.500	.755	.014
1.500	.756	.015
1.500	.757	.016
1.500	.758	.017
1.502	.750 (MMC)	.007

(i) Angularity

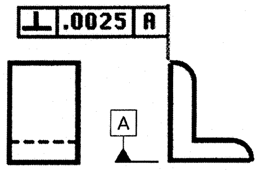
FIGURE 16.31 Summary Fact Data Sheets—*Continues*

SYMBOL

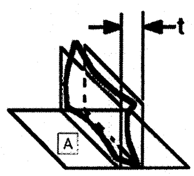
⊥

CONTROL OF
perpendicularity of plane surfaces

DRAWING



TOLERANCE ZONE



two parallel planes

EXPLANATION
The surface of the feature shall lie between 2 parallel planes separated by the tolerance (t) which are at 90° to the datum.

SIZE RULE
Feature shall be separately within locational (&/or size) tolerance where perpendicularity is specified.

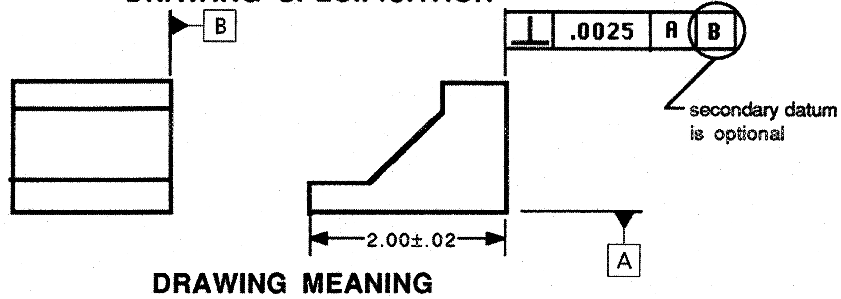
DATUM/MODIFIER
Normally applies RFS as it must to single plane surfaces.

Ⓜ may be applied to datums or features which have a size tolerance (not a single plane surface).

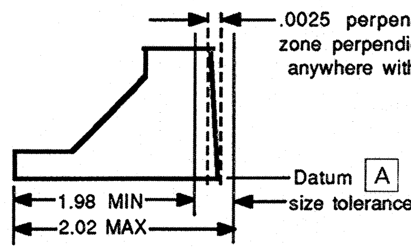
A datum must be specified with perpendicularity even though it may be obvious.

ORIENTATION TOLERANCE PERPENDICULARITY

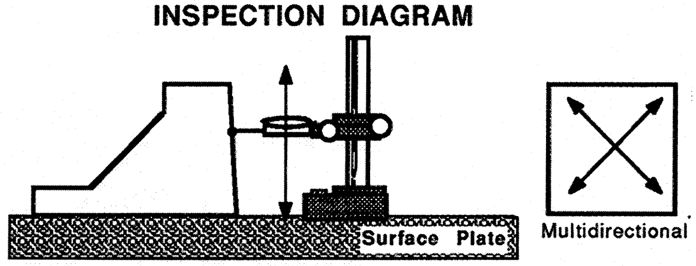
DRAWING SPECIFICATION



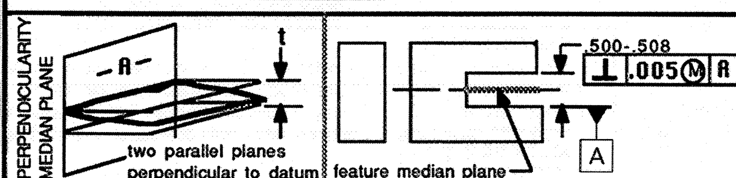
DRAWING MEANING



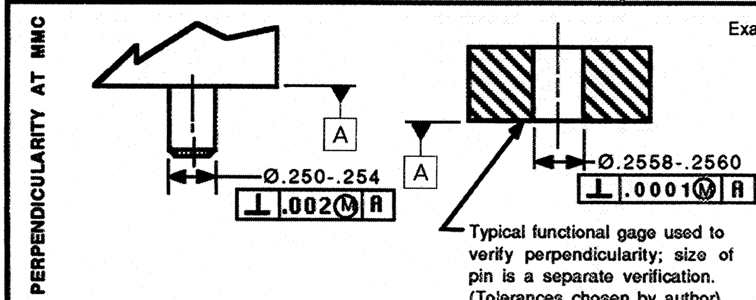
INSPECTION DIAGRAM



INSPECTION AT MMC
Where features or datums (or both) of size are specified perpendicular at MMC the perpendicularity tolerance increases as the feature or datum (or both) depart from MMC by a like amount.
(e.g. A male feature .502 ±.002 with an actual size of .502; and a male datum of .754±.004 with an actual size of .754 which are ⊥ .005Ⓜ AⓂ may be .002+.004+.005 out of perpendicularity (.011).



Feature median plane shall be within .005 at MMC of the feature -as the feature departs from MMC the perpendicularity zone for the median plane increases a like amt.



Typical functional gage used to verify perpendicularity; size of pin is a separate verification. (Tolerances chosen by author)

Examples of feature sizes vs. allowable Perpendicularity

Actual Feature Diameter	Allowable Perpendicularity	Virtual Condition
.254 (MMC)	.002	.256
.253	.003	.256
.252	.004	.256
.251	.005	.256
.250	.006	.256
.2535	.0025	.256

(j) Perpendicularity

IGURE 16.31 Summary Fact Data Sheets—*Continued*

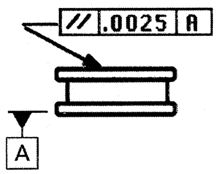
SYMBOL

//

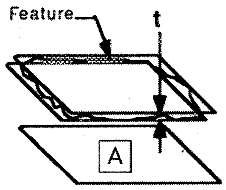
CONTROL OF

parallelism
of plane surfaces

DRAWING



TOLERANCE ZONE



EXPLANATION

The surface of the feature shall lie between 2 parallel planes separated by the tolerance (t) which are parallel to the datum.

F **SIZE RULE**

Entire feature shall lie within size/locational limits. (Rule 1 applies)

DATUM/MODIFIER

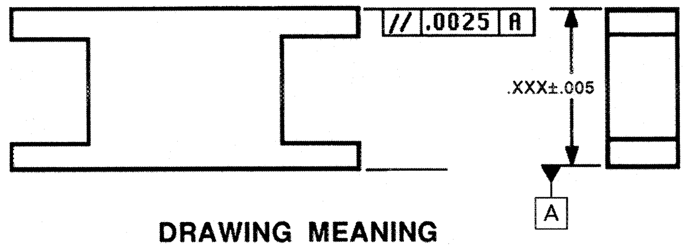
Normally applies RFS as it must to single plane surfaces.

Ⓜ may be applied to datums or features which have a size tolerance (not a single plane surface).

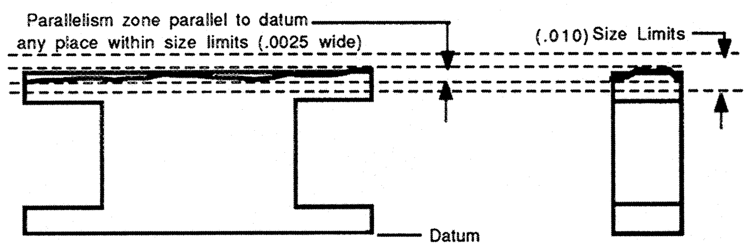
A datum must be specified with parallelism even though it may be obvious.

ORIENTATION TOLERANCE PARALLELISM

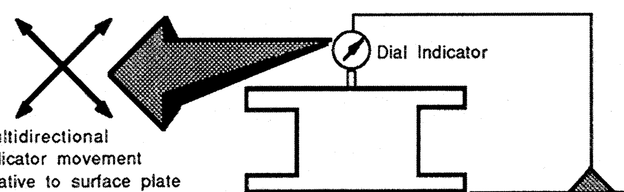
DRAWING SPECIFICATION



DRAWING MEANING



INSPECTION DIAGRAM



Multidirectional indicator movement relative to surface plate or deviations measured from least squares plane relative to datum on CMM (Coord. Meas. Mach.).

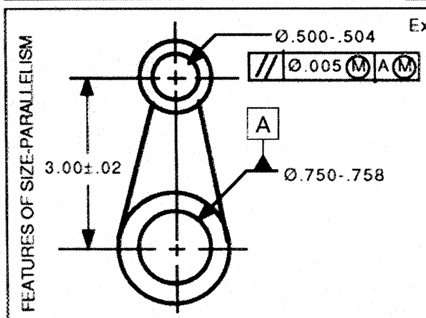
INSPECTION NOTES:

Note: Parallelism must be relative to a datum (point, line, or plane) therefore (unlike the dictionary definition) curved equidistant lines are NOT considered parallel.

INSPECTION AT MMC

Where features or datums (or both) of size are specified parallel at MMC the parallelism tolerance increases as the feature or datum (or both) depart from MMC by a like amount. (e.g. A male feature .502 ±.002 with an actual size of .502; and a male datum of .754±.004 with an actual size of .754 which are // .005 Ⓜ A Ⓜ may be .002+.004+.005 out of parallel (.011).

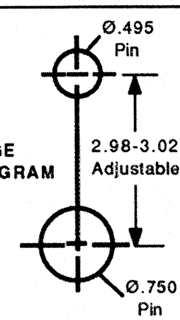
FEATURES OF SIZE-PARALLELISM



Examples of feature/datum sizes vs. allowable parallelism

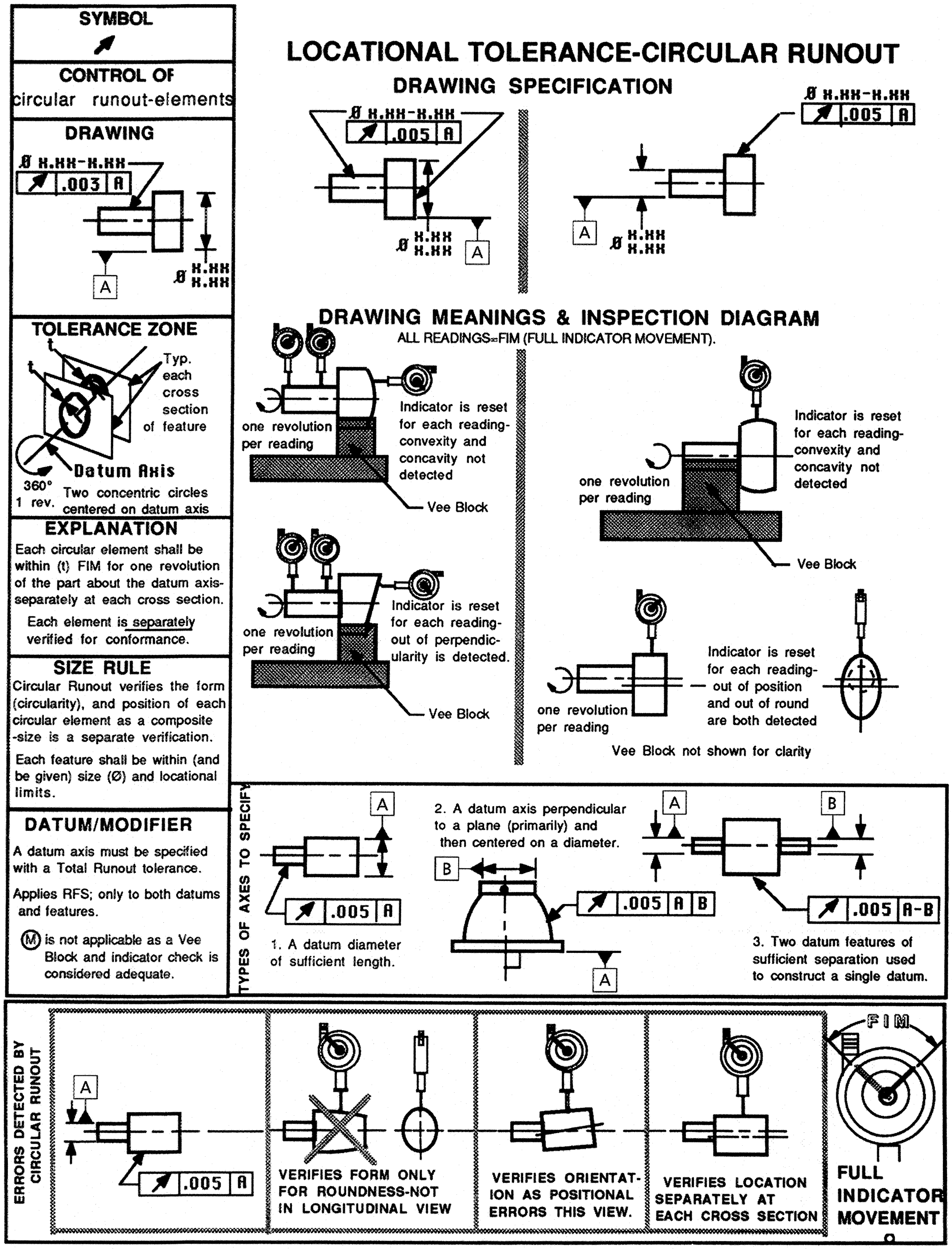
Actual Datum Diameter	Actual Feature Diameter	Allowable Parallelism
.500 (MMC)	.750 (MMC)	.005
.501	.751	.007
.502	.752	.009
.503	.753	.011
.504	.754	.013
.504	.755	.014
.504	.756	.015
.504	.757	.016
.504	.758	.017
.502	.750 (MMC)	.007

GAGE DIAGRAM



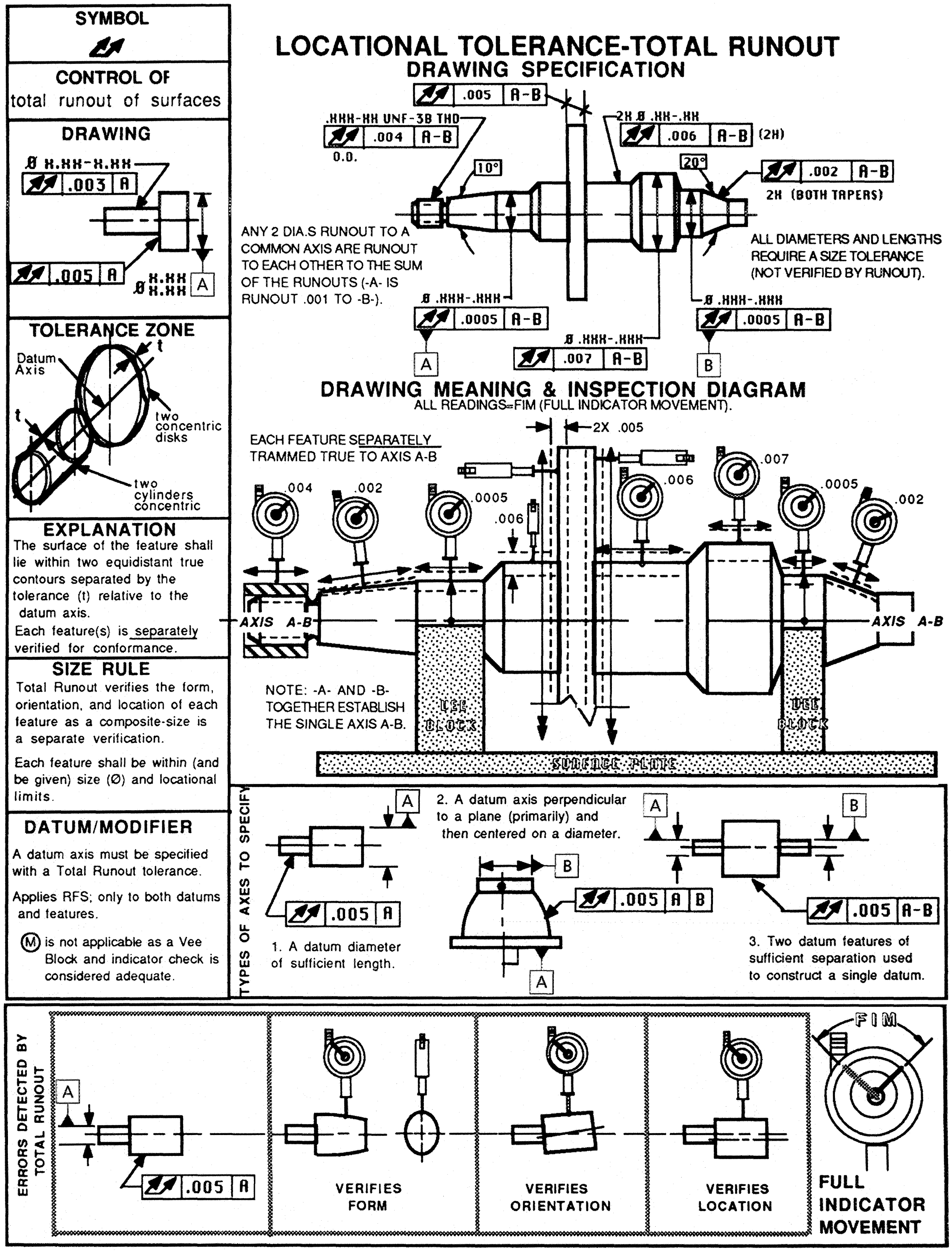
(k) Parallelism

FIGURE 16.31 Summary Fact Data Sheets—*Continues*



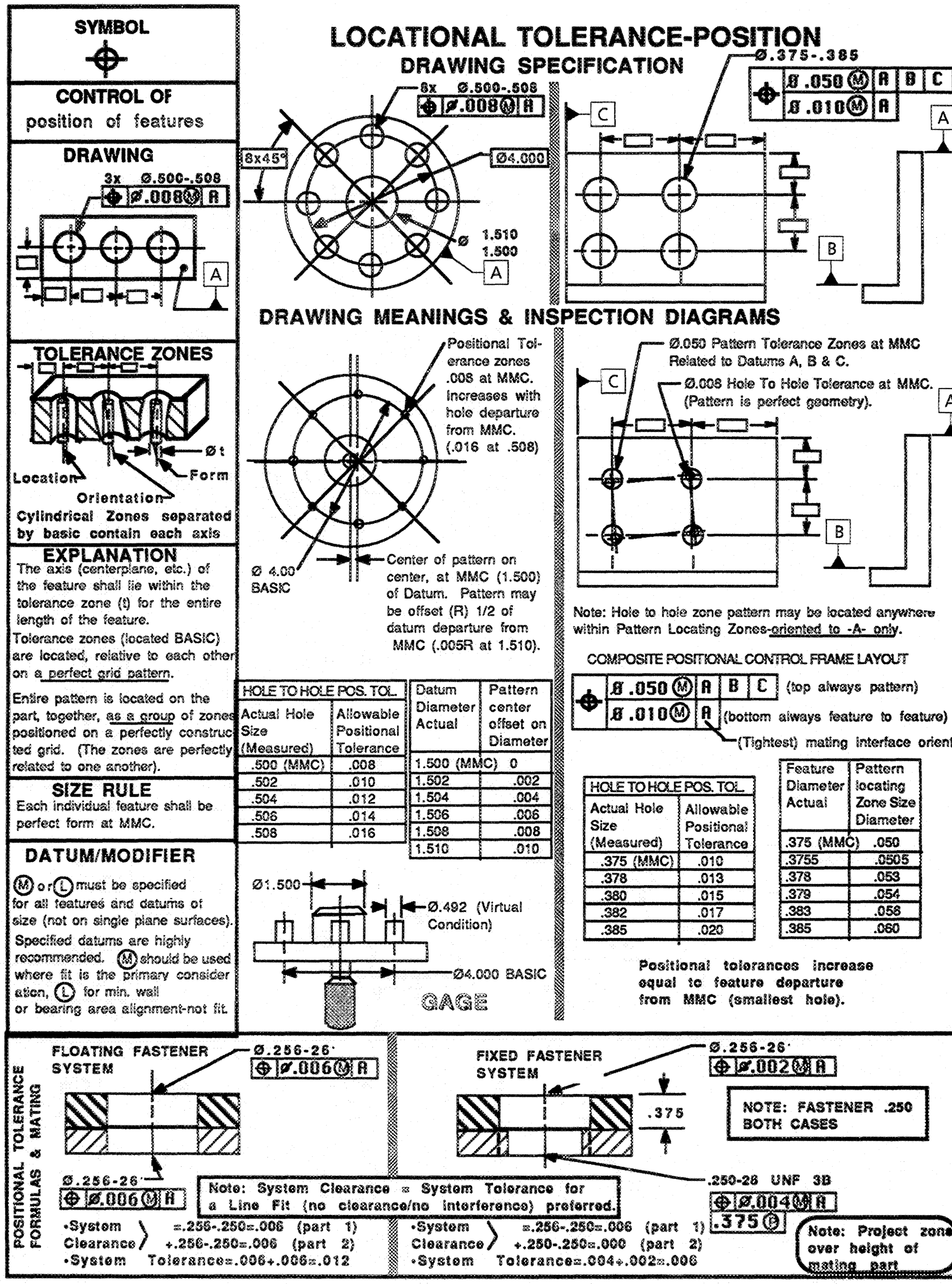
(l) Circular runout

FIGURE 16.31 Summary Fact Data Sheets—*Continued*



(m) Total runout

FIGURE 16.31 Summary Fact Data Sheets—*Continues*



LOCATIONAL TOLERANCE-POSITION

SYMBOL

CONTROL OF
position of features

DRAWING

DRAWING SPECIFICATION

DRAWING MEANINGS & INSPECTION DIAGRAMS

EXPLANATION

The axis (centerplane, etc.) of the feature shall lie within the tolerance zone (t) for the entire length of the feature.

Tolerance zones (located BASIC) are located, relative to each other on a perfect grid pattern.

Entire pattern is located on the part, together, as a group of zones positioned on a perfectly constructed grid. (The zones are perfectly related to one another).

SIZE RULE

Each individual feature shall be perfect form at MMC.

DATUM/MODIFIER

Ⓜ or Ⓛ must be specified for all features and datums of size (not on single plane surfaces). Specified datums are highly recommended. Ⓜ should be used where fit is the primary consideration, Ⓛ for min. wall or bearing area alignment-not fit.

HOLE TO HOLE POS. TOL.	
Actual Hole Size (Measured)	Allowable Positional Tolerance
.500 (MMC)	.008
.502	.010
.504	.012
.506	.014
.508	.016

Datum Diameter Actual	Pattern center offset on Diameter
1.500 (MMC)	0
1.502	.002
1.504	.004
1.506	.006
1.508	.008
1.510	.010

COMPOSITE POSITIONAL CONTROL FRAME LAYOUT

Ø.050 Ⓜ A B C (top always pattern)
Ø.010 Ⓜ A (bottom always feature to feature)
(Tightest) mating interface orient

HOLE TO HOLE POS. TOL.	
Actual Hole Size (Measured)	Allowable Positional Tolerance
.375 (MMC)	.010
.378	.013
.380	.015
.382	.017
.385	.020

Feature Diameter Actual	Pattern locating Zone Size Diameter
.375 (MMC)	.050
.3755	.0505
.378	.053
.379	.054
.383	.058
.385	.060

Positional tolerances increase equal to feature departure from MMC (smallest hole).

POSITIONAL TOLERANCE FORMULAS & MATING

NOTE: FASTENER .250 BOTH CASES

Note: System Clearance = System Tolerance for a Line Fit (no clearance/no interference) preferred.

Floating fastener system:
•System Clearance = .256-.250=.006 (part 1) +.256-.250=.006 (part 2)
•System Tolerance=.006+.006=.012

Fixed fastener system:
•System Clearance = .256-.250=.006 (part 1) +.250-.250=.000 (part 2)
•System Tolerance=.004+.002=.006

Note: Project zone over height of mating part

(n) position

FIGURE 16.31 Summary Fact Data Sheets—*Continued*

LOCATIONAL TOLERANCE CONCENTRICITY

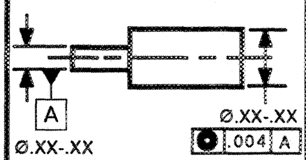
SYMBOL

◎

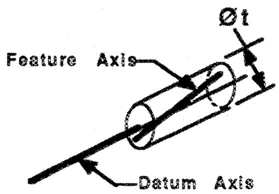
CONTROL OF

location of the axis of a feature to the axis of a datum

DRAWING



TOLERANCE ZONE



(cylindrical zone)

EXPLANATION

The axis of the feature shall be within a cylindrical zone (Øt) which is collinear to the datum axis.

The Least Squares Axes shall be used unless otherwise specified.

SIZE RULE

The feature (and datum) must be within the specified size limits. (Rule 1 applies to each.)

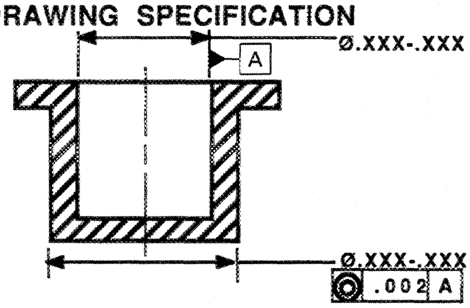
DATUM/MODIFIER

Always applies RFS;

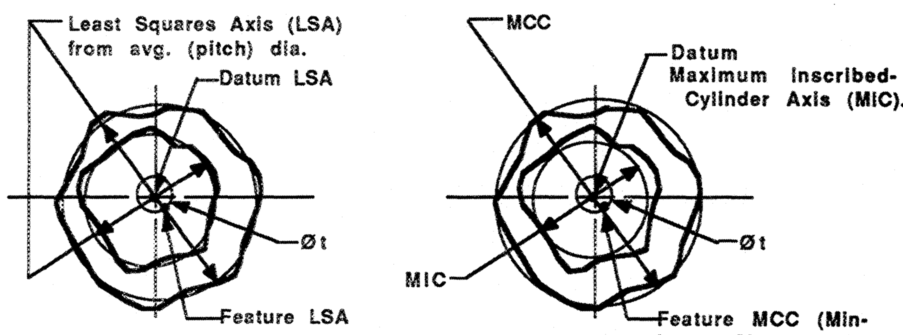
Ⓜ may be not be added as only the axis of the feature and the datum are involved and these are not subject to variations of size.

A datum (axis) must be specified in conjunction with concentricity; this is required even where the datum would be obvious.

DRAWING SPECIFICATION



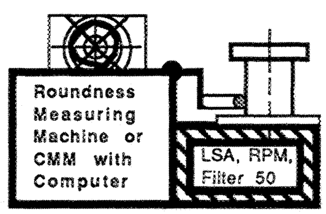
DRAWING MEANING



ALTERNATIVE-BY SPECIFICATION

INSPECTION DIAGRAM

LSA FOUND FROM LEAST SQUARES CENTERS AT VARIOUS CROSS SECTIONS.



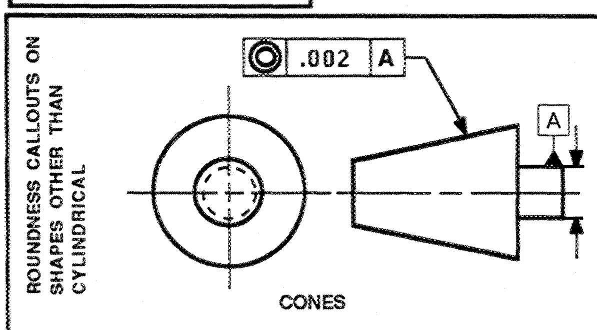
RECOMMENDED METHOD OF INSPECTION

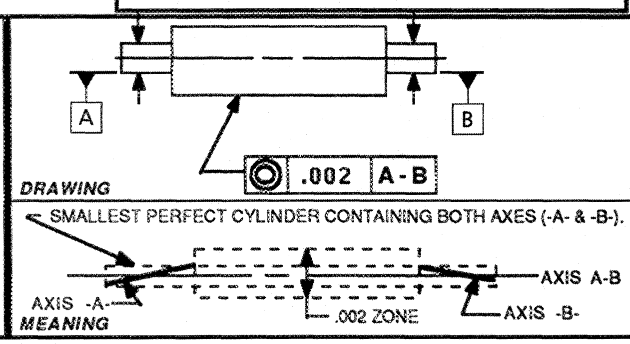
1. THE ALTERNATIVE METHOD OF INSPECTION FINDS THE AXIS OF A MALE FEATURE WITH A COMPARITOR, ADJUSTABLE RING (COLLETT) (OR BY CMM) AND THE INTERNAL BY USING AN EXPANDING MANDREL, FIT PIN, COMPARITOR (OR BY CMM OR ROUNDNESS MEASURING MACHINE).

2. THIS METHOD IS INVOKED BY DIRECT SPECIFICATION ON THE DRAWING SUCH AS: "X." DETERMINE AXIS BY MIC (MAXIMUM INSCRIBED CYLINDER) METHOD.

ALTERNATIVE METHOD OF INSPECTION

ROUNDNESS CALLOUTS ON SHAPES OTHER THAN CYLINDRICAL



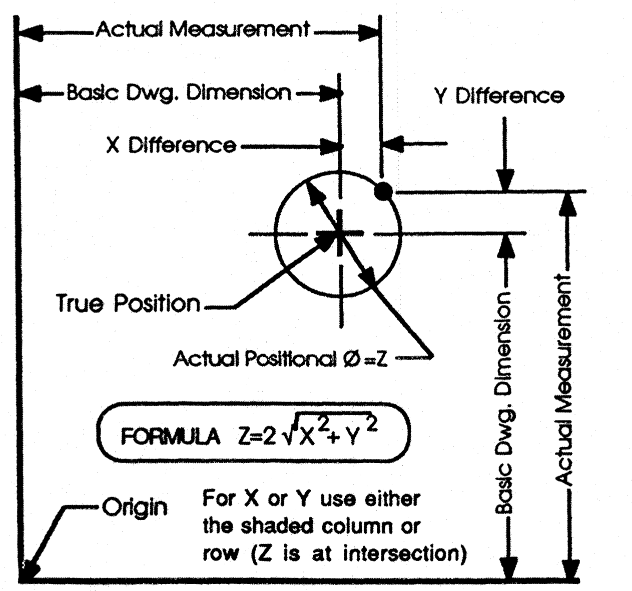


(o) Concentricity

FIGURE 16.31 Summary Fact Data Sheets—*Continues*

.020	.040200	.040450	.040790	.041230	.041760	.042380	.043080	.043860	.044720	.045650	.046650	.047710	.048830	.050000	.051220	.052500	.053810	.055170	.056570
.019	.038210	.038470	.038830	.039290	.039850	.040500	.041230	.042050	.042940	.043910	.044940	.046040	.047200	.048410	.049680	.050990	.052350	.053740	.055170
.018	.036220	.036500	.036880	.037360	.037950	.038630	.039400	.040250	.041180	.042190	.043270	.044410	.045610	.046860	.048170	.049520	.050910	.052350	.053810
.017	.034230	.034530	.034930	.035440	.036060	.036770	.037580	.038470	.039450	.040500	.041620	.042800	.044050	.045340	.046690	.048080	.049520	.050990	.052500
.016	.032250	.032560	.032980	.033530	.034180	.034930	.035780	.036720	.037740	.038830	.040000	.041230	.042520	.043860	.045250	.046690	.048170	.049680	.051220
.015	.030270	.030590	.031050	.031620	.032310	.033110	.034000	.034990	.036060	.037200	.038420	.039700	.041040	.042430	.043860	.045340	.046860	.048410	.050000
.014	.028280	.028640	.029120	.029730	.030460	.031300	.032250	.033290	.034410	.035610	.036880	.038210	.039600	.041040	.042520	.044050	.045610	.047200	.048830
.013	.026310	.026680	.027200	.027860	.028640	.029530	.030530	.031620	.032800	.034060	.035380	.036770	.038210	.039700	.041230	.042800	.044410	.046040	.047710
.012	.024330	.024740	.025300	.026000	.026830	.027780	.028840	.030000	.031240	.032560	.033940	.035380	.036880	.038420	.040000	.041620	.043270	.044940	.046650
.011	.022360	.022800	.023410	.024170	.025060	.026080	.027200	.028430	.029730	.031110	.032560	.034060	.035610	.037200	.038830	.040500	.042190	.043910	.045650
.010	.020400	.020880	.021540	.022360	.023320	.024410	.025610	.026910	.028280	.029730	.031240	.032800	.034410	.036060	.037740	.039450	.041180	.042940	.044720
.009	.018440	.018970	.019700	.020590	.021630	.022800	.024080	.025460	.026910	.028430	.030000	.031620	.033290	.034990	.036720	.038470	.040250	.042050	.043860
.008	.016490	.017090	.017890	.018870	.020000	.021260	.022630	.024080	.025610	.027200	.028840	.030530	.032250	.034000	.035780	.037580	.039400	.041230	.043080
.007	.014560	.015230	.016120	.017200	.018440	.019800	.021260	.022800	.024410	.026080	.027780	.029530	.031300	.033110	.034930	.036770	.038630	.040500	.042380
.006	.012650	.013420	.014420	.015620	.016970	.018440	.020000	.021630	.023320	.025060	.026830	.028640	.030460	.032310	.034180	.036060	.037950	.039850	.041760
.005	.010770	.011660	.012810	.014140	.015620	.017200	.018870	.020590	.022360	.024170	.026000	.027860	.029730	.031620	.033530	.035440	.037360	.039290	.041230
.004	.008940	.010000	.011310	.012810	.014420	.016120	.017890	.019700	.021540	.023410	.025300	.027200	.029120	.031050	.032980	.034930	.036880	.038830	.040790
.003	.007210	.008490	.010000	.011660	.013420	.015230	.017090	.018970	.020880	.022800	.024740	.026680	.028640	.030590	.032560	.034530	.036500	.038470	.040450
.002	.005660	.007210	.008940	.010770	.012650	.014560	.016490	.018440	.020400	.022360	.024330	.026310	.028280	.030270	.032250	.034230	.036220	.038210	.040200
.001	.002	.003	.004	.005	.006	.007	.008	.009	.010	.011	.012	.013	.014	.015	.016	.017	.018	.019	.020

Coordinate (CMM) to Positional Zone Conversion (Chart Above).



FORMULA $Z=2\sqrt{X^2+Y^2}$

For X or Y use either the shaded column or row (Z is at intersection)

THE RULES IN PLAIN ENGLISH

Rule 1: For individual features of size the MMC boundary of perfect form is respected over the entire feature length,the other (LMC) limit is verified locally by 2 point (caliper) measurement. (Not totally reliable in practice-form tolerance may be req'd).

RULE 2 : Position Tolerance Only- Modifier for MMC, RFS or LMC must be specified for features and datums of size in the feature control frame. Not for single plane surfaces.

Rule 3 : All Geometric Tolerances Except Position-RFS understood w/o specification MMC specified where allowed by design for economy/producibility.

Tolerances of Orientation or Position or Datum References are understood to apply to the Pitch Diameter of a thread unless otherwise specified (e.g. Ø MAJOR or Ø MINOR)

Tolerances apply at 68°F (20°C) unless otherwise specified.

Datum features establishing a Datum Reference Frame without orientation tolerances are considered oriented to 0 at MMC for Virtual Conditions used in Gagemaking.

VIRTUAL CONDITION
EXTERNAL FEATURE:
MMC Size + Geometric Tolerance
INTERNAL FEATURE:
MMC Size - Geometric Tolerance

⌴ Counterbore and Spotface
⌵ Countersink
↧ Depth/Deep
All Around
S Ø Spherical Dia
□ Square
Conical Taper
⌒ Arc Length
Origin Dimension
R Radius
Datum Target
Slope

(p) Conversion chart

FIGURE 16.31 Summary Fact Data Sheets—*Continued*

form, and orientation. Profile tolerances are specified as follows:

1. An appropriate view or section is drawn showing the desired basic profile.
2. Depending on design requirements, the tolerance may be divided bilaterally to both sides of the true profile or applied unilaterally to either side of the true profile. If an equal bilateral tolerance is intended, it is necessary to show only the feature control frame, with a leader directed to the surface. For a unilateral tolerance, phantom lines are drawn parallel to the true profile to indicate the line is extended to the feature control frame. If some segments of the profile are controlled by a profile tolerance and other segments by individually toleranced dimensions, the extent of the profile tolerance must be indicated.

16.7.1 Profile of a Line

A **profile of a line tolerance** specifies the limits of the boundaries of individual line elements of a surface [Fig. 16.31(g)]. The tolerance zone is two dimensional. BASIC dimensions define the true profile. The shape of the tolerance zone is two parallel boundaries offset above and below the true profile.

16.7.2 Profile of a Surface

A **profile of a surface tolerance** specifies the limits of all surface elements at the same time [Fig. 16.31(h)]. If the surface is made up of one or more basic curves, arcs, straight lines, or other shapes, all are described by BASIC dimensions. Some segments of a surface may be controlled by profile tolerancing and other segments by different tolerances. Reference letters serve to define the extent of a controlled segment, such as **FROM A TO B**. Points **A** and **B** are directed to the appropriate location on the surface. If the relationship of features is to be zero at MMC, specify by placing a 0 in the control frame or with a note, e.g., **PERFECT ORIENTATION REQUIRED AT MMC**. The orientation tolerance may then be equal to or less than the amount the feature deviates from MMC.

16.8 ORIENTATION TOLERANCES

Orientation tolerances cover parallelism, perpendicularity, and angularity. All require at least one datum specification. Some, such as perpendicularity, may involve an additional (secondary) datum. Orientation tolerances may employ element controls rather than surface requirements, in which case *each element of each radial element* is specified below the feature control frame.

Angularity, parallelism, perpendicularity, and in some instances profile are orientation tolerances applicable to related features. These tolerances control the orientation of features to one another. They are sometimes referred to as **attitude tolerances.** Relation to more than one datum feature should be considered if required to stabilize the tolerance zone in more than one direction. Note that angularity, perpendicularity, and parallelism, when applied to plane surfaces, control flatness if a flatness tolerance is not specified. Tolerance zones require an axis, or all elements of the surface, to fall within this zone.

16.8.1 Angularity

Angularity is the condition in which a surface or an axis is at a specific angle other than 90° from a datum plane or axis [Fig. 16.31(i)]. An **angularity tolerance** specifies one of the following:

- A tolerance zone defined by two parallel planes at the specified basic angle from a datum plane or axis within which the surface of the considered feature must lie
- A tolerance zone defined by two parallel planes at the specified basic angle from a datum plane or axis within which the axis the feature must lie

16.8.2 Perpendicularity

Perpendicularity is the condition in which a surface, center plane, or axis is at a right angle to a datum plane or axis [Fig. 16.31(j)]. A **perpendicularity tolerance** specifies one of the following:

- A tolerance zone defined by two parallel planes perpendicular to a datum plane or axis within which the surface or center plane of the considered feature must lie
- A tolerance zone defined by two parallel planes perpendicular to a datum axis within which the axis of the considered feature must lie
- A cylindrical tolerance zone perpendicular to a datum plane within which the axis of the considered feature must lie
- A tolerance zone defined by two parallel lines perpendicular to a datum plane or axis within which an element of the surface must lie

16.8.3 Parallelism

Parallelism is the condition in which a surface or axis is equidistant at all points from a datum plane or in which an axis is equidistant along its length from a datum axis [Fig. 16.31(k)]. A **parallelism tolerance** specifies one of the following:

- A tolerance zone defined by two planes or lines parallel to a datum plane or axis within which the line elements of the surface or axis of the feature must lie
- A cylindrical tolerance zone whose axis is parallel to a datum axis within which the axis of the feature must lie

16.9 RUNOUT TOLERANCES

A **runout tolerance** controls the functional relationship of one or more features of a part to a datum axis. The types of features controlled by runout tolerances include those surfaces constructed around a datum axis and those constructed at right angles to a datum axis.

Runout tolerances control the composite form, orientation, and position relative to a datum axis. Each feature must be within its runout tolerance when the part is rotated about the datum axis. The tolerance specified for a controlled surface is the total tolerance, or full indicator movement (FIM).

The two types of runout control are circular and total. The type specified depends on design requirements and manufacturing considerations. Circular runout is normally a less complex requirement than total runout.

16.9.1 Circular Runout

Circular runout is the condition of a circular element on the surface with respect to a fixed point during one complete revolution of the part about the datum axis [Fig. 16.31(l)]. Circular runout controls circular elements of a surface. The tolerance is applied independently at any circular cross section as the part is rotated 360°. If applied to surfaces constructed around a datum axis, circular runout can control the cumulative variations of circularity and coaxiality. If applied to surfaces at right angles to the datum axis, circular runout controls circular elements of a plane surface (*wobble*).

16.9.2 Total Runout

Total runout [Fig. 16.31(m)] is the condition of a surface with respect to a perfect counterpart of itself, perfectly oriented and positioned. The indicator is moved across the feature, relative to the desired geometry, as the part is rotated about the datum axis.

Total runout provides composite control of all surface elements. The tolerance is applied simultaneously to all circular and profile-measuring positions as the part is rotated 360°.

16.9.3 Position

Position is a total zone specification, such as a diameter or total width centered on the basic location of the axis, center plane, or center point of a feature, from the true position with respect to datum(s) [Fig. 16.31(n)].

Locating a hole with rectangular coordinates and plus–minus tolerances yields a square or rectangular zone. The worst-case location for the axis of the mating features is at the diagonal. Inscribing this square with a circle does not change the mating relationship, but it does yield a 58% greater area.

Another improvement in the positional tolerancing system is the change from a "chain" (feature-to-feature) basis to a "basic grid" system. Tolerance accumulations are therefore avoided (Fig. 16.32). The grid for a pattern of zones is perfect in all respects. The locations of each of these zones are in perfect relationship to each other. A grid is established by placing **BASIC dimensions** between the features. Figure

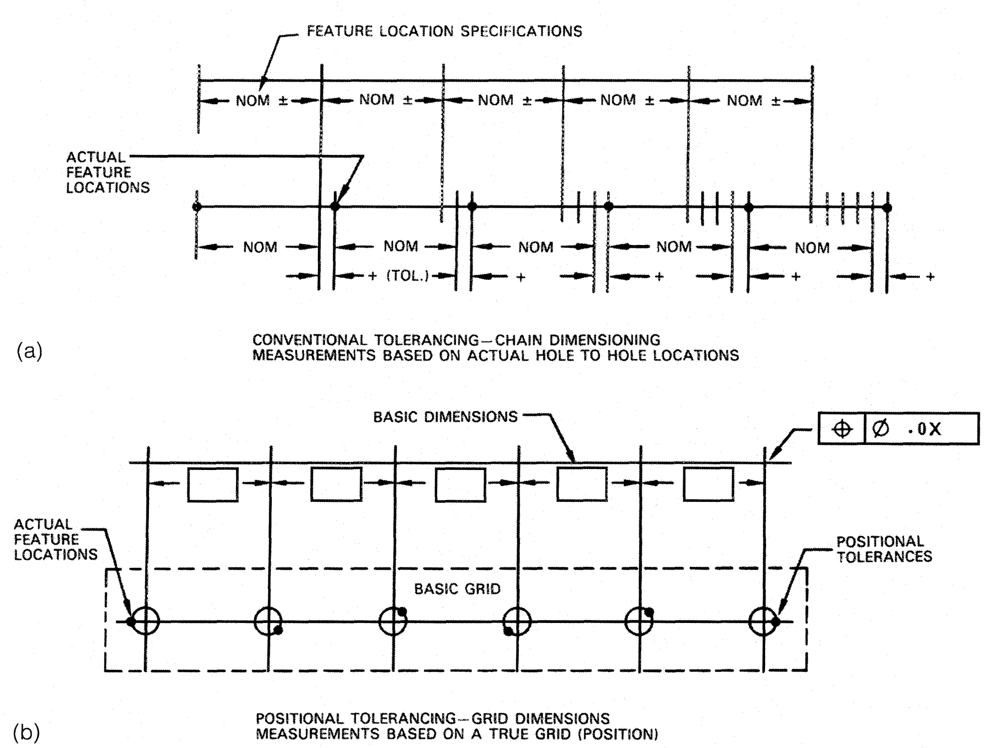


FIGURE 16.32
Conventional Chain Versus the Positional Grid System

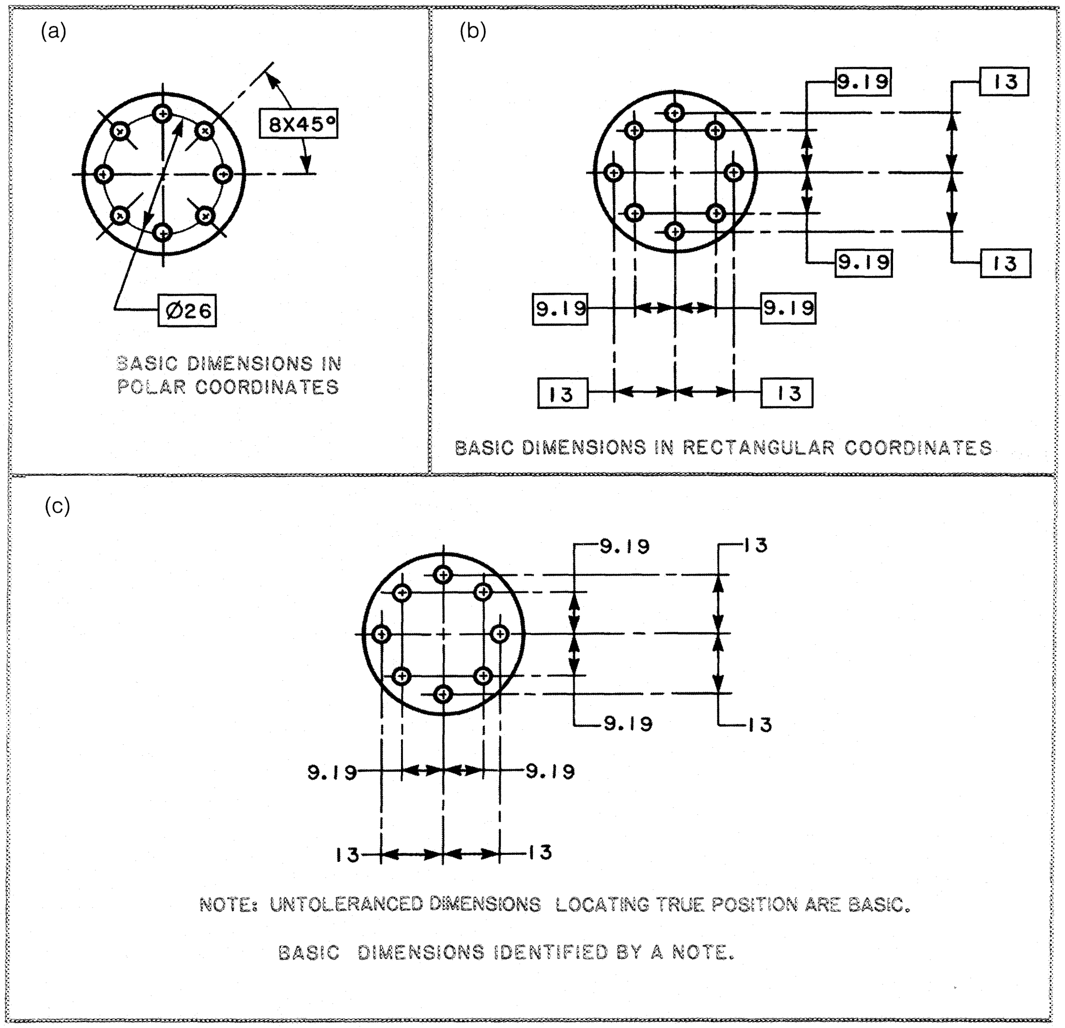


FIGURE 16.33 Identifying Basic Dimensions

16.33 shows examples of how to identify basic dimensions for patterns.

16.9.4 Positional Patterns

Figure 16.34 shows how **patterns** are located on parts. The preference is for composite positional tolerancing, as shown in the figure. The use of plus–minus dimensions to locate a pattern is *not* recommended.

16.10 LIMITS OF SIZE

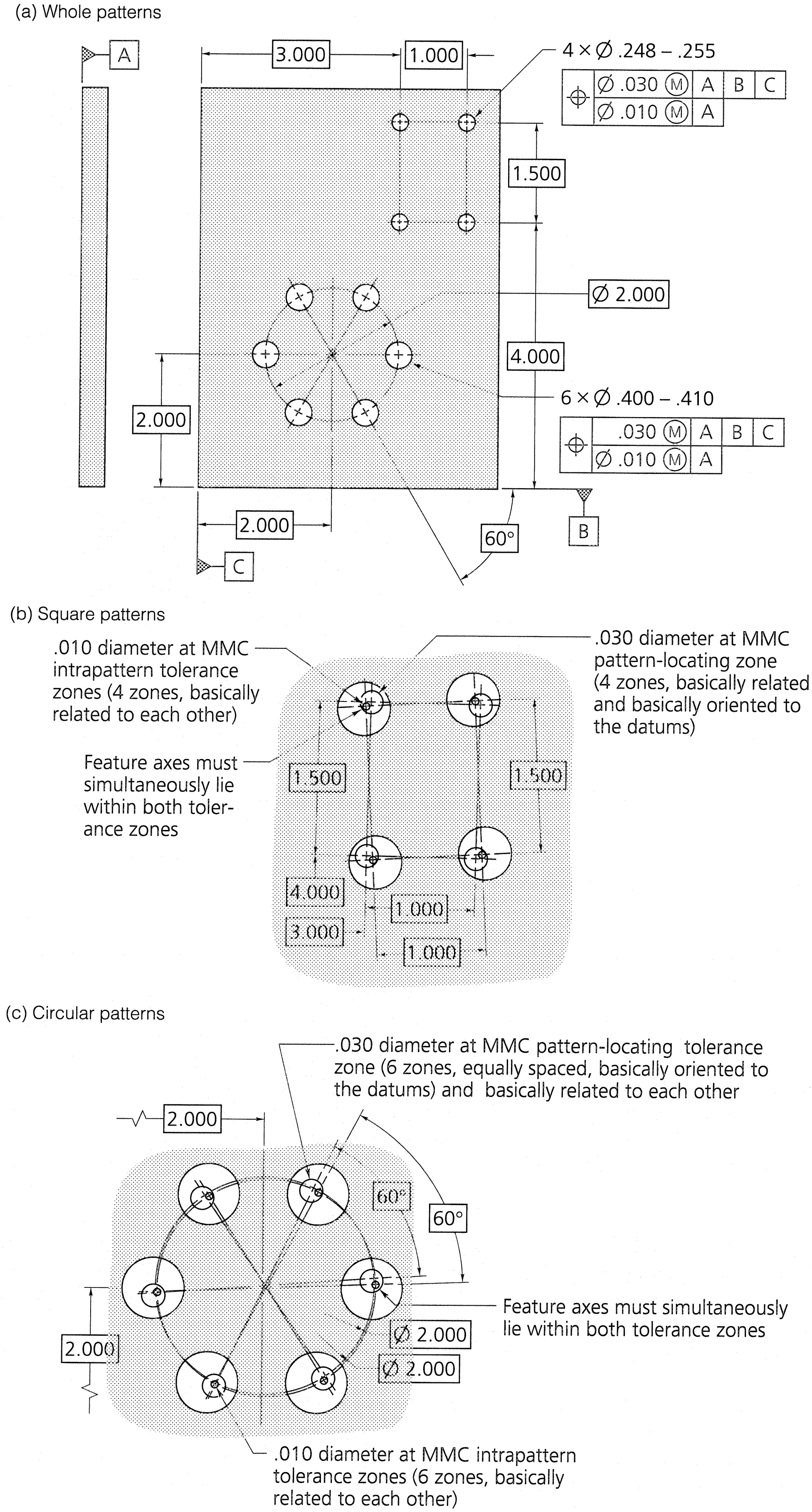
The **limits of size** of a feature describe the extent within which variations of geometric form are allowed. Where only a size tolerance is specified, the limits of size of an individual feature describe the extent to which variations in its geometric form, as well as size, are allowed.

The **actual size** of an individual feature at any cross section must be within the specified tolerance of size. The form of an individual feature is controlled by its limits of size.

The surface or surfaces of a feature must not extend beyond a boundary of perfect form at MMC. This boundary is the true geometric form represented by the drawing. No variation in form is permitted if the feature is produced at its MMC limit of size. Where the actual size of a feature has departed from MMC toward LMC, a variation in form is allowed equal to the amount of such departure. There is no requirement for a boundary of perfect form at LMC. Thus, a feature produced at its LMC limit of size is permitted to vary from true form to the maximum variation allowed by the boundary of perfect form at MMC. The control of geometric form by limits of size does not apply to the following:

- Stock such as bars, sheets, tubing, structural shapes, and other items produced to established industry or government standards that prescribe limits for straightness, flatness, or other geometric characteristics
- Parts subject to free state variation in the unrestrained condition

FIGURE 16.34 Composite Positional Tolerance to Datum Reference Frame



The limits of size do not control the orientation or location relationship among individual features. Features shown perpendicular, coaxial, or symmetrical to each other must be controlled for location or orientation. If it is necessary to establish a boundary of perfect form at MMC to control the relationship between features, use the following:

1. Specify a zero tolerance of orientation at MMC, including a datum reference (at MMC, if applicable), to control the angularity, perpendicularity, or parallelism of the feature.
2. Specify a zero positional tolerance at MMC, including a datum reference at MMC, to control coaxial or symmetrical features.
3. Indicate this control for the features involved with a note such as

   PERFECT ORIENTATION (or COAXIALITY or SYMMETRY) AT MMC REQUIRED FOR RELATED FEATURES.

4. Relate dimensions to a datum reference frame.

16.10.1 ISO Interpretation of Limits of Size

Where datums are specified in the ISO system, measurements are taken from the datums and made relative to them. Where datums are not specified, linear dimensions are intended to apply on a point-to-point basis or directly between the points indicated on the drawing. Unfortunately, caliper measurements float relative to one another and the exact shape is not known (Fig. 16.35). If the configuration is controlled, a form tolerance such as straightness or flatness is given.

Additionally, the direction of measurement can be a problem for a geometry that is not ideal. In Figure 16.36, the

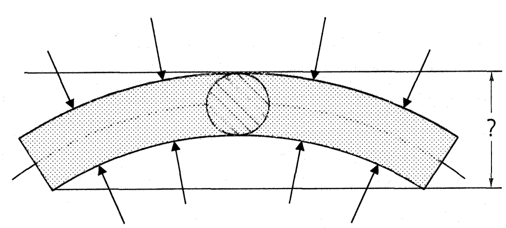


FIGURE 16.35 Caliper Measurements Do Not Measure Form. Illustration shows a cylindrical part.

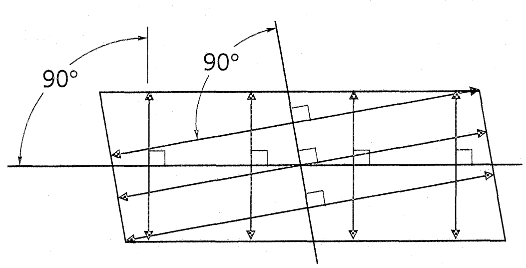


FIGURE 16.36 Measuring Orientation for Caliper Measurement. Illustration shows a rectangular part.

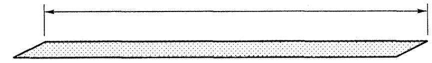


FIGURE 16.37 Measuring Thin Parts

vertical measurements are not perpendicular to the horizontal ones. If the sides of a part are not parallel, finding the center plane to orient measurements is another problem.

For thin parts, the rule changes to taking measurements parallel to the base (Fig. 16.37). Furthermore, to make this system work, a rule of independence was devised:

> Every requirement on a drawing is intended to be applied independently, without reference to other dimensions, conditions, or characteristics, unless a particular relationship is specified.

This rule is voided, however, when "limits and fits" are specified, in which case the Taylor principle applies (the basis of ANSI). Rule 1 of the ANSI standard regarding limits of size follows:

> *Rule 1:* The surface(s) of a feature shall not extend beyond a boundary (envelope) of perfect form at MMC. This boundary is the true geometric form represented by the drawing. There is no requirement for a boundary of perfect form at LMC (Fig. 16.38).

Rule 1 does not apply to stock materials that use established industry or government standards; to parts specified in the "free state" (nonrigid); and to those specifically excluded, such as straightness of the axis or where the note **PERFECT FORM AT MMC NOT REQUIRED** is specified. Rule 1 does apply to individual features and, although they control the form of an individual feature of size, they do not control the orientation, location, or runout of features to each other. These relationships are defined by tolerances, notes, or other specifications called out directly on the drawing.

Advances in metrology—especially with computerized mathematical models and algorithms such as on coordinate measuring machines (CMMs)—lend themselves to the ANSI version of limits of size. A ring gage made to the MMC size of a shaft, and as long as the shaft, can verify a shaft for Rule 1 compliance. A micrometer or caliper can verify the LMC limit. A plug gage, made to the MMC size of a hole, and as long as the hole, can verify a hole for Rule 1 compliance. An inside micrometer or caliper is used to verify the LMC limit.

16.10.2 U.S. Interpretation of Limits of Size

In the United States, the **limits of size** of a feature describe the extent within which variations of geometric form, as well as size, are allowed. This control applies solely to individual features of size. **Feature of size** refers to one cylindrical or

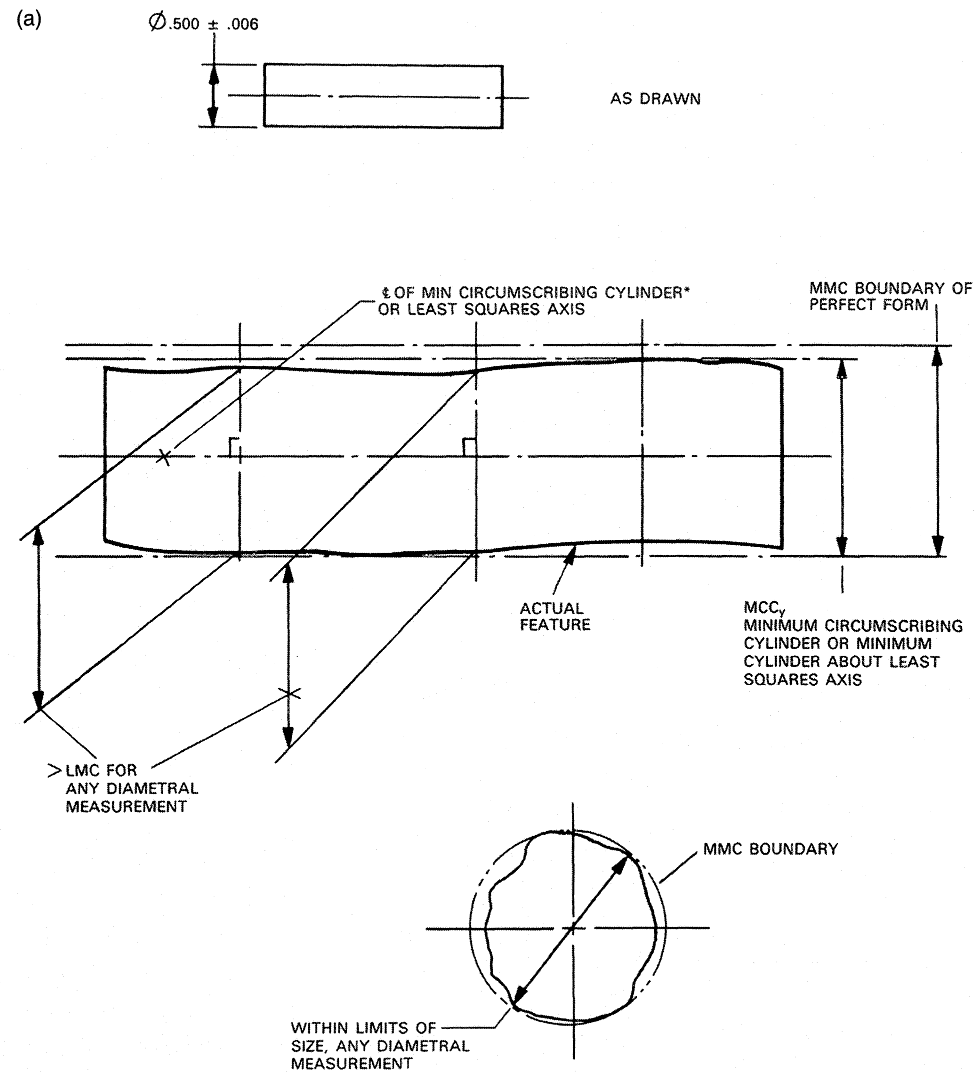


FIGURE 16.38 Limits of Size Interpretation for Individual Features

*Where a preference for a least squares or ℄ of envelope axis exists, it must be specified on the drawing.

spherical surface, or to a set of two plane parallel surfaces each of which is associated with a size dimension. Where only a **tolerance of size** is specified, the limits of size of an individual feature describe the extent to which variations in its geometric form, as well as size, are allowed (Fig. 16.39). **The actual size** of an individual feature at any cross section is within the specified tolerance of size.

16.11 GENERAL TOLERANCING RULES

Rules have been established to ensure uniform interpretation and to avoid costly errors and misunderstandings. Study the following six rules carefully.

- The system of indicating tolerances (whether size, location, or geometry) does not necessarily require any particular method of production or quality.
- Regardless of the number of places involved, all toleranced limits are considered to be absolute. Each limit is considered to be continued with trailing zeros. For example:

1.22 = 1.220 000 000 . . .
1.20 = 1.200 000 000 . . .

1.2 = 1.200 000 000 . . .
1.0 = 1.000 000 000 . . .

1.20 + .02 = 1.220 000 000 . . .
1.20 − .00 = 1.200 000 000 . . .

This rule applies to all limits (plus–minus or limit dimensioned), including those where title block tolerances are applied.

- All dimensions and tolerances are at 68°F (20°C) unless otherwise specified.
- Surfaces drawn at 90° are subject to the title block tolerance specified for angles or by a note, such as

PERFECT ORIENTATION REQUIRED AT MMC.

This rule also applies to features that have a common centerline or axis of revolution. Where function or inter-

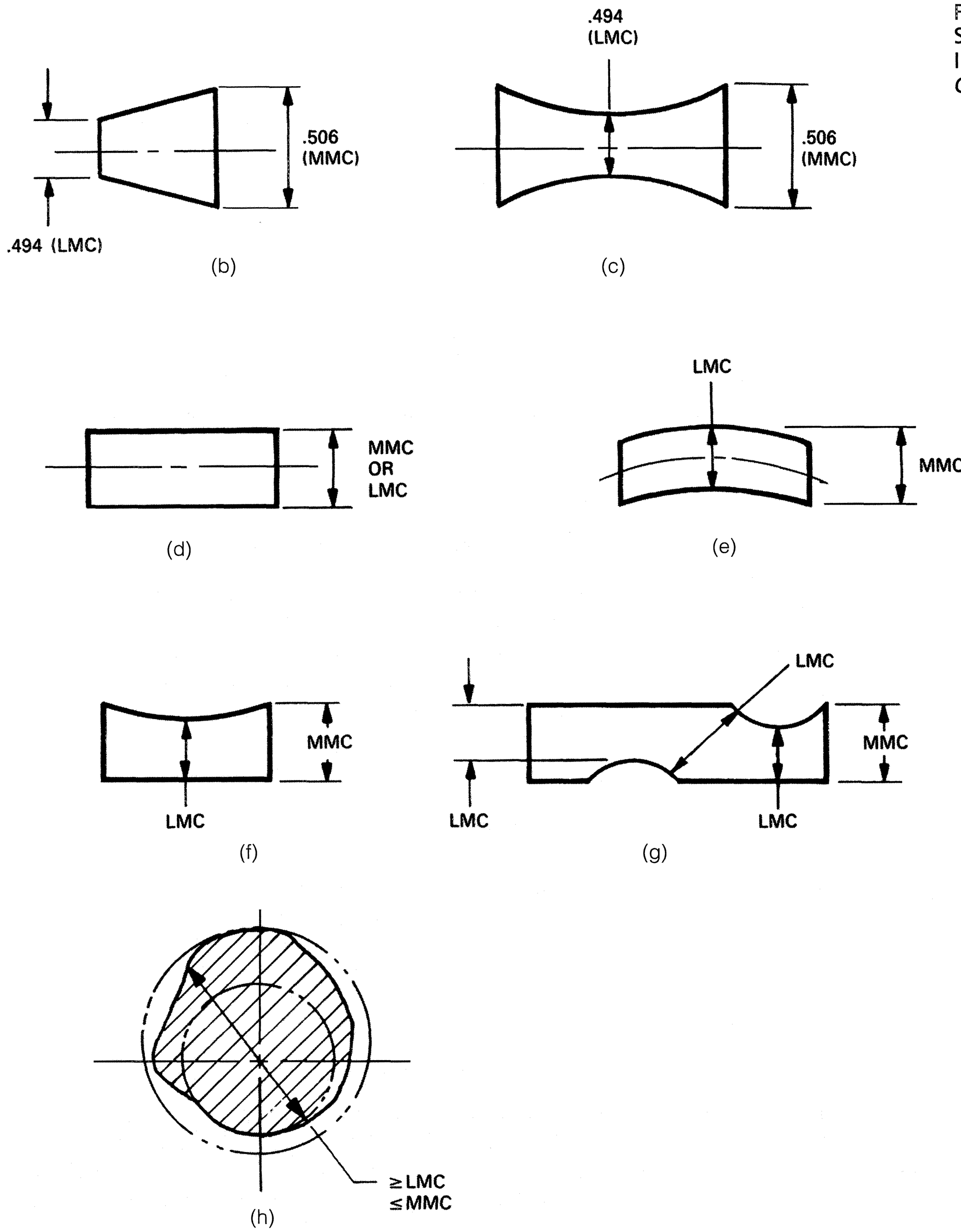


FIGURE 16.38 **Limits of Size Interpretation for Individual Features—*Continued***

changeability is affected, the tolerances are to be specified.

- Theoretical constructions such as centerlines or planes, shown at right angles, and from which features such as holes or pins are dimensioned, are considered to be at 90° BASIC. Variations in the inspection setup are subtracted from the allowable tolerances during the verification process.
- The tolerance specified on the drawing is the total amount allowable, including manufacturing, inspection, and gaging variations. To ensure rapid part acceptance, manufacturing usually does not use more than 90° of the available tolerance.

16.11.1 General Rules of Geometric Tolerancing

Use the following rules to apply geometric tolerancing to a part.

- The surface(s) of a feature must not extend beyond a boundary (envelope) of perfect form at MMC. There is no requirement for a boundary of perfect form at LMC.
- Position tolerance requirements for modifiers are specified in the feature control frame. A modifier, M or L, is specified after the feature tolerance and after each datum for features and datums of size. No modifier is specified for a single-plane surface.

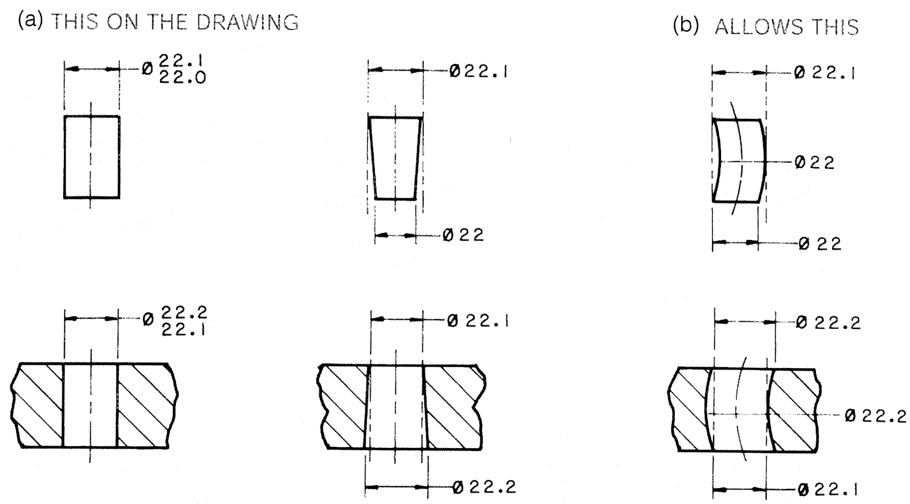
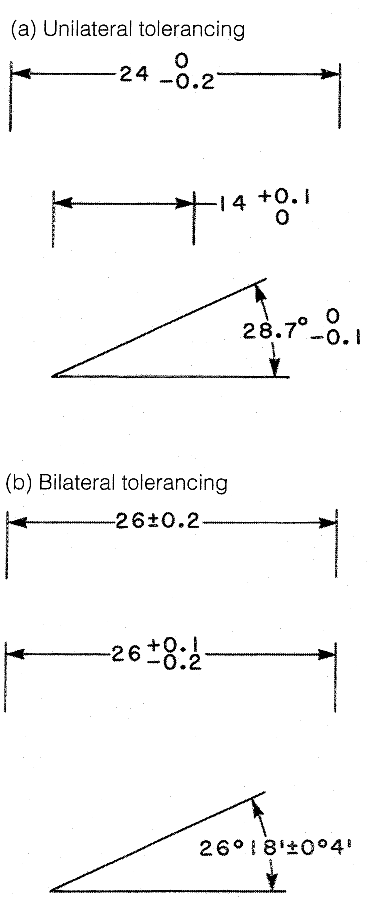


FIGURE 16.39 Tolerance Variations Allowed for Individual Features

- Requirements for modifiers of tolerances other than position are specified in the feature control frame. RFS applies, unless another modifier is specified, for all features and datums. The RFS modifier is not shown in the feature control frame. MMC is specified for features and datums of size where the design allows.



IGURE 16.40 Plus-or-Minus Tolerancing on Dimensions

16.11.2 Setting Tolerances

The **nominal size** is often referred to as the **basic size** or **design size.** The nominal and the associated tolerances have the same number of decimal places, except in the metric system. The "plus" value is shown above the "minus" value. European drawings also use +, + and −, and − tolerances.

In **bilateral tolerancing** [Fig. 16.40(b)], the tolerance is applied in both directions from the nominal:

$$.500 \pm .005 \qquad \text{or} \qquad 1.200 \begin{array}{l} + .002 \\ - .005 \end{array}$$

In **unilateral tolerancing** [Fig. 16.40(a)], the tolerance is applied in one direction; the other value is zero:

$$1.200 \begin{array}{l} + .002 \\ - .000 \end{array} \qquad 1.200 \begin{array}{l} + .000 \\ - .006 \end{array}$$

16.11.3 Limit Dimensioning

In **limit dimensioning,** the maximum value is placed above the minimum value (Fig. 16.41). In note form, the larger value is placed to the right of the lesser value, separated by a dash. Both limits have the same number of decimal places:

$$\begin{array}{l} .750 \\ .748 \end{array} \qquad \text{or} \qquad .748 - .750$$

Even values are preferred. Although there is usually a trailing zero, the number of decimal places is minimized. Plus-or-minus and limit dimensions may appear on the same drawing. Generally, limit dimensions specify the size of features, and plus-or-minus dimensions specify the location of features. Plus-or-minus dimensions, in bilateral form, are preferred for numerical control production, where the mean is used.

16.11.4 Title Block Tolerances

Title block tolerances are used where there is a uniformity in tolerances (Fig. 16.42). In Figure 16.43, the nominal

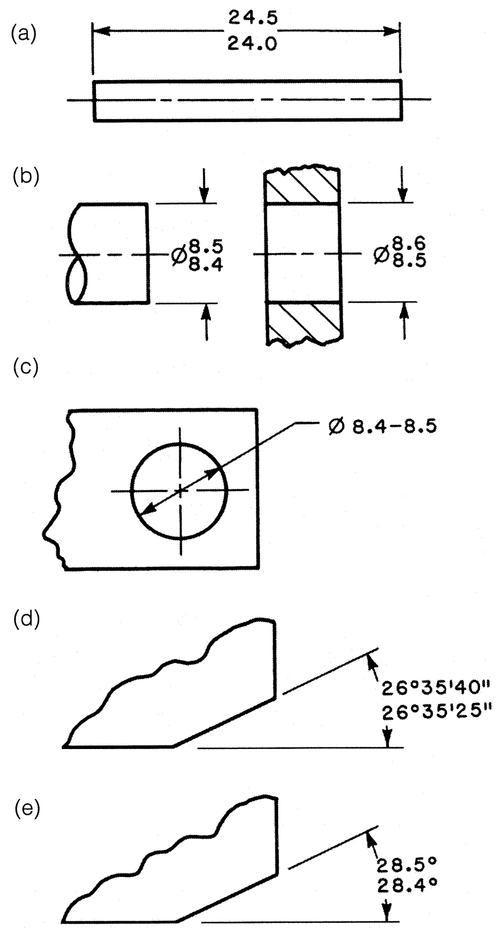


FIGURE 16.41 Limit Dimensioning on Drawings

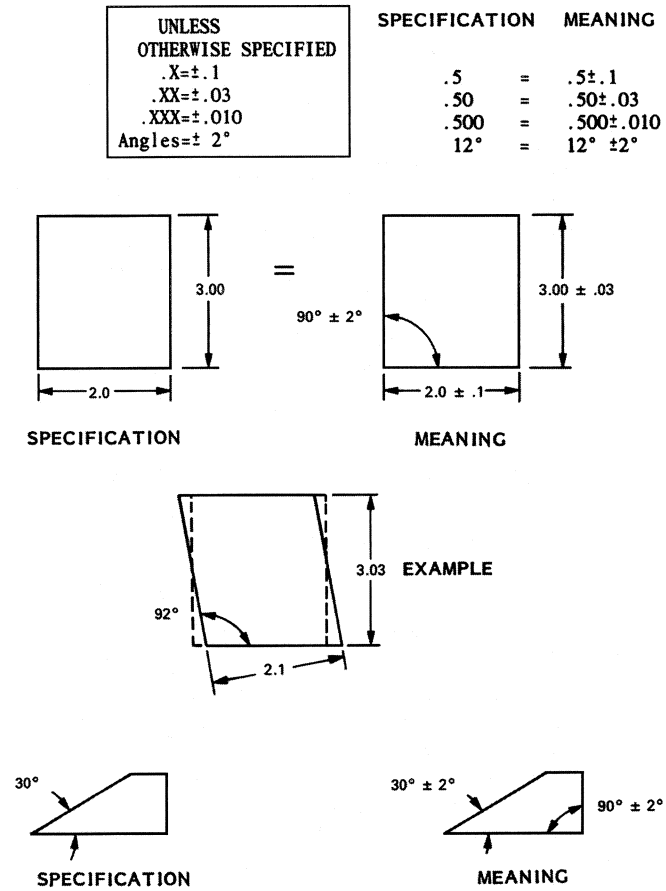


FIGURE 16.43 Application of Title Block Tolerance

dimension is given alone on the face of the drawing and a bilateral tolerance is shown in the title block. If larger tolerances are allowed for a particular feature, they should be specified. In European title blocks, tolerances are based on feature size; in the United States they are based on the number of decimal places specified (Fig. 16.43).

16.11.5 Tolerance Accumulation

Figure 16.44 compares the tolerance values from three methods of dimensioning.

Chain dimensioning The maximum variation between two features is equal to the tolerances on the intermediate

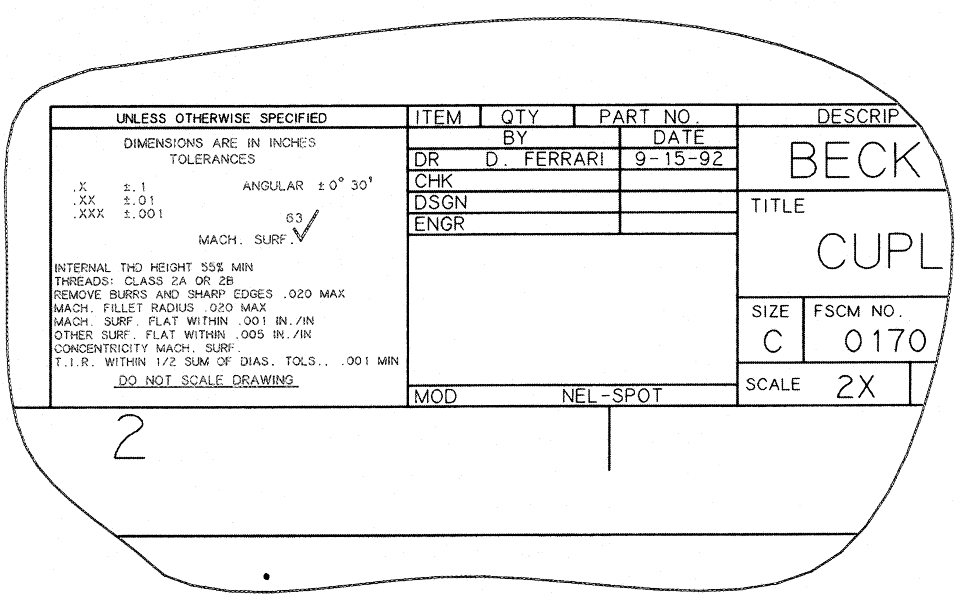
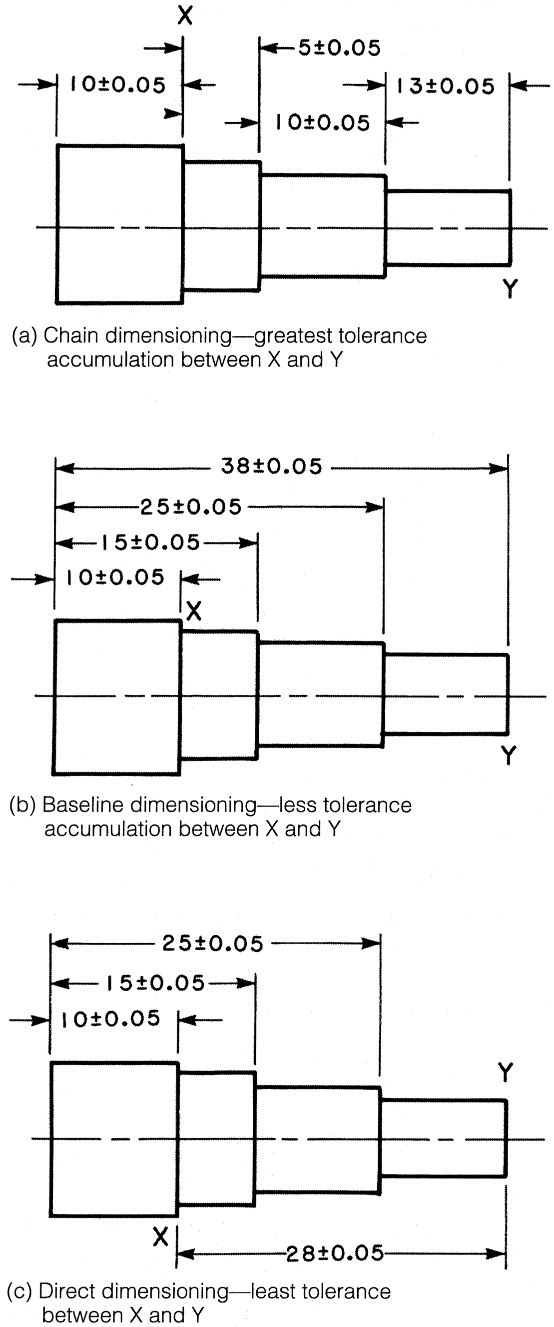


FIGURE 16.42 Title Block Tolerances



(a) Chain dimensioning—greatest tolerance accumulation between X and Y

(b) Baseline dimensioning—less tolerance accumulation between X and Y

(c) Direct dimensioning—least tolerance between X and Y

FIGURE 16.44 Tolerance Accumulations

listances. This method results in the *greatest tolerance ıccumulation.* In Figure 16.44(a), the tolerance accumulation ɔetween surfaces **X** and **Y** is ±.15.

ʒaseline dimensioning The maximum variation between two eatures is equal to the sum of the tolerances on the two limensions from their origin to the features. This reduces he tolerance accumulation. In Figure 16.44(b), the toler-ınce accumulation between surfaces **X** and **Y** is ±.1.

Direct dimensioning The maximum variation between two eatures is controlled by the tolerance on the dimension between the features. This results in the *least tolerance.* In Figure 16.44(c), the tolerance between surfaces **X** and **Y** is ±.05.

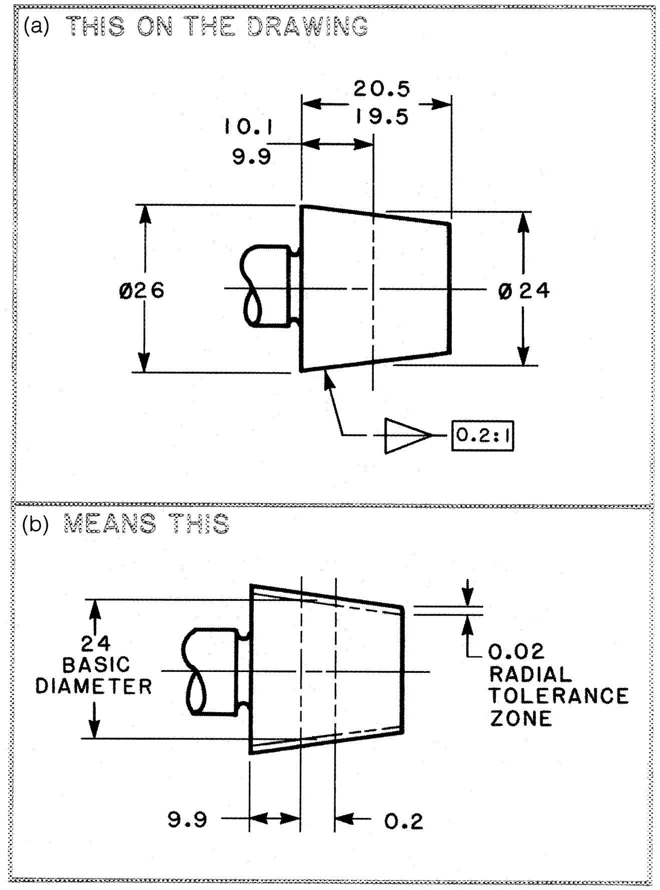


FIGURE 16.45 Specifying a Basic Taper and a Basic Diameter

16.11.6 Tolerances for Flat and Conical Tapers

Taper is defined as the ratio of the difference between the diameters of two sections, perpendicular to the axis of a cone, to the distance between these sections. A **conical taper** is specified by one of the following methods:

- A basic taper and a basic diameter (Fig. 16.45)
- A size tolerance combined with a profile of a surface tolerance applied to the taper
- A toleranced diameter at both ends, or a taper and a toleranced length

A **flat taper** is defined by specifying a toleranced slope and a toleranced height at one end (Fig. 16.46). **Slope** is defined as the inclination of a surface expressed as a ratio of the difference between the heights at each end, above and at right angles to a baseline, to the distance between those heights. Flat and conical tapers are toleranced as shown in Figure 16.47.

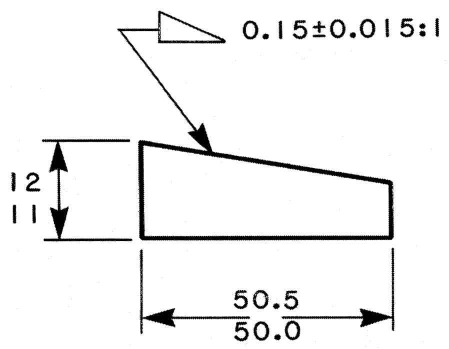


FIGURE 16.46 Slope Designation on a Drawing

16.11.7 Single Limits: Min and Max

The unspecified limit in minimum dimensions (the maximum limit) approaches infinity. Therefore overall lengths are not specified as minimum. The unspecified value in maximum dimensions (the minimum limit) approaches zero.

16.11.8 Reference Dimensions

Reference dimensions, as discussed in Chapter 15, are specified by enclosing the dimension in parentheses, e.g., **(.500)**. No tolerance is given. Reference dimensions are not intended to govern production or inspection; that is, they are informational, not controlling.

16.11.9 Functional Dimensions

Tolerances accumulate (Fig. 16.44). An example of how to control this situation is shown for a firing pin in Figure 16.48. The most important function of the firing pin is to project far enough to detonate the primer, but not far enough to pierce the primer. Also, the point must be fully below the bolt face, in the retracted position, to prevent premature detonation in the cartridge. This function is controlled by dimension **A**, a direct dimension from the point face to the interface with the bolt in the full forward position. Dimension **B** is established similarly. Dimensions that affect function should be dimensioned directly to avoid tolerance accumulation. Conversely, dimension C_1 was replaced by **(C)** during a producibility team review. This dimension must be long enough so that the hammer will drive the pin to its full forward position; but the length is not critical, since the pin has plenty of overtravel. Dimension **(C)**, for ease of manufacture, was taken to the end of the spherical surface that is contacted by the hammer. The tapered section doesn't have to be accurate, but it must be located. This is accomplished by dimension **D**.

16.11.10 Nonmandatory Dimensions

If practical, the finished part is defined without specifying the manufacturing method. For example, the diameter of a hole is given without indicating whether it should be drilled, reamed, punched, or made by any other operation. If manufacturing, processing, verification, or environmental

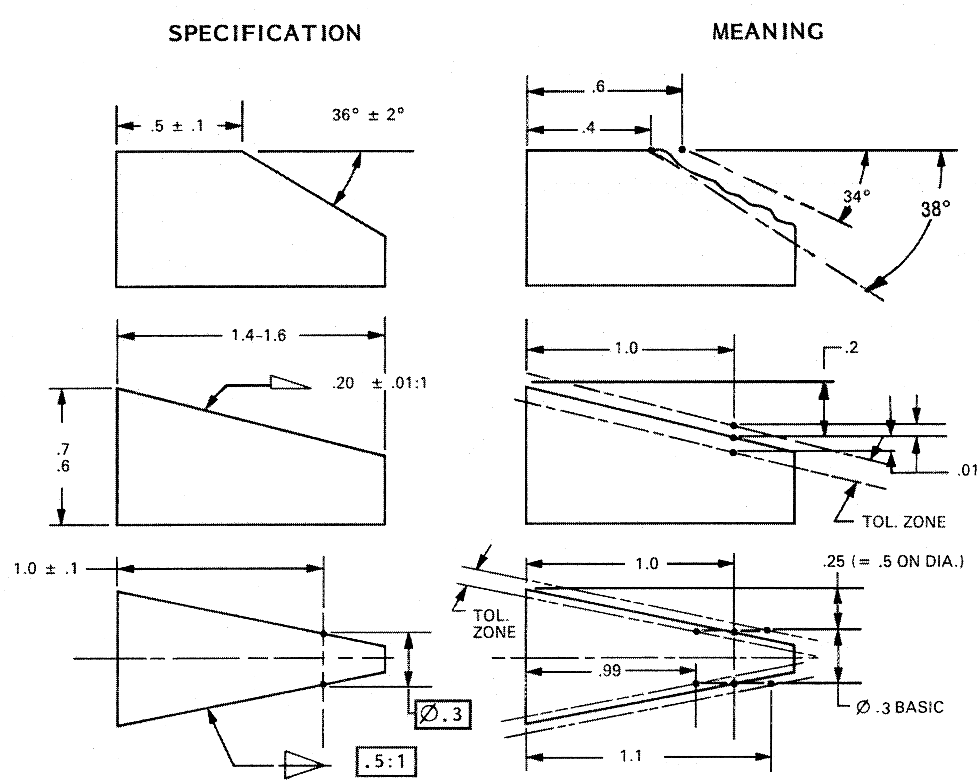


FIGURE 16.47 Flat and Conical Taper Tolerance Zones

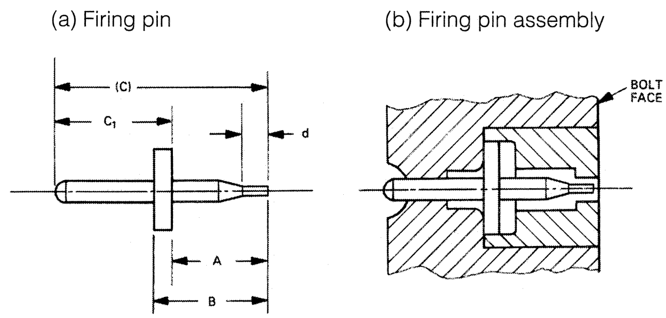


FIGURE 16.48 Functional Dimensioning: Firing Pin Assembly

information is essential to the definition, it is specified on the drawing. The affected dimensions are identified as **NONMANDATORY (MFG DATA)**. This allows improved or superior methods to be used at the discretion of the manufacturing or quality-control departments.

16.11.11 Coordination, Interface Control, and Correlation Dimensions/ Tolerances

On large projects or programs, dimensions and tolerances are agreed on by all parties. A coordination drawing has agreement on function, mating, shipping, equipment removal, etc. These dimensions are then "flagged" on the hardware drawings. This protects the design from inadvertent changes. Interface-control and correlation drawings are handled the same way. The only difference is that coordination drawings are prepared for the total system, whereas correlation and interface-control drawings are prepared for major subsystems. Dimensions on these drawings are flagged to be in compliance with the coordination drawings.

16.12 LIMITS AND FITS

Production and inspection benefit from standard limits. ANSI B4.2 and ISO 286 describe these systems. The ISO system has more than 500 possible tolerance zones for holes and shafts; ANSI has about 150. Many products can be standardized via the system of limits and fits: drills, reamers, clevis pins, bushings, keys, keyways, gages, and bolts. **Renard preferred numbers** are used in the metric system with limits and fits to maximize standardization. The tables presented are not restricted to the preferred numbers. Sizes in design are often determined by factors other than cost, such as mechanical and thermal stress, and weight.

There are three types of **fits**: **clearance**, where there is always clearance; **transition**, where there may be clearance or interference; and **interference**, where there is always interference (Fig. 16.49). The Taylor principle is applied to limits and fits; that is, each individual feature is in perfect form at MMC.

Limits and directly applied tolerance values are specified as follows:

Limit dimensioning The high limit (maximum value) is placed above the low limit (minimum value). When expressed in a single line, the low limit precedes the high limit and a dash separates the two values.

Plus-or-minus tolerancing The dimension is given first and is followed by a plus-or-minus expression of tolerance.

16.12.1 Single Limits

For **single limits, MIN** or **MAX** is placed after a dimension where other elements of the design definitely determine the other unspecified limit (depth of holes, length of threads, corner radii, chamfers, etc.). Single limits are used where the intent is clear and the unspecified limit can be zero or can approach infinity without interfering with the designed function of the part.

16.12.2 Tolerance Expression

The conventions regarding the number of decimal places carried are different for metric and inch drawings. For millimeter dimensions use the following rules.

For **unilateral tolerancing,** when either the plus or the minus value is nil, a single zero is shown without a plus or a minus sign:

Example:

$32\,^{0}_{-.02}$ or $32\,^{+.02}_{0}$

With **bilateral tolerancing,** both the plus and the minus

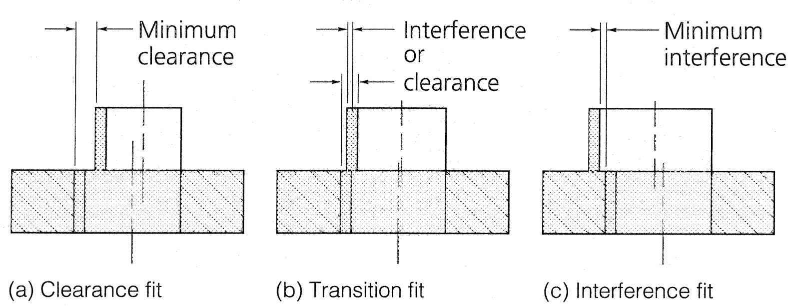


FIGURE 16.49 Basic Types of Fits

values have the same number of decimal places, using zeros where necessary:

Example:

$32\,^{+.25}_{-.10}$ not $32\,^{+.25}_{-.1}$

Where **limit dimensioning** is used and either the maximum or minimum value has digits following a decimal point, the other value has zeros:

Example:

$\begin{matrix}25.45\\25.00\end{matrix}$ not $\begin{matrix}25.45\\25\end{matrix}$

For **inch dimensions,** both limit dimensions, or the plus-or-minus tolerance and its dimension, are expressed with the same number of decimal places:

Examples:

.5 + .005 not .50 + .005

$\begin{matrix}+.005\\-.000\end{matrix}$ not $\begin{matrix}+.005\\0\end{matrix}$

25.0 + .2 not 25 + .2

16.12.3 Preferred Metric Fits

For metric application of limits and fits, the tolerance may be indicated by a basic size and tolerance symbol. See ANSI B4.2 for complete information on this system. The preferred metric fits are defined as follows:

Loose Running (H11c11) Suitable for wide commercial tolerances or allowances on external members

Free running (H9/d9) Not suitable for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures

Close running (H8/f7) Suitable for running on accurate machines and for accurate location at moderate speeds and journal pressures

Sliding fit (H7/g6) Not intended to run freely, but to move and turn freely and to locate accurately

Locational clearance (H7/h6) Provides snug fit for locating stationary parts, but can be freely assembled and disassembled

Locational transition (H7/k6) Suitable for accurate location; a compromise between clearance and interference

Locational transition (H7/n6) For more accurate location where greater interference is permissible

Locational interference (H7/p6) Suitable for parts requiring rigidity and alignment, with prime accuracy of location but without special bore pressure required

Medium drive (H7/s6) Suitable for ordinary steel parts or for shrink fits on light sections; tightest fit usable with cast iron

Force fit (H7/u6) Suitable for parts that may be highly stressed or for shrink fits where the heavy pressing forces required may be impractical

16.12.4 Preferred Inch Fits

There are three general groups of fits: running and sliding fits, locational fits, and force fits. **Running and sliding fits** provide similar running performance, with a suitable lubrication allowance, throughout the range of sizes. The first ten preferences for inch fits are as follows: RC 4, RC 7, RC 9, LC 2, LC 5, LT3, LT6, LN 2, FN2, and FN4. Running and sliding fits are defined as follows:

RC 1 Close sliding fits are intended for the accurate location of parts that must be assembled without perceptible play.

RC 2 Sliding fits are intended for accurate location, but with greater maximum clearance than class RC 1. Parts made to this fit move and turn easily, but are not intended to run freely; in the larger sizes, they may seize with small temperature changes.

RC 3 Precision running fits are about the closest fits that can be expected to run freely, and they are intended for precision work at slow speeds and light journal pressures, but are not suitable where appreciable temperature differences are likely to be encountered.

RC 4 Close running fits are intended chiefly for running fits on accurate machinery with moderate surface speeds and journal pressures, where accurate location and minimum play are desired.

RC 5 and RC 6 Medium running fits are intended for higher running speeds, or heavy journal pressures, or both.

RC 7 Free running fits are intended for use where accuracy is not essential, or where large temperature variations are likely to be encountered, or under both these conditions.

RC 8 and RC 9 Loose running fits are intended for use where wide commercial tolerances may be necessary, together with an allowance, on the external member.

16.12.5 Locational Fits

Locational fits are intended to determine only the location of the mating parts; they may provide rigid or accurate location, as with interference fits, or provide some freedom of location, as with clearance fits. They are divided into three groups: **clearance fits (LC), transition fits (LT),** and **interference fits (LN).**

LC Locational clearance fits are intended for parts that are usually stationary but that can be freely assembled or disassembled. They range from snug fits for parts requiring accuracy of location, through the medium-clearance fits for parts such as spigots, to the looser fastener fits where freedom of assembly is the prime consideration.

Applying Parametric Design . . .

TOLERANCING AND PARAMETRIC FEATURES

The manufacturing of parts and assemblies requires a degree of precision determined by **tolerances**. A typical parametric design system supports three types of tolerances:

- **Dimensional** tolerancing specifies allowable variation of size (see Fig. A)
- **Geometric** tolerancing controls form, profile, orientation, and runout
- **Surface finish** tolerancing controls the deviation of a part surface from its normal value

When you design a part, you specify dimensional tolerance—**allowable variations in size**. All dimensions are controlled by tolerances. The exception applies only to "**basic**" dimensions, which for the purpose of reference are considered to be exact (see Fig. B). The radius value in this example was changed from a "limits" format to "basic." You may also modify the upper or lower tolerance value (see Fig. C). **Dimensional tolerances** on a drawing can be expressed in two forms:

- As **general tolerances**—presented in a tolerance table. These apply to those dimensions that are displayed in nominal format, that is, without tolerances.
- As **individual tolerances**—specified for individual dimensions.

You can use general tolerances given as defaults in a table or set individual tolerances by modifying default values of selected dimensions. Default tolerance values are used at the moment you start to create a model (see Fig. D); therefore, default tolerances must be set prior to creating geometry. The system recognizes six decimal places for which you can specify the tolerance values. When you start to create a part (see Fig. E), the table at the bottom of the window will display the current defaults for tolerances (Fig. F). If you have not specified tolerances, the system defaults are assumed, and the

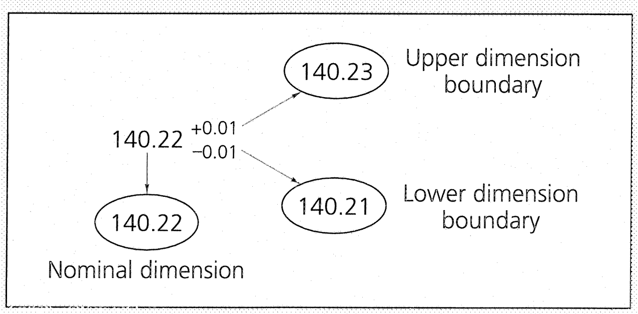


FIGURE A Dimensional Tolerance

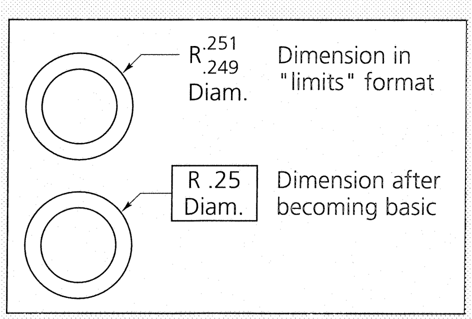


FIGURE B Limits Dimension and Basic Dimension

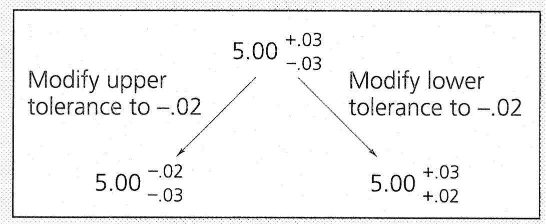


FIGURE C Modifying Upper and Lower Tolerance Values

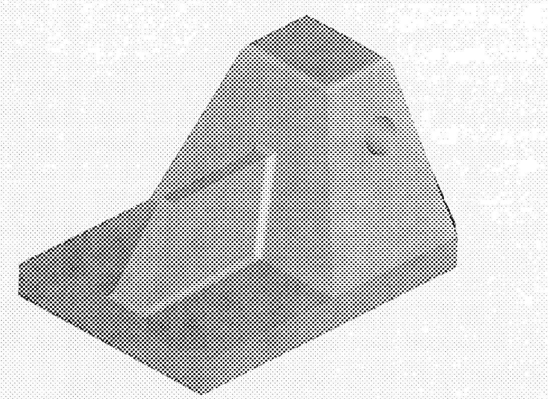


FIGURE D Shaded Model of Part

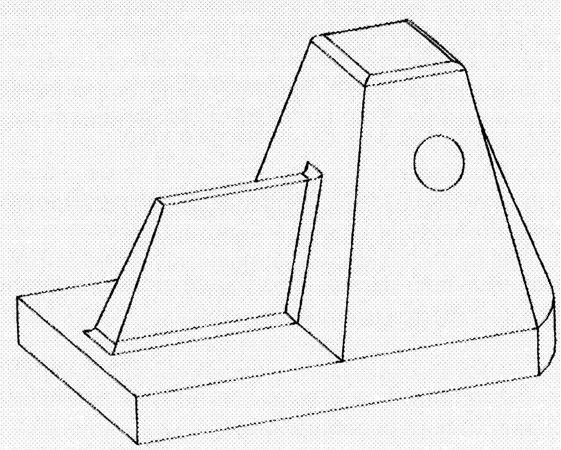


FIGURE E Part Model

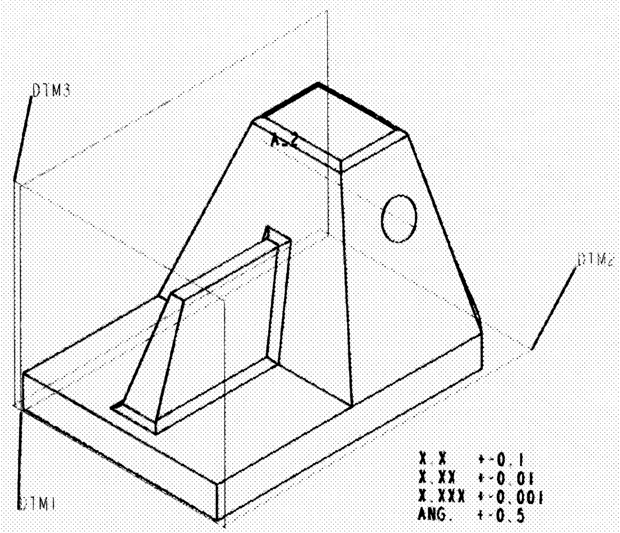


FIGURE F Part with Default Tolerances Displayed

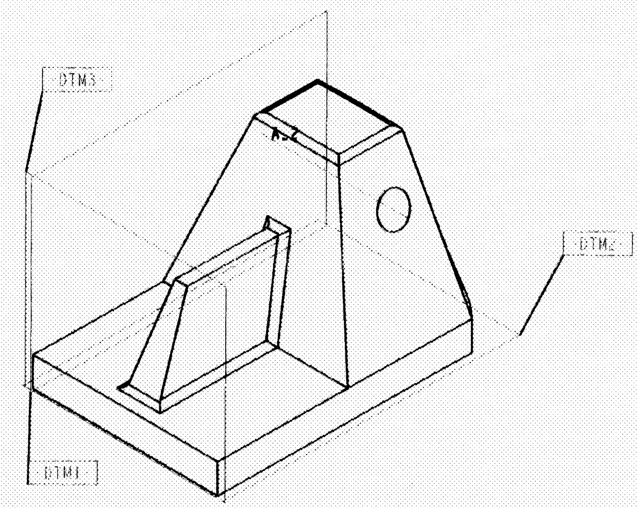


FIGURE G Tolerances Not Displayed; Datums Set as Basic

table will look as follows.

*X.X	±0.1
*X.XX	±0.01
*X.XXX	±0.001
*ANG.	±0.5

You have a choice of displaying or blanking tolerances. Even if tolerances are not displayed, the system still stores dimensions with their default tolerances. You can specify geometric tolerances, create "basic" dimensions, and set selected datums as reference datums for geometric tolerancing (Fig. G). Figure H provides an example of the part dimensions in the limits format, and Figure I shows the same dimensions as basic.

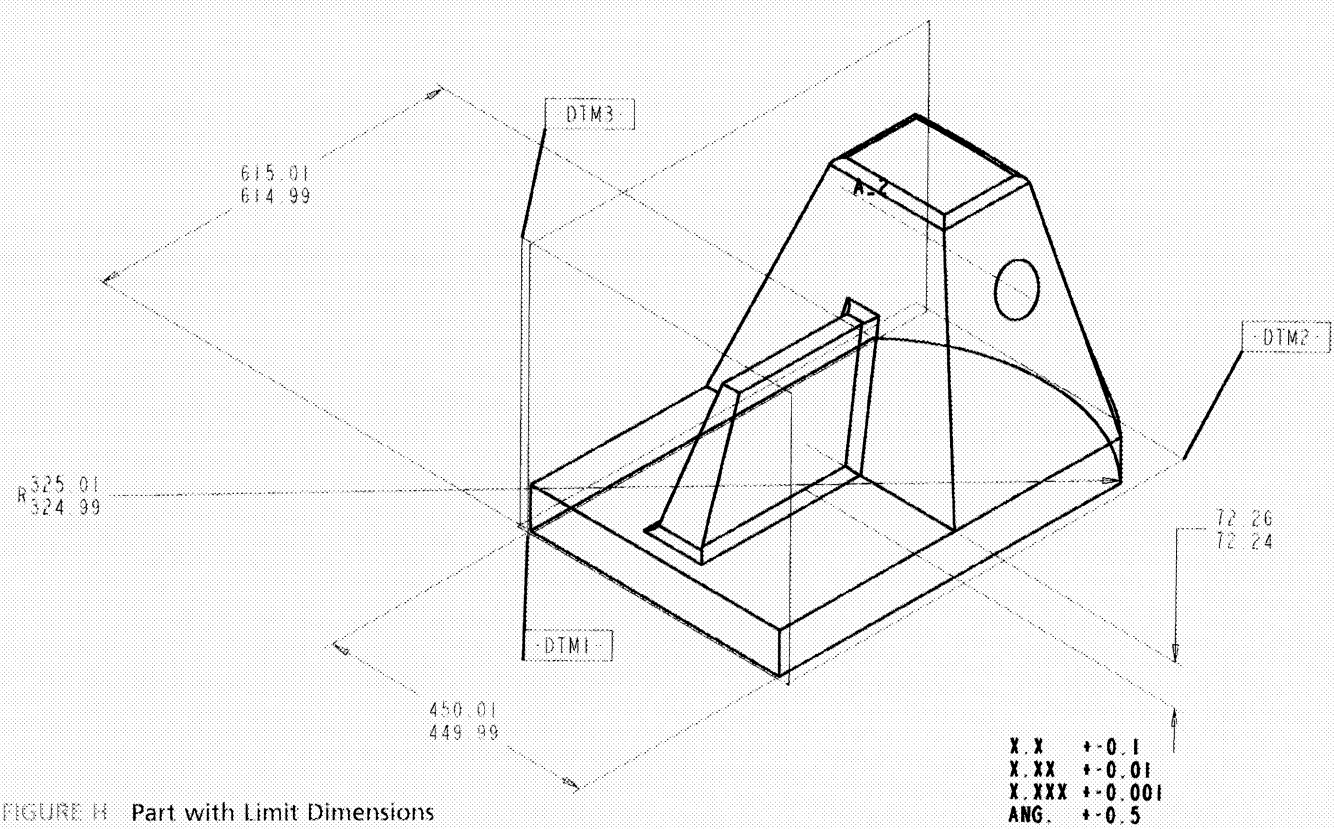
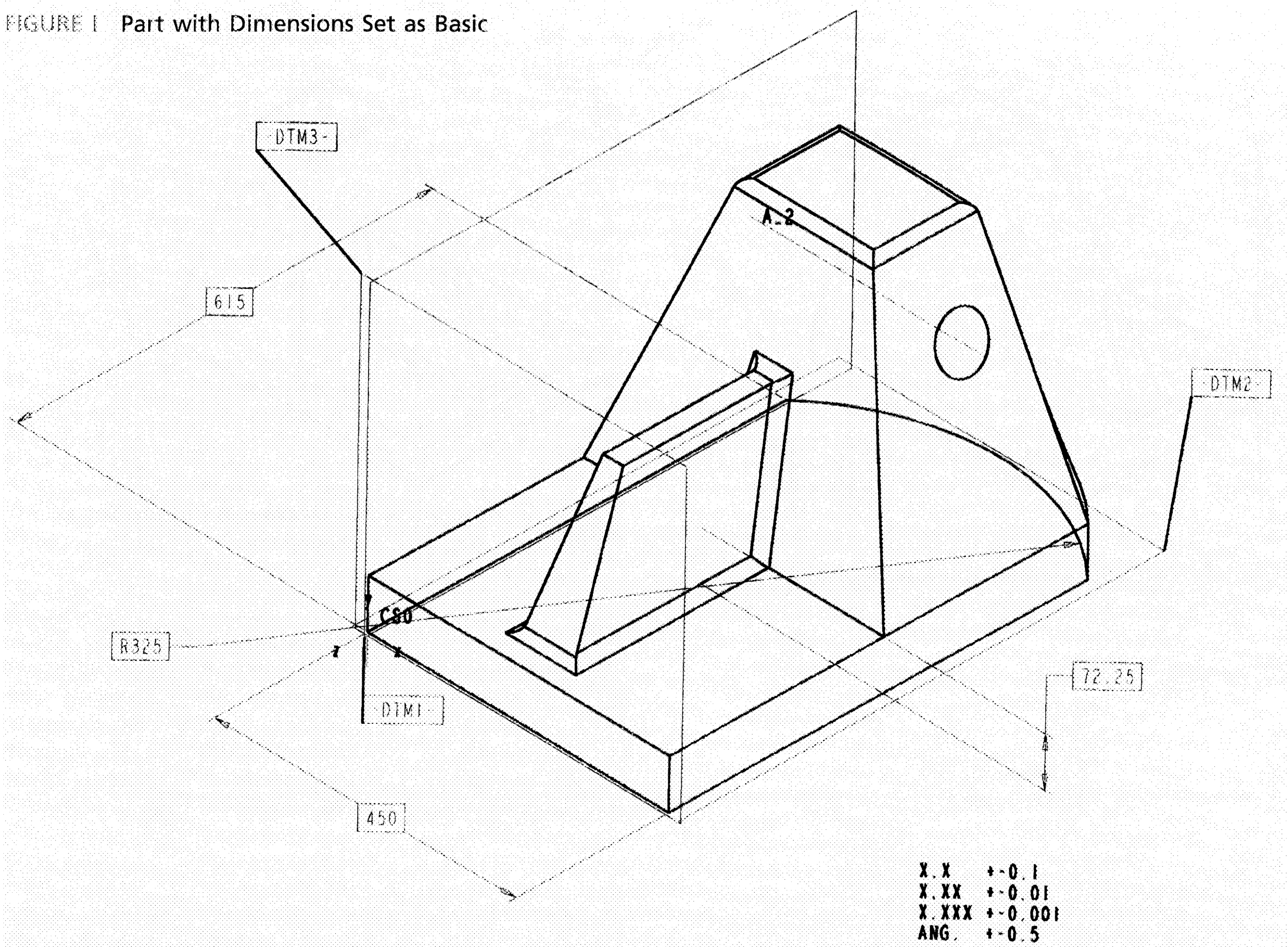


FIGURE H Part with Limit Dimensions

FIGURE I Part with Dimensions Set as Basic



The available tolerance formats are (Fig. J):

Normal Dimensions are displayed without tolerances.
Limits Tolerances are displayed as upper and lower limits.
Plus–Minus Tolerances are displayed as nominal with plus-or-minus tolerance. The positive and negative values are independent.
±Symmetric Tolerances are displayed as nominal, with a single value for both the positive and the negative tolerances.

Geometric tolerances provide a method for controlling the location, form, profile, orientation, and runout of features (see Fig. K). You are able to add geometric tolerances to the model

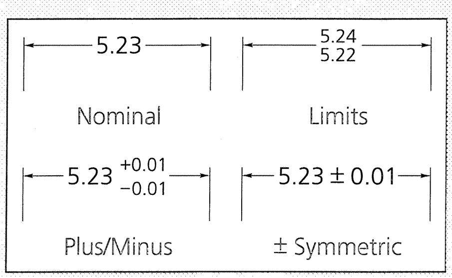


FIGURE J Tolerance Formats

from drawing mode. The geometric tolerances are treated by the system as annotations, and they are always associated with the model. *Unlike dimensional tolerances, geometric tolerances do not have any effect on part geometry.*

When adding a geometric tolerance to the model, you can attach it to existing dimensions, edges, or existing geometric tolerances (Fig. L), or you can display it as a note without a leader.

Before you can reference a datum (see Figs. D–I, sequence) in a geometric tolerance, you must first indicate your attention by **"setting"** the datum. Once a datum is set, the datum name is prefixed and appended by hyphens, and it is enclosed in a rectangle (see Fig. G). You can change the name of a datum either before or after it has been "set" by using the **Name** item on the SET UP menu in Part mode. You can choose any datum feature as a reference datum for a geometric tolerance; the system will warn you if you pick an inappropriate datum, but your selection will be accepted. To set a reference datum:

1. Choose **Set Datum** from the GEOM TOL menu.
2. Select the datum plane or axis to be set.
3. The datum is enclosed in a feature control frame (Fig. M).

A geometric tolerance for individual features is specified by means of a **feature control frame** (a rectangle) divided into compartments containing the geometric tolerance symbol followed by the tolerance value (see Fig. M). Where applicable, the

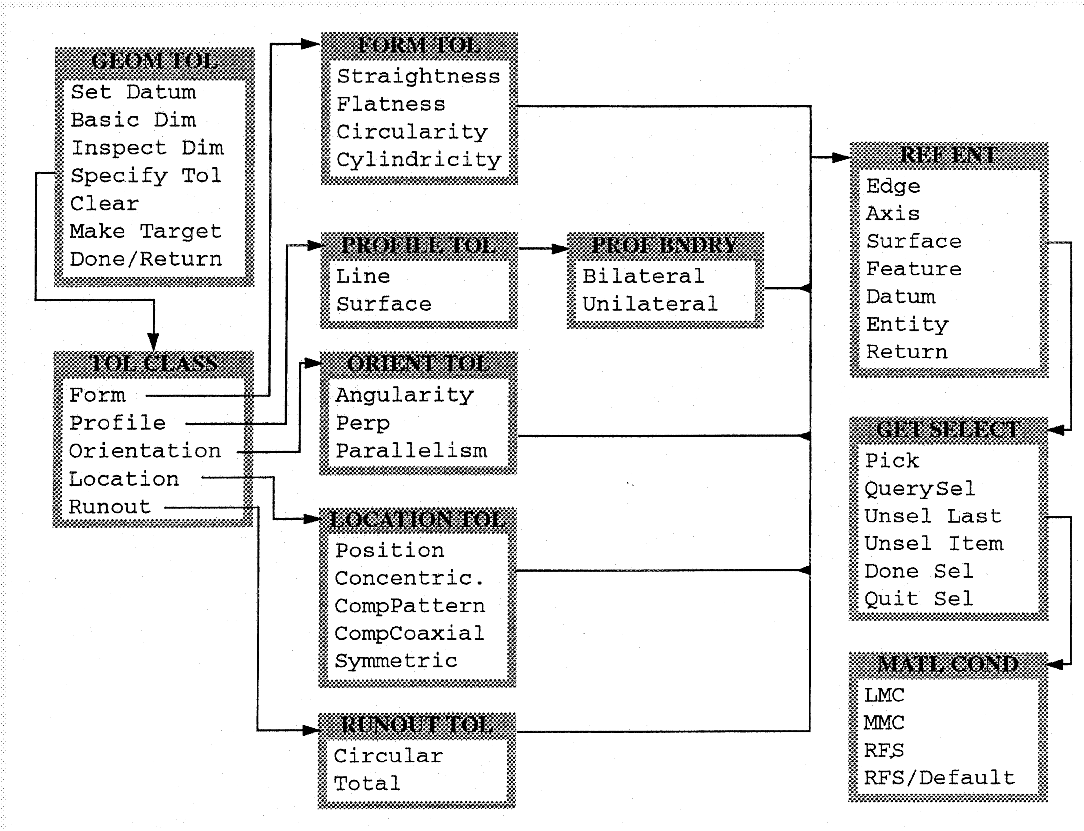


FIGURE K Geometric Tolerancing Menu

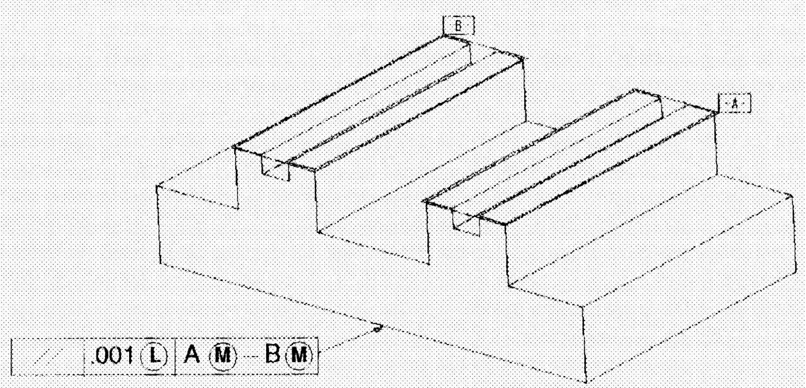


FIGURE L Displaying Geometric Tolerance with a Leader

tolerance is followed by a **material condition symbol**.

Where a geometric tolerance is related to a datum, the reference datum name is placed in a compartment following the tolerance value. Where applicable, the datum reference letter is followed by a material condition symbol.

For each class of tolerance, the types of tolerances available and the appropriate types of entities can be referenced (see Fig. N). The available material condition symbols are also shown in Figure N.

The system guides you in building a geometric tolerance by requesting each piece of information. You respond by making menu choices, entering a tolerance value, and selecting entities and datums (see Fig. O). As the tolerance is built, the choices are limited to those items that make sense in the context of the information you have already provided. For example, if the

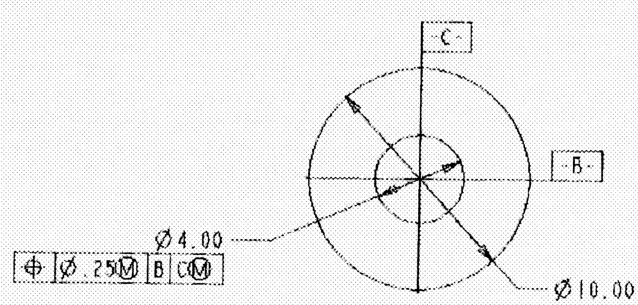


FIGURE M Feature Control Frame

FIGURE N **Class, Type, Symbol, and Material Condition**

Class	Type	Symbols	Entities
Form	Straightness	—	Surface of revolution, axis, straight edge
	Flatness	⏥	Plane surface (not datum plane)
	Circularity	○	Cylinder, cone, sphere
	Cylindricity	⌭	Cylindrical surface
Profile	Line	⌒	Edge
	Surface	⌓	Surface (not datum plane)
Orientation	Angularity	∠	Plane, surface, axis
	Parallelism	//	Cylindrical surface, axis, planar surface
	Perpendicularity	⊥	Cylindrical surface, axis, planar surface
Runout	Circular	↗	Cone, cylinder, sphere, plane
	Total	⌰	Cone, cylinder, sphere, plane
Location	Position	⌖	Any
	Concentricity	◎	Axis, surface of revolution
	Symmetry	⌯	Any

LMC	Ⓛ	Least material condition
MMC	Ⓜ	Maximum material condition

geometric characteristic is one that does not require a datum reference, you will not be prompted for one. Other checks are made to help prevent mistakes in the selection of entities and datums.

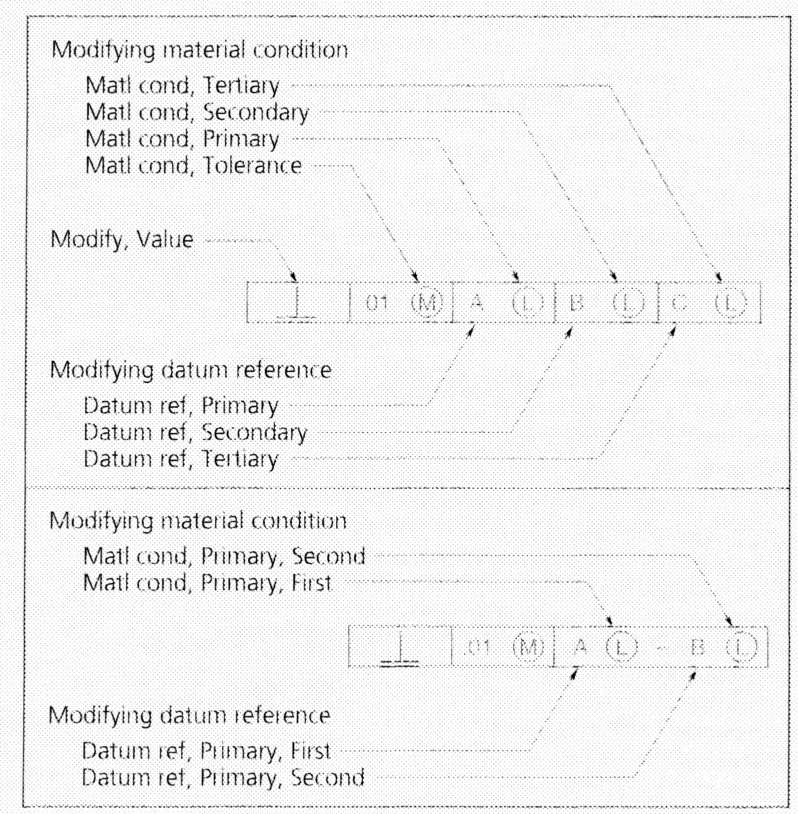


FIGURE O **Building a Geometric Tolerance**

LT Locational transition fits are a compromise between clearance and interference fits, for applications where accuracy of location is important but either a small amount of clearance or interference is permissible.

LN Locational interference fits are used where accuracy of location is of prime importance and for parts requiring rigidity and alignment with no special requirements for bore pressure. Such fits are not intended for parts designed to transmit frictional loads (these are covered by force fits).

16.12.6 Force or Shrink Fits

Force or **shrink fits** are a special type of interference fit, normally characterized by maintenance of constant bore pressures throughout the range of sizes. The interference varies almost directly with diameter, and the difference between its minimum and maximum values is small, to maintain the resulting pressures within reasonable limits.

FN 1 Light drive fits are those requiring light assembly pressures, and they produce more or less permanent assemblies. They are suitable for thin sections or long fits, or in cast-iron external members.

FN 2 Medium drive fits are suitable for ordinary steel parts or for shrink fits on light sections. They are the tightest fits that can be used with high-grade cast-iron external members.

FN 3 Heavy drive fits are suitable for heavier steel parts or for shrink fits in medium sections.

FN 4 and FN 5 Force fits are suitable for parts that can be highly stressed or for shrink fits where the heavy pressing forces required are impractical.

16.12.7 Preferred Tolerance Zones

A profile tolerance may be applied to an entire surface or to individual profiles taken at various cross sections through the part. **Preferred tolerance zones** are shown in Figure 16.50.

Preferred hole tolerance zones

							H1		JS1									
							H2		JS2									
							H3		JS3									
							H4		JS4									
						G5	H5		JS5	K5	M5	N5	P5	R5	S5	T5	U5	V5
					F6	G6	H6	J6	JS6	K6	M6	N6	P6	R6	S6	T6	U6	V6
			D7	E7	F7	G7	H7	J7	JS7	K7	M7	N7	P7	R7	S7	T7	U7	V7
		C8	D8	E8	F8	G8	H8	J8	JS8	K8	M8	N8	P8	R8	S8	T8	U8	V8
A9	B9	C9	D9	E9	F9	G9	H9		JS9	K9	M9	N9	P9	R9	S9	T9	U9	V9
A10	B10	C10	D10	E10	F10	G10	H10		JS10	K10	M10	N10	P10	R10	S10	T10	U10	V10
A11	B11	C11	D11	E11	F11		H11		JS11									
A12	B12	C12	D12	E12			H12		JS12									
A13	B13	C13					H13		JS13									
A14	B14						H14		JS14									
							H15		JS15									
							'H16		JS16									

1. First choice zones are ellipsed
2. Second choice zones framed
3. Third choice zones open

Preferred shaft tolerance zones

							h1		js1									
							h2		js2									
							h3		js3									
						g4	h4		js4	k4	m4	n4	p4	r4	s4	t4	u4	v4
					f5	g5	h5	j5	js5	k5	m5	n5	p5	r5	s5	t5	u5	v5
				e6	f6	g6	h6	j6	js6	k6	m6	n6	p6	r6	s6	t6	u6	v6
			d7	e7	f7	g7	h7	j7	js7	k7	m7	n7	p7	r7	s7	t7	u7	v7
		c8	d8	e8	f8	g8	h8		js8	k8	m8	n8	p8	r8	s8	t8	u8	v8
a9	b9	c9	d9	e9	f9	g9	h9		js9	k9	m9	n9	p9	r9	s9	t9	u9	v9
a10	b10	c10	d10	e10	f10		h10		js10									
a11	b11	c11	d11	e11			h11		js11									
a12	b12	c12	d12				h12		js12									
a13	b13	c13					h13		js13									
a14	b14						h14		js14									
							h15		js15									
							'h16		js16									

1. First choice zones are ellipsed
2. Second choice zones framed
3. Third choice zones open

FIGURE 16.50 Preferred (Standardized) Tolerance Zones

A **positional tolerance** defines a zone within which the center, axis, or center plane of a feature of size is permitted to vary from true position. Basic dimensions establish the true position from specified datum features and between interrelated features. A positional tolerance is indicated by the position symbol, a tolerance, and appropriate datum references placed in a feature control frame.

16.12.8 Metric Preferred Sizes

Metric **preferred sizes** are based on the Renard series of preferred numbers. The first choice is rounded from the R10 series, where succeeding numbers each increase by 25%. The second choice is rounded from the R20 series, which has 12% increments. The rationale for first-choice sizes is the selection of every second number in the series, such as 1, 1.6, 2.5. This series is rounded from the R5 series of preferred numbers, in which the increments are 60%. Preferred sizes from 1 to 300 are given in metric (Table 16.2) and .01 to 20.00 in inches (Table 16.3). The **hole basis system** is the preferred system for selecting standard tools and gages.

16.12.9 Standardized Tolerances

Standardized **metric tolerances** are given in Table 16.4. International tolerance (IT) grade values are used. The basis for these is the tolerance unit i, which is defined as follows:

$$i = .45\sqrt[3]{D} + 0.001D$$

where D = the nominal dimension in millimeters.

Standardized **inch tolerances** are given in Table 16.5. The equivalents to IT values are based on the following formula (in inches):

$$i = .052\sqrt[3]{D} + .001D$$

where D = the nominal dimension in inches.

IT grades for manufacturing processes are shown in Figure 16.51. Production costs may be reduced by limiting dimensions or grades to those for which gaging equipment is available. This allows production personnel to apply "Go" (MMC) and "Not Go" (LMC) gages to the inspection of small parts.

16.13 CALCULATING LIMITS AND FITS

Metric tolerancing makes extensive use of limits and fits and symbology. In the United States, the symbology is supplemented by the limits, or the limits are specified and the symbology referenced on drawings, to prevent misinterpretation. Tables 16.6 and 16.7 are, respectively, metric and inch shaft position tables. These are used in conjunction with Tables 16.4 and 16.5 to calculate limits.

TABLE 16.2 Preferred Metric Basic Sizes (B.S.4318)

Choice			Choice			Choice		
1st	*2nd*	*3rd*	*1st*	*2nd*	*3rd*	*1st*	*2nd*	*3rd*
1					23			122
	1.1				24	125		
1.2			25					128
		1.3			26	130		
	1.4			28				132
		1.5	30				135	
1.6				32				138
		1.7			34	140		
	1.8		35					142
		1.9			36		145	
2				38				148
		2.1	40			150		
	2.2			42				152
		2.4			44		155	
2.5			45					158
		2.6			46	160		
	2.8			48				162
3			50				165	
		3.2		52				168
	3.5				54	170		
		3.8	55				175	
4					56			178
		4.2		58		180		
	4.5		60					182
		4.8		62			185	
5					64			188
		5.2	65			190		
	5.5				66			192
		5.8		68			195	
6			70					198
		6.2		72		200		
	6.5			74				205
		6.8	75				210	
	7				76			215
		7.5		78		220		
8			80					225
		8.5			82		230	
	9			85				235
		9.5			88	240		
10			90					245
	11				92		250	
12				95				255
		13			98	260		
	14		100					265
		15			102		270	
16				105				275
		17	110					285
	18				108		290	
		19			112			295
20				115		300		
		21			118			
	22		120					

TABLE 16.3 Preferred Basic Sizes in Inches

Decimal			Fractional					
0.010	2.00	8.50	$\frac{1}{64}$	0.015625	$2\frac{1}{4}$	2.2500	$9\frac{1}{2}$	9.5000
0.012	2.20	9.00	$\frac{1}{32}$	0.03125	$2\frac{1}{2}$	2.5000	10	10.0000
0.016	2.40	9.50	$\frac{1}{16}$	0.0625	$2\frac{3}{4}$	2.7500	$10\frac{1}{2}$	10.5000
0.020	2.60	10.00	$\frac{3}{32}$	0.09375	3	3.0000	11	11.0000
0.025	2.80	10.50	$\frac{1}{8}$	0.1250	$3\frac{1}{4}$	3.2500	$11\frac{1}{2}$	11.5000
0.032	3.00	11.00	$\frac{5}{32}$	0.15625	$3\frac{1}{2}$	3.5000	12	12.0000
0.040	3.20	11.50	$\frac{3}{16}$	0.1875	$3\frac{3}{4}$	3.7500	$12\frac{1}{2}$	12.5000
0.05	3.40	12.00	$\frac{1}{4}$	0.2500	4	4.0000	13	13.0000
0.06	3.60	12.50	$\frac{5}{16}$	0.3125	$4\frac{1}{4}$	4.2500	$13\frac{1}{2}$	13.5000
0.08	3.80	13.00	$\frac{3}{8}$	0.3750	$4\frac{1}{2}$	4.5000	14	14.0000
0.10	4.00	13.50	$\frac{7}{16}$	0.4375	$4\frac{3}{4}$	4.7500	$14\frac{1}{2}$	14.5000
0.12	4.20	14.00	$\frac{1}{2}$	0.5000	5	5.0000	15	15.0000
0.16	4.40	14.50	$\frac{9}{16}$	0.5625	$5\frac{1}{4}$	5.2500	$15\frac{1}{2}$	15.5000
0.20	4.60	15.00	$\frac{5}{8}$	0.6250	$5\frac{1}{2}$	5.5000	16	16.0000
0.24	4.80	15.50	$\frac{11}{16}$	0.6875	$5\frac{3}{4}$	5.7500	$16\frac{1}{2}$	16.5000
0.30	5.00	16.00	$\frac{3}{4}$	0.7500	6	6.0000	17	17.0000
0.40	5.20	16.50	$\frac{7}{8}$	0.8750	$6\frac{1}{2}$	6.5000	$17\frac{1}{2}$	17.5000
0.50	5.40	17.00	1	1.0000	7	7.0000	18	18.0000
0.60	5.60	17.50	$1\frac{1}{4}$	1.2500	$7\frac{1}{2}$	7.5000	$18\frac{1}{2}$	18.5000
0.80	5.80	18.00	$1\frac{1}{2}$	1.5000	8	8.0000	19	19.0000
1.00	6.00	18.50	$1\frac{3}{4}$	1.7500	$8\frac{1}{2}$	8.5000	$19\frac{1}{2}$	19.5000
1.20	6.50	19.00	2	2.000	9	9.0000	20	20.0000
1.40	7.00	19.50	—	—	—	—	—	—
1.60	7.50	20.00						
1.80	8.00	—	All dimensions are given in inches.					

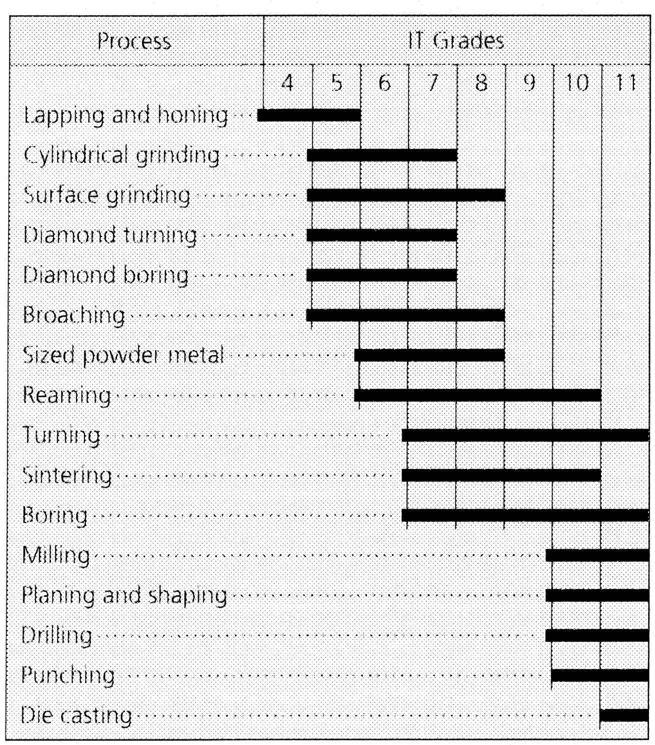


FIGURE 16.51 IT Grades for Manufacturing Processes

16.13.1 Calculating Metric Hole Limits

The hole basis system places the low limit of a hole at exactly the basic or design size. The high limit is calculated by adding to it the IT value (Table 16.4). For example:

∅ 80 H7 = 80 + 0.03, − 0

(.03 is from Table 16.4.)

16.13.2 Calculating Inch Hole Limits

The low limit of the hole is at the basic or design size. The high limit is calculated by adding to it the IT value (Table 16.5). For example:

∅ 3.1500 H7 = 3.1500 + 0.0012, − .0000

(.0012 is from Table 16.6.)

TABLE 16.4 Metric Table—Standard Tolerances

Nominal[1] Size		IT Tolerance Grade												IT = ISO Series of Tolerances					
Over (mm)	***Up to and includ-ing (mm)***	***IT01***	***IT0***	***IT1***	***IT2***	***IT3***	***IT4***	***IT5***	***IT6[3]***	***IT7***	***IT8***	***IT9***	***IT10***	***IT11***	***IT12***	***IT13***	***IT14[2]***	***IT15[2]***	***IT16[2]***
—	3	0·3	0·5	0·8	1·2	2	3	4	6	10	14	25	40	60	100	140	250	400	600
3	6	0·4	0·6	1	1·5	2·5	4	5	8	12	18	30	48	75	120	180	300	480	750
6	10	0·4	0·6	1	1·5	2·5	4	6	9	15	22	36	58	90	150	220	360	580	900
10	18	0·5	0·8	1·2	2	3	5	8	11	18	27	43	70	110	180	270	430	700	1100
18	30	0·6	1	1·5	2·5	4	6	9	13	21	33	52	84	130	210	330	520	840	1300
30	50	0·6	1	1·5	2·5	4	7	11	16	25	39	62	100	160	250	390	620	1000	1600
50	80	0·8	1·2	2	3	5	8	13	19	30	46	74	120	190	300	460	740	1200	1900
80	120	1	1·5	2·5	4	6	10	15	22	35	54	87	140	220	350	540	870	1400	2200
120	180	1·2	2	3·5	5	8	12	18	25	40	63	100	160	250	400	630	1000	1600	2500
180	250	2	3	4·5	7	10	14	20	29	46	72	115	185	290	460	720	1150	1850	2900
250	315	2·5	4	6	8	12	16	23	32	52	81	130	210	320	520	810	1300	2100	3200
315	400	3	5	7	9	13	18	25	36	57	89	140	230	360	570	890	1400	2300	3600
400	500	4	6	8	10	15	20	27	40	63	97	155	250	400	630	970	1550	2500	4000
500	630	—	—	—	—	—	—	—	44	70	110	175	280	440	700	1100	1750	2800	4400
630	800	—	—	—	—	—	—	—	50	80	125	200	320	500	800	1250	2000	3200	5000
800	1000	—	—	—	—	—	—	—	56	90	140	230	360	560	900	1400	2300	3600	5600
1000	1250	—	—	—	—	—	—	—	66	105	165	260	420	660	1050	1650	2600	4200	6600
1250	1600	—	—	—	—	—	—	—	78	125	195	310	500	780	1250	1950	3100	5000	7800
1600	2000	—	—	—	—	—	—	—	92	150	230	370	600	920	1500	2300	3700	6000	9200
2000	2500	—	—	—	—	—	—	—	110	175	280	440	700	1100	1750	2800	4400	7000	11000
2500	3150	—	—	—	—	—	—	—	135	210	330	540	860	1350	2100	3300	5400	8600	13500

Tolerance unit 0.001 mm
[1]Standard tolerance in microns (1μ = 0.001 mm)
[2]Not applicable to sizes below 1 mm
Not recommended for fits in sizes above 500 mm
ISO tolerance grade 6 in abbreviated form is IT6

16.13.3 Calculating Metric Shaft Limits

With a hole basis system, unless the shaft is at position *h* (see figure at the bottom right of Table 16.6), the value in Table 16.6 must be added algebraically to calculate the upper limit

TABLE 16.5 Inch Values—Standard Tolerances

IT Grade	01	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14*	15*	16*
≤ 0.12	0.012	0.02	0.03	0.05	0.08	0.12	0.15	0.25	0.4	0.6	1.0	1.6	2.5	4.0	6.0	10.0	16.0	25.0
> 0.12 to 0.24	0.015	0.025	0.04	0.06	0.10	0.15	0.2	0.3	0.5	0.7	1.2	1.8	3.0	5.0	7.0	12.0	18.0	30.0
> 0.24 to 0.40	0.015	0.025	0.04	0.06	0.10	0.15	0.25	0.4	0.6	0.9	1.4	2.2	3.5	6.0	9.0	14.0	22.0	35.0
> 0.40 to 0.71	0.02	0.03	0.05	0.08	0.12	0.2	0.3	0.4	0.7	1.0	1.6	2.8	4.0	7.0	10.0	16.0	28.0	40.0
> 0.71 to 1.19	0.025	0.04	0.06	0.10	0.15	0.25	0.4	0.5	0.8	1.2	2.0	3.5	5.0	8.0	12.0	20.0	35.0	50.0
> 1.19 to 1.97	0.025	0.04	0.06	0.10	0.15	0.3	0.4	0.6	1.0	1.6	2.5	4.0	6.0	10.0	16.0	25.0	40.0	60.0
> 1.97 to 3.15	0.03	0.05	0.08	0.12	0.2	0.3	0.5	0.7	1.2	1.8	3.0	4.5	7.0	12.0	18.0	30.0	45.0	70.0
> 3.15 to 4.73	0.04	0.06	0.1	0.15	0.25	0.4	0.6	0.9	1.4	2.2	3.5	5.0	9.0	14.0	22.0	35.0	50.0	90.0
> 4.73 to 7.09	0.05	0.08	0.12	0.2	0.3	0.5	0.7	1.0	1.6	2.5	4.0	6.0	10.0	16.0	25.0	40.0	60.0	100.0
> 7.09 to 9.85	0.08	0.12	0.2	0.3	0.4	0.6	0.8	1.2	1.8	2.8	4.5	7.0	12.0	18.0	28.0	45.0	70.0	120.0
> 9.85 to 12.41	0.10	0.15	0.25	0.3	0.5	0.6	0.9	1.2	2.0	3.0	5.0	8.0	12.0	20.0	30.0	50.0	80.0	120.0
> 12.41 to 15.75	0.12	0.2	0.3	0.4	0.5	0.7	1.0	1.4	2.2	3.5	6.0	9.0	14.0	22.0	35.0	60.0	90.0	140.0
> 15.75 to 19.69	0.15	0.25	0.3	0.4	0.6	0.8	1.0	1.6	2.5	4.0	6.0	10.0	16.0	25.0	40.0	60.0	100.0	160.0

Table shows standard tolerances in 0.001 inches for diameter steps in inches.
*Up to .04 in., grades 14 to 16 are not provided.

TABLE 16.6 Metric Tolerance Zone Position—Standard Fits

METRIC-TOLERANCE ZONE POSITION TABLE-SHAFT UPPER LIMITS FROM ZERO LINE (BASIC SIZE) SEE GRAPHIC AT LOWER RIGHT.

Loose, free & close running, sliding, locational, drive, and force fits

Over-To	c11	d9	f7	g6	h6	k6	n6	p6	s6	u6
≤3	−60	−20	−6	−2	0	6	10	12	20	24
3 to 6	−70	−30	−10	−4	0	9	16	20	27	31
6 to 10	−80	−40	−13	−5	0	10	19	24	32	37
10 to 14	−95	−50	−16	−6	0	12	23	29	39	44
14 to 18	−95	−50	−16	−6	0	12	23	29	39	44
18 to 24	−110	−65	−20	−7	0	15	28	35	48	54
24 to 30	−110	−65	−20	−7	0	15	28	35	48	61
30 to 40	−120	−80	−25	−9	0	18	33	42	59	76
40 to 50	−130	−80	−25	−9	0	18	33	42	59	86
50 to 65	−140	−100	−30	−10	0	21	39	51	72	106
65 to 80	−150	−100	−30	−10	0	21	39	51	78	121
80 to 100	−170	−120	−36	−12	0	25	45	59	93	146
100 to 120	−180	−120	−36	−12	0	25	45	59	101	166
120 to 140	−200	−145	−43	−14	0	28	52	68	117	195
140 to 160	−210	−145	−43	−14	0	28	52	68	125	215
160 to 180	−230	−145	−43	−14	0	28	52	68	133	235
180 to 200	−240	−170	−50	−15	0	33	60	79	151	265
200 to 225	−260	−170	−50	−15	0	33	60	79	159	287
225 to 250	−280	−170	−50	−15	0	33	60	79	169	313
250 to 280	−300	−190	−56	−17	0	36	66	88	190	347

1. Basic hole system (unilateral hole basis) employed (table covers shafts).
2. Values are in thousandths of a millimeter (microns); sizes in mm.
3. Values represent the upper limit (relative to the zero line) of shafts.
4. The selected fits indicated are recommended in ANSI B4.1. These are somewhat similar to those in (UK standard) BS 4500.
5. Add value from table algebraically to basic size for upper shaft limit.

Note:
- Uppercase letters represent holes/bores. (e.g., H7).
- Lowercase letters represent shafts (e.g., f6).
- The letter (location symbol) represents the position/distance to the "zero line"/Basic size.
- H and h are on the zero line.
- The number (quality no.) represents the tolerance grade. Higher nos. yield coarser fits.

* See tolerance grade (IT) table for limits not shown in these tables.
* Add IT to hole for max limit (Basic size is min hole)
* Subtract IT from shaft limit calc. from table for min.

METHODS OF INDICATING
1. Ø30 f7
2. Ø29.980 +.000, 2.021 (Ø30 f7)
3. Ø29.959–29.980 (Ø30 f7)

INDICATING FITS
1. Ø.30 H8/f7 (hole first)
2. Ø30 H8 (30.000−30.033)
   f7 (29.959−29.980)

Clearance Fits

FIT	HOLE	SHAFT
loose	H11	c11
free	H9	d9
close	H8	f7
sliding	H7	g6
locational	H7	h6

Transition Fits

FIT	HOLE	SHAFT
locational	H7	k6
locational	H7	n6

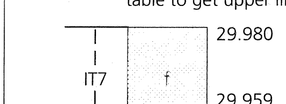
Interference Fits

FIT	HOLE	SHAFT
locational	H7	p6
med. drive	H7	s6
force	H7	u6

USING THE TABLE ABOVE
Ø30 f7 (Shaft)

zero line 30.00

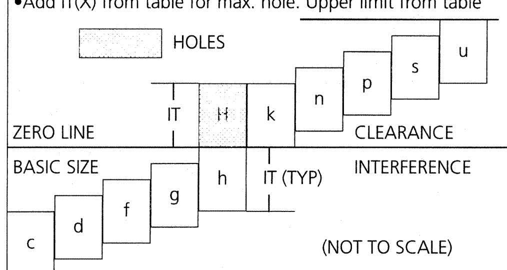
Basic size "−20 (.02) from above" table to get upper limit



IT7 (other table) = .021

30.000 2 .020 5 29.980
29.980 2 .021 5 29.959

•BASIC size is min. hole size.
•Add IT(X) from table for max. hole. Upper limit from table



SHAFTS

•Subtract value from table above for one (high) limit of shaft.
•Subtract IT(X) from high limit of shaft for low limit.

POSITIONS OF TOLERANCE ZONES TO ZERO LINE.

of a shaft; subtracting the IT value (Table 16.4) from the upper limit yields the lower limit.

Example:

∅ 80 k6 = 80 + .021, + .002

or

80.002–80.021 (.021 – .019 = .002)

(.021 is from Table 16.6 and .019 from Table 16.4.)

TABLE 16.7 Inch Tolerance Zone Position—Standard Fits

INCH-TOLERANCE ZONE POSITION TABLE-SHAFT UPPER LIMITS FROM ZERO LINE (BASIC SIZE) SEE GRAPHIC ON METRIC TABLE PAGE.

RUNNING, SLIDING, CLEARANCE, TRANSITION, AND INTERFERENCE LOCATIONAL FITS

Over-To	c9&c10	d8	d9	e7&E8	e9	f6,f7	f8	g4,g5	g6	h5,6,7&9	s6	s7	k6	k7	n5	n6	n7	p6	r6	Sp-10	Sp-11	Sp-12
0-.12	−2.50	−1.0	−1.0	−.6	−.6	−.3	−.3	−.10	−.10	0	0.1	0.2	-	-	0.45	0.5	0.65	0.65	0.75	−4.0	−4.0	−5
.12-.24	−2.80	−1.2	−1.2	−.8	−.8	−.4	−.4	−.15	−.15	0	0.15	0.25	-	-	0.5	0.6	0.8	0.8	0.9	−4.5	−4.5	−6
.24-.40	−3.00	−1.6	−1.6	−1.0	−1.0	−.5	−.5	−.20	−.20	0	0.2	0.3	0.5	0.7	0.65	0.8	1	1	1.2	−5.0	−5.0	−7
.40-.71	−3.50	−2.0	−2.0	−1.2	−1.2	−.6	−.6	−.25	−.25	0	0.2	0.35	0.5	0.8	0.8	0.9	1.2	1.1	1.4	−6.0	−6.0	−8
.71-1.19	−4.50	−2.5	−2.5	−1.6	−1.6	−.8	−.8	−.30	−.30	0	0.25	0.4	0.6	0.9	1	1.1	1.4	1.3	1.7	−7.0	−7.0	−10
1.19-1.97	−5.00	−3.0	−3.0	−2.0	−2.0	−1.0	−1.0	−.40	−.40	0	0.3	0.5	0.7	1.1	1.1	1.3	1.7	1.6	2	−8.0	−8.0	−12
1.97-3.15	−6.00	−4.0	−4.0	−2.5	−2.5	−1.2	−1.2	−.40	−.40	0	0.3	0.6	0.8	1.3	1.3	1.5	2	2.1	2.3	−9.0	−10.0	−14
3.15-4.73	−7.00	−5.0	−5.0	−3.0	−3.0	−1.4	−1.4	−.50	−.50	0	0.4	0.7	1	1.5	1.6	1.9	2.4	2.5	2.9	−10.0	−11.0	−16
4.73-7.09	−8.00	−6.0	−6.0	−3.5	−3.5	−1.6	−1.6	−.60	−.60	0	0.5	0.8	1.1	1.7	1.9	2.2	2.8	2.8	3.5	−12.0	−12.0	−18
7.09-9.85	−10.00	−7.0	−7.0	−4.0	−4.0	−2.0	−2.0	−.60	−.60	0	0.6	0.9	1.4	2	2.2	2.6	3.2	3.2	4.2	−15.0	−16.0	−22
9.85-12.41	−12.00	−8.0	−7.0	−5.0	−4.5	−2.5	2.2	−.80	−.70	0	0.6	1	1.4	2.2	2.3	2.6	3.4	3.4	4.7	−18.0	−20.0	−28
12.41-15.75	−14.00	−10.0	−8.0	6	−5.0	−3.0	2.5	−1.00	−.70	0	0.7	1	1.6	2.4	2.6	3	3.8	3.9	5.9	−22.0	−22.0	−30

1. Basic hole system employed (table covers shafts).
2. Values are in thousandths of an inch.
3. Values represent the upper limit (relative to the zero line) of shafts.
4. Values indicated "Sp-X" are not used in the ISO (International ISO 286).
*5. Add value from table algebraically to basic size for upper shaft limit.

FORCE AND SHRINK FITS

Over-To	s6	t6	u6	x7	Sp-5
0-.12	0.85	-	0.95	1.3	0.5
.12-.24	1	-	1.2	1.7	0.6
.24-.40	1.4	-	1.6	2	0.75
0.4-.56	1.6	-	1.8	2.3	0.8
.56-.71	1.6	-	1.8	2.5	0.9
.71-.95	1.9	-	2.1	3	1.1
.95-1.19	1.9	2.1	2.3	3.3	1.2
1.19-1.58	2.4	2.6	3.1	4	1.3
1.58-1.97	2.4	2.8	3.4	5	1.4
1.97-2.56	2.7	3.2	4.2	6.2	1.8
2.56-3.15	2.9	3.7	4.7	7.2	1.9
3.15-3.94	3.7	4.4	5.9	8.4	2.4
3.94-4.73	3.9	4.9	6.9	9.4	2.6
4.73-5.52	4.5	6	8	11.6	2.9
5.52-6.30	5	6	8	13.6	3.2
6.30-7.09	5.5	7	9	13.6	3.5
7.09-7.88	6.2	8.2	10.2	15.8	3.8
7.88-8.86	6.2	8.2	11.2	17.8	4.3
8.86-9.85	7.2	9.2	13.2	17.8	4.3
9.85-11.03	7.2	10.2	13.2	20	4.9
11.03-12.41	8.2	10.2	15.2	22	4.9
12.41-13.98	9.4	11.4	17.4	24.2	5.5
13.98-15.75	9.4	13.4	19.4	27.2	6.1
$15.75-17.72	10.6	13.6	21.6	30.5	7
17.72-19.69	11.6	15.6	23.6	32.5	7

Running and Sliding Fits

FIT	HOLE	SHAFT
RC1	H5	g4
RC2	H6	g5
RC3	H7	f6
RC4	H8	f7
RC5	H8	e7
RC6	H9	e8
RC7	H9	d8
RC8	H10	c9
RC9	H11	Sp-10

Clearance Locational Fits

FIT	HOLE	SHAFT
LC1	H6	h5
LC2	H7	h6
LC3	H8	h7
LC4	H10	h9
LC5	H7	g6
LC6	H9	f8
LC7	H10	e9
LC8	H10	d9
LC9	H11	c10
LC10	H12	Sp11
LC11	H13	Sp12

Clearance Locational Fits

FIT	HOLE	SHAFT
LT1	H7	ls6
LT2	H8	ls7
LT3	H7	k6
LT4	H8	k7
LT5	H7	n6
LT6	H7	n7

Interference Locational Fits

FIT	HOLE	SHAFT
LN1	H6	n5
LN2	H7	p6
LN3	H7	r6

Force and Shrink Fits

FIT	HOLE	SHAFT
FN1	H6	Sp5
FN2	H7	s6
FN3	H7	t6
FN4	H7	u6
FN5	H8	x7

Note:
- Uppercase letters represent holes/bores (e.g., H7).
- Lowercase letters represent shafts (e.g., f6).
- The letter (location symbol) represents the position/distance to the "zero line"/Basic size.
- H and h are on the zero line.
- The number (quality no.) represents the tolerance grade. Higher nos. yield coarser fits.

* See tolerance grade (IT) table for limits not shown in these tables.
* Add IT(x) for hole limit (from 0). (Basic hole is min size.)
* Subtract IT from max shaft limit calculated from basic size and value in table for other limit.

METHODS OF INDICATING
1. Ø.30 f7
2. Ø.2995 +.000, −.0006 (Ø.30f7)
3. Ø2989–.2995 (Ø.30f7)

INDICATING FITS
1. Ø.30 H8/f7 (hole first)
2. Ø.30 H8 (.3000–.3009)
f7 (.2989–.2995)

Location on zero line (H) — IT Grade (7)
Ø30 H7
Basic Size (Ø30) — Hole Designation

Location to zero line (f) — IT Grade (7)
Ø30 f7
Basic Size (Ø30) — Shaft Designation

Example:

∅ 80 c11 = .80 − .150, − 340

or

79.660 − 79.850 (−.150 − .190) = −.340

(−.150 is from Table 16.6 and .190 from Table 16.4.)

16.13.4 Calculating Inch Shaft Limits

With a hole basis system, unless the shaft is at position *h* (see figure at the bottom right of Table 16.7), the value in Table 16.7 must be added algebraically to arrive at the upper limit of a shaft; subtracting the IT value (Table 16.6) from the upper limit yields the lower limit.

Example

∅ 3.15 k6 = 3.1500 − .0008, − .0015

or

3.1485−3.14992 (−.0008 − .0007 = − .0015)

(− .0008 is from Table 16.7 and .0007 from Table 16.6.)

Example

∅ 12.00 d9 = 12.00 − .007, − .012

or

11.988−11.993 (−.007 − .005 = − .012)

(.007 is from Table 16.7 and .005 from Table 16.6.)

page 633

page 634

page 635

page 636

page 637

page 638

page 639

page 640

page 641

page 643

page 644

{end of chapter 16}