1.0 SCOPE

1.1 General - This is a multiple variation specification (multiple revisions of ASME Y14.5) providing a uniform interpretation of size requirements, datums, form, location, and profile tolerances called for on Caterpillar drawings/models and specifications. This specification is based on American Society of Mechanical Engineers (ASME) Standard Y14.5 and International Organization for Standardization (ISO) Standards R1101, and 1660.

2.0 DRAWING DESIGNATIONS

Current Designation	Former Designation
1E0012	1E0012
1E0012A	None

Figure 1

3.0 APPLICATION

3.1 1E0012 - The interpretations provided in this specification are applicable to all Caterpillar drawings as part of specification **1E0011 INTERPRETATION AND TOLERANCE**. Specification 1E0012 will not appear on drawings/models.

3.1.1 Dimensions and tolerances shown in the illustrations may be in either millimeters or inches. The values shown are for illustrative purposes only.

3.1.2 Unless otherwise specified, all geometric tolerances apply for full depth, length and width of the feature.

3.2 1E0012A - The interpretations provided in ASME Y14.5-2009 are applicable to all Caterpillar drawings having 1E0012A listed within the drawing Specification Block with the exception of Datum Targets. See Paragraph 8.5 (this document) for Datum Target interpretation.

4.0 TERMS AND DEFINITIONS

4.1 Datum - 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 component are established. (See Article 8.0).

4.2 Datum Target - A specified point, line, or area on a component used to establish a datum (See Article 8.0).

4.3 Feature - The general term applied to a physical portion of a component, such as a surface, pin, tab, hole, or slot.

4.4 Feature Of Size - One cylindrical or spherical surface, or a set of two opposed elements or opposed parallel surfaces, associated with a size dimension.

4.5 Datum Feature - An actual feature of a component that is used to establish a datum. (See Article 8.0).

4.6 Basic Dimension - A numerical value used to describe 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. Toleranced dimensions that locate features are considered basic when they are also used to establish basic profile for profile of surface tolerances such as cast or forged surfaces.

4.7 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, minimum hole diameter, maximum shaft diameter. (See Article 7.0).

4.8 Least Material Condition (LMC) - The condition in which a feature of size contains the least amount of material within the stated limits of size - for example, maximum hole diameter, minimum shaft diameter. (See Article 7.0).

4.9 Regardless Of Feature Size (RFS) - The term used to indicate that a geometric tolerance or datum reference applies at any increment of size of the feature within its size tolerance. (See Article 7.0).

4.10 Form Tolerance - The maximum permissible form variation from the perfect form indicated (When applicable) by basic dimensions. (See Article 9.0).

4.11 Positional Tolerance - The maximum permissible position variation from the true position indicated by basic dimensions. (See Article 10.0).

4.12 Profile Tolerance - The maximum permissible variation in the profile of a surface from the perfect profile indicated by basic dimensions. (See Article 11.0).

4.13 Size Tolerance - The difference between maximum and minimum limits of size.

4.14 Inner Boundary - A worst case boundary (that is, locus) generated by the smallest feature (MMC for an internal feature and LMC for an external feature) minus the stated geometric tolerance and any additional geometric tolerance (if applicable) from the feature's departure from its specified material condition.

4.15 Outer Boundary - A worst case boundary (that is, locus) generated by the largest feature (LMC for an internal feature and MMC for an external feature) plus the geometric tolerance and any additional geometric tolerance (if applicable) from the feature's departure from its specific material condition.

4.16 Datum Feature Simulator - A surface of adequately precise form (such as a surface plate, a gage surface, or mandrel) contacting the datum feature(s) and used to establish the simulated datum(s). (See Article 8.0).

4.17 Simulated Datum - A point, axis, or plane established by processing or inspection equipment, such as the following simulators: a surface plate, a gage surface, or a mandrel. (See Article 8.0).

4.18 Actual Mating Envelope - This term is defined according to the type of feature, as follows:

4.18.1 For An External Feature - A similar perfect feature counterpart of smallest size that can be circumscribed about the feature so that it just contacts the surface at the highest points.

4.18.1.1 For example, a smallest cylinder of perfect form or two parallel planes of perfect form at minimum separation that just contact(s) the highest points of the surface(s).

4.18.1.2 For features controlled by orientation or positional tolerances, the actual mating envelope is oriented relative to the appropriate datum(s), for example, perpendicular to a primary datum plane.

4.18.2 For An Internal Feature - A similar perfect feature counterpart of largest size that can be inscribed within the feature so that it just contacts the surface at the highest points.

4.18.2.1 For example, a largest cylinder of perfect form or two parallel planes of perfect form at maximum separation that just contact(s) the highest points of the surface(s).

4.18.2.2 For features controlled by orientation or positional tolerances, the actual mating envelope is oriented relative to the appropriate datum(s).

4.19 Axis Of Feature - A straight line that coincides with the axis of the true geometric counterpart of the specified feature.

4.20 Center Plane Of Feature - A plane that coincides with the center plane of the true geometric counterpart of the specified feature.

4.21 Derived Median Plane Of Feature - An imperfect plane (abstract) that passes through the center points of all line segments bounded by the feature. These line segments are normal to the actual mating envelope.

4.22 Derived Median Line Of Feature - An imperfect line (abstract) that passes through the center points of all cross sections of the feature. These cross sections are normal to the axis of the actual mating envelope.

4.23 Tangent Plane - A theoretically exact plane derived from the true geometric counterpart of the specified feature surface.

4.24 Resultant Condition - The variable boundary generated by the collective effects of a size feature's specified MMC or LMC material condition, the geometric tolerance for the material condition, the size tolerance, and the additional geometric tolerance derived from the feature's departure from its specified material condition.

4.25 Actual Size - The general term for the size of a produced feature. This term includes the actual mating size and the actual local sizes.

4.26 Actual Local Size - The value of any individual distance at any cross section of a feature.

4.27 Actual Mating Size - The dimensional value of the actual mating envelope.

4.28 Limits Of Size - The specified maximum and minimum sizes.

4.29 Nominal Size - The designation used for purposes of general identification.

4.30 Resultant Condition Size - The actual value of the resultant condition boundary.

4.31 Virtual Condition Size - The actual value of the virtual condition boundary.

4.32 True Geometric Counterpart - The theoretically perfect boundary (virtual condition or actual mating envelope) or best-fit (tangent) plane of a specified datum feature.

4.33 Mathematically Defined Datum Feature - A contoured surface mathematically defined and related to a datum reference system. The theoretically true geometric counterpart of the shape is used to establish the datum.

5.0 GENERAL RULES

5.1 Limits Of Size - Unless otherwise specified, the limits of size of a feature prescribe the extent within which variations of geometric form, as well as size, are allowed. This control applies solely to individual features of size as defined in Paragraph 5.1.1.

5.1.1 Individual Feature Of Size - Where only a tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its geometric form, as well as size, are allowed.

5.1.1.1 Variations Of Size - The actual local size of an individual feature at any cross section shall be within the specified limits of size.

5.1.1.2 Variations Of Form (Boundary Principle) - The form of an individual feature is controlled by its limits of size to the extent prescribed in the following paragraphs and illustrated in Figure 2.

The surface or surfaces of a feature shall not extend beyond a boundary of perfect form at MMC. This boundary is the true geometric form represented by the drawing/model. No variation in form is permitted if the feature is produced at its MMC limit of size.

Where the actual local 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.

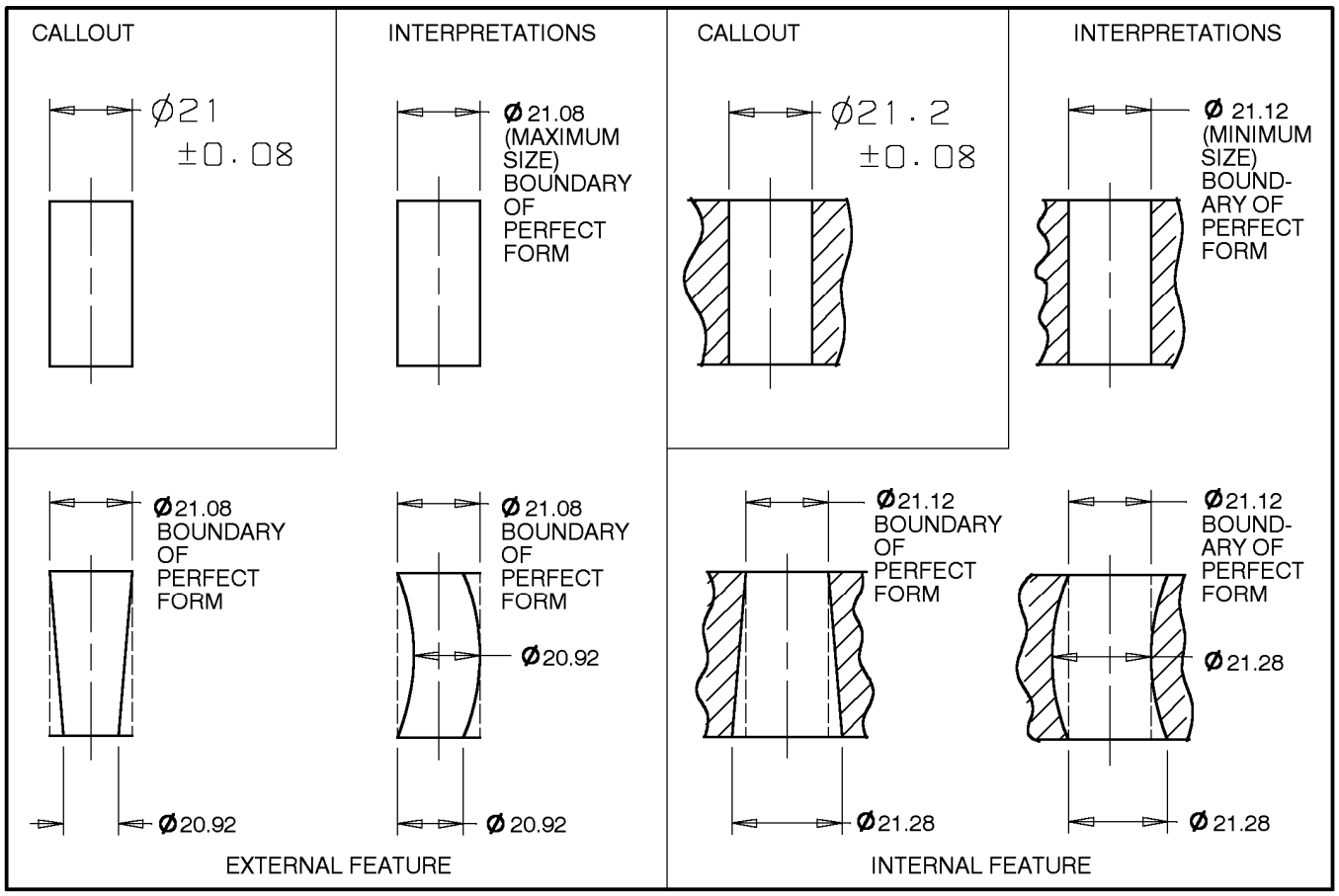


Figure 2 - Size Dimensions For Individual Feature

5.1.2 Commercial Stock Exception - Form tolerances for as furnished commercial stock such as bars, sheet, and tubing are controlled by mill tolerance specifications such as 1E2177, 1E2315, 1E2324, and 1E2325, or established industry standards.

5.1.3 Free State Exception - Form tolerances for non-rigid components are controlled using the free state modifier symbol to indicate the measurements are to be taken in the unrestrained condition (free state).

5.1.4 Perfect Form At MMC Not Required - Where a note such as **PERFECT FLATNESS NOT REQD AT MAX SIZE** Is specified related to a form tolerance and a size dimension, the surface is permitted to exceed the boundary of perfect form at MMC.

5.2 Positional Tolerances - Positional tolerances were formerly called true position tolerances (See Paragraph 10.3). The positional tolerances shall be specified with the appropriate material condition modifier.

5.3 Other Geometric Tolerances - RFS applies with respect to the individual tolerance, datum reference, or both, where no modifying symbol is specified. MMC or LMC shall be specified on the drawing/model where it is required.

5.3.1 Size, form, and location tolerances are checked independently and each shall conform.

5.3.2 For straightness (surface elements), flatness, circularity (roundness), cylindricity, and parallelism: The specified tolerance value may be reduced to zero due to the effect of size or position relative to the boundary of perfect form requirement.

5.3.3 For perpendicularity, angularity, runout, concentricity, and symmetry: The full specified tolerance value is available regardless of feature size (RFS) unless modified as shown in Paragraph 7.1.

5.4 Screw Threads - Geometric tolerances and datum references on a screw thread apply to the pitch diameter unless noted otherwise on the drawing/model.

5.5 Gears and Splines - A gear or spline pitch diameter may be designated as a datum as shown in Figure 3. The datum is established on the involute surfaces of the gear or spline as close to the midpoint of the involute as practical. See Specification 1E0392 for gears. See specification 1E2028 for splines.

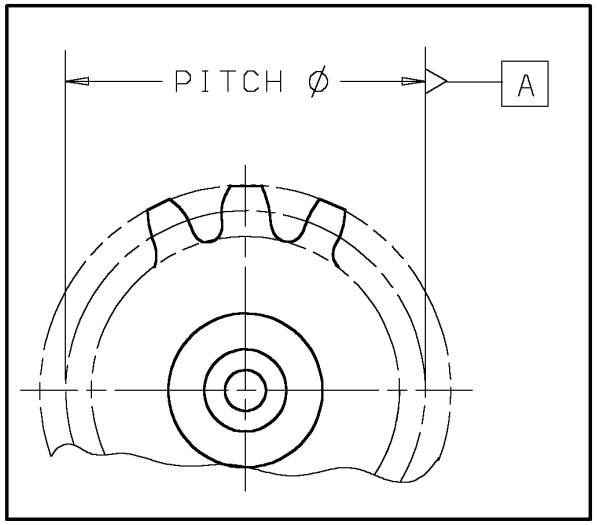


Figure 3 - Datum on Gear

5.6 Virtual Condition - Depending upon its function, a feature is controlled by tolerances such as size, form, orientation, and location; MMC or RFS may also be applicable. Consideration shall be given to the collective effect of these factors in determining the clearance between mating parts and in establishing gage feature sizes. From such consideration, a net resultant boundary is derived, termed virtual condition, as illustrated by the following:

5.6.1 Where a straightness of the axis tolerance of 0.04 diameter is specified at MMC or RFS for a pin whose size limits are 15.89-16.00 diameter, the virtual condition boundary is 16.04 diameter.

5.6.2 Where a 0.4 diameter positional tolerance is specified at MMC for a hole whose size limits are 14.0-14.8 diameter, the virtual condition boundary is 13.6 diameter.

5.6.3 Where a perpendicularity tolerance of 0.05 diameter is specified at MMC for a pin whose size limits are 15.966-15.984 diameter, the virtual condition boundary is 16.034 diameter and is perpendicular to the datum.

5.6.4 Datum Features At Virtual Condition - Depending on whether it is used as a primary, secondary, or tertiary datum, a virtual condition exists for a datum feature of size where its axis or center plane is controlled by a geometric tolerance. In such a case, the datum feature applies at its virtual condition even though it is referenced in a feature control frame at MMC or LMC.

6.0 GENERAL INTERPRETATION

6.1 Feature Control Frames - Features to be controlled are identified by feature control frames which consist of the geometric characteristic symbol, the tolerance value, modifiers (where applicable) and datum reference letters (where applicable) as shown in Figure 4.

Other information in a feature control frame may be specified in the original or international sequence with no difference in interpretation. The international sequence is used in illustrations in this specification.

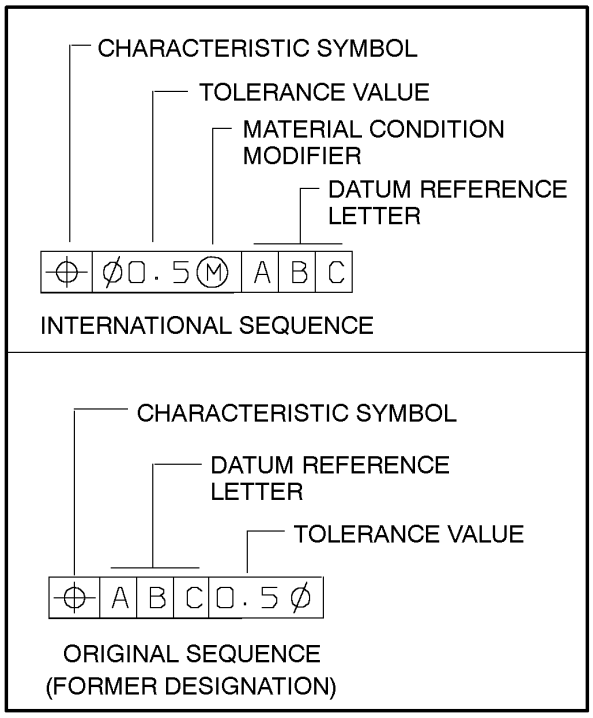


Figure 4 - Feature Control Frame

6.1.1 Feature Control Frames May Be Combined - Composite or multiple single segment feature control frames may be used where more than one tolerance of a given geometric characteristic applies to the same feature. Each tolerance is measured independently and each shall conform. (See Figure 5).

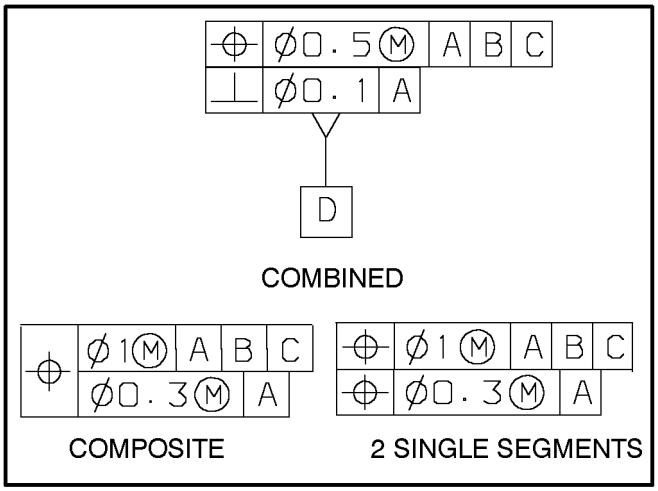


Figure 5 - Combined, Composite And Multiple Feature Control Frames

6.1.2 Feature control frames and datum symbols are connected to the feature being toleranced as shown in Figure 6 by one of the following methods:

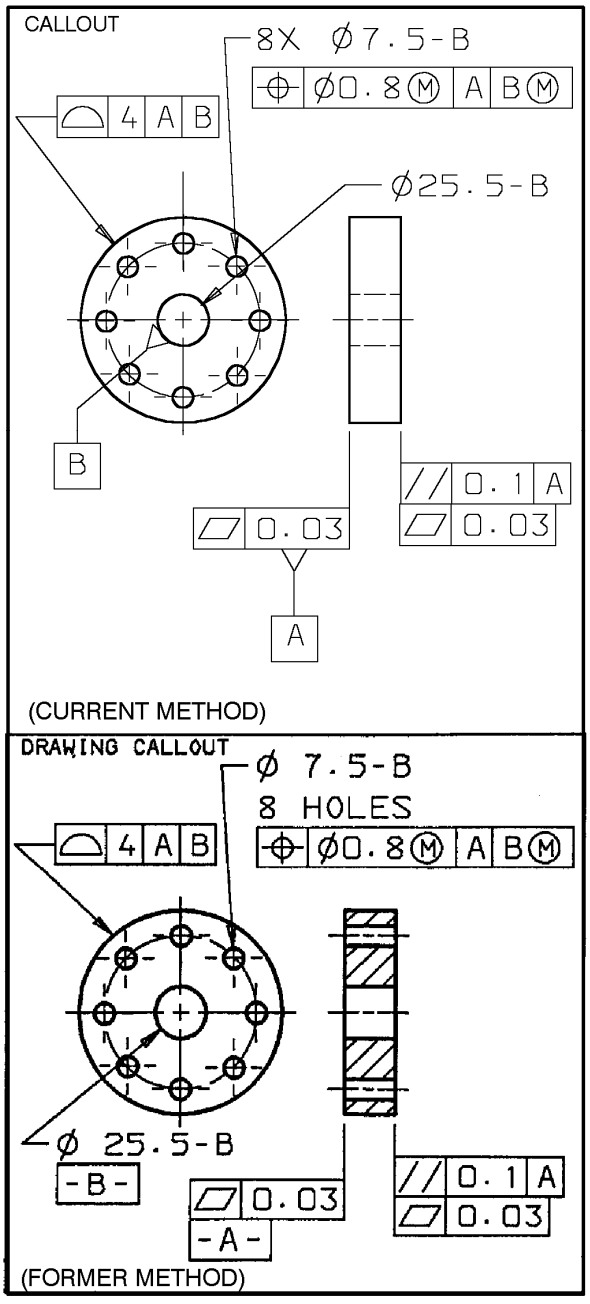


Figure 6 - Feature Control Frames

6.1.2.1 The feature control frame is part of a note or dimension pertaining to the feature.

6.1.2.2 A leaderline is connected from the feature to the center of the feature control frame.

6.1.2.3 A side or end of the feature control frame is connected to a dimension or extension line from the feature.

6.1.2.4 A datum feature symbol is attached to a feature, feature control frame, leader line, or extension line.

6.2 Notes - Notes are used to describe or supplement geometric requirements where symbols will not provide adequate control.

7.0 EFFECT OF MODIFIERS

7.1 Regardless of Feature Size - Referencing a datum feature on an RFS basis means that a centering about its axis or center plane is necessary, regardless of the actual size of the feature. Where a geometric tolerance is applied on an RFS basis, the specified tolerance is independent of the size of the considered feature. The tolerance is limited to the specified value regardless of the actual size of the feature.

7.1.1 Where the modifier symbol Ⓢ is specified with the datum reference, the datum feature size variation does not allow increased tolerance. (See Figure 7, Example A).

7.1.2 Where the modifier symbol Ⓢ is specified with the tolerance, the controlled feature size variation does not allow increased tolerance. (See Figure 7, Example B).

7.1.3 Where the modifier symbol Ⓢ is specified with both the datum reference and the tolerance, the tolerance applies with no increase. (See Figure 7, Example C).

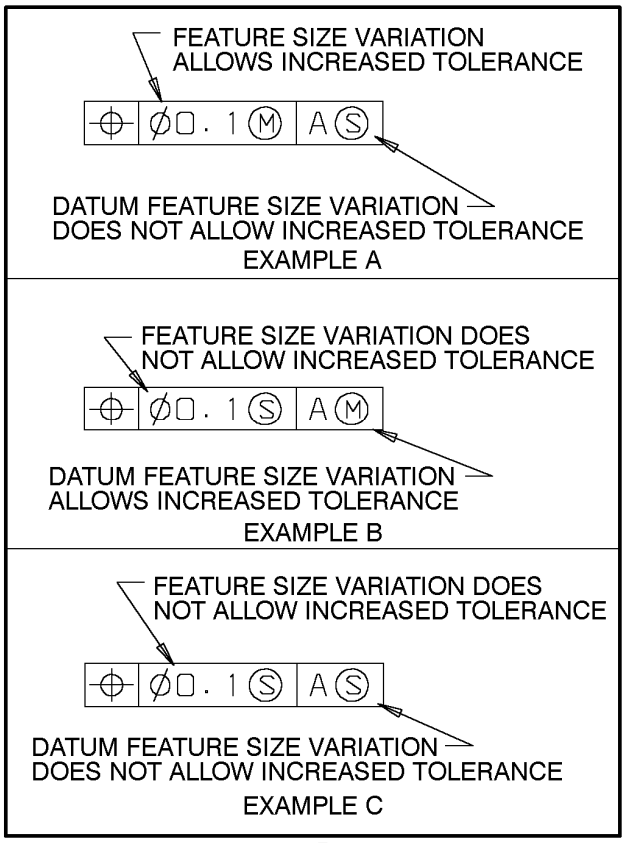


Figure 7 - Ⓢ Modifier

7.2 Maximum Material Condition - Referencing a datum feature on an MMC basis means the datum is the axis or center plane of the feature at the MMC limit. Where the actual size of the datum feature has departed from MMC, a deviation is allowed between its axis or center plane and the axis or center plane of the datum. Where a geometric tolerance is applied on an MMC basis, the specified tolerance is dependent on the size of the considered feature. Where the actual size of the feature has departed from MMC, an increase in the tolerance is allowed equal to the amount of such departure. The total permissible variation in the specific geometric characteristic is maximum when the feature is at LMC. The tolerance is limited to the specified value if the feature is produced at its MMC limit of size.

7.2.1 Where the modifier symbol Ⓜ is specified with the datum reference, the datum feature size variation allows increased tolerance. (See Figure 8, Example A). The relationship of the controlled feature, with respect to the datum, may shift within a tolerance zone located at the datum axis or center plane. The size of the zone equals the amount:

7.2.1.1 An external feature decreases from maximum size (MMC), within the specified limits of size.

7.2.1.2 An internal feature increases from minimum size (MMC), within the specified limits of size.

7.2.2 Where the modifier symbol Ⓜ is specified with the tolerance, the controlled feature size variation allows increased tolerance. (See Figure 8, Example B).

7.2.2.1 Increased (bonus) tolerance applies to the controlled feature if the size departs from MMC. The tolerance specified in the feature control frame increases by the amount:

An external controlled feature decreases from maximum size (MMC), within the specified limits of size.

An internal controlled feature increases from minimum size (MMC), within the specified limits of size.

7.2.2.2 The tolerance (No bonus) applies to the controlled feature where:

An external feature is at maximum size (MMC), such as maximum shaft size.

An internal feature is at minimum size (MMC), such as minimum hole size.

7.2.3 Both interpretations (Paragraphs 7.2.1 and 7.2.2) apply where the modifier symbol Ⓜ is specified with the tolerance and the datum reference, as shown in Figure 8, Example C.

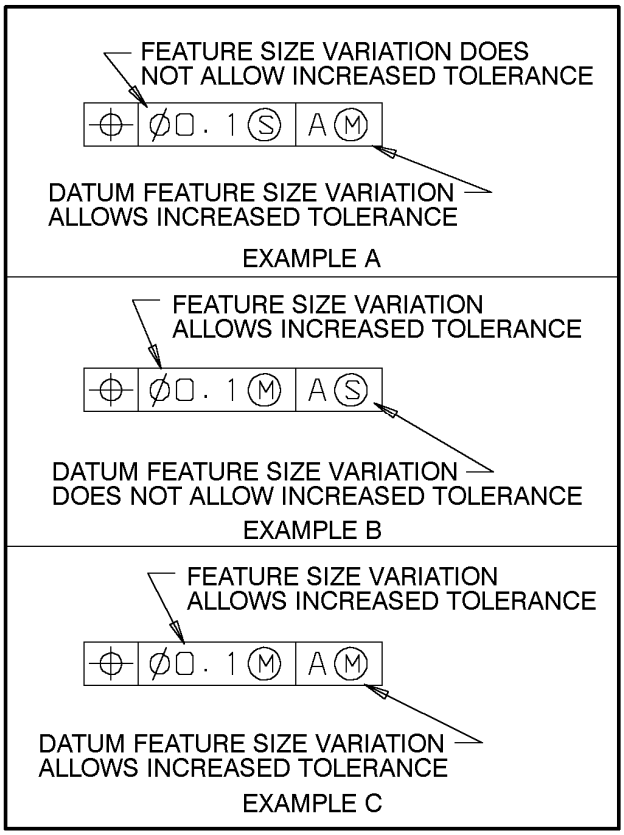


Figure 8 - Ⓜ Modifier

7.3 Least Material Condition - Referencing a datum feature on an LMC basis means the datum is the axis or center plane of the feature at the LMC limit. Where the actual size of the datum feature has departed from LMC, a deviation is allowed between its axis or center plane and the axis or center plane of the datum. Where a geometric tolerance is applied on an LMC basis, the specified tolerance is dependent on the size of the considered feature. Where the actual size of the feature has departed from LMC, an increase in the tolerance is allowed equal to the amount of such departure. The total permissible variation in specific geometric characteristic is maximum when the feature is at MMC. The specified tolerance is limited to the specified value if the feature is produced at its LMC limit of size.

7.3.1 Where the modifier symbol Ⓛ is specified with the datum reference, the datum feature size variation allows increased tolerance. (See Figure 9, Example A).

The relationship of the controlled feature, with respect to the datum, may shift within a tolerance zone located at the datum axis or center plane. The size of the zone equals the amount:

7.3.1.1 An external feature increases from minimum size (LMC), within the specified limits of size.

7.3.1.2 An internal feature decreases from maximum size (LMC), within the specified limits of size.

7.3.2 Where the modifier symbol Ⓛ is specified with the tolerance, the controlled feature size variation allows increased tolerance. (See Figure 9, Example B).

7.3.2.1 Increased (Bonus) tolerance applies to the controlled feature if the size departs from LMC. The tolerance specified in the feature control frame increases by the amount:

An external controlled feature increases from minimum size (LMC), within the specified limits of size.

An internal controlled feature decreases from maximum size (LMC), within the specified limits of size.

7.3.2.2 The tolerance (No bonus) applies to the controlled feature where:

An external feature is at minimum size (LMC), such as minimum shaft size.

An internal feature is at maximum size (LMC), such as a maximum hole size.

7.3.3 Both interpretations (Paragraphs 7.3.1 and 7.3.2) apply where the modifier symbol Ⓛ is specified with the tolerance and the datum reference, as shown in Figure 9, Example C.

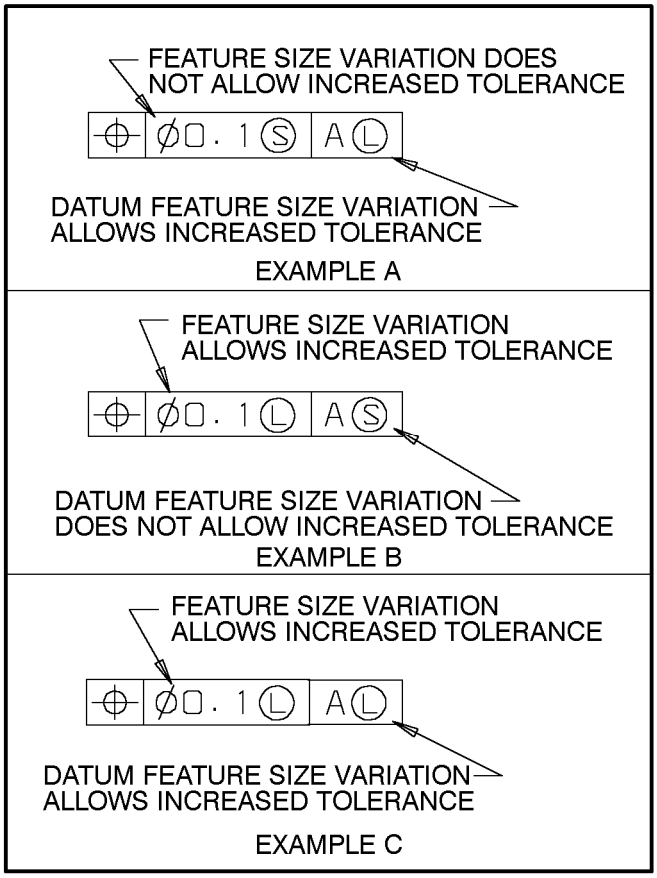


Figure 9 - Ⓛ Modifier

7.4 Projected Tolerance Zone - The modifier symbol Ⓟ indicates a projected tolerance zone. Where the symbol Ⓟ and a value are specified within or below a positional tolerance feature control frame, the tolerance zone is projected to the area of the mating component. The tolerance zone does not apply within the feature being controlled. (See Figure 10 and Paragraph 10.3).

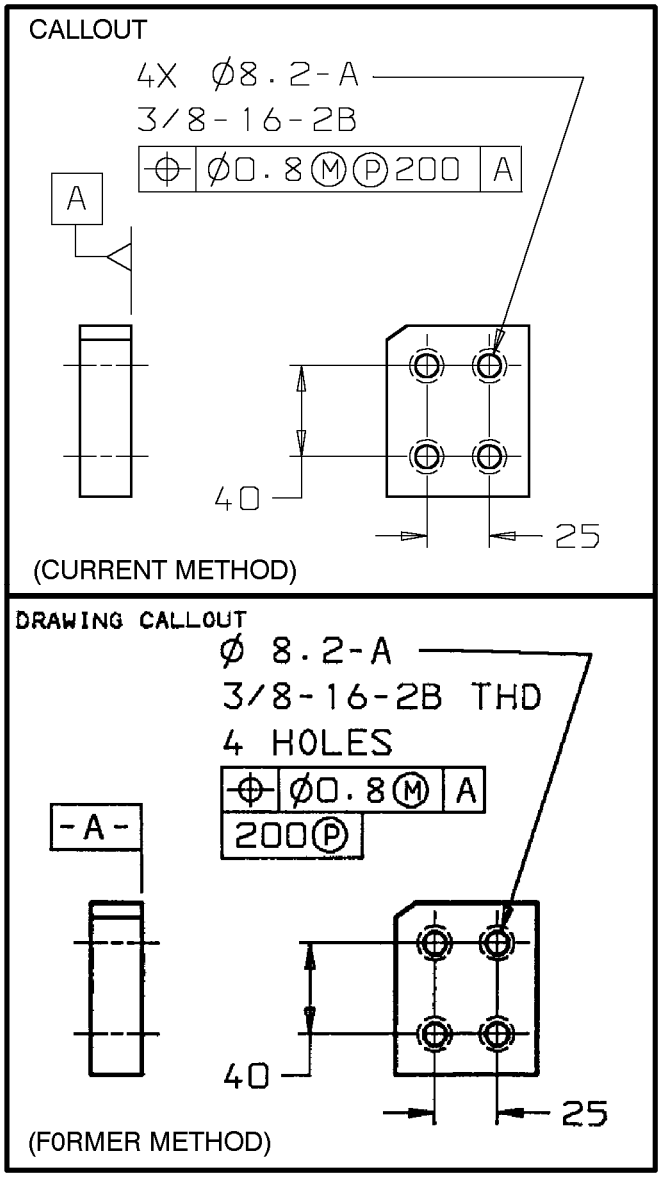


Figure 10 - Ⓟ Modifier

7.5 Tangent Plane - The modifier symbol Ⓣ is specified in the feature control frame after the stated tolerance. The tangent plane concept may be used with straightness, flatness, parallelism, perpendicularity, and angularity. (See Paragraph 9.5).

7.6 Free State - The modifier symbol Ⓕ indicates a free state variation exists. This is used with non-rigid components to indicate measurements are to be taken in the unrestrained condition. (See Paragraph 9.3).

8.0 DATUMS

8.1 Datum Concept - Because measurements cannot be made from true geometric counterparts, datums are assumed to exist in associated processing equipment. For example, machine tables and surface plates are of such quality that they are used to simulate datum planes for purposes of dimensional verification.

8.2 Datum Feature Symbols - Datum features are identified by the datum feature symbols in Figure 11.

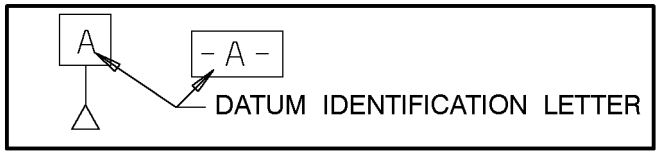


Figure 11 - Datum Feature Symbols

8.3 Datum Identification Letters - Datum identification letters are used in feature control frames. Datums are listed in the order in which they are to be applied for positioning a component for inspection of functional requirements (these are not necessarily the datums used for machining purposes). See Figures 12 thru 19.

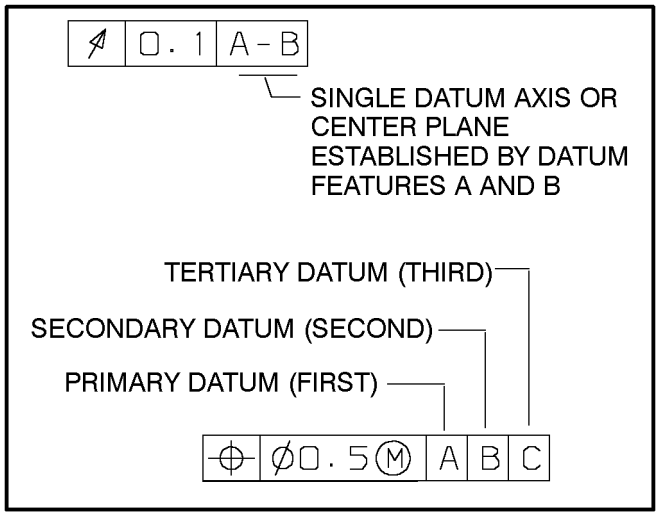


Figure 12 - Datum Identification

8.4 Datum Reference System - The datum reference system consists of three mutually perpendicular datum planes which are established by datum features. Surfaces or other features most important to the design of a component are selected as datum features. Sufficient features are chosen to position the component in relation to the three datum planes. All related measurements of the component originate from these planes. On many components, a single datum reference system is adequate. Multiple datum reference systems are used where the separation of features require additional datum reference systems at other locations on the component. Each feature control frame designates the applicable datums.

8.4.1 Datums specified on drawings/models do not dictate the specific method of manufacturing (including gaging). Processing and/or inspection may be accomplished by using other datums. However, the method used to inspect finished components shall be capable of validating (directly or indirectly) all design requirements including references to design datums.

8.4.2 For a datum system involving a rectangular component, the three datum planes are established by three mutually perpendicular surfaces from which measurements are taken and tolerances applied. (See Figure 13).

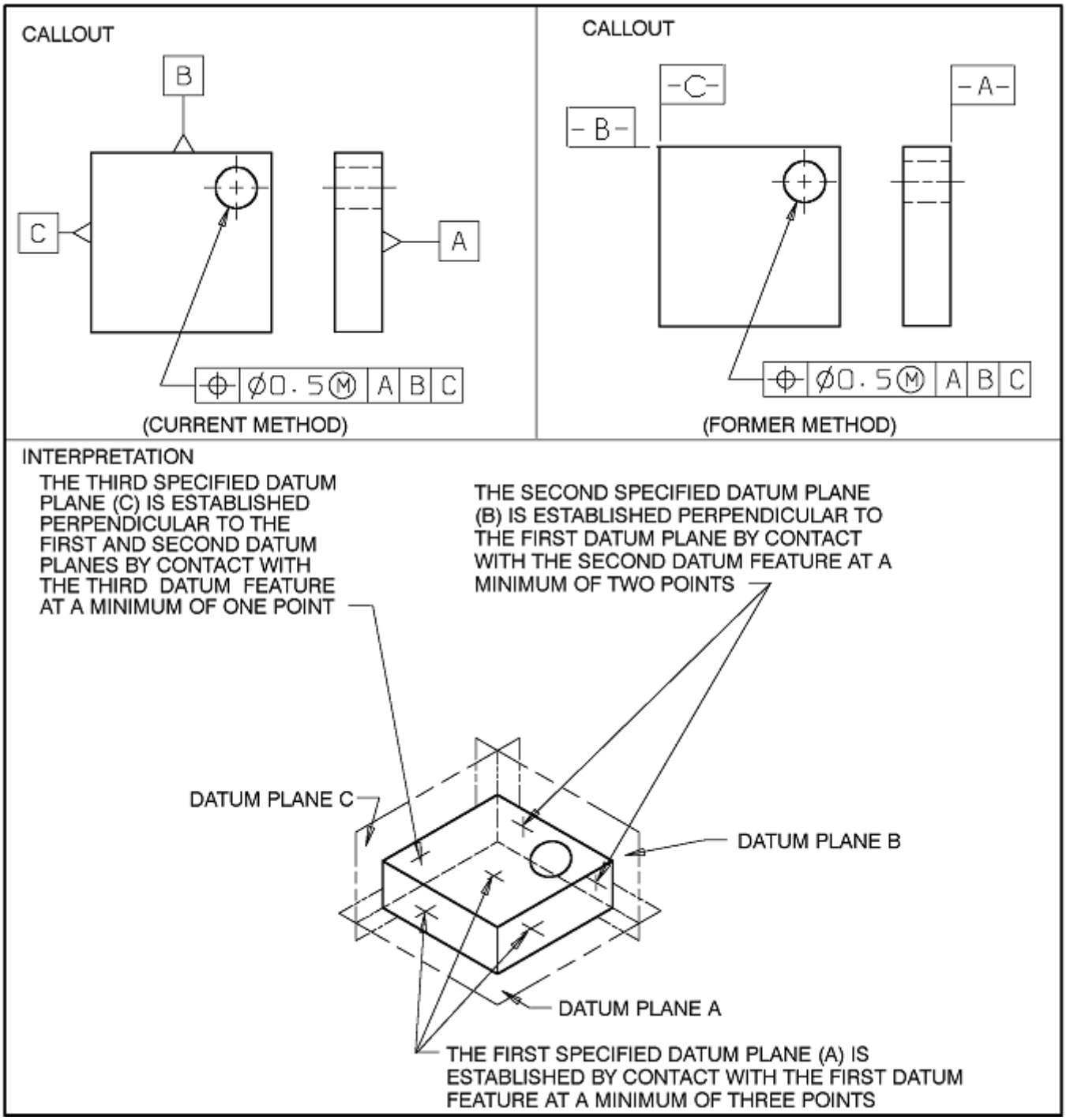


Figure 13 - Datum System for a Rectangular Component

8.4.3 For a datum system involving a circular locating datum feature, the datum axis is formed at the intersection of two planes usually represented by perpendicular center-planes intersecting at the axis of the circular datum feature. Thus, one circular datum feature may be considered as associated with two datum planes. The other datum plane is perpendicular to the planes intersecting the datum axis and is established by either one or three point minimum contact. See Figures 14 through 17.

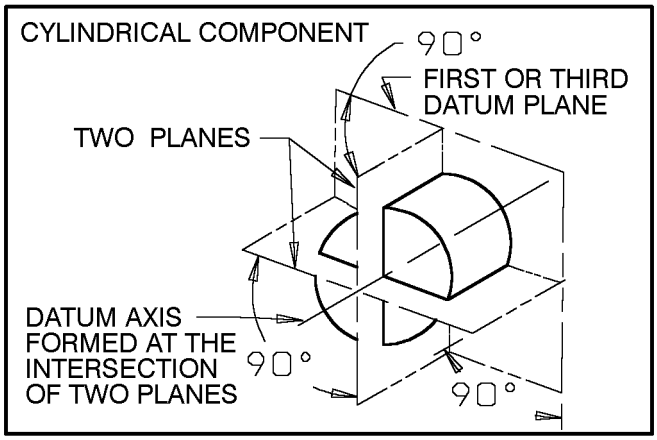


Figure 14 - Datum System For Circular Component

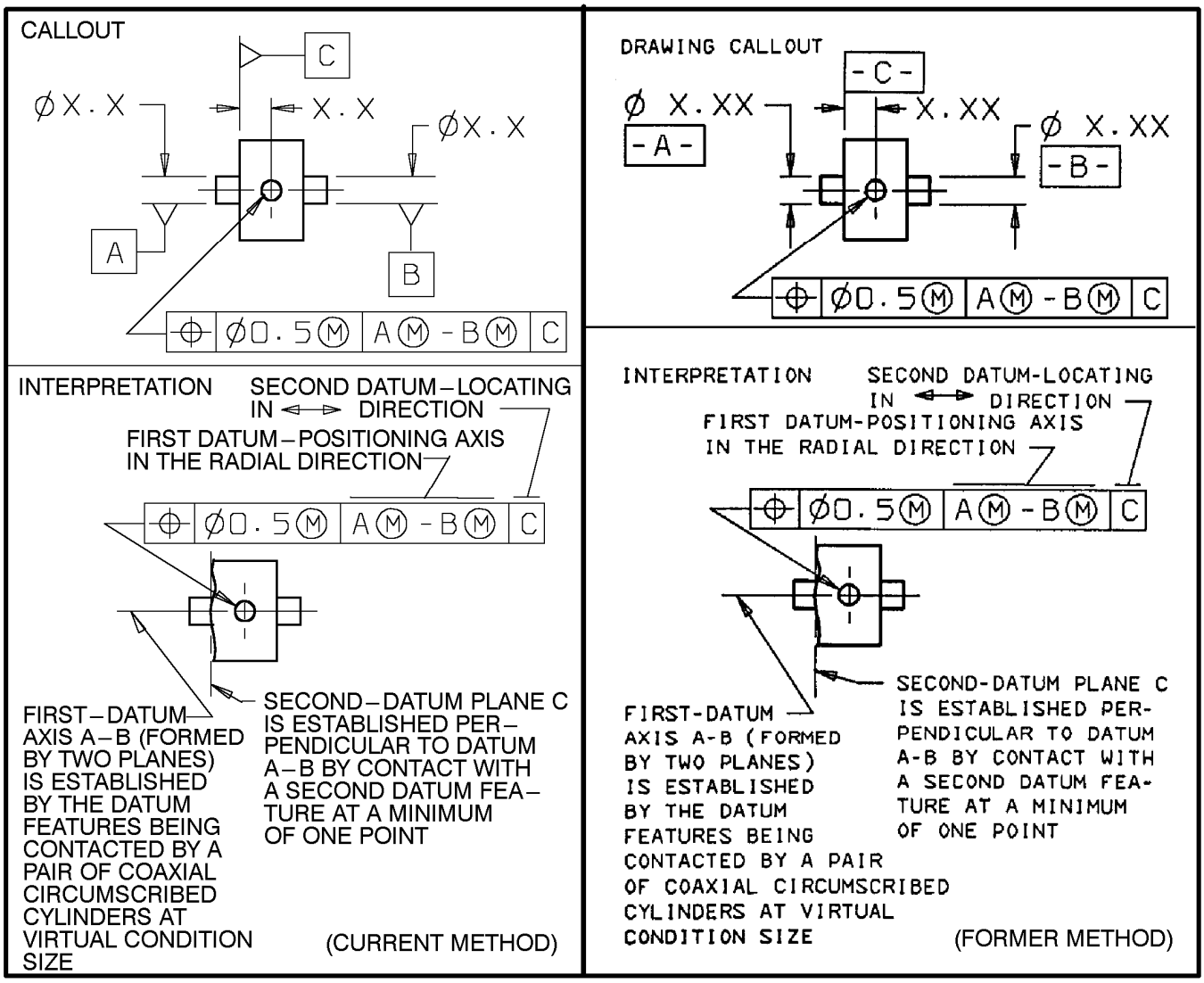


Figure 15 - Datum System (With Axis Established By Two Diameters)

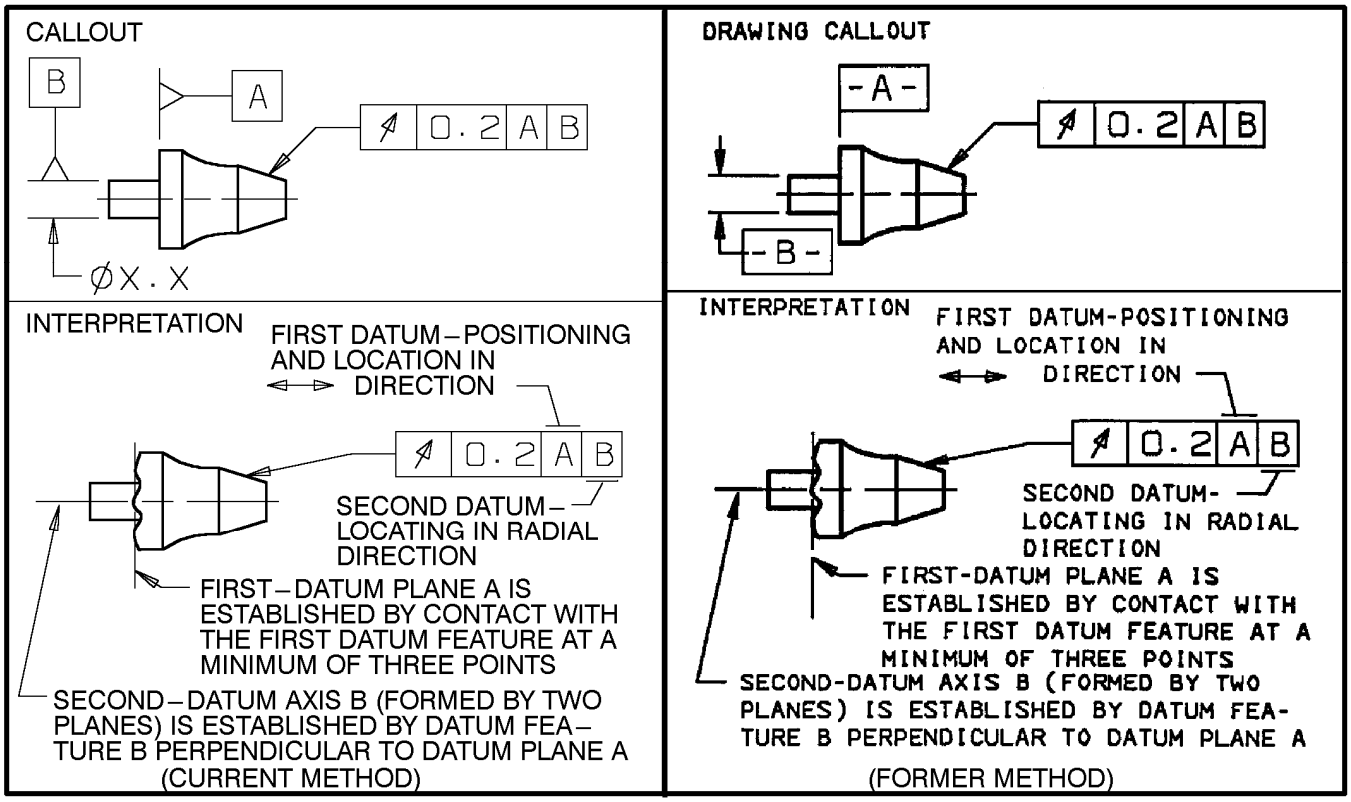


Figure 16 - Datum System (Plane Surface As First Datum)

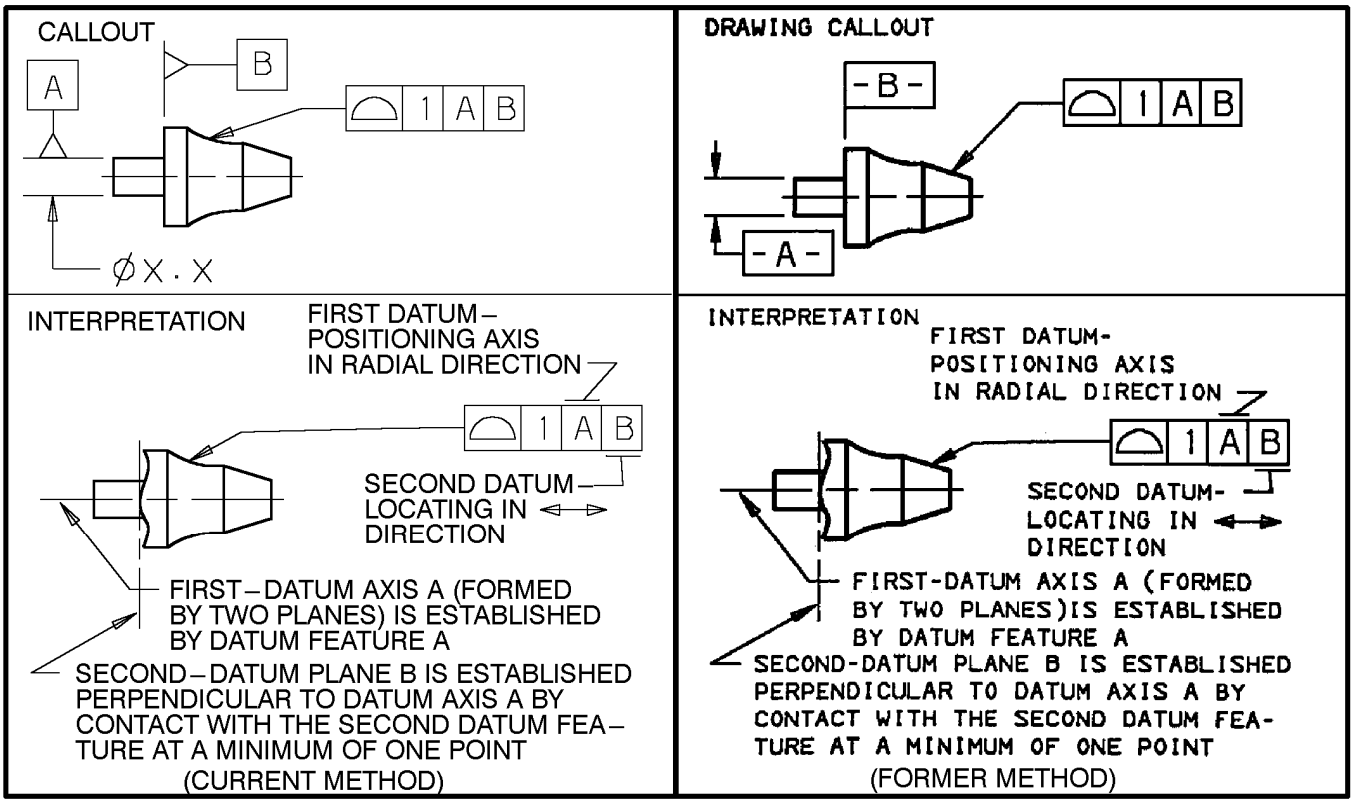


Figure 17 - Datum System (Cylinder As First Datum)

8.4.4 For a datum system involving a conical taper datum feature, the datum axis is formed at the intersection of two planes intersecting at the axis of the conical datum feature and a third datum which is perpendicular to the datum axis is established at the gage diameter. The three datum planes are established simultaneously by the conical taper feature. If a second datum is specified in the feature control frame, it stops rotation about the datum axis and is established by one point contact. (See Figure 18).

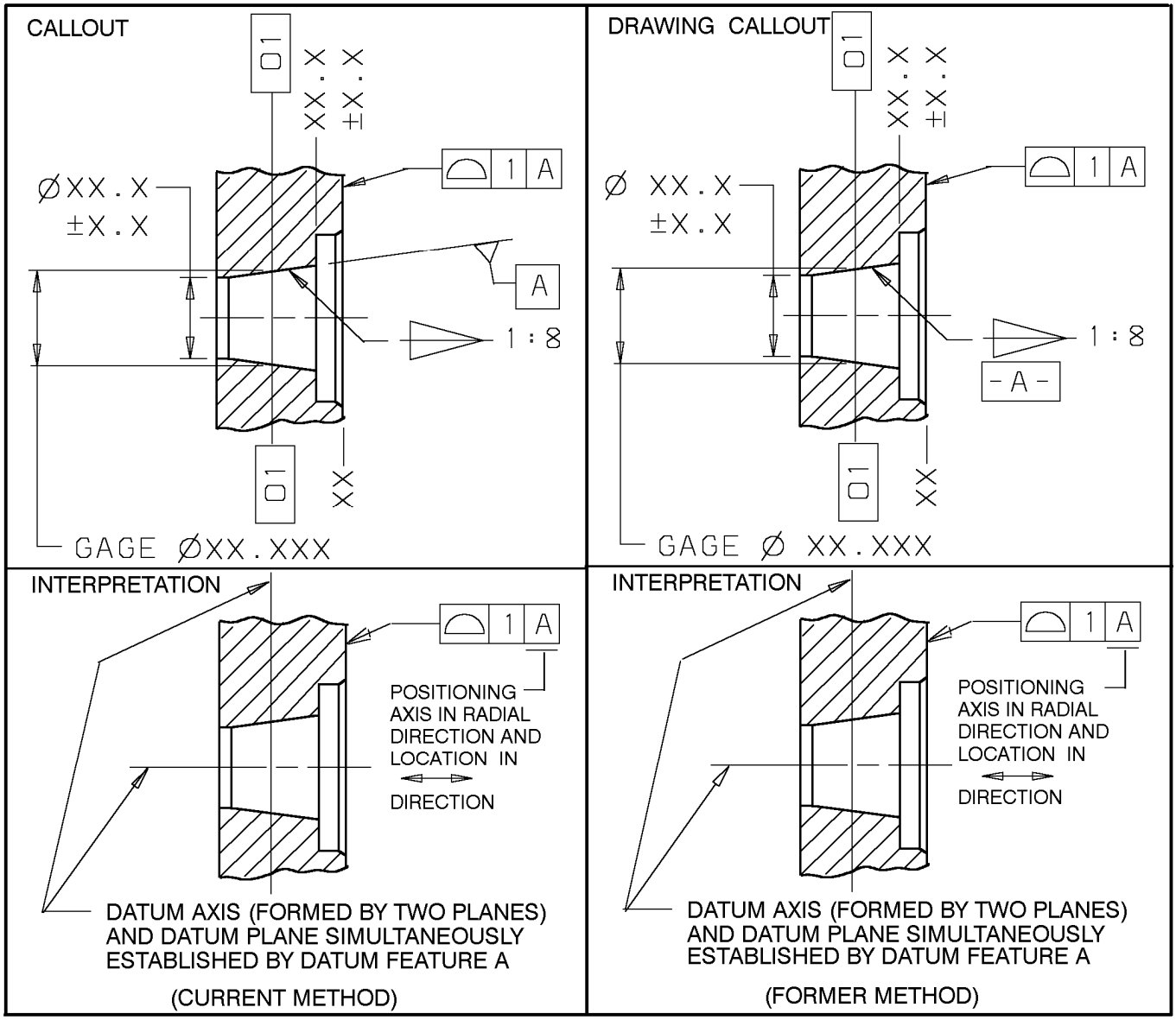


Figure 18 - Datum System For Conical Taper Feature

8.4.5 For a datum system involving an inclined surface datum feature, a true contacting plane is oriented at the basic angle of the inclined datum feature. The corresponding plane at the datum reference frame is rotated through this same basic angle to be mutually perpendicular to the other two planes. (See Figure 19).

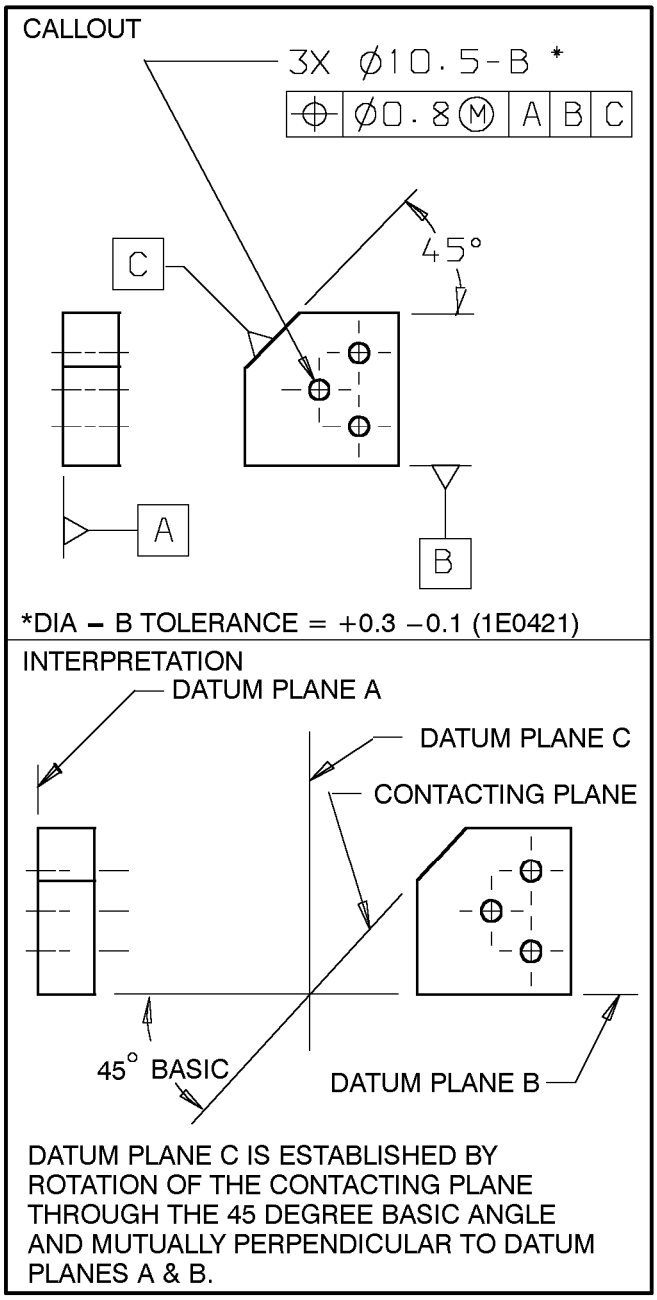


Figure 19 - Datum System For Inclined Datum Feature

8.5 Datum Targets - Datum systems may be established by datum targets (formerly called locators) for components with irregular shapes such as castings, forgings, fabricated assemblies (weldments), and flat metal parts.

8.5.1 Datums established by datum targets may be either functional or non-functional (process). Functional datums are used as a basis for completed component requirements. Non-functional (process) datums are used as a basis for acceptance of the rough component and for initial machining. Non-functional (process) datums are normally designated X, Y, and Z. Datum feature symbols are attached to extension lines that lie in the plane passing through at least one of the datum targets. For equalizing and vee datum targets, the datum feature symbol is attached to the axis or center plane established by the datum targets.

8.5.2 Dimensions from surfaces, on which datum targets have been established, shall be interpreted as originating from the datum targets or a plane established by the datum targets. All dimensions locating the datum targets are basic (untoleranced). Targets shall be 12 to 15 mm diameter unless otherwise specified on the drawing/model.

8.5.3 The letter specified in the lower section of the datum target symbol is determined by the datum plane or axis that the datum targets establish. The number **1** is placed in the lower section of the first symbol for a given datum plane or axis; the number **2** in the second; the number **3** in the third. (See Figures 20 through 23).

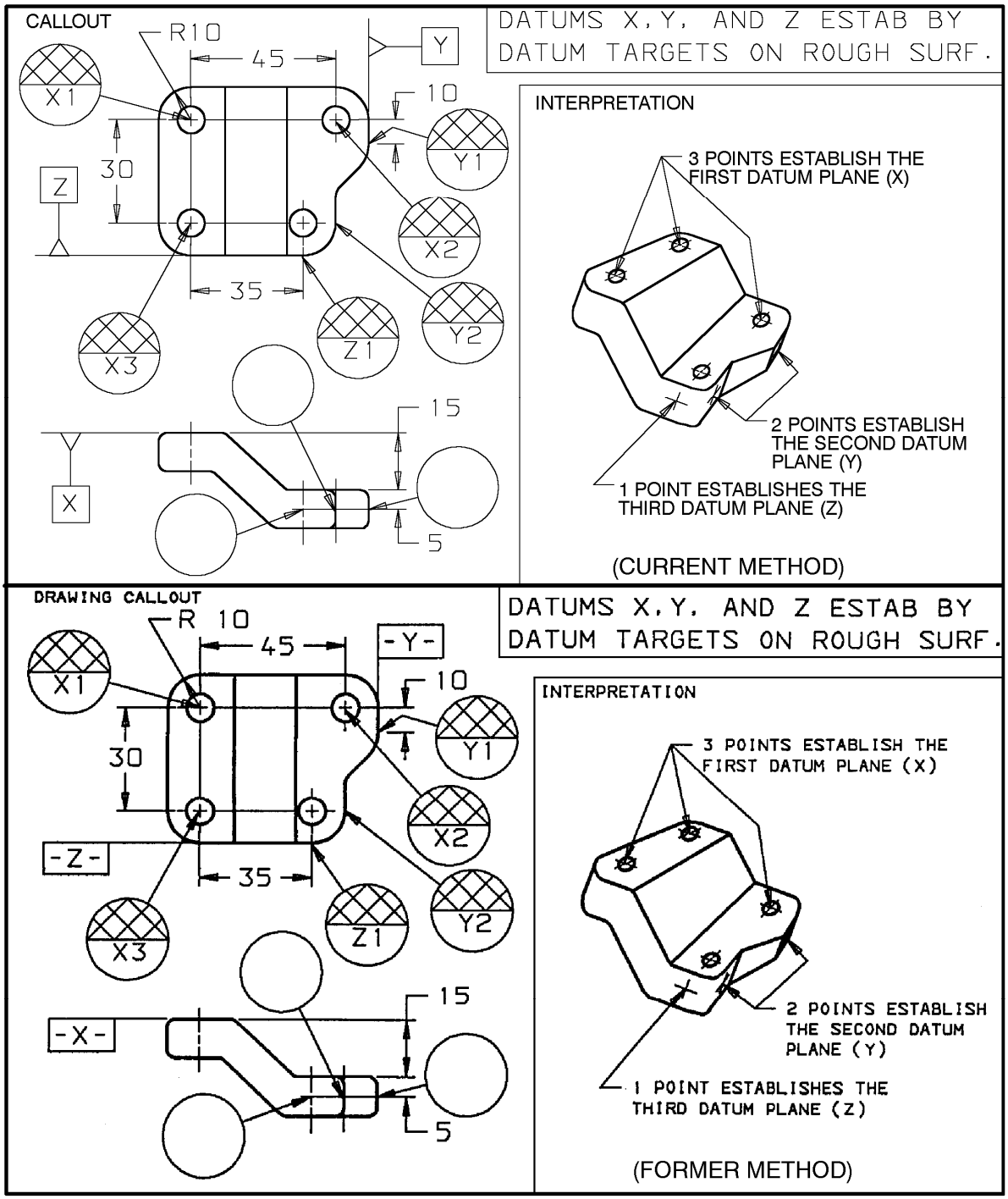


Figure 20 - Datums Established By Datum Targets

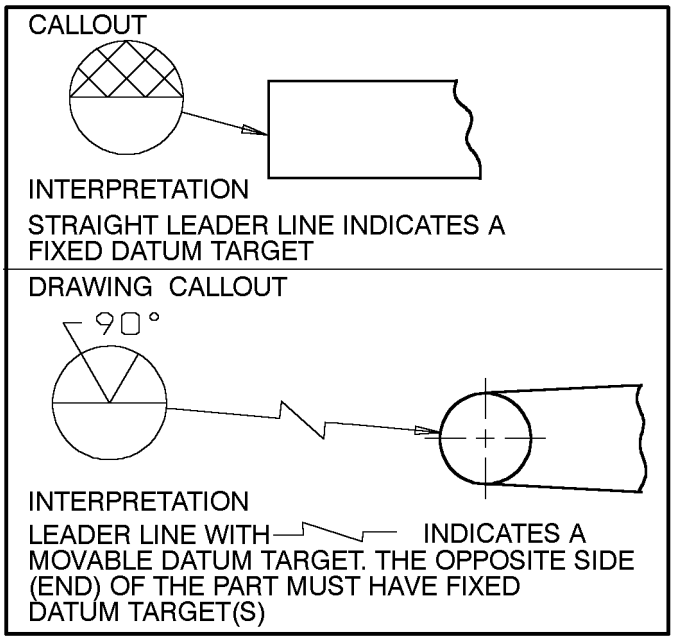


Figure 21 - Leader Lines

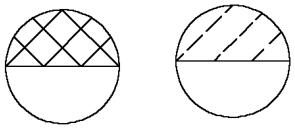
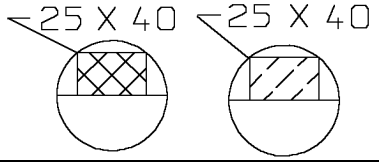
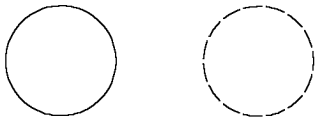
SINGLE CONTACT DATUM TARGET SYMBOLS	
CALLOUT **VISIBLE HIDDEN**	**INTERPRETATION AND INSTRUCTIONS**
CIRCULAR 	**THIS SYMBOL REPRESENTS AN INDIVIDUAL DATUM TARGET. THE LEADER LINE POINTS TO THE CENTER OF THE DATUM TARGET ON THE COMPONENT.**
RECTANGULAR 	**THIS SYMBOL REPRESENTS AN INDIVIDUAL DATUM TARGET OR IT MAY BE USED IN CONJUNCTION WITH A MULTIPLE TARGET SYMBOL TO INDICATE THE SIZE AND SHAPE OF THE DATUM TARGET. THE DIMENSIONS OF THE RECTANGULAR AREA ARE INDICATED ADJACENT TO THE SYMBOL. THE LEADER LINE POINTS TO THE CENTER OF THE DATUM TARGET ON THE COMPONENT.**
SUPPLEMENTAL 	**THIS SYMBOL IS USED AS A SUPPLEMENTAL SYMBOL TO THE MULTIPLE TARGET SYMBOLS AND TO INDIVIDUAL TARGET SYMBOLS TO INDICATE THE SECONDARY DIMENSION OF THE CONTACT AREAS. THIS SYMBOL MAY BE USED IN CONJUNCTION WITH ONE OR MORE MULTIPLE TARGET SYMBOLS ON THE SAME DETAIL DRAWING.**

Figure 22 - Datum Target Symbols (Continued on Next Page)

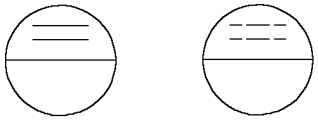
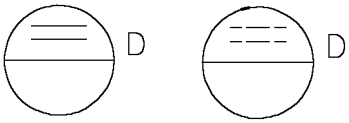
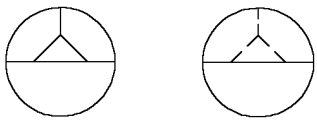
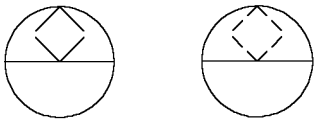
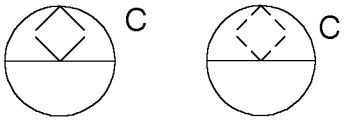
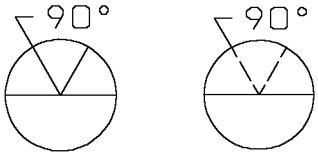
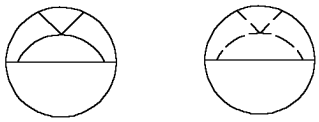
MULTIPLE CONTACT DATUM TARGET SYMBOLS	
CALLOUT **VISIBLE HIDDEN**	**INTERPRETATION AND INSTRUCTIONS**
EQUALIZER - 2 WAY 	**INDICATES EQUALIZING OR CENTERING IN ONE PLANE, WHICH NORMALLY CONSISTS OF TWO POINT CONTACT 180 DEGREES APART. EQUALIZING DIRECTION IN BORES, PORTS, ETC., IS INDICATED BY ONE LEADER LINE POINTING TO THE APPROPRIATE CENTERLINE. EQUALIZING BETWEEN TWO FLAT SURFACES IS INDICATED WITH TWO LEADER LINES POINTING TO THE TWO CONTACT AREAS.**
EQUALIZER (SEPARATED CONTACT POINTS) 	**IF IT IS NOT PRACTICAL TO SHOW TWO LEADER LINES FROM THE SYMBOL TO THE CONTACT POINTS, TWO SYMBOLS EACH WITH A LEADER LINE POINTING TO THE CONTACT SURFACE MAY BE USED. THE TWO SYMBOLS ARE IDENTIFIED BY THE SAME LETTER TO REPRESENT THE TWO TARGETS THAT ARE USED TOGETHER AS ONE EQUALIZER.**
EQUALIZER - 3 WAY 	**INDICATES EQUALIZING OR CENTERING IN TWO PLANES, WHICH NORMALLY CONSISTS OF THREE POINT CONTACT 120 DEGREES APART WHICH MOVE SIMULTANEOUSLY AND EQUALLY TO LOCATE THE CENTER OF A FEATURE. OTHER ANGULAR RELATIONS MAY BE USED BY SPECIFYING THE ANGLE. ONE LEADER LINE POINTS TO THE APPROPRIATE SURFACE INDICATING THE LOCATION OF THE ONE CONTACT AREA WHICH IS DIMENSIONED.**
CENTERING VEES 	**FOUR POINT CONTACT 90 DEGREES APART UNLESS OTHERWISE SPECIFIED. THIS SYMBOL REPRESENTS TWO VEES THAT MOVE SIMULTANEOUSLY AND EQUALLY TO LOCATE THE CENTER OF A FEATURE.**
CENTERING VEES (SEPARATED CONTACT POINTS) 	**THIS SYMBOL REPRESENTS TWO VEES THAT MOVE SIMULTANEOUSLY AND EQUALLY TO ESTABLISH CENTER PLANES FROM TWO FEATURES. EACH VEE HAS TWO CONTACT POINTS 90 DEGREES APART AND EACH VEE CONTACTS A SEPARATE FEATURE. TWO SYMBOLS ARE REQUIRED AND ARE IDENTIFIED BY THE SAME LETTER TO REPRESENT THE TWO VEES THAT SHALL BE USED TOGETHER.**
VEE 	**ONE LEADER LINE IS USED TO INDICATE THE POSITION OF THE VEE ON A DIAMETER BY POINTING TO THE CENTERLINE ON WHICH THE VERTEX OF THE INCLUDED ANGLE OF THE VEE WOULD INTERSECT THE PART. OCCASIONALLY, VEES ARE USED TO EQUALIZE RECTANGULAR SHAPES. THE INCLUDED ANGLE IS DESIGNATED ADJACENT TO THE SYMBOL. IF THE WIDTH OF THE VEE IS A CRITICAL DIMENSION, IT IS INDICATED BY A DIMENSION ADJACENT TO THE VEE SYMBOL.**
ROCKER 	**LEADERS TO THE TWO CONTACT POINTS INDICATE THE TWO POINTS OF CONTACT.**

Figure 22 - Datum Target Symbols (Continued)

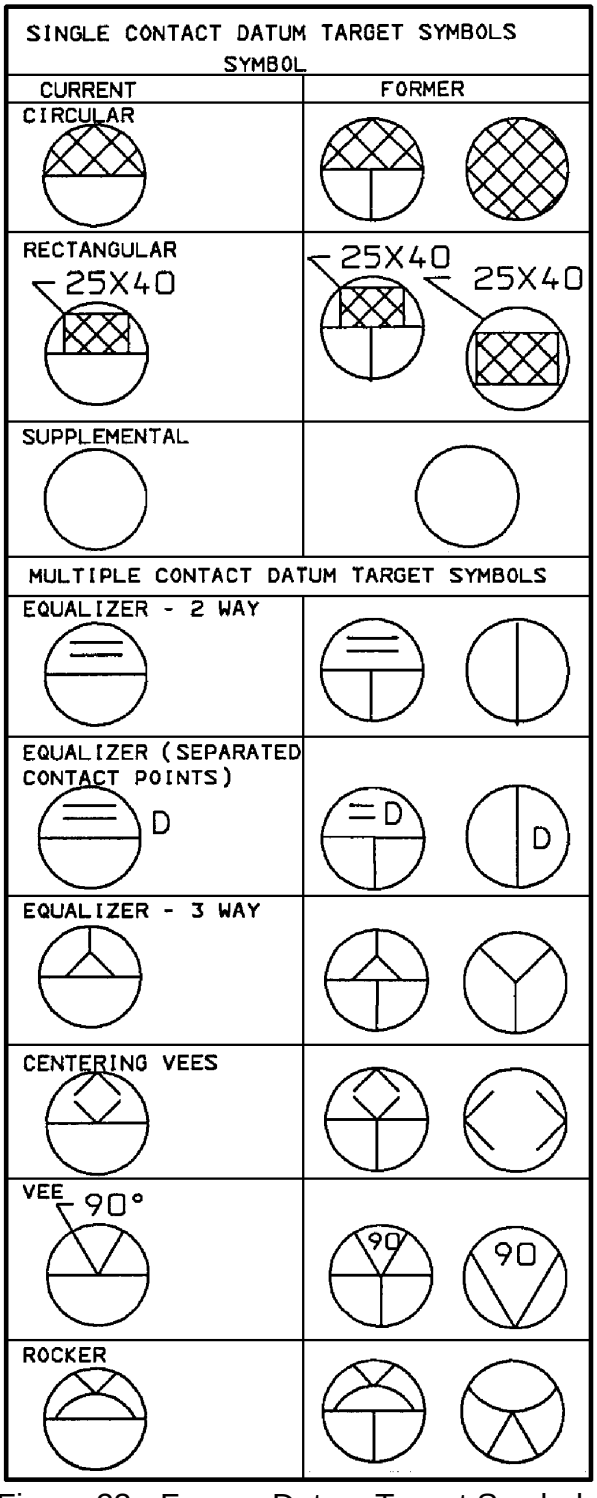


Figure 23 - Former Datum Target Symbols

9.0 STRAIGHTNESS

9.1 Definition - Straightness is a condition where an element of a surface or an axis is a straight line.

9.1.1 Tolerance - Straightness tolerance specifies a tolerance zone within which the considered element or derived median line shall lie as follows:

9.1.1.1 When applied to line elements of a flat surface, the line element of the surface shall lie between two parallel straight lines separated by the amount of the specified straightness tolerance. The straightness tolerance shall be less than the size tolerance and the surface shall be within the boundary of perfect form at MMC size. (See Figure 24).

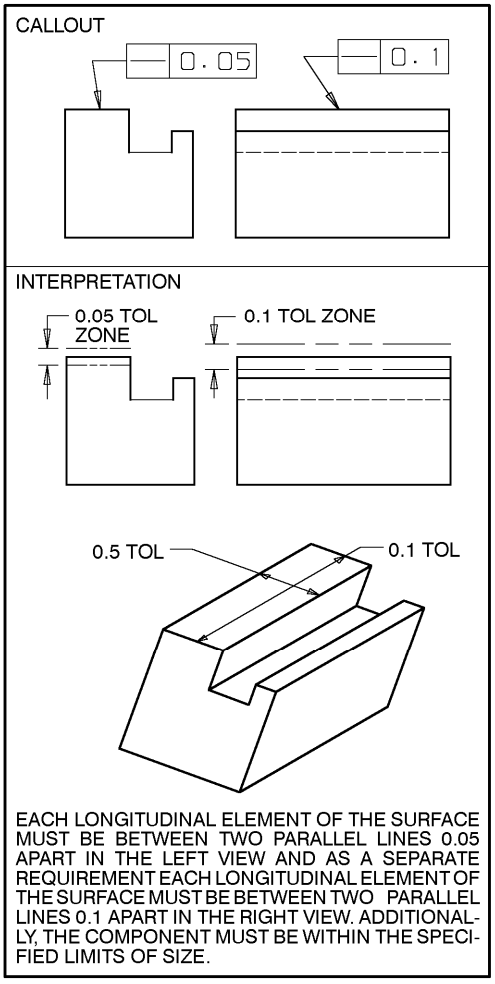


Figure 24 - Straightness - Flat Surface

9.1.1.2 When applied to the longitudinal elements of the surface of a cylindrical feature, each longitudinal element of the surface shall lie between two parallel lines separated by the amount of the specified straightness tolerance and in a plane common with the normal axis of the feature. The straightness tolerance shall be less than the size tolerance and the surface shall be within the boundary of perfect form at MMC size. (See Figure 25).

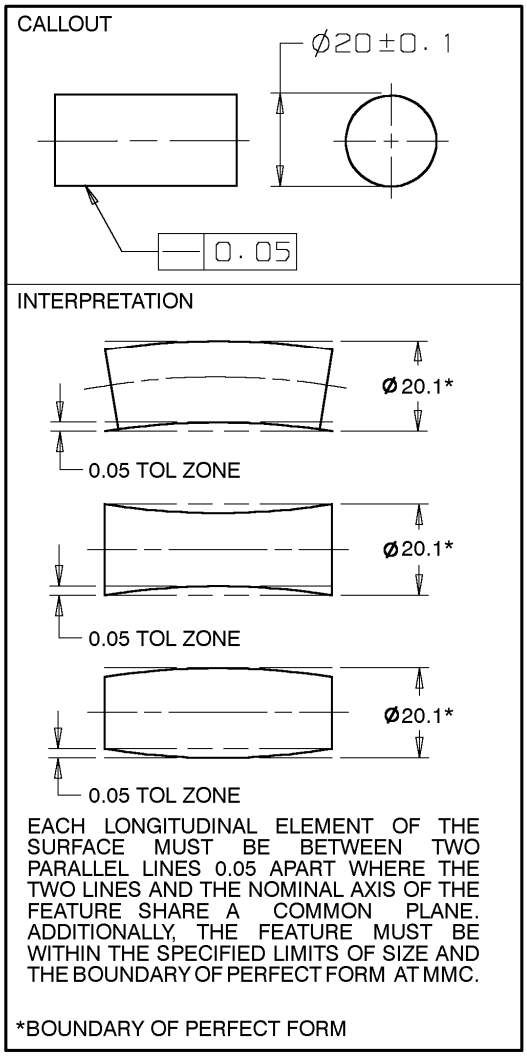


Figure 25 - Straightness - Longitudinal Surface Elements Of A Cylinder

9.1.1.3 When applied to the size dimension of a cylindrical feature, the derived median line of the cylindrical feature shall lie within a cylindrical tolerance zone whose diameter is:

The specified straightness tolerance value when applied on an RFS basis. (See Figure 26).

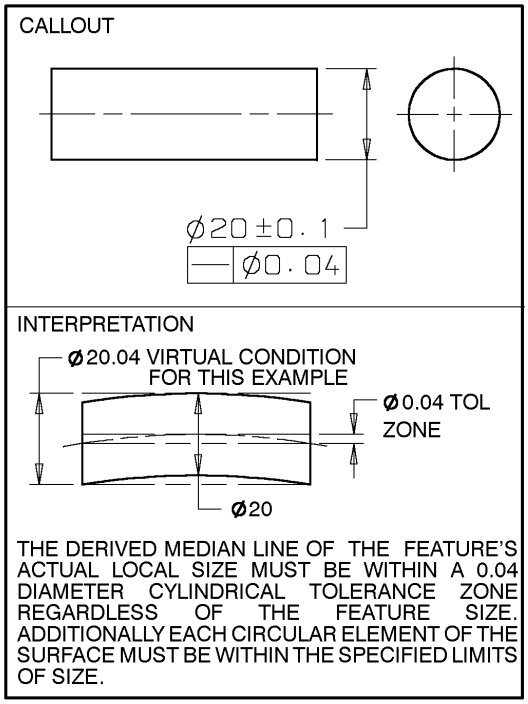
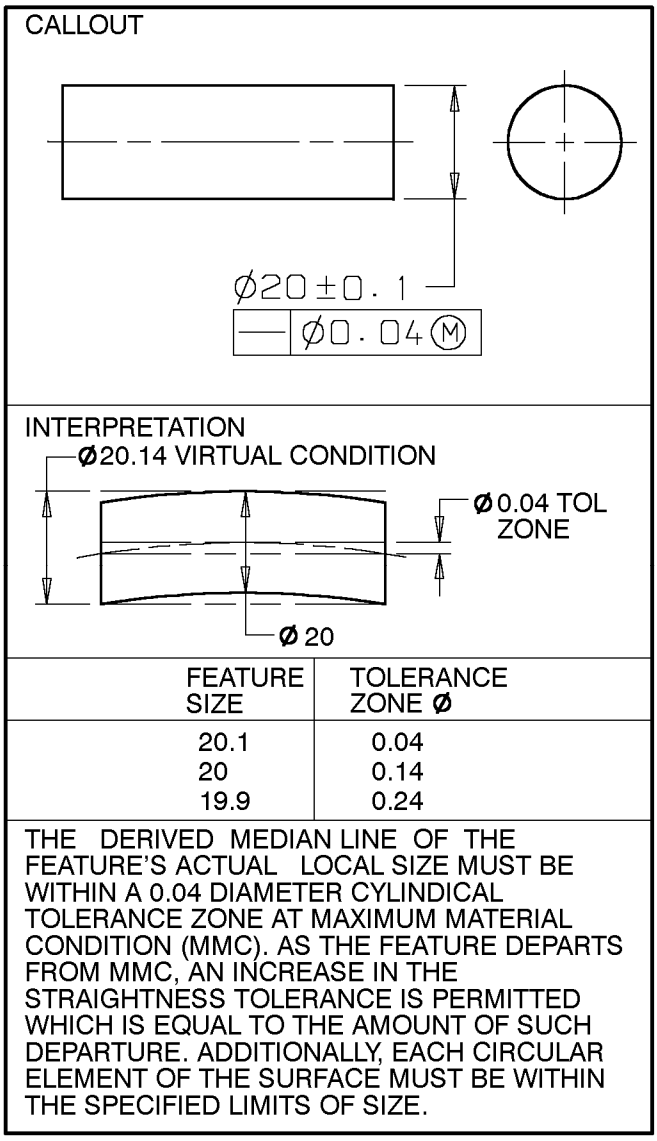


Figure 26 - Straightness RFS

The specified straightness tolerance value plus the amount the controlled feature departs from its MMC size when applied on an MMC basis. (See Figure 27).

When the feature control frame is applied to the size dimension, the boundary of perfect form at MMC may be violated and, where required, the straightness tolerance may be greater than the size tolerance. The collective effect of size and form variation can produce a virtual condition equal to the MMC size plus the straightness tolerance for an external feature and the MMC size minus the straightness tolerance for an internal feature. The tolerance value is preceded by the diameter symbol in the feature control frame.



FEATURE SIZE	TOLERANCE ZONE Ø
20.1	0.04
20	0.14
19.9	0.24

THE DERIVED MEDIAN LINE OF THE FEATURE'S ACTUAL LOCAL SIZE MUST BE WITHIN A 0.04 DIAMETER CYLINDICAL TOLERANCE ZONE AT MAXIMUM MATERIAL CONDITION (MMC). AS THE FEATURE DEPARTS FROM MMC, AN INCREASE IN THE STRAIGHTNESS TOLERANCE IS PERMITTED WHICH IS EQUAL TO THE AMOUNT OF SUCH DEPARTURE. ADDITIONALLY, EACH CIRCULAR ELEMENT OF THE SURFACE MUST BE WITHIN THE SPECIFIED LIMITS OF SIZE.

Figure 27 - Straightness At MMC

9.1.1.4 When applied to the size dimension of a non cylindrical feature, the derived median plane shall lie between two parallel planes separated by:

The amount of the specified straightness tolerance when applied on an RFS basis.

The amount of the specified straightness tolerance plus the amount the controlled feature departs from its MMC size when applied on an MMC basis. (See Figure 28).

When the feature control frame is applied to the size dimension, the boundary of perfect form at MMC may be violated and, where required, the straightness tolerance may be greater than the size tolerance. The collective effect of size and form variation can produce a virtual condition equal to the MMC size plus the straightness tolerance for an external feature or the MMC size minus the straightness tolerance for an internal feature.

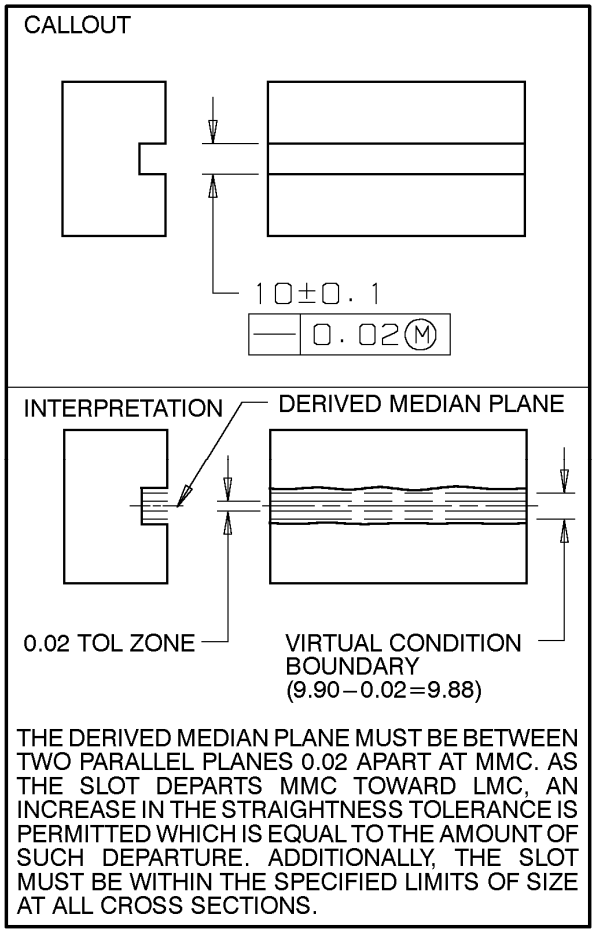


Figure 28 - Straightness At MMC

9.1.2 Straightness may be restricted by a second straightness requirement that is applied to a specified increment of length to prevent an abrupt surface variation within a relatively short length of the feature. The incremental straightness requirement is specified as part of a composite or multiple single segment feature control frame with the total feature straightness requirement as shown in Figure 29.

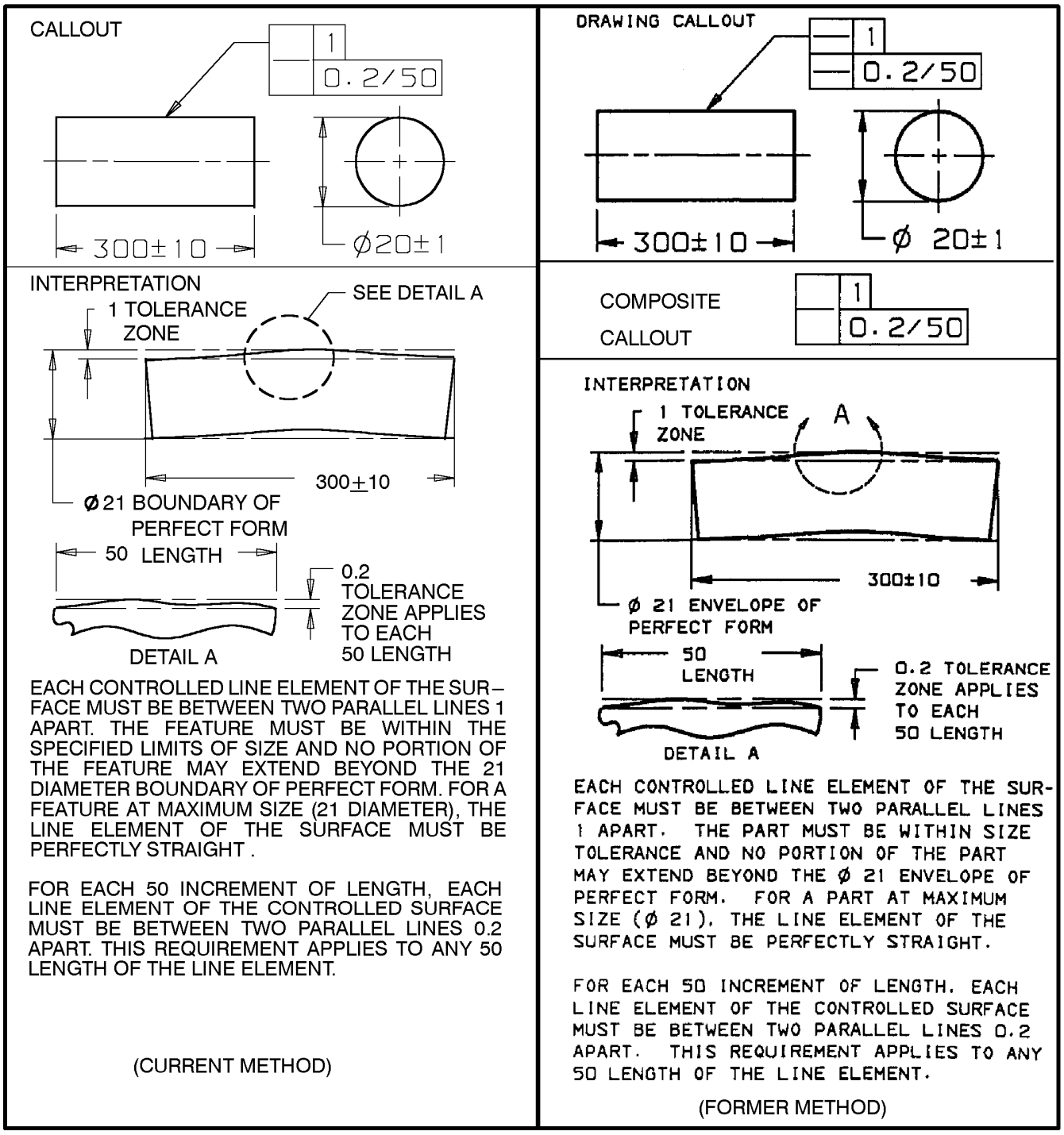


Figure 29 - Incremental Straightness

9.1.3 Former Practice - A feature controlled by a straightness tolerance may extend beyond the boundary of perfect form if one of the following notes is specified in conjunction with the size dimension. (See Figure 30).

PERFECT STRAIGHTNESS NOT REQUIRED AT MAX SIZE, for an external feature.

PERFECT STRAIGHTNESS NOT REQUIRED AT MIN SIZE, for an internal feature.

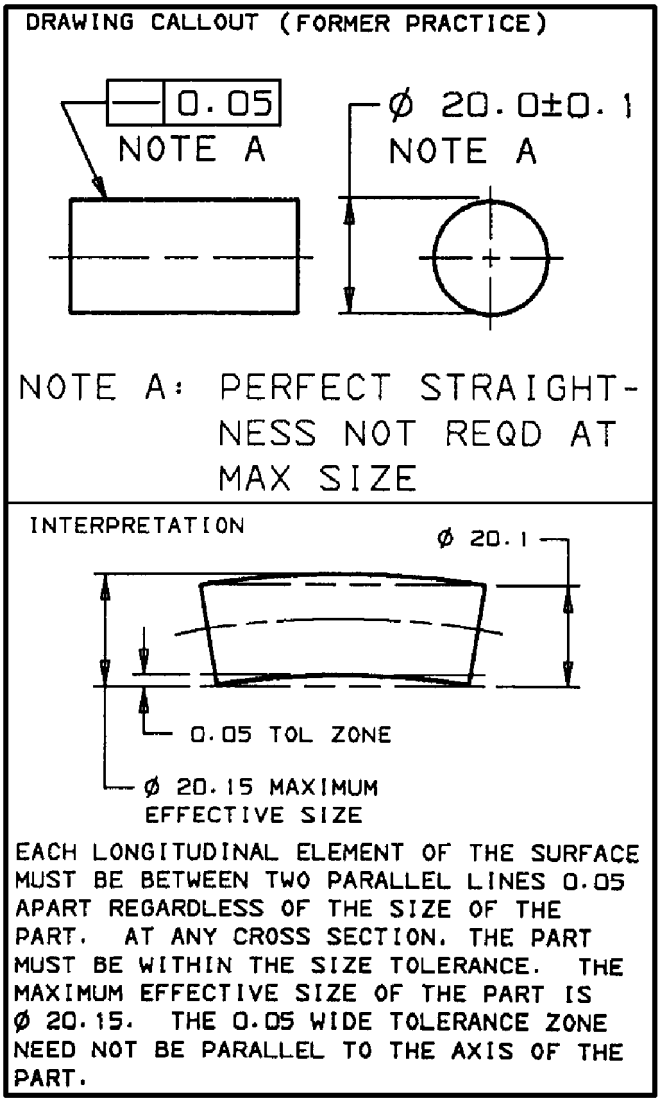


Figure 30 - Straightness (Former Practice)

9.2 Flatness

9.2.1 Definition - Flatness is the condition of a surface having all elements in one plane.

9.2.2 Tolerance - Flatness tolerance is the distance between two parallel planes within which the controlled surface shall lie. (See Figure 31).

9.2.2.1 Unless the size requirement is modified by note, (see Paragraph 9.2.2.2), the specified flatness tolerance shall be within the specified limits of size and the boundary of perfect form created by:

The largest limit of size of an external feature, or

The smallest limit of size of an internal feature.

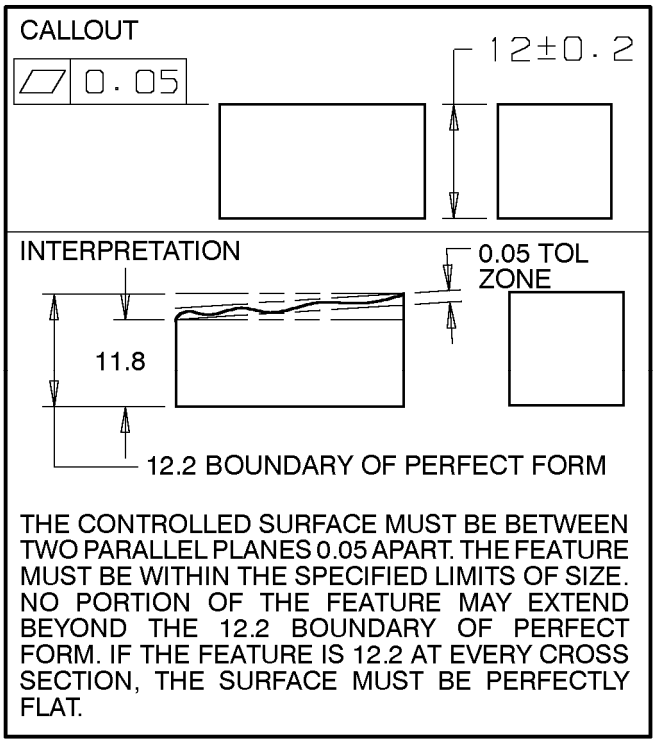


Figure 31 - Flatness

9.2.2.2 A surface controlled by a flatness tolerance may extend beyond the boundary of perfect form if one of the following notes is specified in conjunction with the size dimension and the feature control frame:

PERFECT FLATNESS NOT REQUIRED AT MAX SIZE, for an external feature, or

PERFECT FLATNESS NOT REQUIRED AT MIN SIZE, for an internal feature.

See Figure 32.

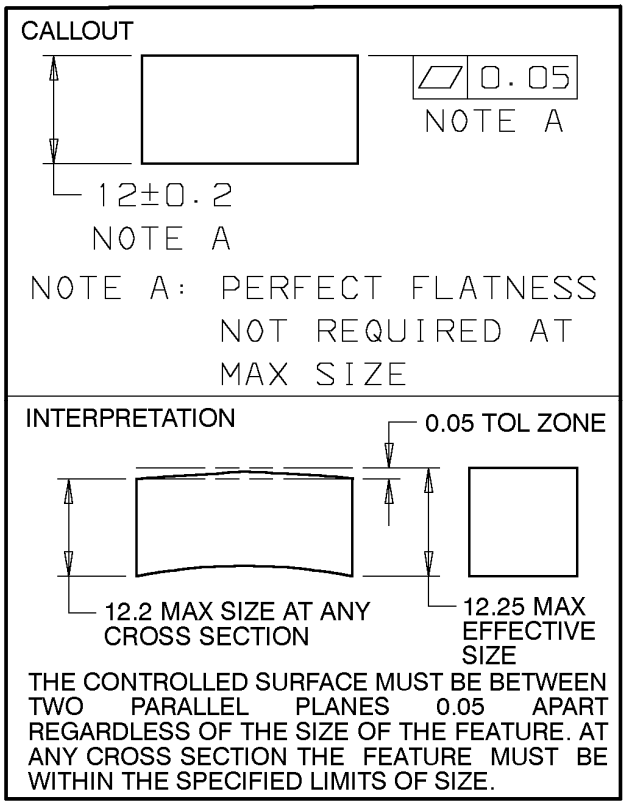


Figure 32 - Flatness

9.2.2.3 Flatness may be further restricted by one of the following notes below the symbol:

MUST NOT BE CONCAVE, or

MUST NOT BE CONVEX

See Figure 33.

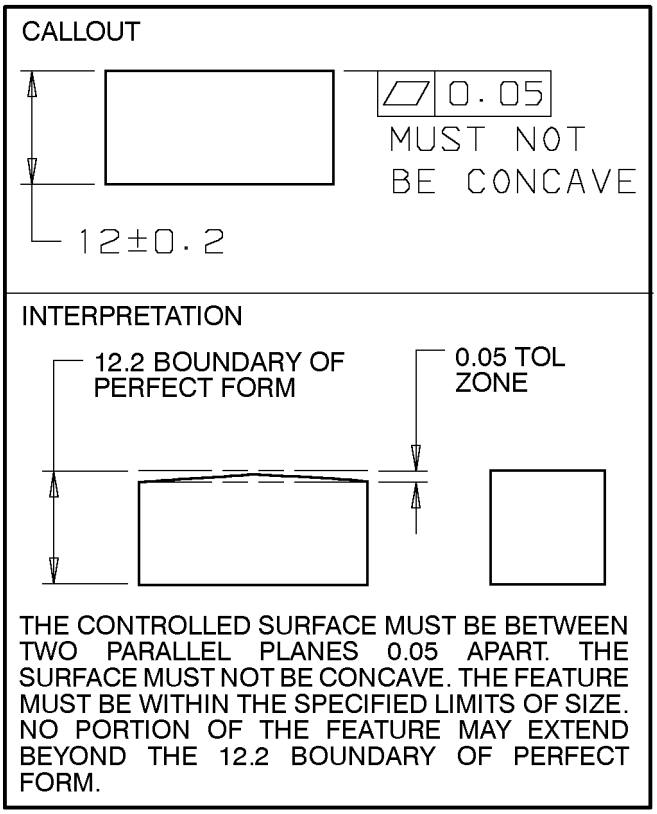


Figure 33 - Flatness

9.2.3 Flatness tolerances specified in mill tolerance specifications such as 1E2177 shall be interpreted in accordance with Figure 32.

9.2.3.1 Flatness may also be restricted by a second flatness requirement that applies to a specified increment of length (or diameter) by specifying **xx/xxlg** or **(xx/Øxx)** in a composite or multiple single segment feature control frame. (See Figure 34).

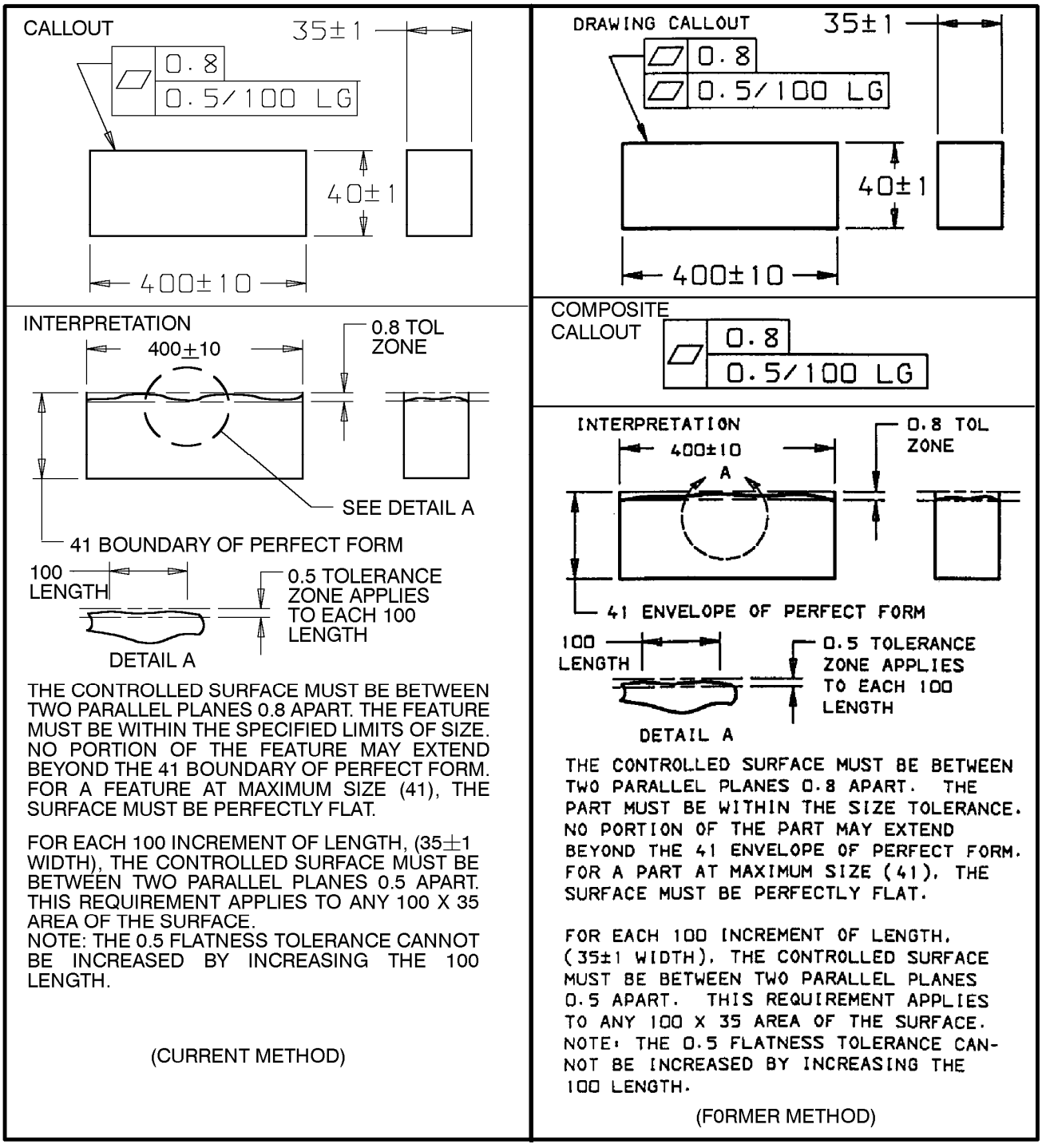


Figure 34 - Incremental Flatness

9.3 Circularity (Roundness)

9.3.1 Definitions - Circularity is a condition of a feature where:

9.3.1.1 For a cylinder or cone, all points of the surface intersected by any plane perpendicular to a common axis are equidistant from that axis.

9.3.1.2 For a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.

9.3.1.3 Average diameter **(ØXX AVG)** is the average of the maximum and minimum diameters at any intersecting plane perpendicular to the common axis of a cylinder or cone, or passing through the center of a sphere.

9.3.1.4 Maximum diameter **(ØXX MAX)** is the maximum diameter at any intersecting plane perpendicular to the axis of a cylinder or cone, or passing through the center of a sphere. Minimum diameter **(ØXX MIN)** is the minimum diameter at any intersecting plane perpendicular to the axis of a cylinder or cone, or passing through the center of a sphere.

9.3.2 Tolerance - Circularity tolerance is the radial distance between two concentric circles in the same plane. Each circular element of the controlled surface shall be within the zone.

9.3.3 Inspection - Circularity, size, and location tolerances of the feature are checked independently and each shall conform.

9.3.4 Control - Methods of circularity control.

9.3.4.1 For circularity control as a restrictive tolerance within the specified limits of size. (See Figure 35).

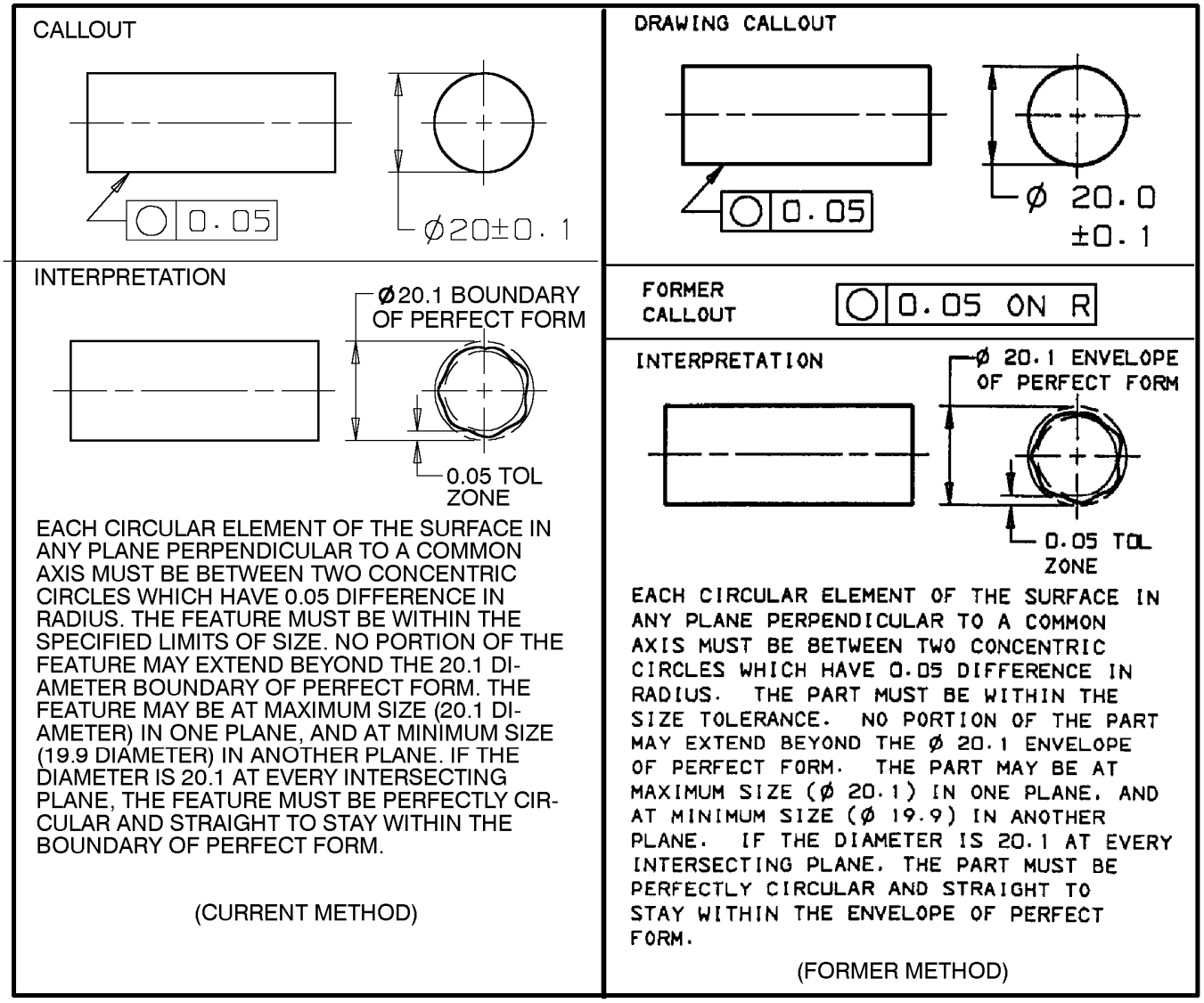


Figure 35 - Circularity Control As A Restrictive Tolerance

9.3.4.2 Circularity may be controlled by one of two methods that use average diameter.

For circularity control and average diameter, see Figure 36. This method provides a fixed circularity tolerance value using the free state modifier symbol.

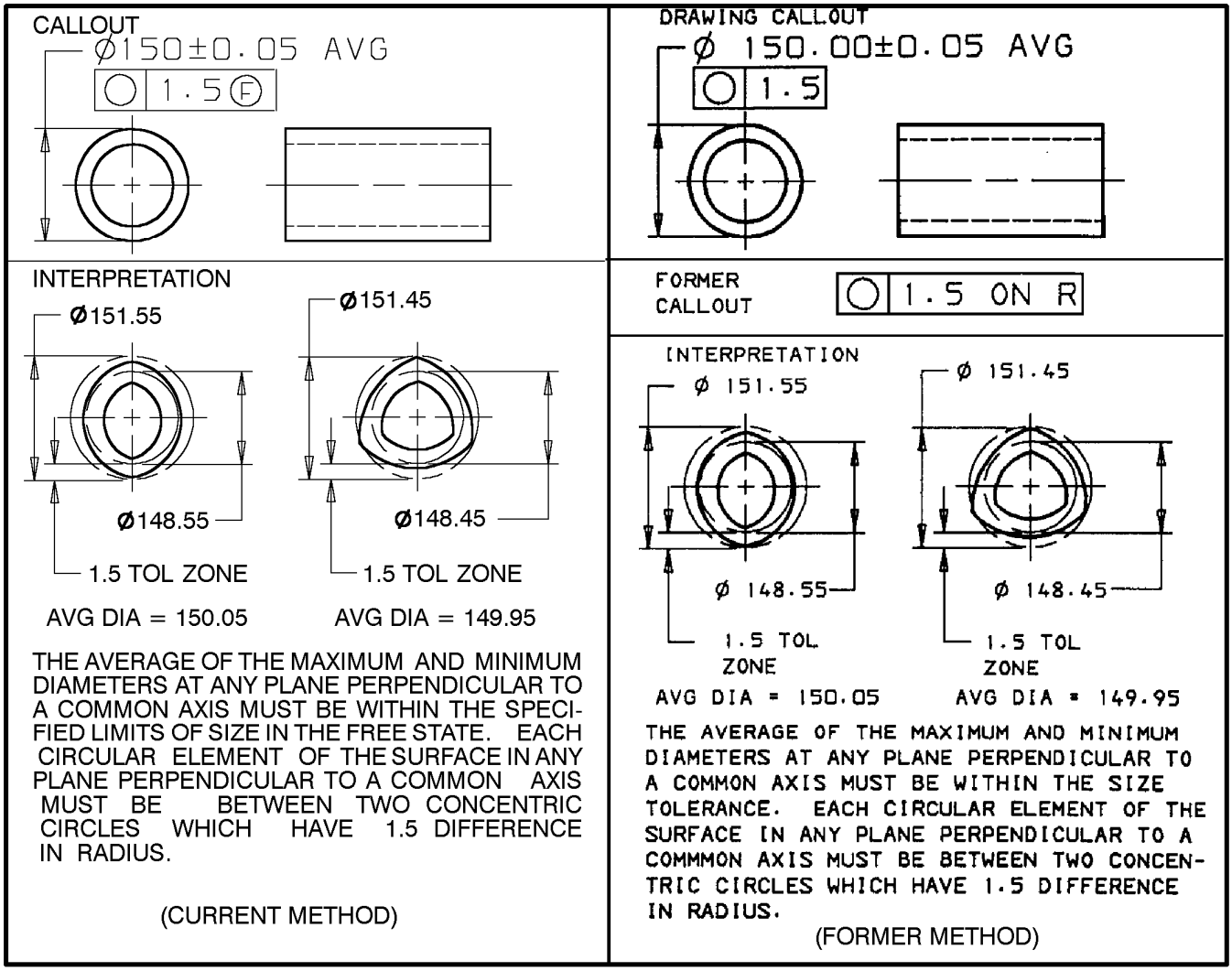


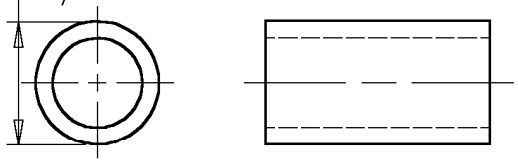
Figure 36 - Circularity Control Average Diameter

For maximum (or minimum) diameter and average diameter, see Figure 37. This method provides a variable circularity tolerance value dependent on the relationship between the maximum and minimum diameters.

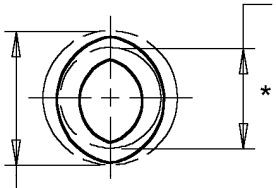
CALLOUT

Ø51±0.05 AVG

Ø51.16 MAX



INTERPRETATION



EXAMPLES		
MAX DIA	51.16	51.16
* MIN DIA	50.74	50.94
SUM	101.90	102.10
AVG DIA	50.95	51.05

THE AVERAGE OF THE MAXIMUM AND MINIMUM DIAMETERS AT ANY PLANE PERPENDICULAR TO A COMMON AXIS MUST BE WITHIN THE SPECIFIED LIMITS OF SIZE. THE MAXIMUM DIAMETER DEFINES THE BOUNDARY OF PERFECT FORM. NO PORTION OF THE FEATURE MAY EXTEND BEYOND THE 51.16 MAXIMUM DIAMETER. NOTE THAT THE CIRCULARITY TOLERANCE VARIES FROM 0.22 TO 0.42 ON THE DIAMETER AS THE AVERAGE DIAMETER VARIES FROM THE HIGH LIMIT TO THE LOW LIMIT.

*THE MINIMUM DIAMETER PERMISSIBLE IS CALCULATED AS FOLLOWS:

MINIMUM DIAMETER FOR AVERAGE DIAMETER AT LOW LIMIT

MIN Ø = MAX Ø – 2(MAX Ø – LOW LIMIT AVG Ø)

MINIMUM DIAMETER FOR AVERAGE DIAMETER AT HIGH LIMIT

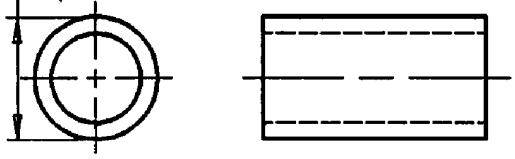
MIN Ø = MAX Ø – 2(MAX Ø – HIGH LIMIT AVG Ø)

(CURRENT METHOD)

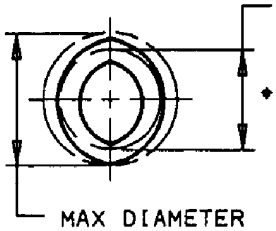
DRAWING CALLOUT

Ø 51.00±0.05 AVG

Ø 51.16 MAX



INTERPRETATION



EXAMPLES		
MAX DIA	51.16	51.16
* MIN DIA	50.74	50.94
SUM	101.90	102.10
AVG DIA	50.95	51.05

THE AVERAGE OF THE MAXIMUM AND MINIMUM DIAMETERS AT ANY PLANE PERPENDICULAR TO A COMMON AXIS MUST BE WITHIN THE SPECIFIED SIZE TOLERANCE. THE MAXIMUM DIAMETER DEFINES THE ENVELOPE OF PERFECT FORM. NO PORTION OF THE FEATURE MAY EXTEND BEYOND THE 51.16 MAXIMUM DIAMETER. NOTE THAT THE CIRCULARITY TOLERANCE VARIES FROM 0.22 TO 0.42 ON THE DIAMETER AS THE AVERAGE DIAMETER VARIES FROM THE HIGH LIMIT TO THE LOW LIMIT.

*THE MINIMUM DIAMETER PERMISSIBLE IS CALCULATED AS FOLLOWS:

MINIMUM DIAMETER FOR AVERAGE DIAMETER AT LOW LIMIT

MIN Ø = MAX Ø-2(MAX Ø-LOW LIMIT AVG Ø)

MINIMUM DIAMETER FOR AVERAGE DIAMETER AT HIGH LIMIT

MIN Ø = MAX Ø-2(MAX Ø-HIGH LIMIT AVG Ø)

(FORMER METHOD)

Figure 37 - Maximum Diameter And Average Diameter

9.4 Cylindricity

9.4.1 Definition - Cylindricity is a condition of a surface of revolution in which all points of the surface are equidistant from a common axis.

9.4.2 Tolerance - Cylindricity tolerance is the radial distance between two concentric cylinders within which the controlled surface shall lie. The tolerance applies simultaneously to both circular and longitudinal elements (the entire surface). See Figure 38.

9.4.3 Inspection - Cylindricity, size, and location (when specified) tolerances of the feature are checked independently and each shall conform.

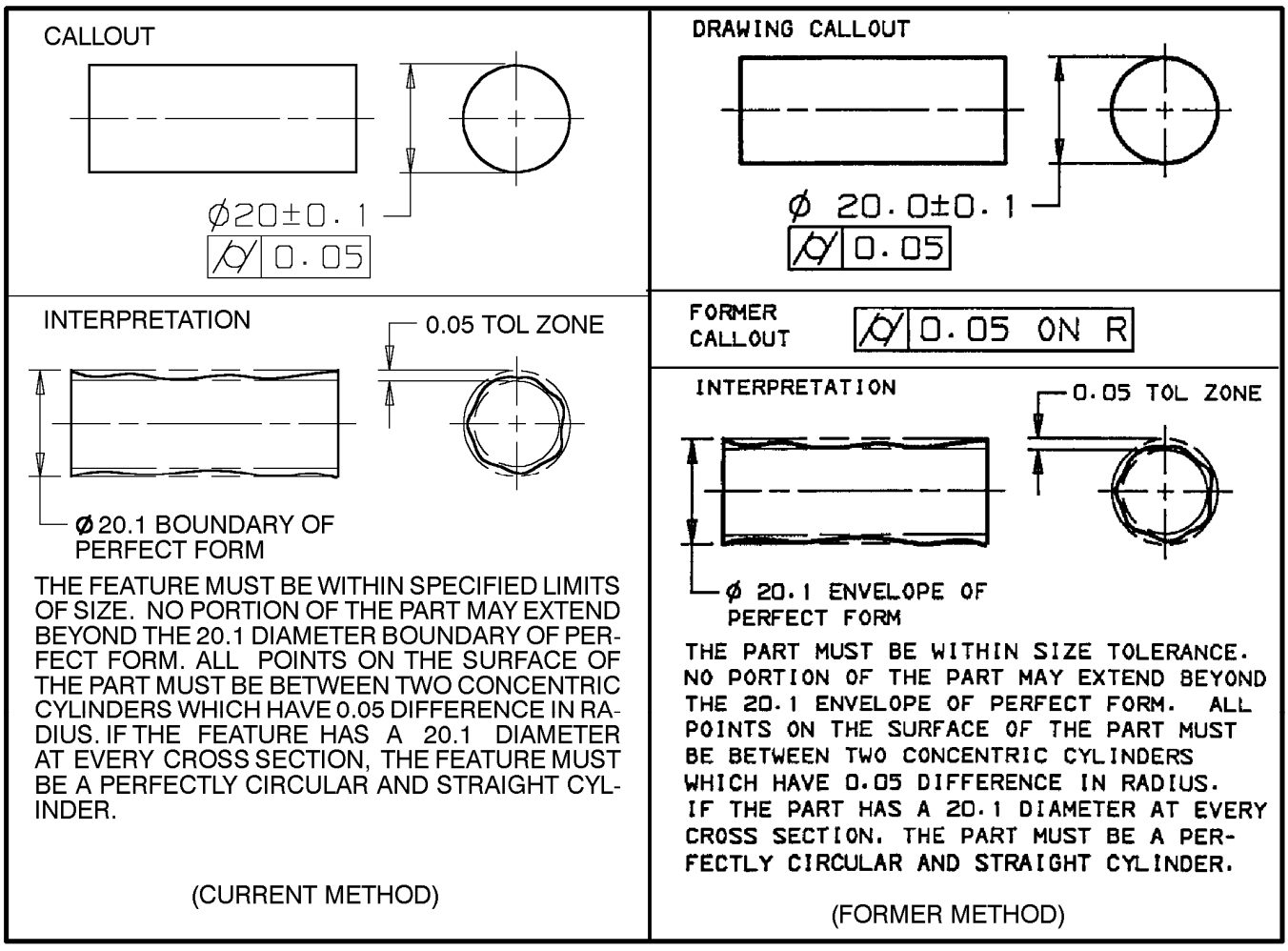


Figure 38 - Cylindricity

9.5 Parallelism

9.5.1 Definition - Parallelism is the condition of a surface or center plane, equidistant at all points from the datum plane; or an axis, equidistant along its length from one or more datum planes or a datum axis.

9.5.2 Tolerance - A parallelism tolerance specifies one of the following:

9.5.2.1 A tolerance zone defined by two parallel planes parallel to a datum plane or axis, within which the surface or center plane of the considered feature shall lie.

9.5.2.2 A tolerance zone defined by two parallel planes parallel to a datum plane or axis, within which axis of the considered feature shall lie.

9.5.2.3 A cylindrical tolerance zone parallel to one or more datum planes or a datum axis, within which the axis of the feature shall lie.

9.5.2.4 A tolerance zone defined by two parallel lines parallel to a datum plane or axis, within that a line element of the surface shall lie.

9.5.3 Inspection - Parallelism, size, and location tolerances of the feature are checked independently and each shall conform. Examples of the parallelism control are illustrated in Figures 39 thru 46.

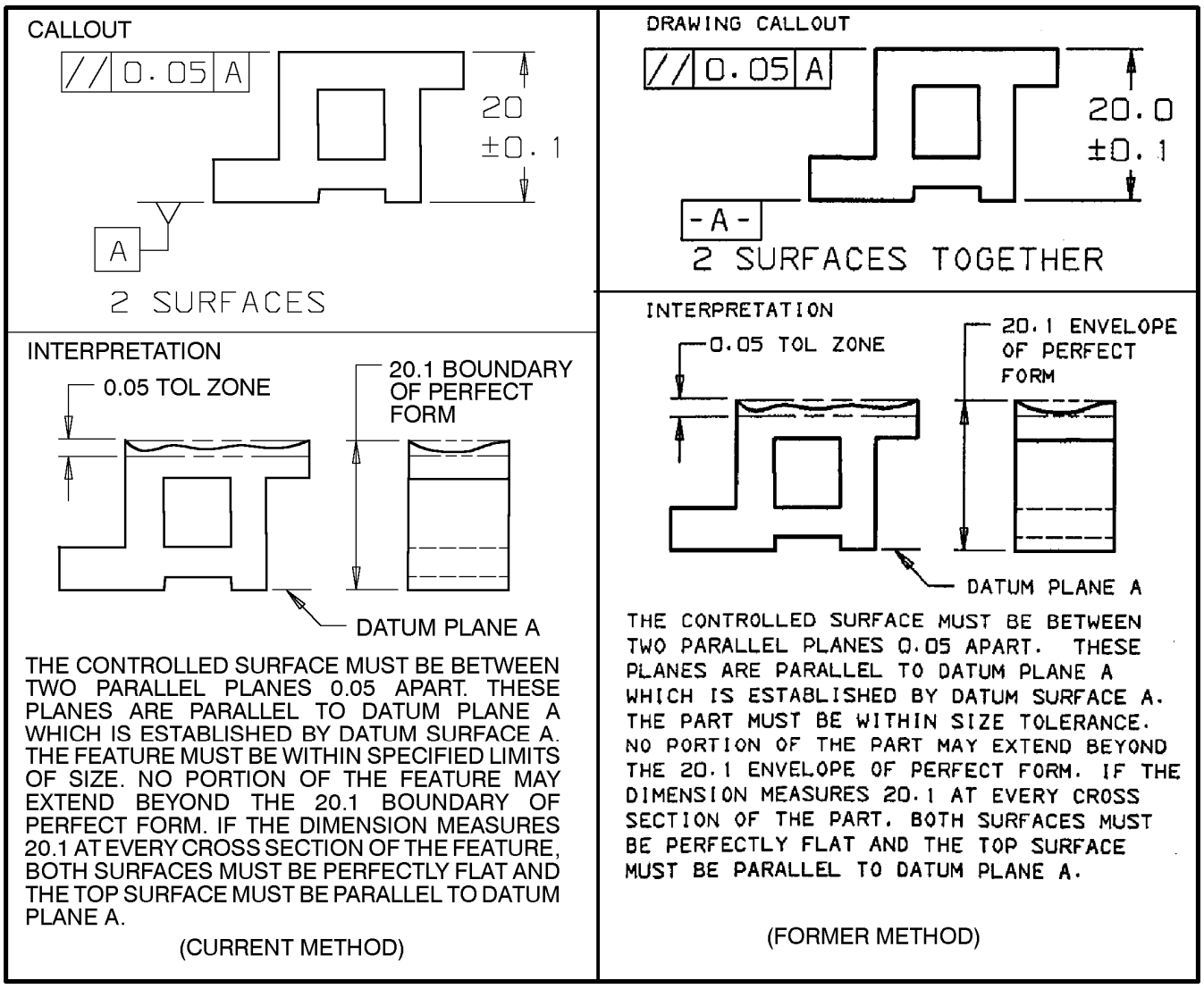


Figure 39 - Parallelism

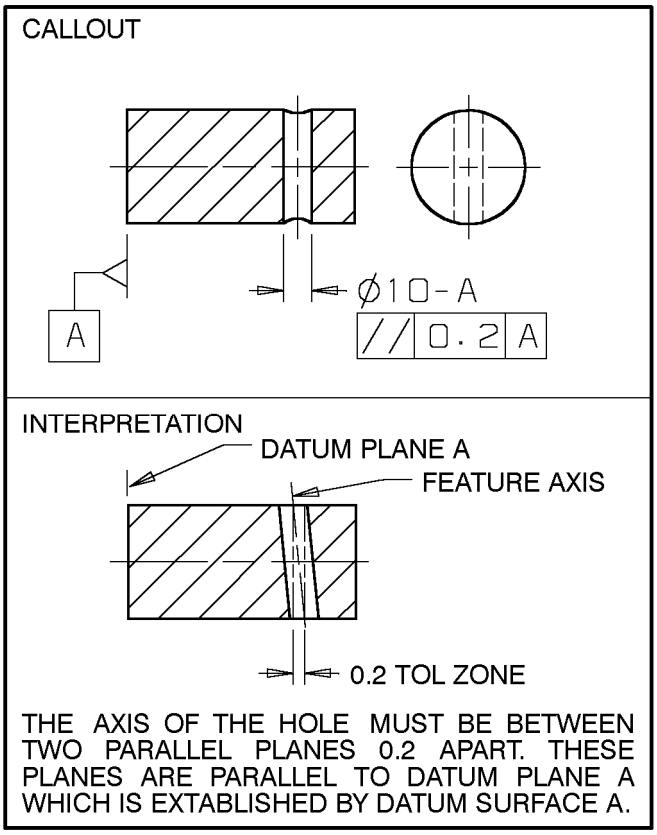


Figure 40 - Parallelism

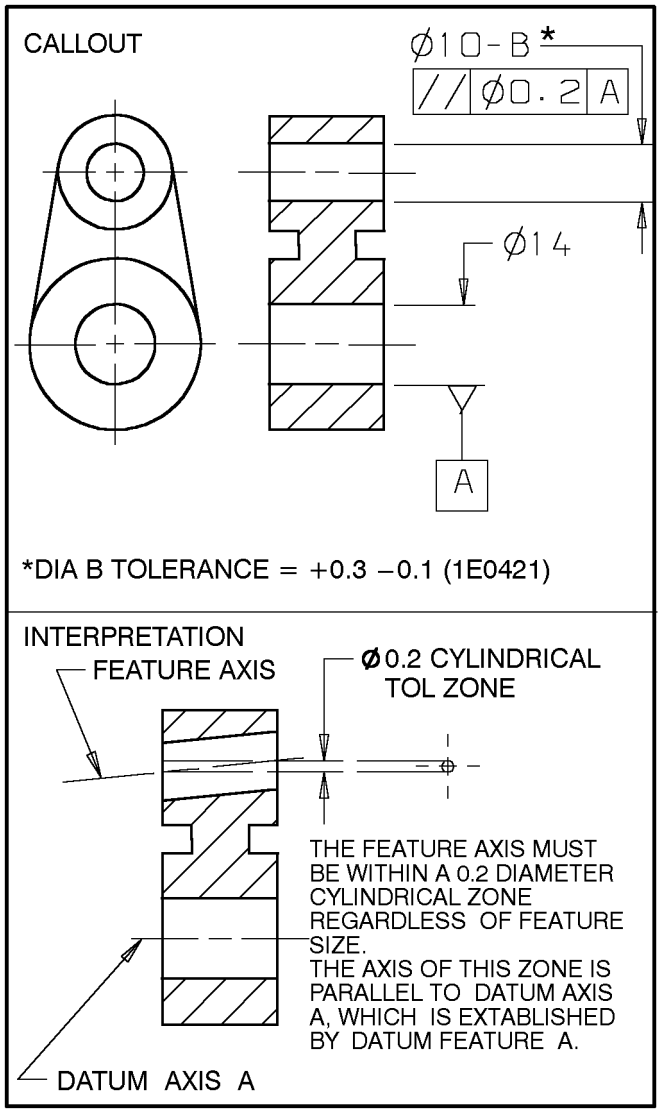


Figure 41 - Parallelism (RFS)

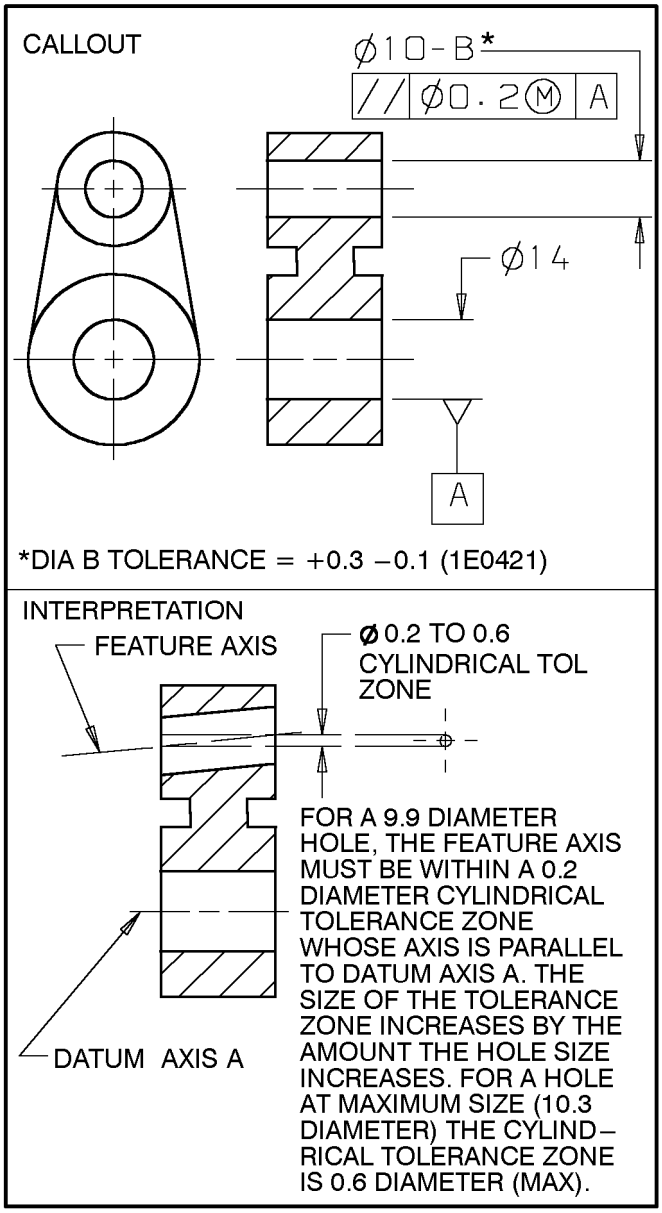


Figure 42 - Parallelism (MMC)

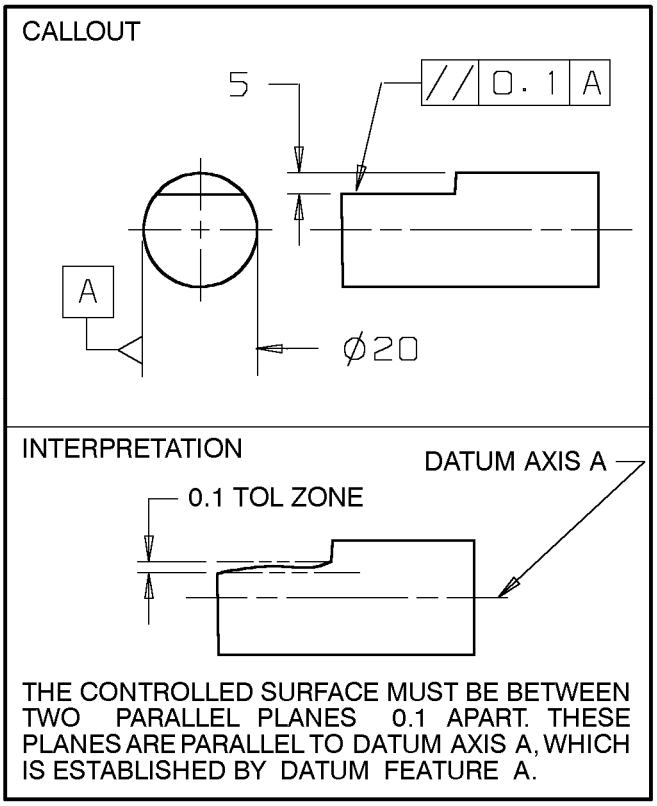


Figure 43 - Parallelism

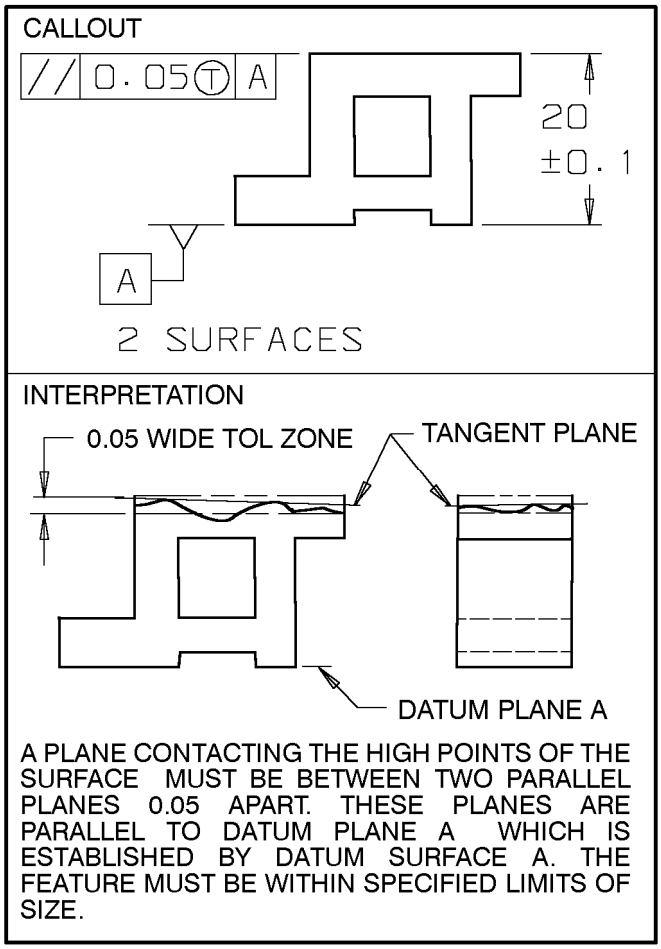


Figure 44 - Parallelism with Tangent Plane

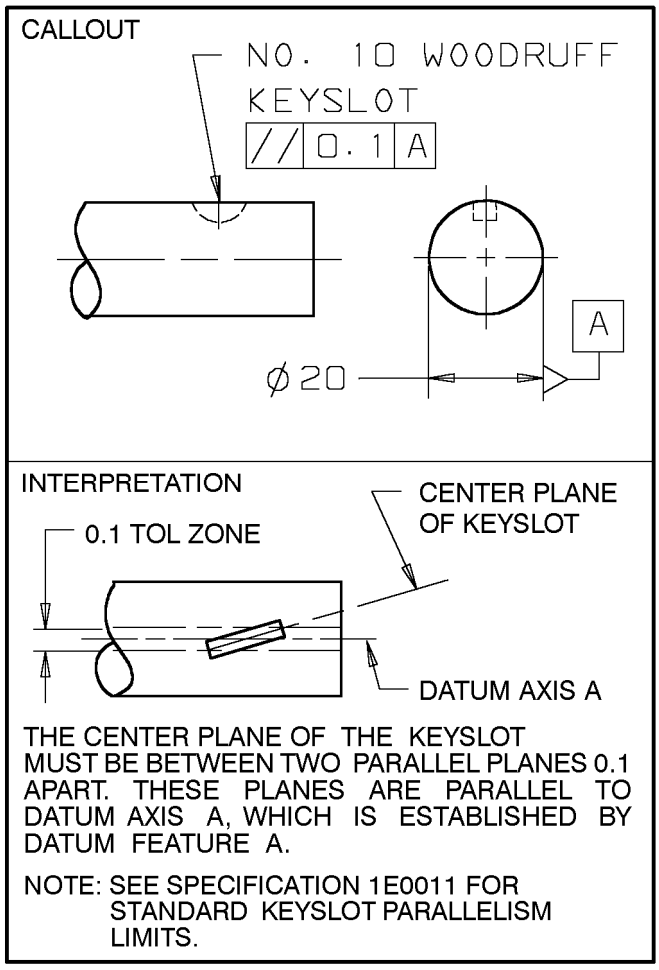


Figure 45 - Parallelism

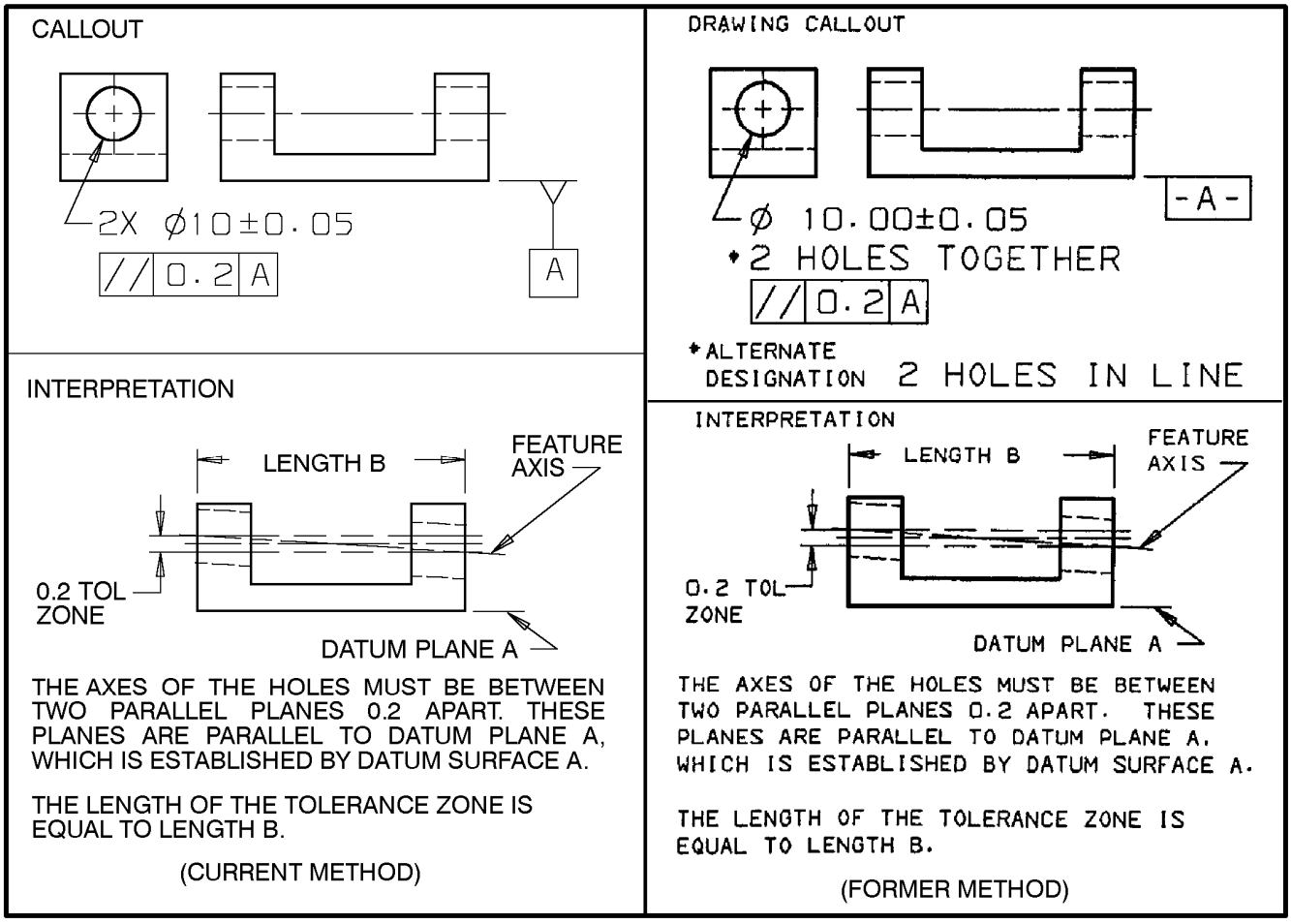


Figure 46 - Parallelism

9.6 Perpendicularity

9.6.1 Definition - Perpendicularity is the condition of a surface, center plane, or axis at a right angle (90°) to a datum plane or axis.

9.6.2 Tolerance - A perpendicularity tolerance specifies one of the following:

9.6.2.1 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 shall lie.

9.6.2.2 A cylindrical tolerance zone perpendicular to a datum plane within which the axis of the considered feature shall lie.

9.6.2.3 Tolerance zone defined by two parallel planes perpendicular to a datum axis within which the axis of the considered feature shall lie.

9.6.2.4 A tolerance zone defined by two parallel lines perpendicular to a datum plane or axis within which a line element of the surface shall lie.

9.6.2.5 A tolerance zone defined by two parallel planes perpendicular to two mutually perpendicular datum planes that are established by datum surfaces (3 point contact on first and 2 point contact on second) within which the surface of the considered feature shall lie.

9.6.2.6 A tolerance zone defined by two parallel planes perpendicular to two mutually perpendicular datum planes, one datum plane established by a datum surface and the other located by one datum axis and oriented by a second datum axis, within which the surface of the considered feature shall lie.

9.6.3 Inspection - Perpendicularity, size, and location tolerances of the feature are checked independently and each shall conform. Examples of perpendicularity control are illustrated in Figures 47 thru 54.

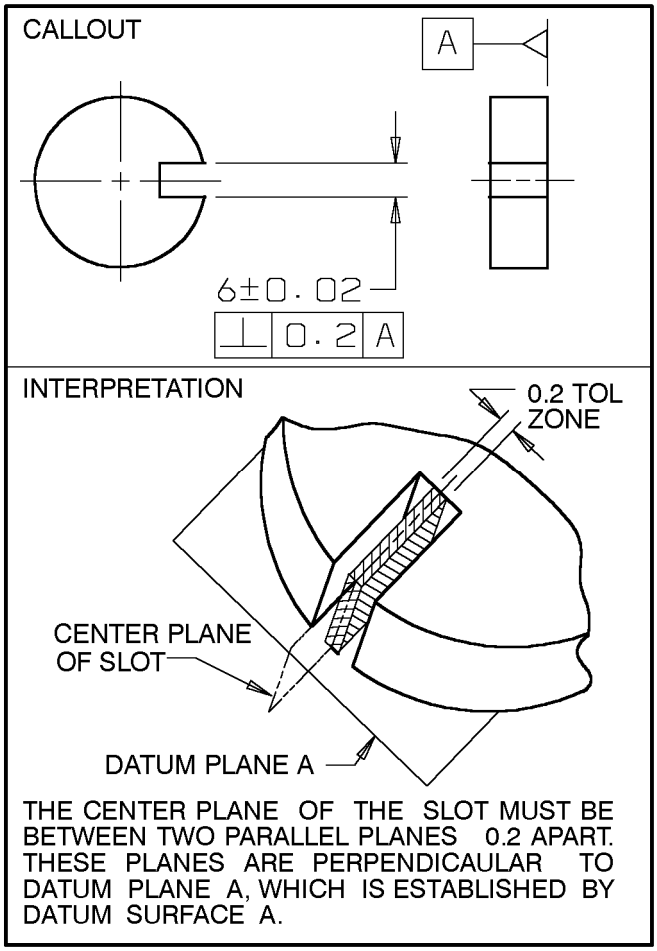


Figure 47 - Perpendicularity

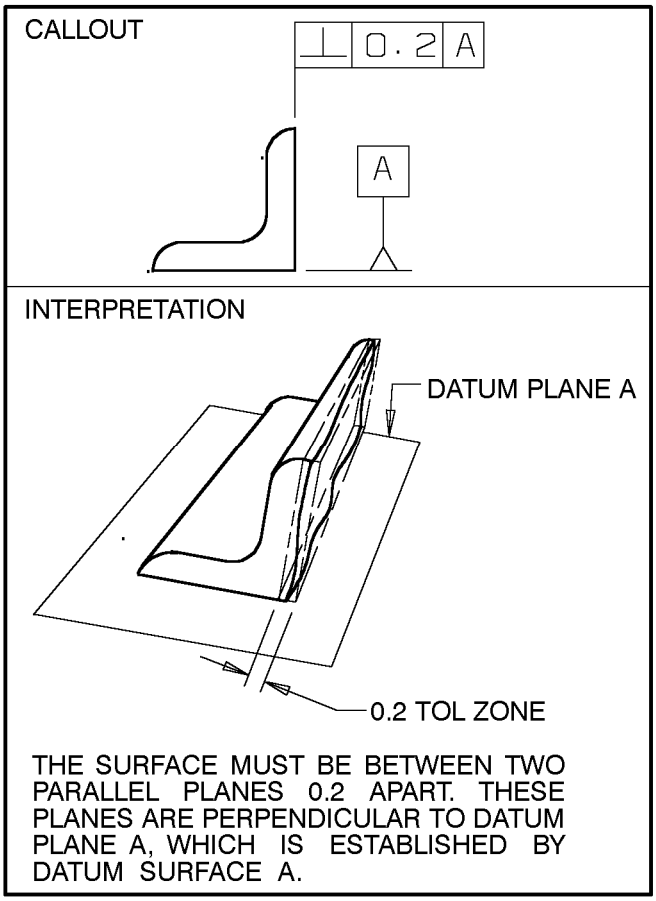


Figure 48 - Perpendicularity

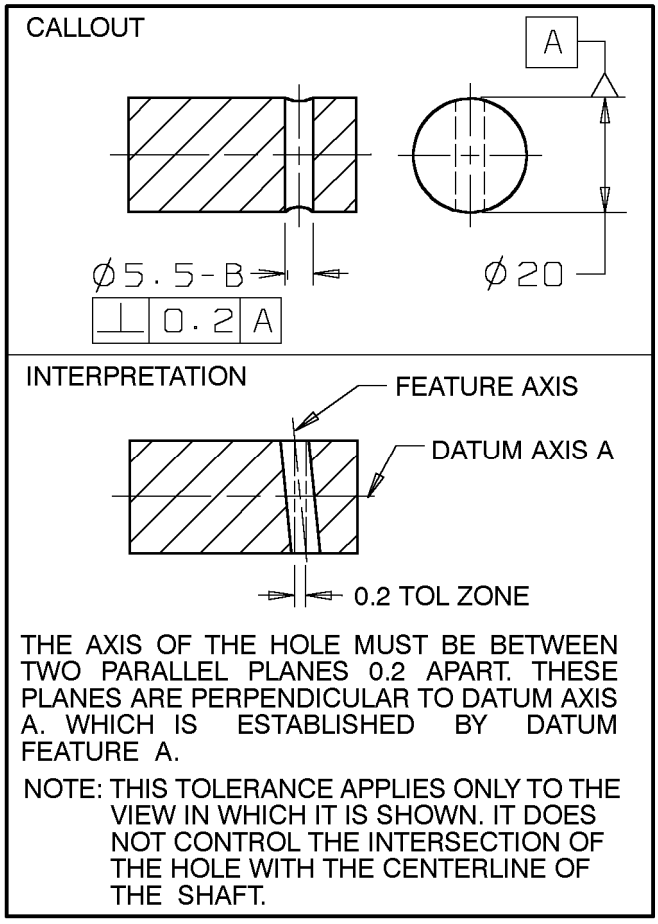


Figure 49 - Perpendicularity

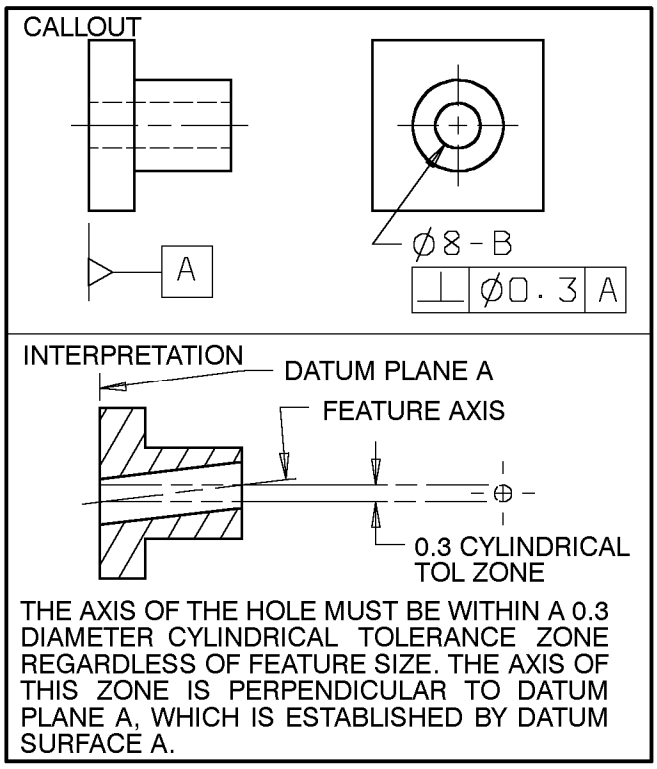


Figure 50 - Perpendicularity (RFS)

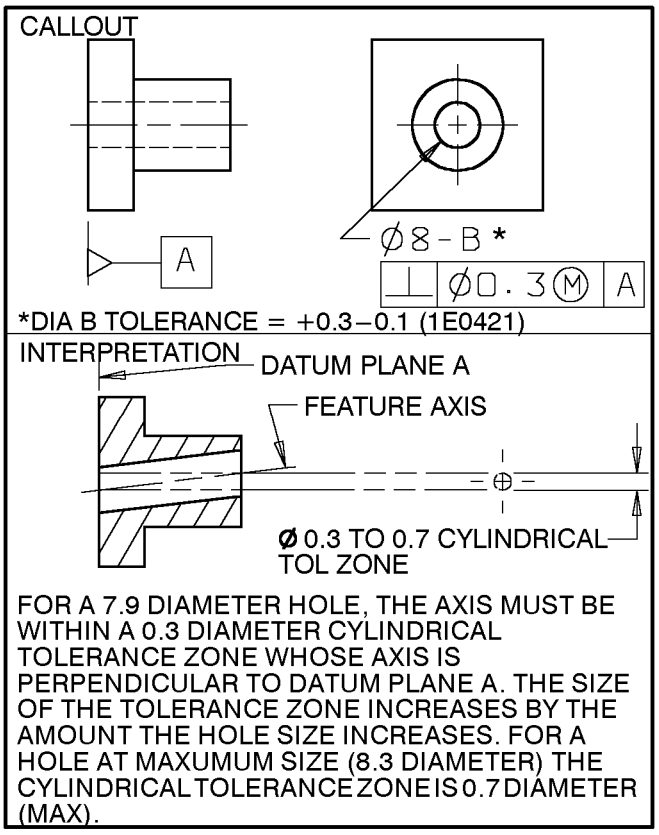


Figure 51 - Perpendicularity (MMC)

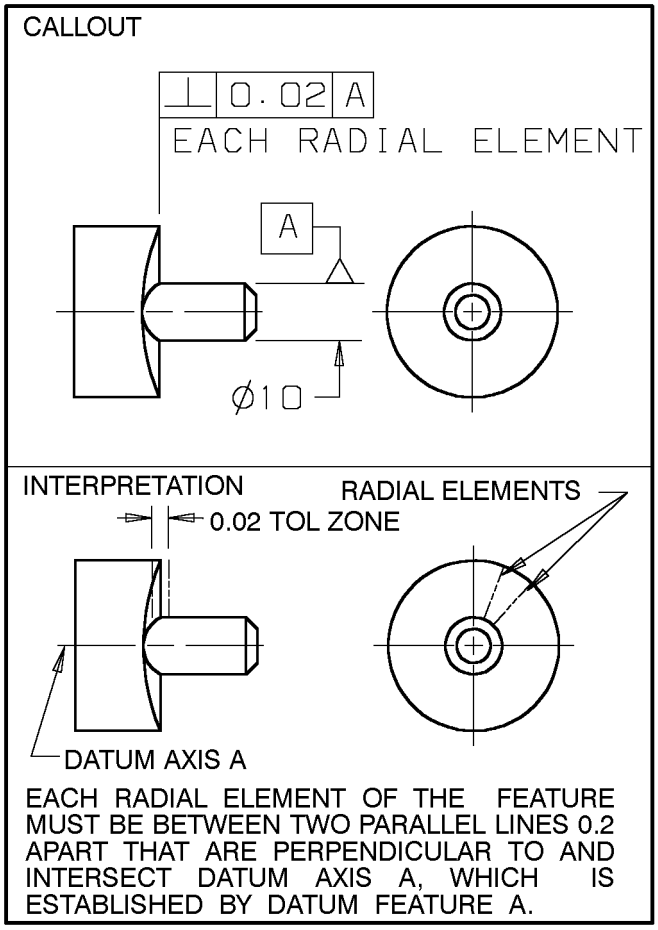


Figure 52 - Perpendicularity

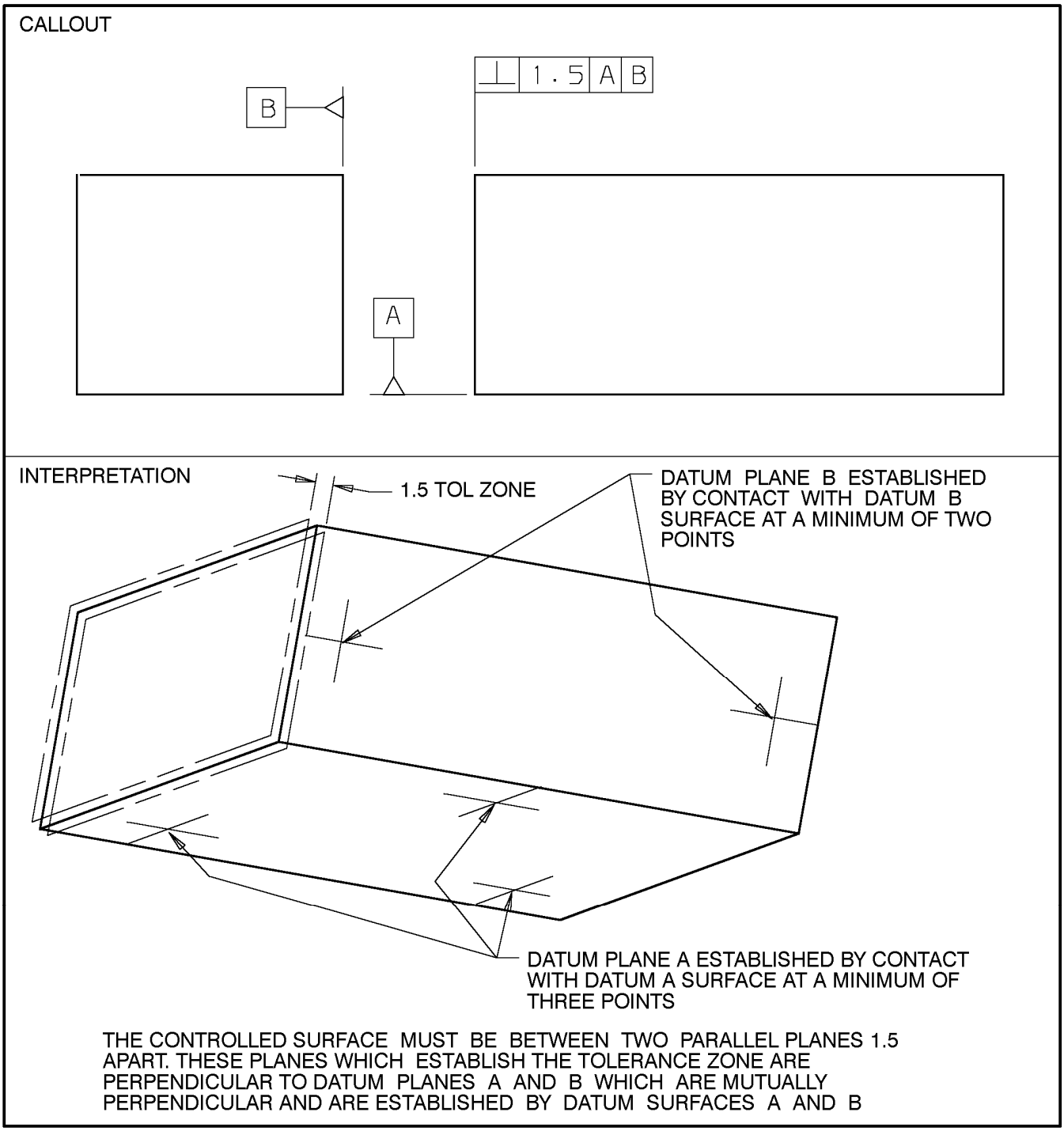


Figure 53 - Perpendicularity

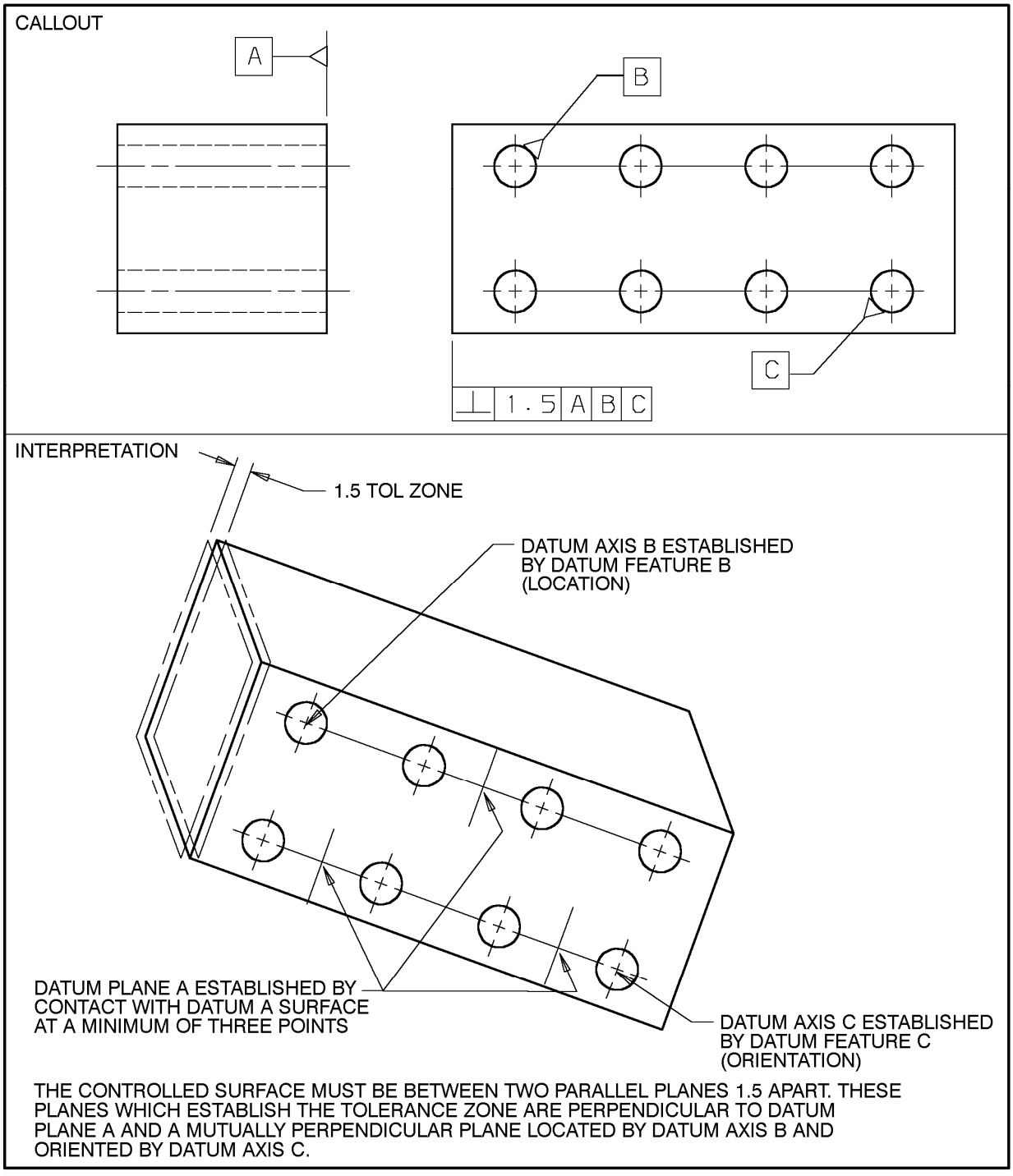


Figure 54 - Perpendicularity

9.7 Angularity

9.7.1 Definition - Angularity is the condition of a surface, center plane, or axis at a specified angle (other than 90 degree) from a datum plane or axis.

9.7.2 Tolerance - An angularity tolerance specifies one of the following:

9.7.2.1 A Tolerance zone defined by two parallel planes at the specified basic angle from one or more datum planes, or axes, within which the surface or center plane of the considered feature shall lie.

9.7.2.2 A tolerance zone defined by two parallel planes at the specified basic angle from one or more datum planes, or axes, within which the axis of the considered feature shall lie.

9.7.2.3 A cylindrical tolerance zone at the specified basic angle from one or more datum planes or axes, within which the axis of the considered feature shall lie.

9.7.2.4 A tolerance zone defined by two parallel lines at the specified basic angle from a datum plane or axis within which a line element of the surface shall lie.

9.7.3 Application - Angularity controls flatness of a plane surface unless a more restrictive flatness tolerance is specified. The angle between the datum and controlled features is basic (untoleranced).

9.7.4 Inspection - Angularity, size, and location tolerances of the feature are checked independently and each shall conform. Examples of angularity control are illustrated in Figures 55 through 63.

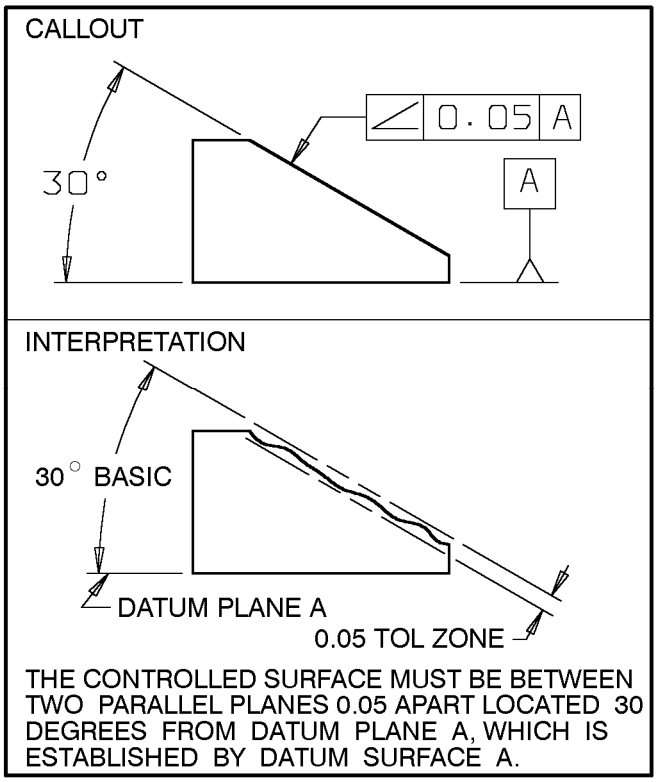


Figure 55 - Angularity

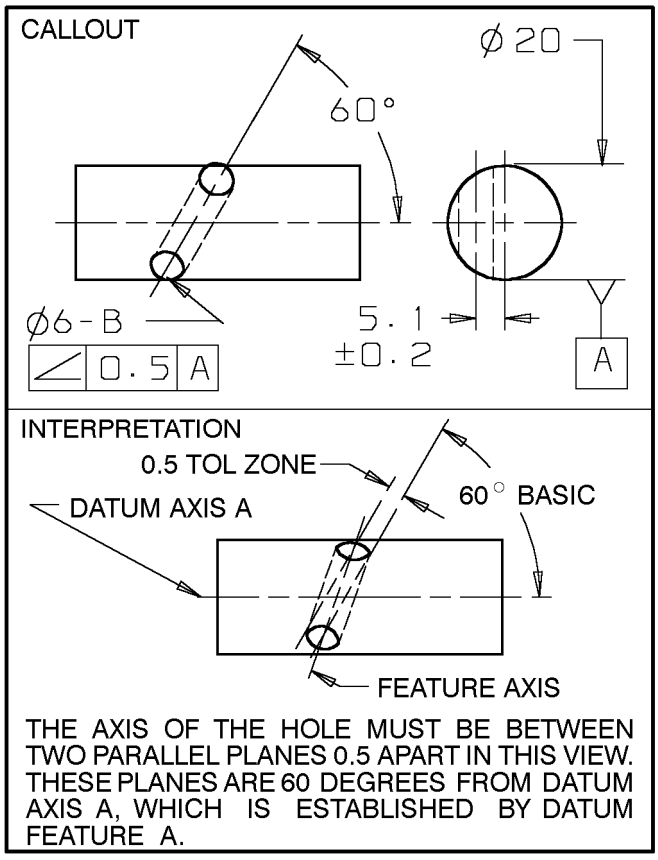


Figure 56 - Angularity

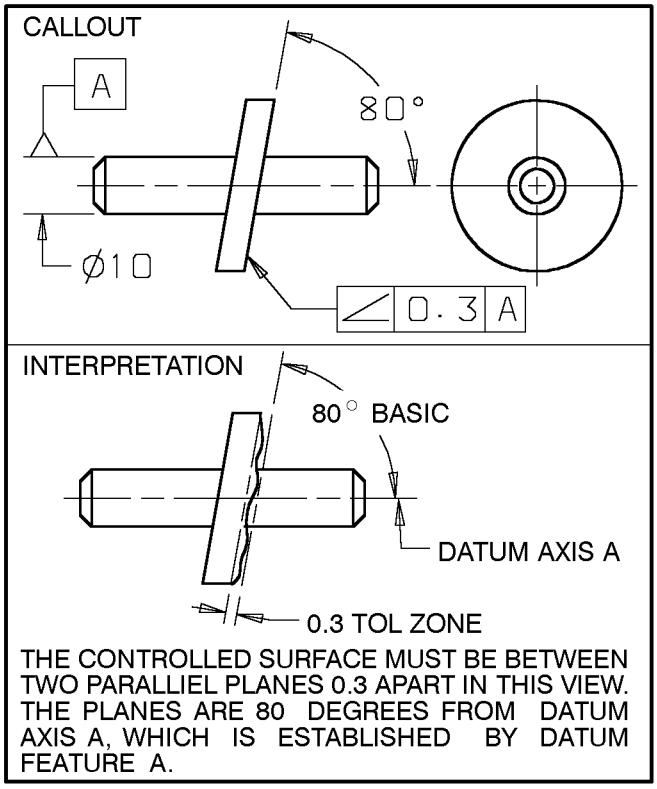


Figure 57 - Angularity

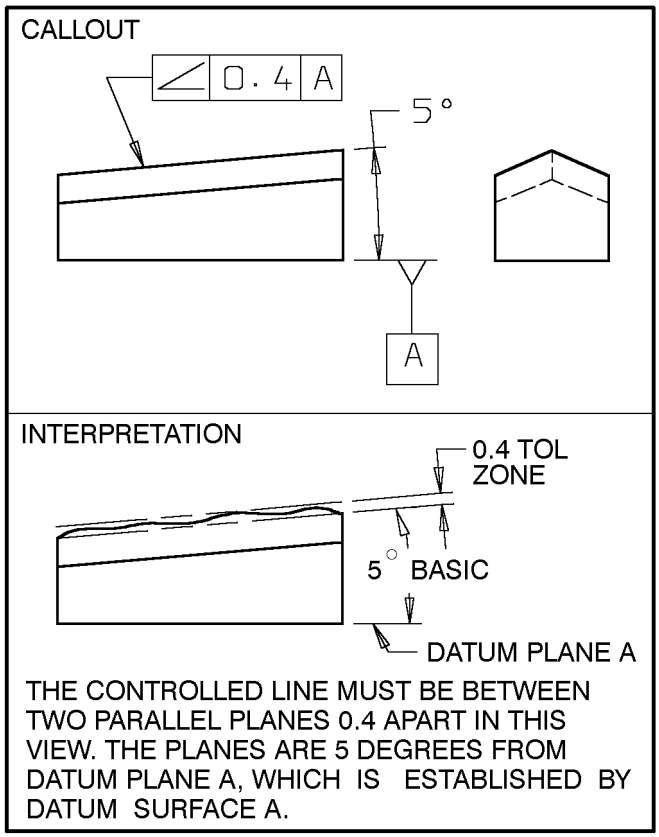


Figure 58 - Angularity

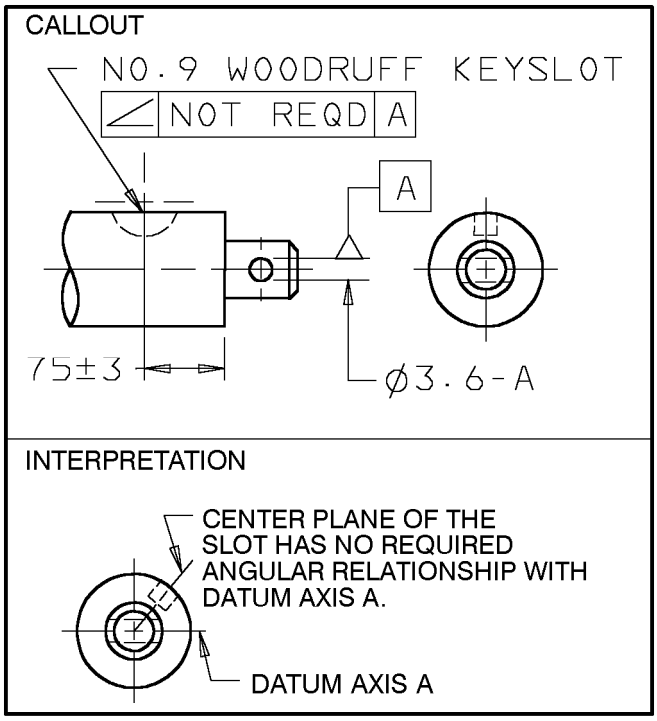


Figure 59 - Angularity

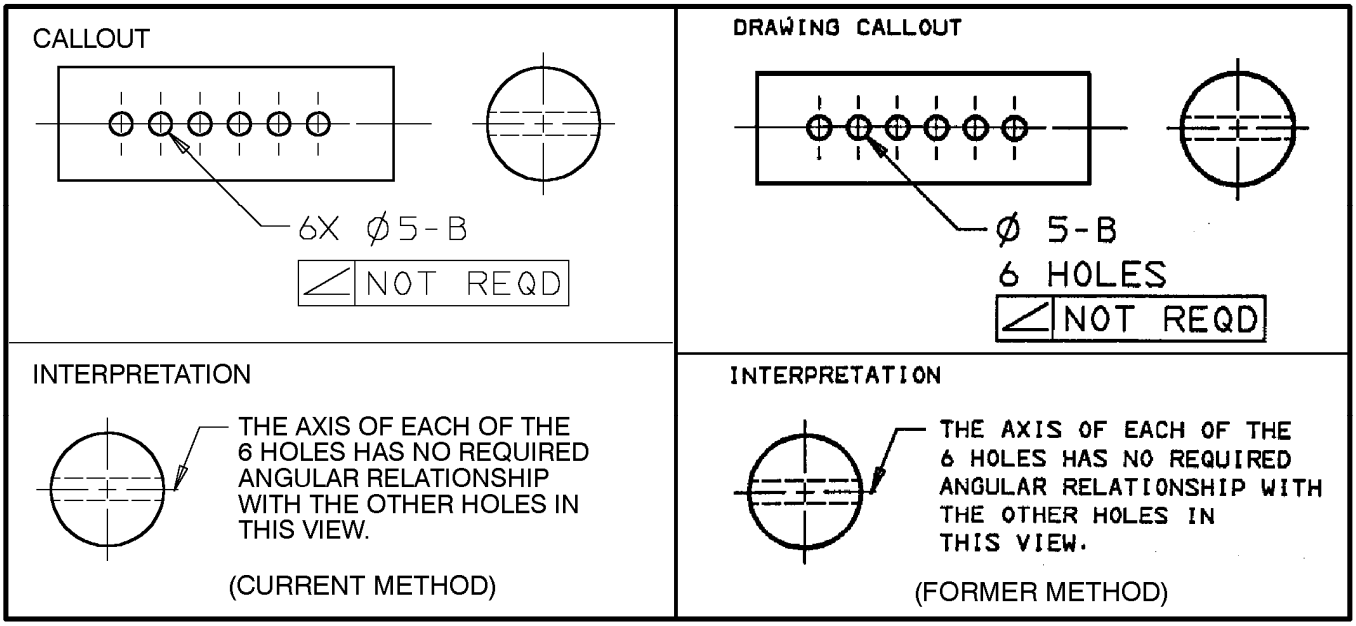


Figure 60 - Angularity

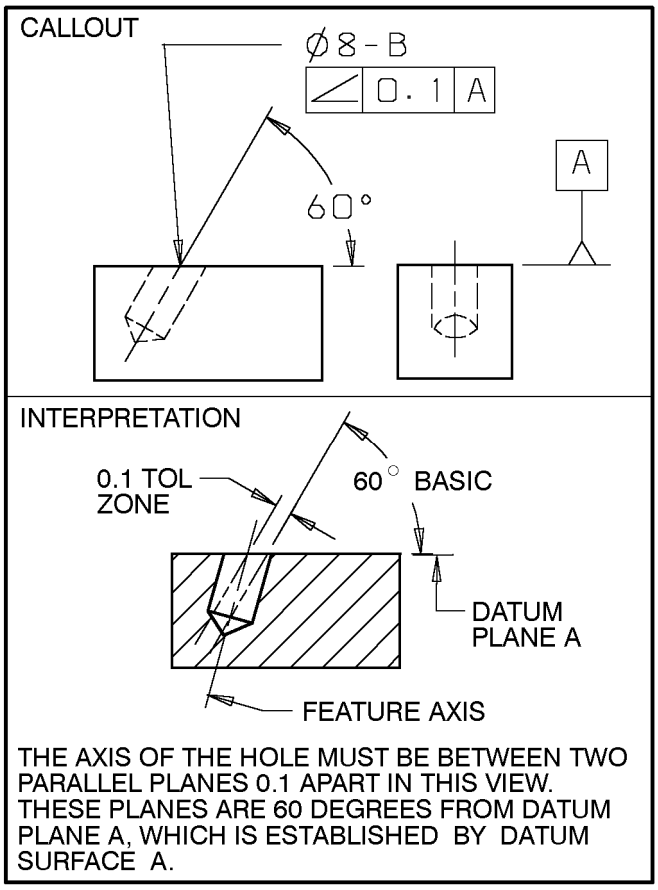


Figure 61 - Angularity

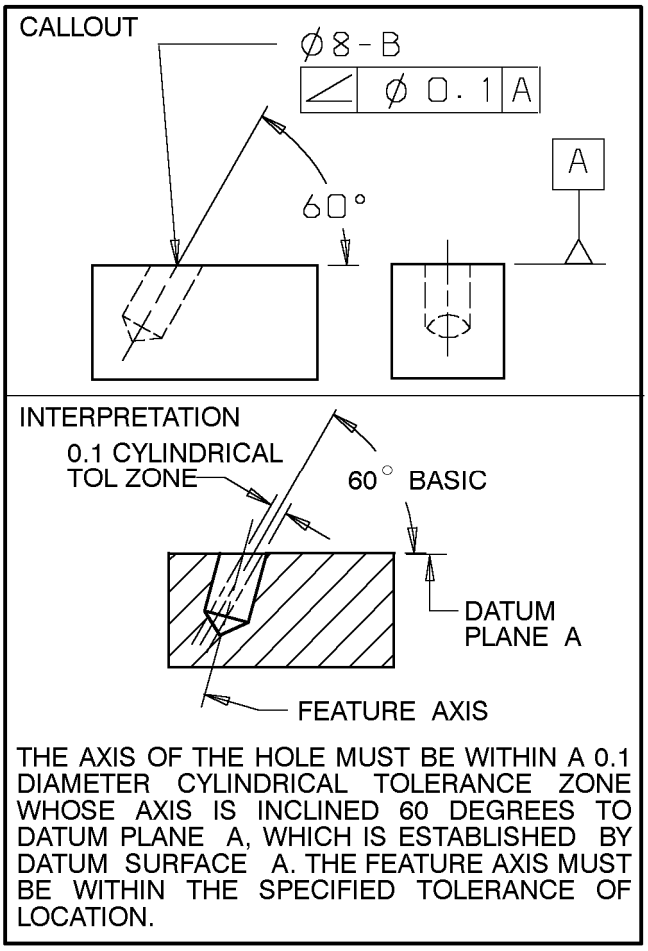


Figure 62 - Angularity (RFS)

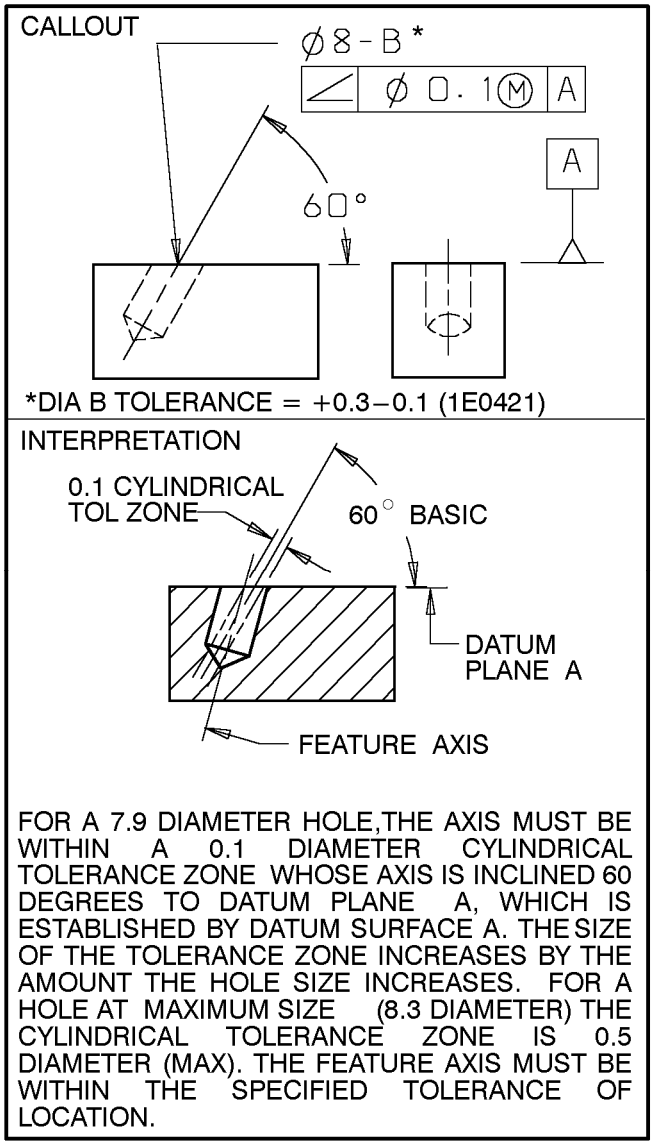


Figure 63 - Angularity (MMC)

9.8 Runout

9.8.1 Definition - Runout is a composite tolerance used to control the functional relationship of one or more features of a component 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.

9.8.2 Control - Each controlled feature shall fulfill the following typical conditions:

9.8.2.1 The full indicator movement (FIM), formerly total indicator reading (TIR), shall not exceed the specified tolerance.

9.8.2.2 The indicator or component is rotated about the datum axis.

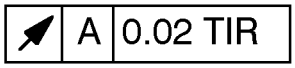
9.8.2.3 The datum axis is established by one of the following:

- A diameter of sufficient length.
- A flat surface and a diameter perpendicular to it.
- Two diameters usually at opposite ends of a component such as bearing journals on a shaft.

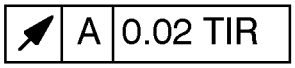
9.8.3 Runout and size tolerance of the feature are checked independently and each shall conform.

9.8.3.1 There are two types of runout control, circular runout and total runout. Symbols are  for circular runout and  for total runout.

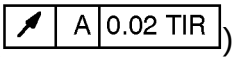
9.8.3.2 Former designations specified the term **CIRC** or **TOTAL** below the feature control frames.



CIRC



TOTAL

For older drawings that do not specify either CIRC or TOTAL (

), the interpretation for circular runout shall apply (see Figures 64, 65, and 66). The former term total indicator reading has been replaced by full indicator movement with no change in interpretation.

9.8.4 Circular Runout - Circular runout provides composite control of the circular elements of a surface. The tolerance is applied independently at any circular measuring position as the component or indicator is rotated 360° about the datum axis. Where applied to circular features, circular runout controls the cumulative variations of form and location characteristics, that is, circularity and concentricity. Where applied to surfaces constructed at right angles to the datum axis, circular runout controls circular elements of the surface (wobble). See Figures 64, 65, and 66.

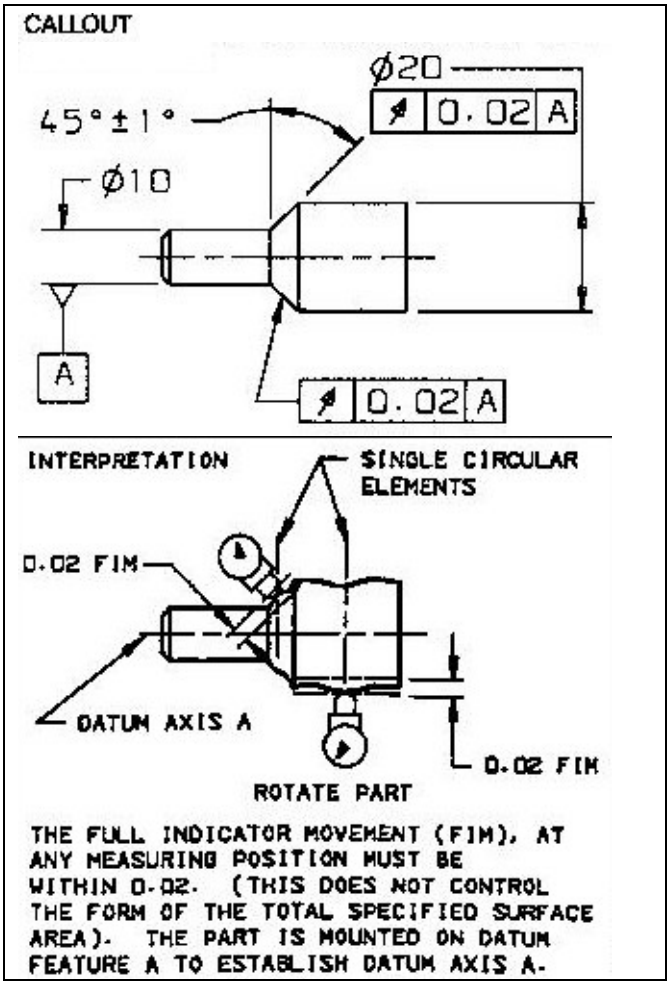


Figure 64 - Circular Runout

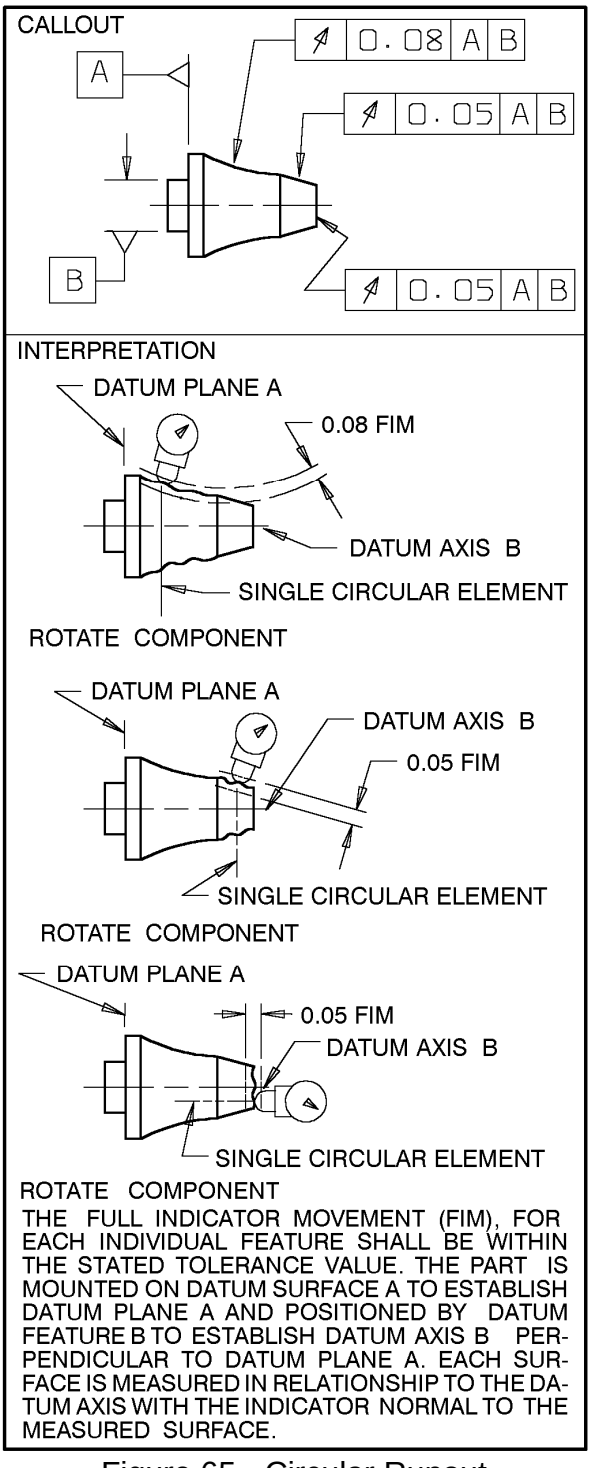


Figure 65 - Circular Runout

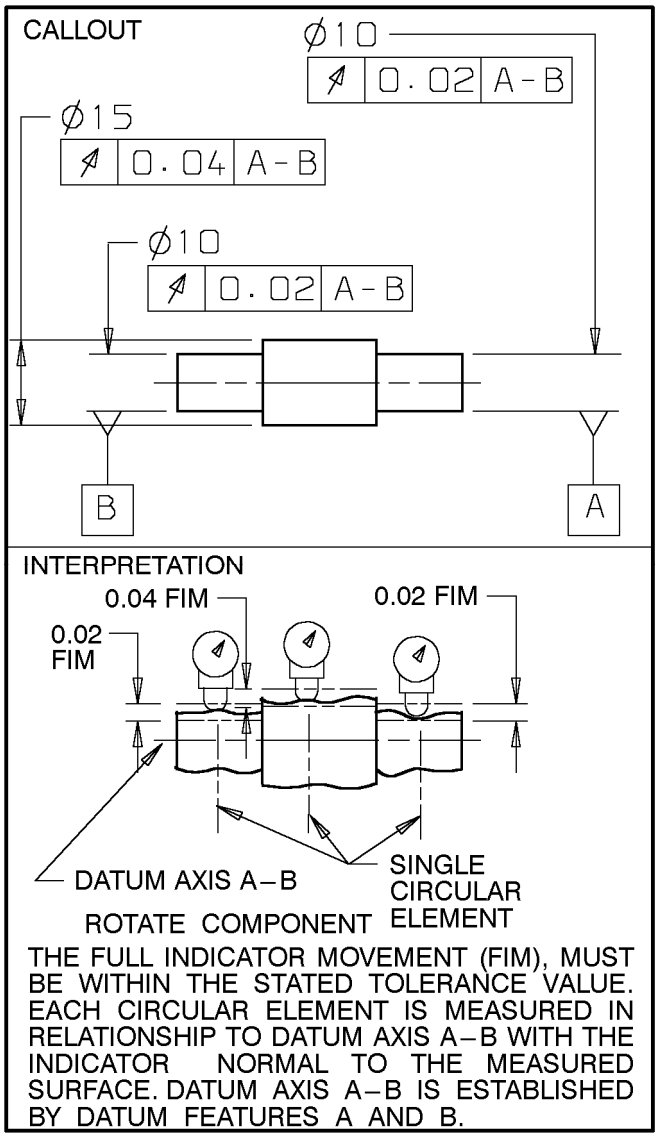


Figure 66 - Circular Runout

9.8.5 Total Runout - Total runout provides composite control of all surface elements. The tolerance is applied simultaneously to all circular and profile-measuring positions as the component or indicator is rotated 360° about the datum axis. Where applied to circular features constructed around the datum axis, total runout controls angularity, taper, straightness of surface elements, and profile of a surface in addition to circularity and concentricity which are controlled by circular runout. Where applied to surfaces constructed at right angles to a datum axis, total run out controls perpendicularity and flatness (convex and concave conditions) in addition to wobble controlled by circular runout. (See Figures 67 and 68).

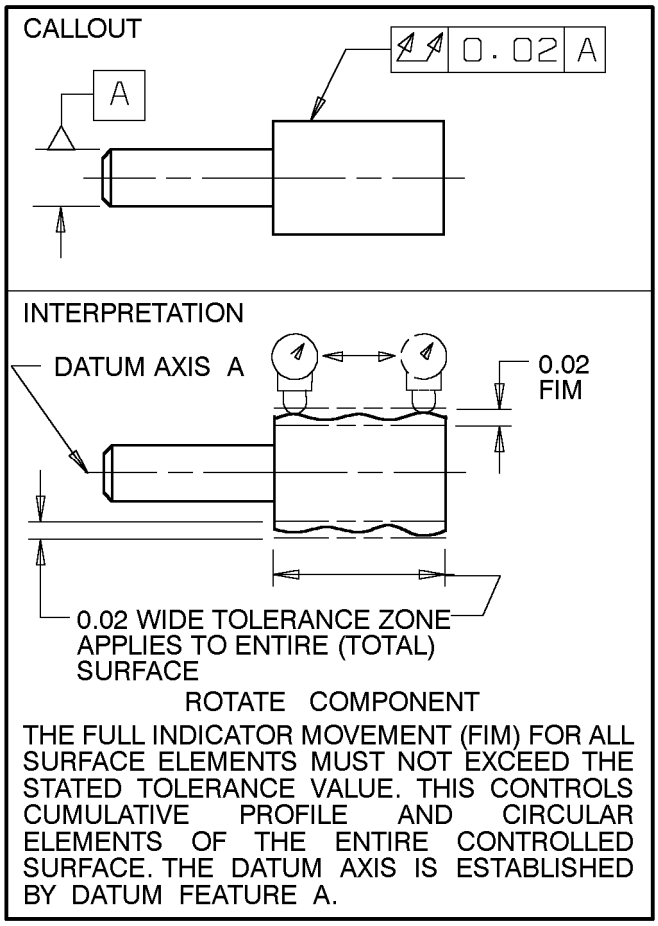


Figure 67 - Total Runout

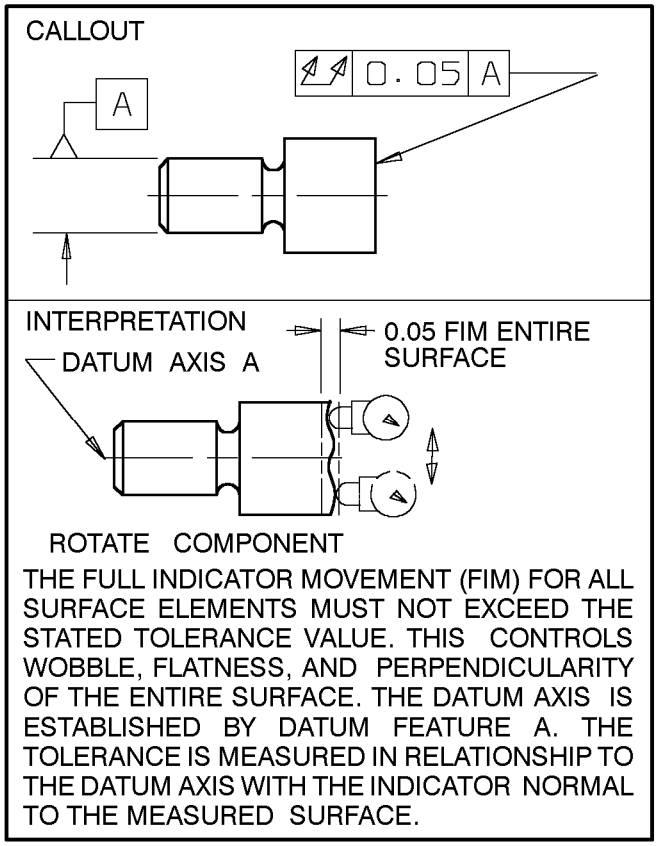


Figure 68 - Total Runout

10.0 POSITIONAL TOLERANCES

10.1 Definition - Position is the theoretically exact position of the axis or center plane of a feature.

10.2 Tolerance - A positional tolerance (formerly called true position tolerance) describes one of the following:

10.2.1 The diameter of a cylindrical zone as illustrated in Figure 69. This interpretation is indicated by specifying the diameter symbol before the tolerance value.

10.2.2 The diameter of a spherical zone as illustrated in Figure 70. This interpretation is indicated by specifying the spherical diameter symbol before the tolerance value.

10.2.3 The distance between two parallel planes as illustrated in Figure 71. This interpretation is indicated by the absence of a symbol before the tolerance value.

10.2.4 The zones defined in a, b, and c can also be converted to a boundary interpretation as defined in Item e where the positional tolerance specifies the difference between the maximum material condition of the controlled feature and the size of the boundary.

10.2.5 The size of a boundary by specifying the word **BOUNDARY** beneath a feature control frame listing the positional tolerance value. An alternate method is by specifying a positional tolerance value followed by the abbreviation **BDY** within the feature control frame. The positional tolerance specifies the difference between the maximum material condition of the controlled feature and the size of the boundary as shown in Figures 72, 99, and 100.

10.2.6 The tolerance zone is located at true position with respect to specified datums. The limits of the tolerance zone are equal to the extent (length or length and width) of the controlled feature. The tolerance zone also limits the angular variation in the axis or center plane of the controlled feature.

10.2.7 Application - Positional tolerance may be applied to either circular or noncircular features.

10.2.8 Basic (untoleranced) dimensions locate the true position of controlled features with respect to each other and from specified datums.

10.3 Interpretation Rules

10.3.1 Material Condition Basis - Positional tolerance is applied on a maximum material condition (MMC), regardless of feature size (RFS), or least material condition (LMC) basis. The appropriate modifier symbol Ⓜ for MMC, Ⓢ for RFS, or Ⓛ for LMC is placed after the tolerance value and applicable datum references in the feature control frame.

10.3.1.1 Maximum Material Condition Basis - It is only when the controlled feature is at its MMC size that the specified positional tolerance applies. Where the actual size of the feature departs from MMC (within the specified limits of size), additional positional tolerance results. This increase of positional tolerance is equal to the difference between the specified MMC limit of size and the actual size of the feature. (See Paragraph 7.2).

Note: Drawings which do not specify a modifier symbol (Ⓜ, Ⓢ, or Ⓛ) in a position tolerance feature control frame shall be interpreted such that the modifier symbol Ⓜ, [Maximum material condition (MMC)] applies (without being specified for both the tolerance value and applicable size datums) unless the modifier symbol Ⓢ is specified in the feature control frame. Before 1 Jan 88 the modifier symbol Ⓛ was not used on Caterpillar drawings and the modifier symbol Ⓢ was used only when it was necessary to restrict the maximum material condition interpretation.

10.3.1.2 Regardless of Feature Size Basis - The specified positional tolerance applies regardless of the size of the controlled feature. The modifier symbol Ⓢ may be applied to a feature only if it has size variation and should be used only where increased tolerance due to feature size variation (MMC basis) would adversely affect design requirements such as gear center locating holes and press fits. (See Paragraph 7.1).

10.3.1.3 Least Material Condition Basis - It is only when the controlled feature is at its LMC size that the specified positional tolerance applies. Specification of LMC further requires perfect form at LMC - perfect form at MMC is not required. Where the actual size of the feature departs from LMC (Within the specified limits of size), additional positional tolerance results. This increase in positional tolerance is equal to the difference between the specified LMC limit and the actual size of the feature. LMC is used to maintain a desired relationship between the surface of a feature and its true position at tolerance extremes such as maintaining a minimum wall thickness between a boss and a hole. (See Paragraph 7.3).

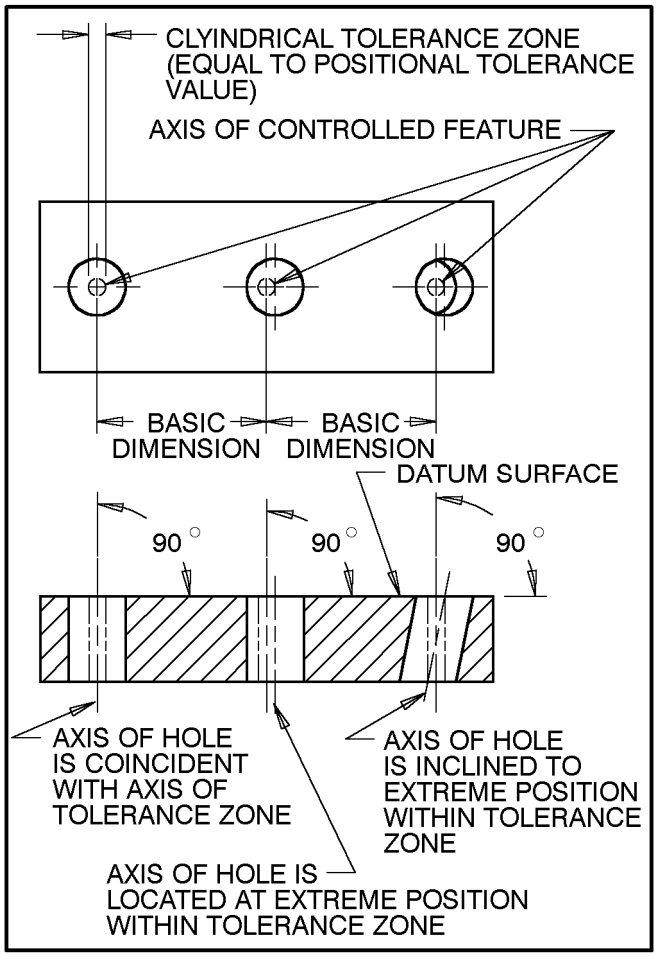


Figure 69 - Cylindrical Tolerance Zone

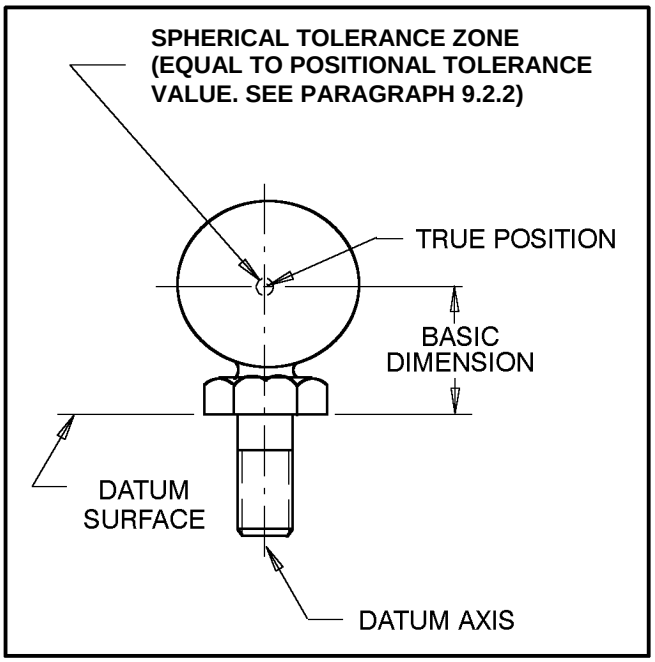


Figure 70 - Spherical Tolerance Zone

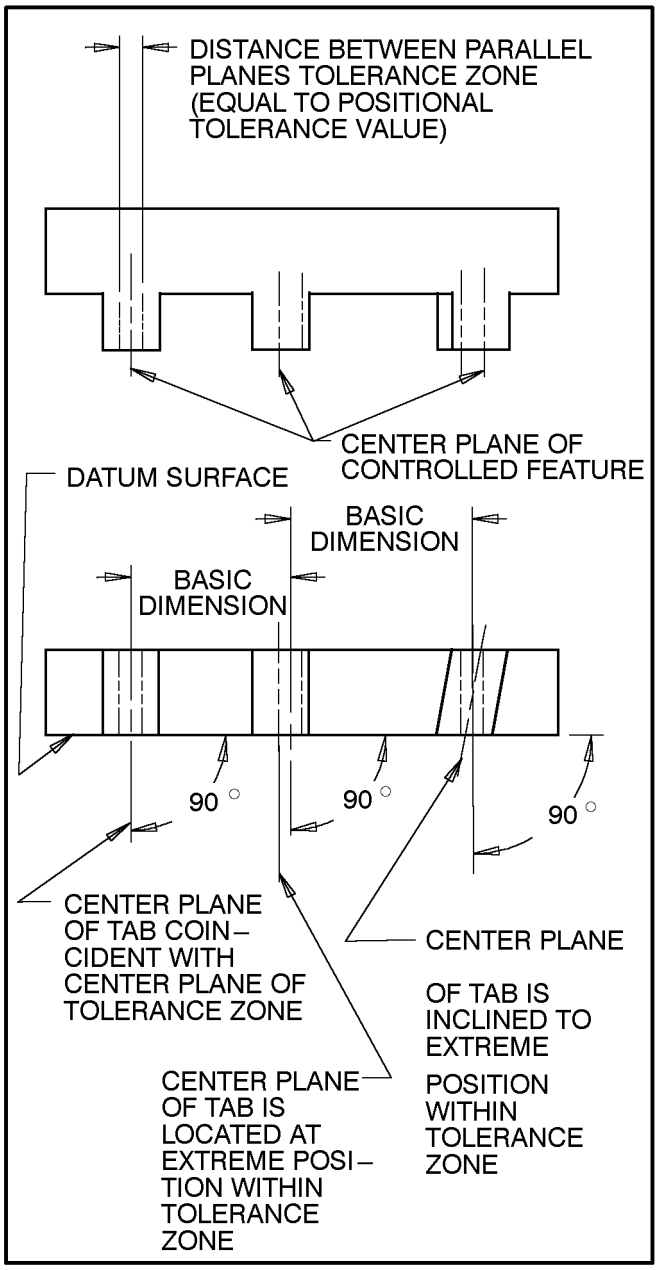


Figure 71 - Parallel Planes Tolerance Zone

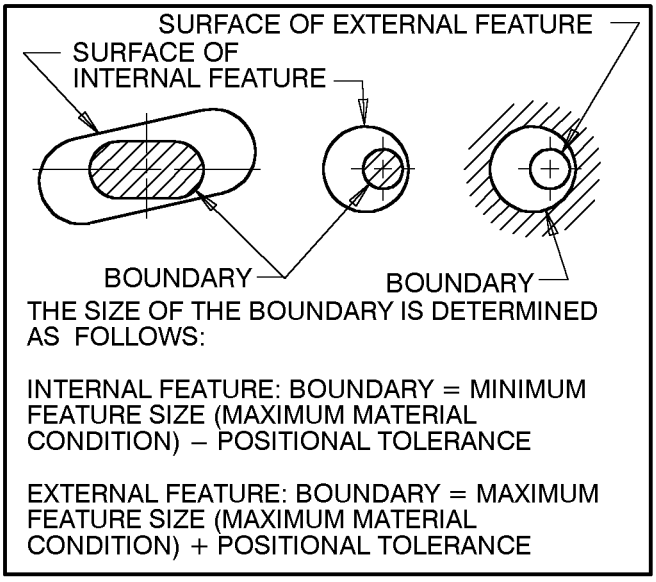


Figure 72 - Boundary Interpretation

10.3.2 Combination Positional Tolerancing And Toleranced Locating Dimensions (Former Method) - The coordinate toleranced dimensions establish rectangular tolerance zones within which only the axes of the position tolerance zones shall lie. Each feature axis shallbe within its respective cylindrical positional tolerance zone whose diameter is equal to the specified tolerance where an external feature is at maximum size or an internal feature is at minimum size. This permits the axis of the feature to be outside the zone established by the coordinate dimensions by an amount equal to one-half the applicable positional tolerance. (See Figure 105).

10.3.3 Inspection - Size, form, and location tolerances of the feature are checked independently and each shall conform.

10.3.4 Examples of positional tolerance control are shown in Figures 73 through 105.

10.3.5 The modifier symbol Ⓟ indicates a projected tolerance zone. The modifier symbol Ⓟ and a value (projected length) are specified within or below a positional tolerance feature control frame, the tolerance zone is projected to the area of the mating component. The tolerance zone does not apply within the feature being controlled. (See Figure 102).

10.3.6 Feature Pattern Location - Composite positional tolerancing is used to permit a feature-relating tolerance zone framework (FRTZF) to be located and oriented within limits imposed upon it by a pattern-locating tolerance zone framework (PLTZF).

10.3.6.1 Where composite controls are used, the upper segment is referred to as the pattern-locating tolerance zone framework. The PLTZF is located from specified datums by basic dimensions. It specifies the larger positional tolerance for the location of the pattern of features as a group. Applicable datums are specified in the desired order of precedence, and serve to relate the PLTZF to the datum reference frame.

10.3.6.2 The lower segment is referred to as the feature-relating tolerance zone framework. It governs the smaller positional tolerance for each feature within the pattern (feature-to-feature relationship). Basic dimensions used to relate the PLTZF to specified datums are not applicable to the FRTZF. Where datum references are not specified in the lower segment of the composite feature control frame, the FRTZF is free to be located and oriented (Shift and/or tilt) within the boundaries established and governed by the PLTZF. If datums are specified in the lower segment, they govern the orientation of the FRTZF relative to the PLTZF. Where datum references are specified, one or more of the datums specified in the upper segment of the composite feature control frame are repeated, as applicable, and in the same order of precedence, to govern the orientation of the FRTZF.

10.3.6.3 Where the primary datum A is a plane and is repeated in the lower segment of the composite feature control frame, the axes of both the PLTZF and FRTZF tolerance cylinders are perpendicular to datum plane A and therefore, parallel to each other. In certain instances, portions of the smaller zones may fall beyond the peripheries of the larger tolerance zones. However, these portions of the smaller tolerances zones are not usable because the axes of the features must not violate the boundaries of the larger tolerance zones. The axes of the actual holes may vary obliquely (out of perpendicularity) only within the confines of the respective smaller positional tolerance zones (FRTZF). (See Figure 103).

10.3.6.4 Where both the primary and secondary datums are planes and are repeated in the lower segment of the composite feature control frame, the tolerance cylinders of the FRTZF may be displaced from the true position locations (as a group) as governed by the tolerance cylinders of the PLTZF, while remaining perpendicular to datum plane A and parallel to datum plane B. The actual axes of the holes in the actual feature pattern shall reside within both the tolerance cylinders of the FRTZF and the PLTZF. (See Figure 104).

10.3.7 Size charts in Figure 106 (mm) and Figure 107 (inch) may be used as an inspection aid to determine conformance to positional tolerances. These charts are based on the coordinate difference determined as shown in the example.

Note: These charts are not to be used with dimensions that have specified coordinate tolerances.

10.4 Concentricity

10.4.1 Definition - Concentricity is that condition where the median points of all diametrically opposed elements of a figure of revolution (or correspondingly-located elements of two or more radially-disposed features) are congruent with the axis (or center point) of a datum feature.

10.4.2 Tolerance - Concentricity tolerance is the diameter of a cylindrical tolerance zone within which the axis of the controlled feature shall lie. The axis of the tolerance zone coincides with the axis of the datum feature. See Figure 88 for the interpretation of concentricity.

10.5 Symmetry

10.5.1 Definition - Symmetry is that condition where the median points of all opposed or correspondingly-located elements of two or more feature surfaces are congruent with the axis or center plane of a datum feature.

10.5.2 Tolerance - Symmetry tolerance is the distance between two parallel planes within which the center plane of the controlled feature shall lie. See Figure 86 for the interpretation of symmetry.

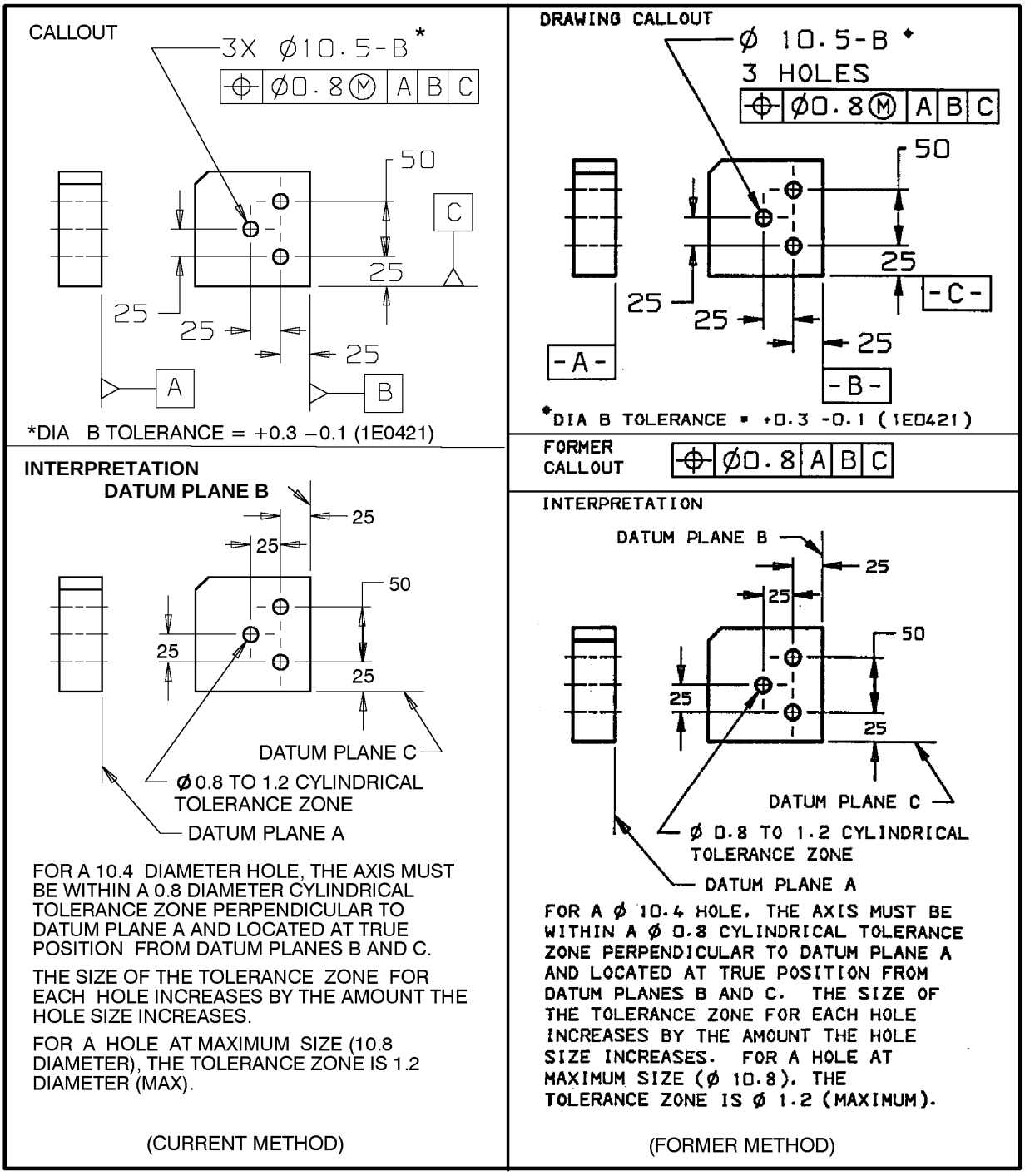


Figure 73 - Positional Tolerance

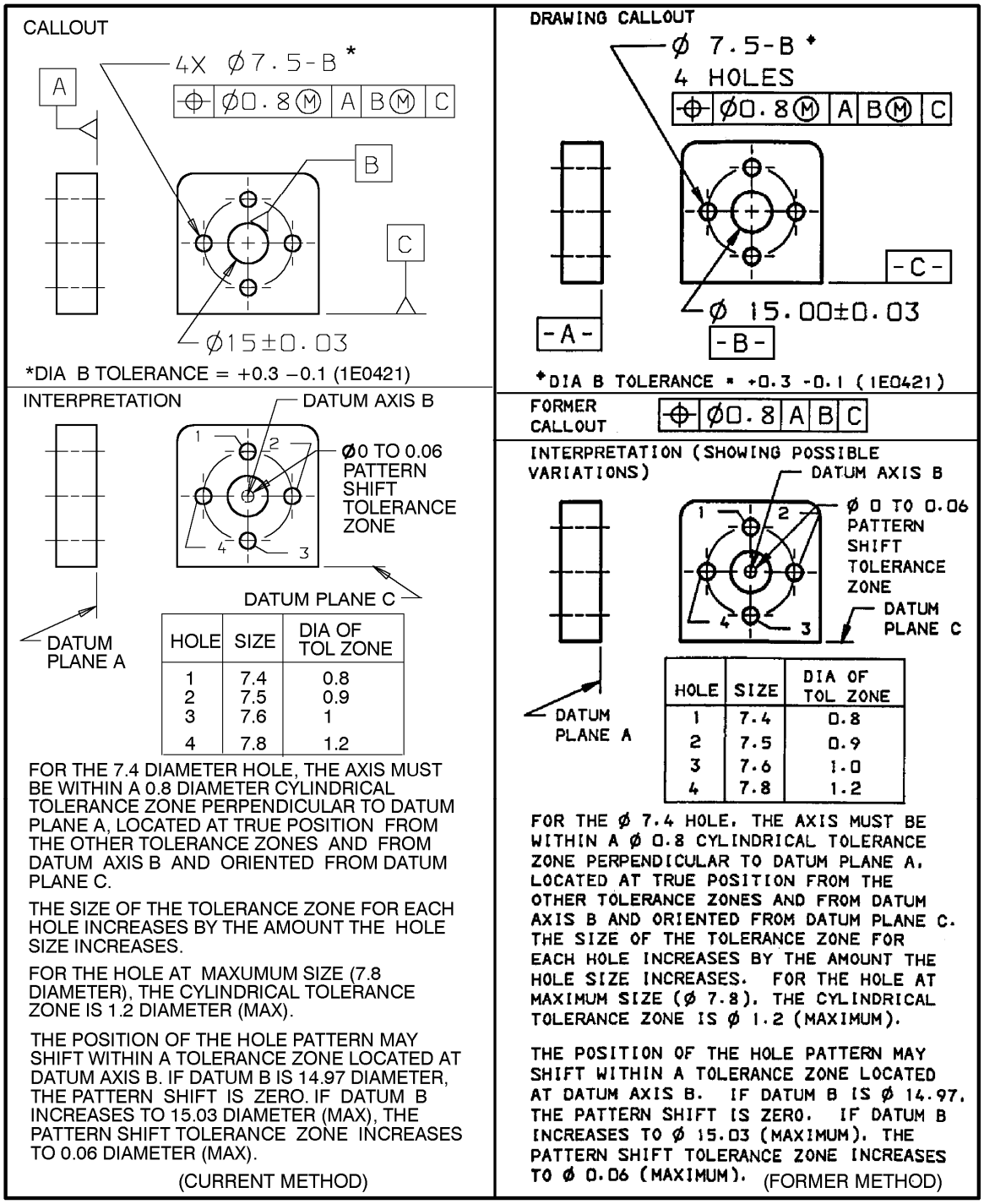
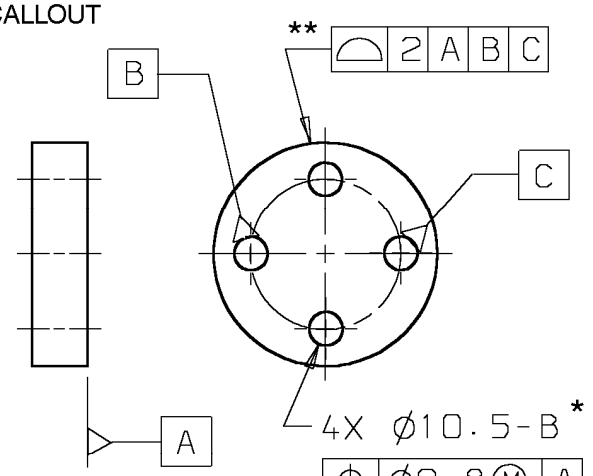


Figure 74 - Positional Tolerance

INTERPRETATION - GEOMETRIC TOLERANCE	DATE	CHG NO	NUMBER
	26 MAR 2013	10	1E0012

CALLOUT



*DIA B TOLERANCE = +0.3 −0.1 (1E0421)

INTERPRETATION

FOR A 10.4 DIAMETER HOLE, THE AXIS MUST BE WITHIN A 0.8 DIAMETER CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A AND LOCATED AT TRUE POSITION.

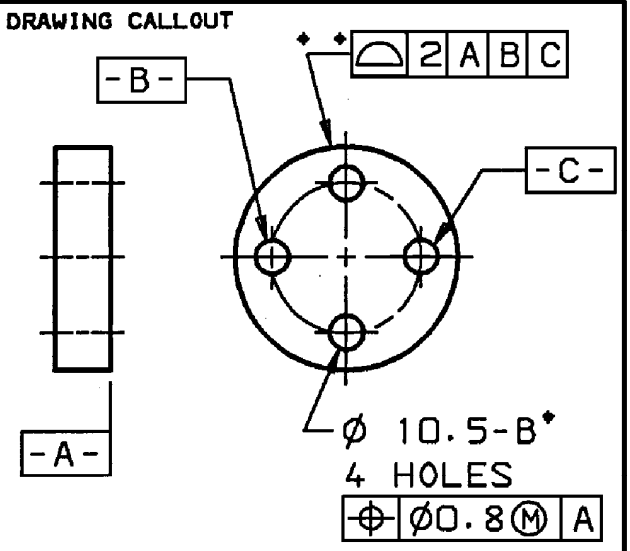
THE SIZE OF THE TOLERANCE ZONE FOR EACH HOLE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES.

FOR A HOLE AT MAXIMUM SIZE (10.8 DIAMETER), THE TOLERANCE ZONE IS 1.2 DIAMETER (MAX).

**THE EXTERNAL CIRCULAR SURFACE IS CONTROLLED BY A PROFILE OF A SURFACE TOLERANCE FROM DATUMS A, B, AND C. REFER TO ARTICLE 10.0

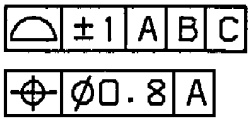
(CURRENT METHOD)

DRAWING CALLOUT



* DIA B TOLERANCE = +0.3 -0.1 (1E0421)

FORMER CALLOUT



INTERPRETATION

FOR A Ø 10.4 HOLE, THE AXIS MUST BE WITHIN A Ø 0.8 CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A AND LOCATED AT TRUE POSITION. THE SIZE OF THE TOLERANCE ZONE FOR EACH HOLE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES. FOR A HOLE AT MAXIMUM SIZE (Ø 10.8), THE TOLERANCE ZONE IS Ø 1.2 (MAXIMUM).

**THE EXTERNAL CIRCULAR SURFACE IS CONTROLLED BY A PROFILE OF A SURFACE TOLERANCE FROM DATUMS A, B, AND C. REFER TO ARTICLE 10.0.

(FORMER METHOD)

Figure 75 - Positional Tolerance

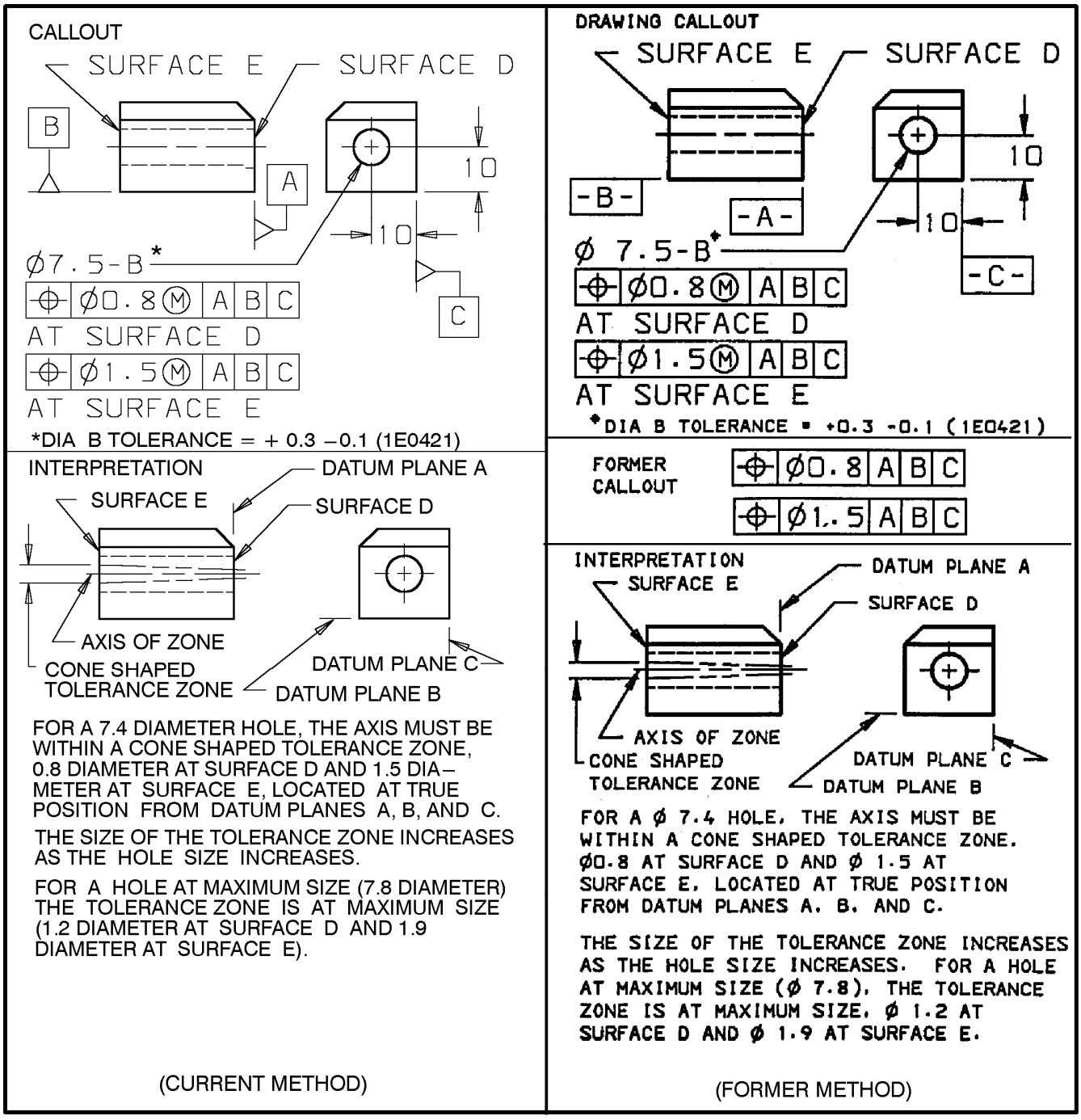


Figure 76 - Feature Controlled Closer At One End

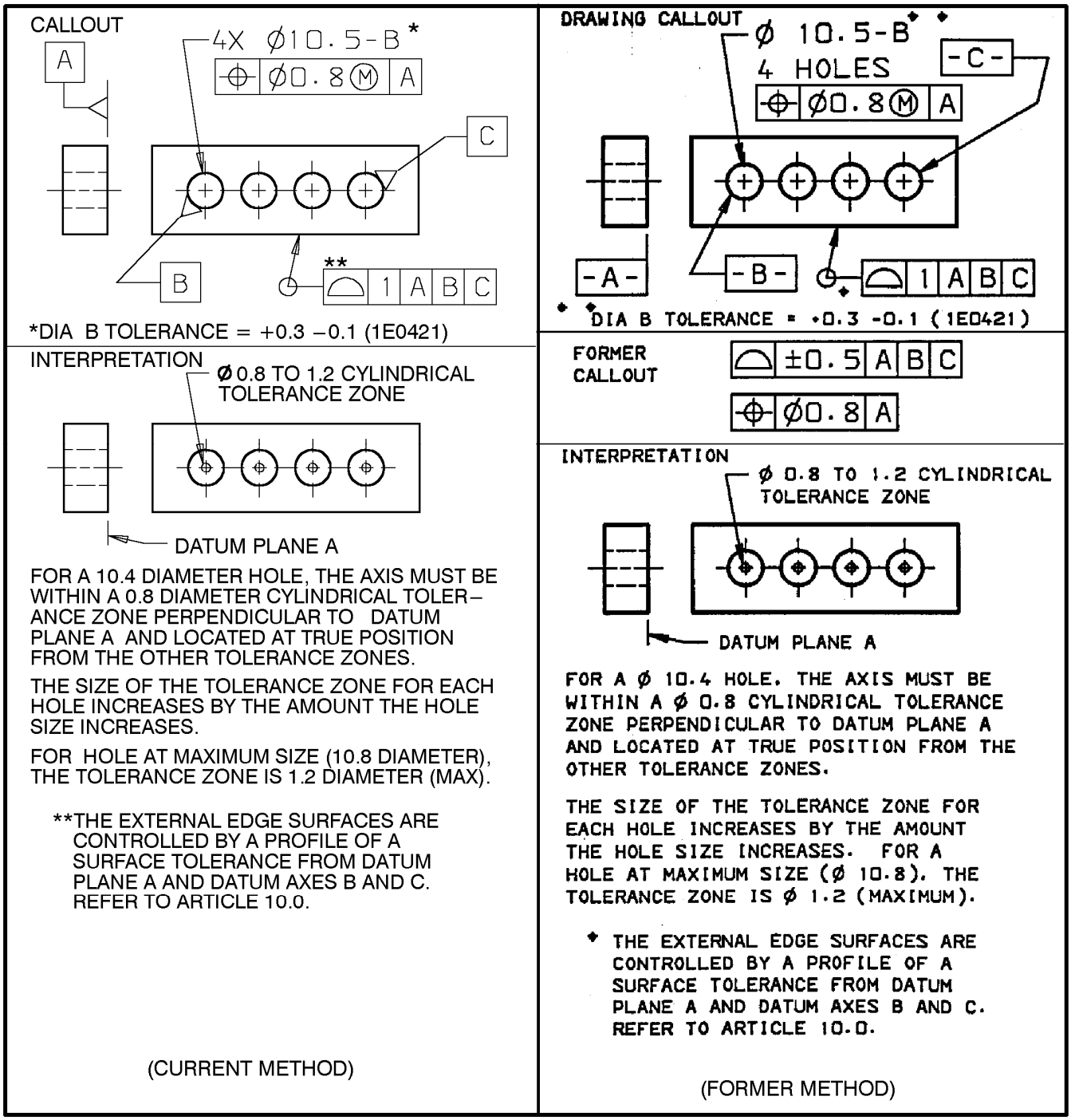


Figure 77 - Positional Tolerance

INTERPRETATION - GEOMETRIC TOLERANCE	DATE	CHG NO	NUMBER
	26 MAR 2013	10	1E0012

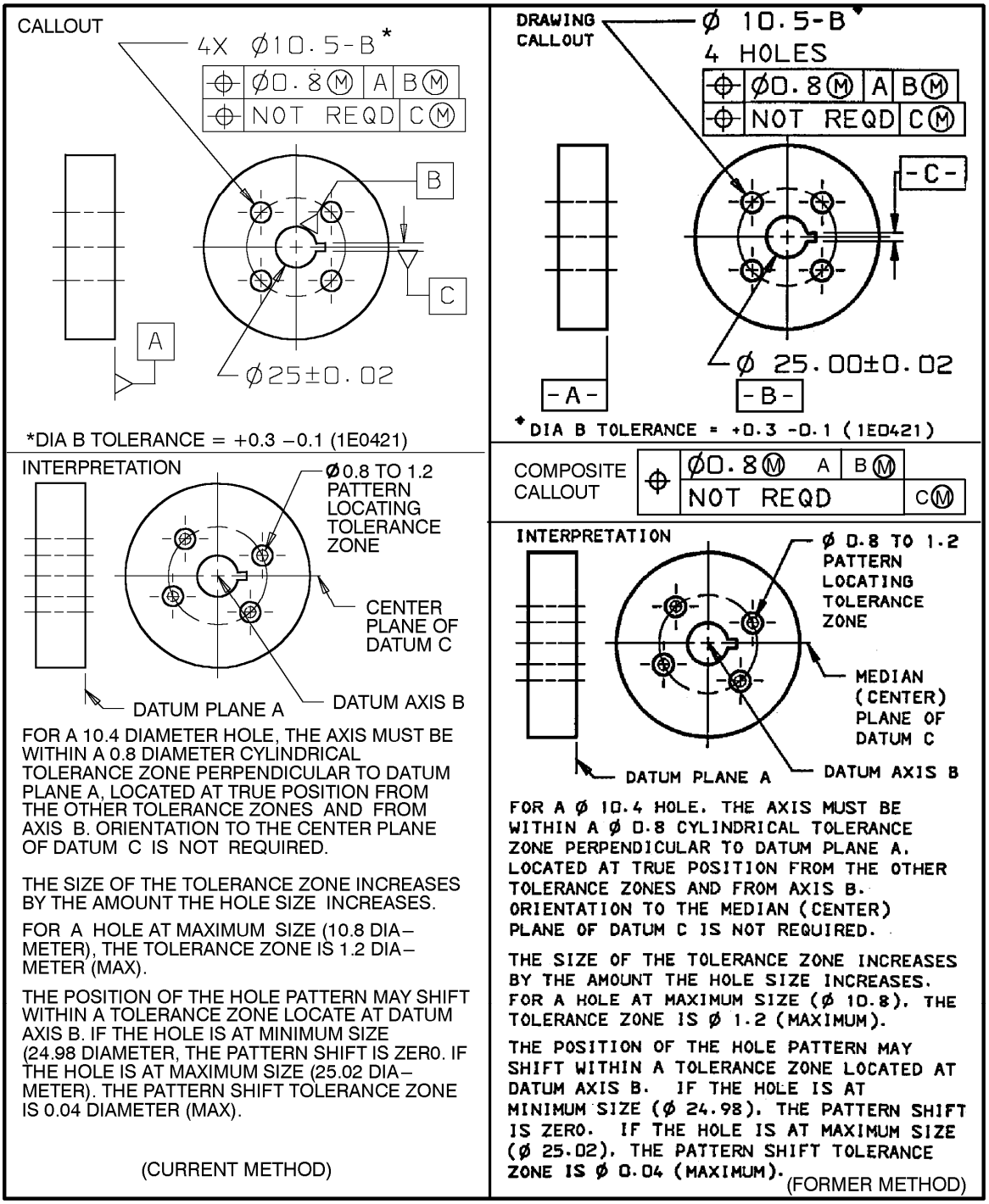


Figure 78 - Multiple Segment Positional Tolerance (Control Not Req)

INTERPRETATION - GEOMETRIC TOLERANCE	DATE	CHG NO	NUMBER
	26 MAR 2013	10	1E0012

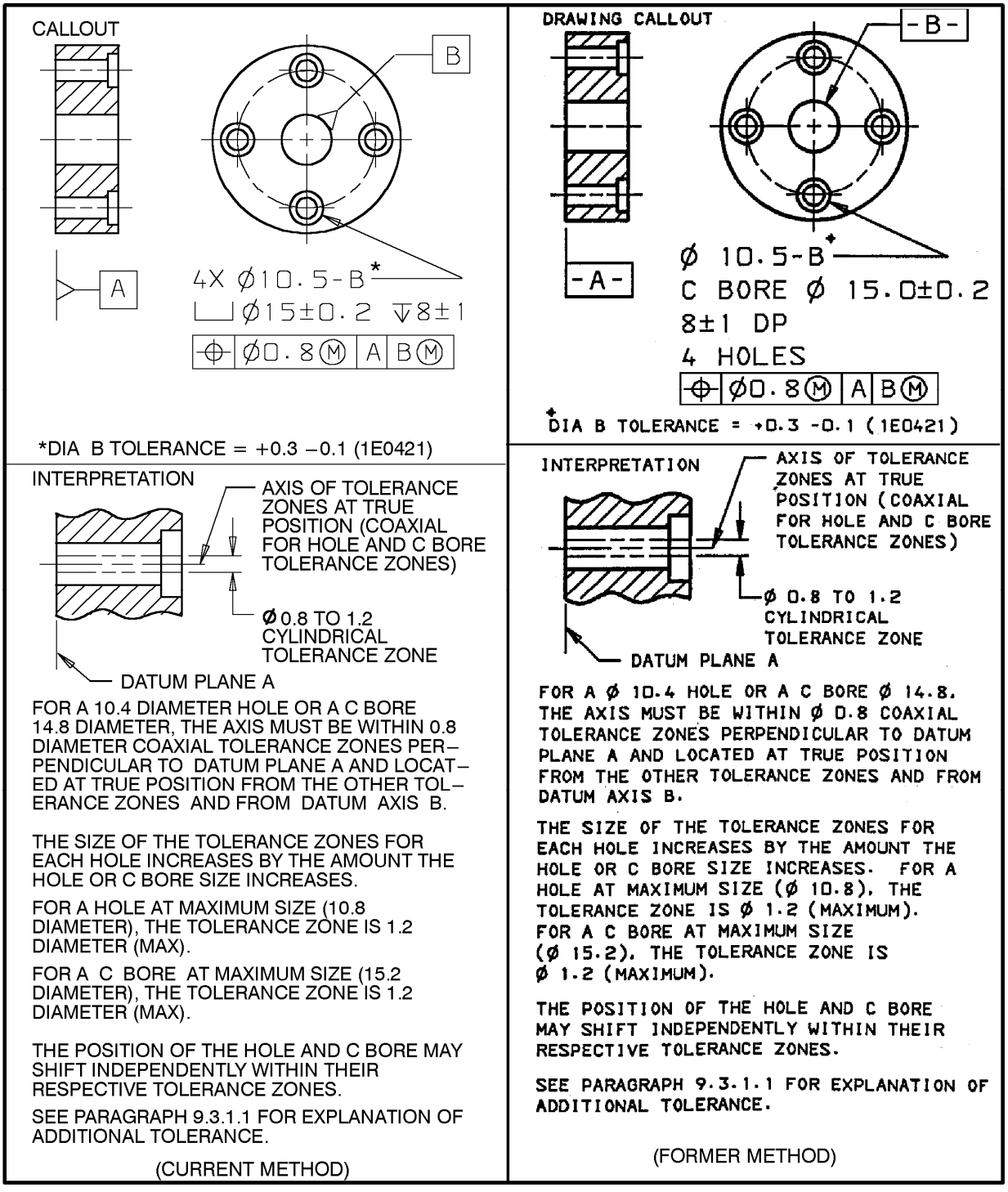


Figure 79 - Coaxial Features (Same Positional Tolerance)

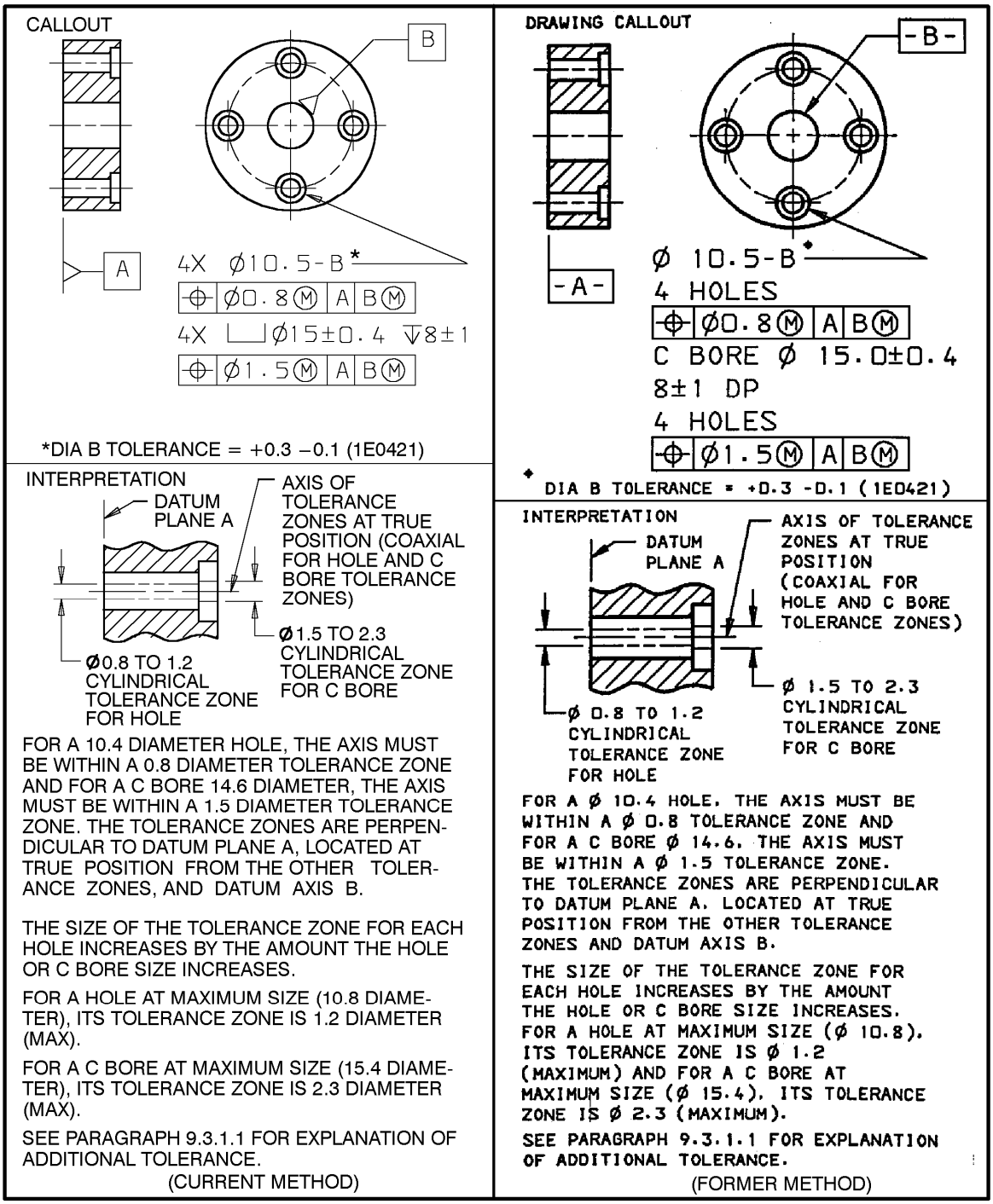
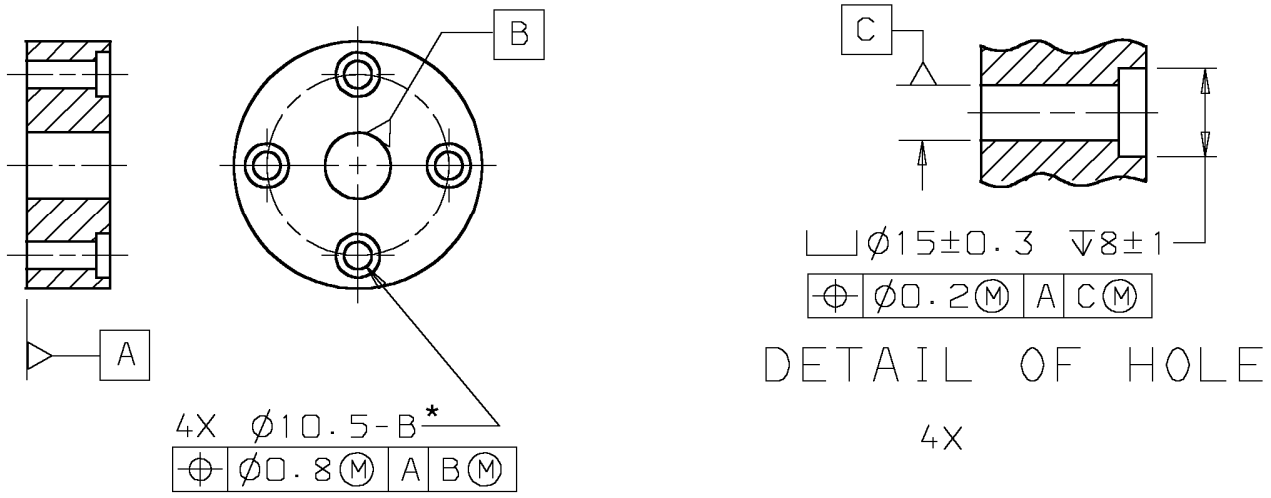


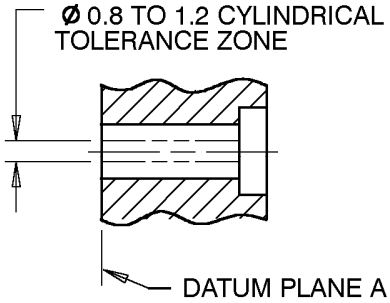
Figure 80 - Coaxial Features (Same Datums Different Tolerances)

CALLOUT



*DIA B TOLERANCE = +0.3 −0.1 (1E0421)

INTERPRETATION

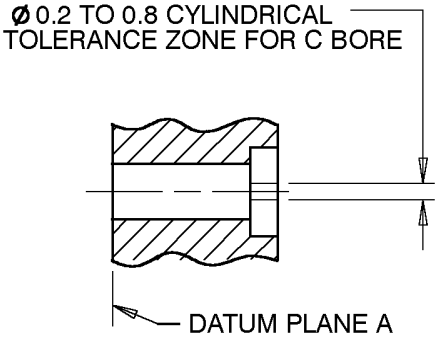


FOR A 10.4 DIAMETER HOLE, THE AXIS MUST BE WITHIN A 0.8 DIAMETER TOLERANCE ZONE THAT IS PERPENDICULAR TO DATUM PLANE A AND LOCATED AT TRUE POSITION FROM THE OTHER TOLERANCE ZONES AND FROM DATUM AXIS B.

THE SIZE OF THE TOLERANCE ZONE FOR EACH HOLE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES.

FOR A HOLE AT MAXIMUM SIZE (10.8 DIAMETER) THE TOLERANCE ZONE IS 1.2 DIAMETER (MAX).

SEE PARAGRAPH 9.3.1.1 FOR EXPLANATION OF ADDITIONAL TOLERANCE.



FOR A C BORE 1.47 DIAMETER, THE AXIS MUST BE WITHIN A 0.2 DIAMETER TOLERANCE ZONE THAT IS PERPENDICULAR TO DATUM PLANE A AND LOCATED AT TRUE POSITION FROM DATUM AXIS C. THE TOLERANCE ZONE AND DATUM AXIS C ARE ESTABLISHED FOR EACH OF THE FOUR HOLES.

THE SIZE OF THE TOLERANCE ZONE INCREASES BY THE AMOUNT THE C BORE SIZE INCREASES.

FOR A C BORE AT MAXIMUM SIZE (15.3 DIAMETER) THE TOLERANCE ZONE IS 0.8 DIAMETER (MAX).

(CURRENT METHOD)

Figure 81 - Coaxial Features (Different Tolerances And Datums) (Continued on Next Page)

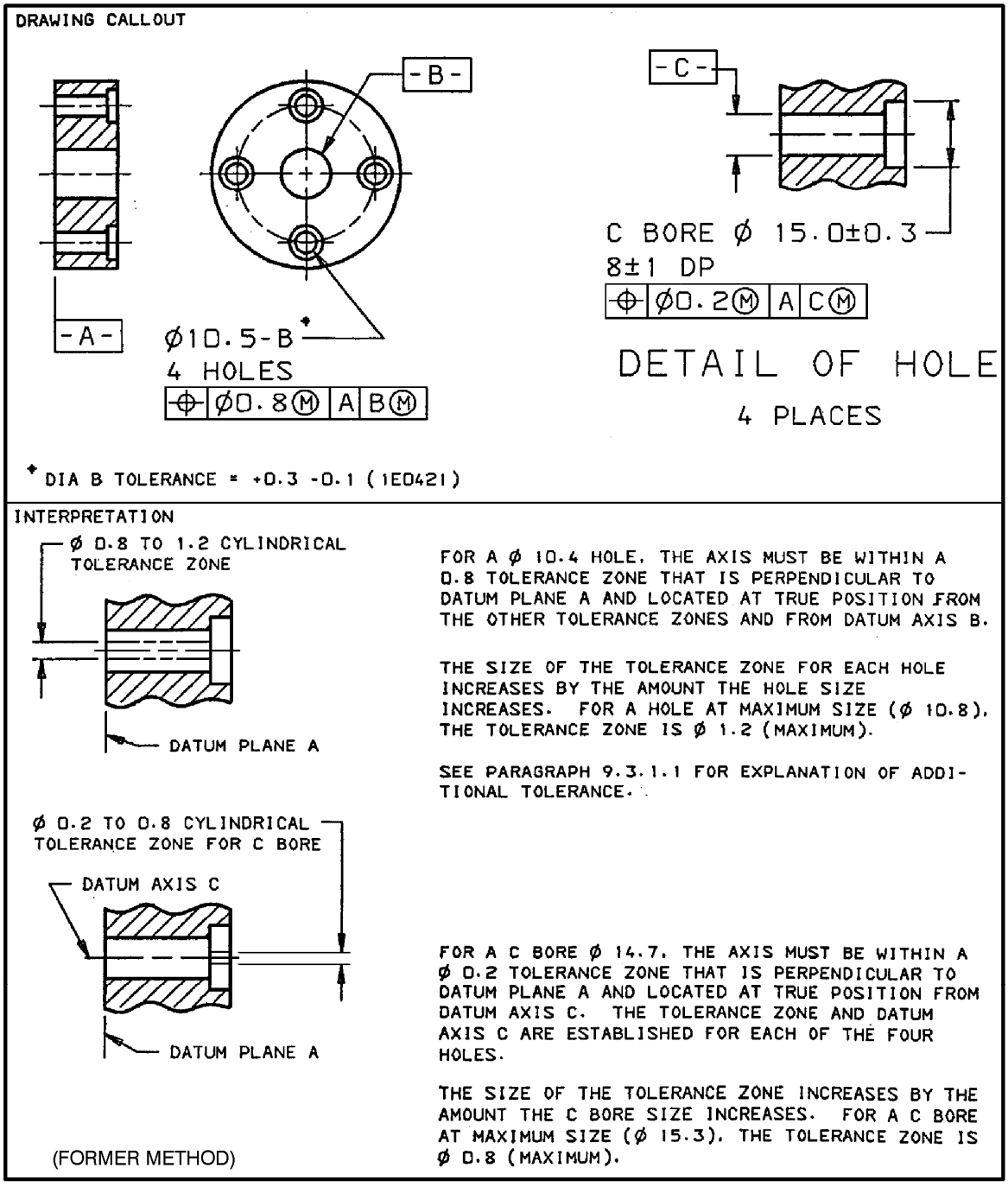


Figure 81 - Coaxial Features (Different Tolerance And Datums) (Continued)

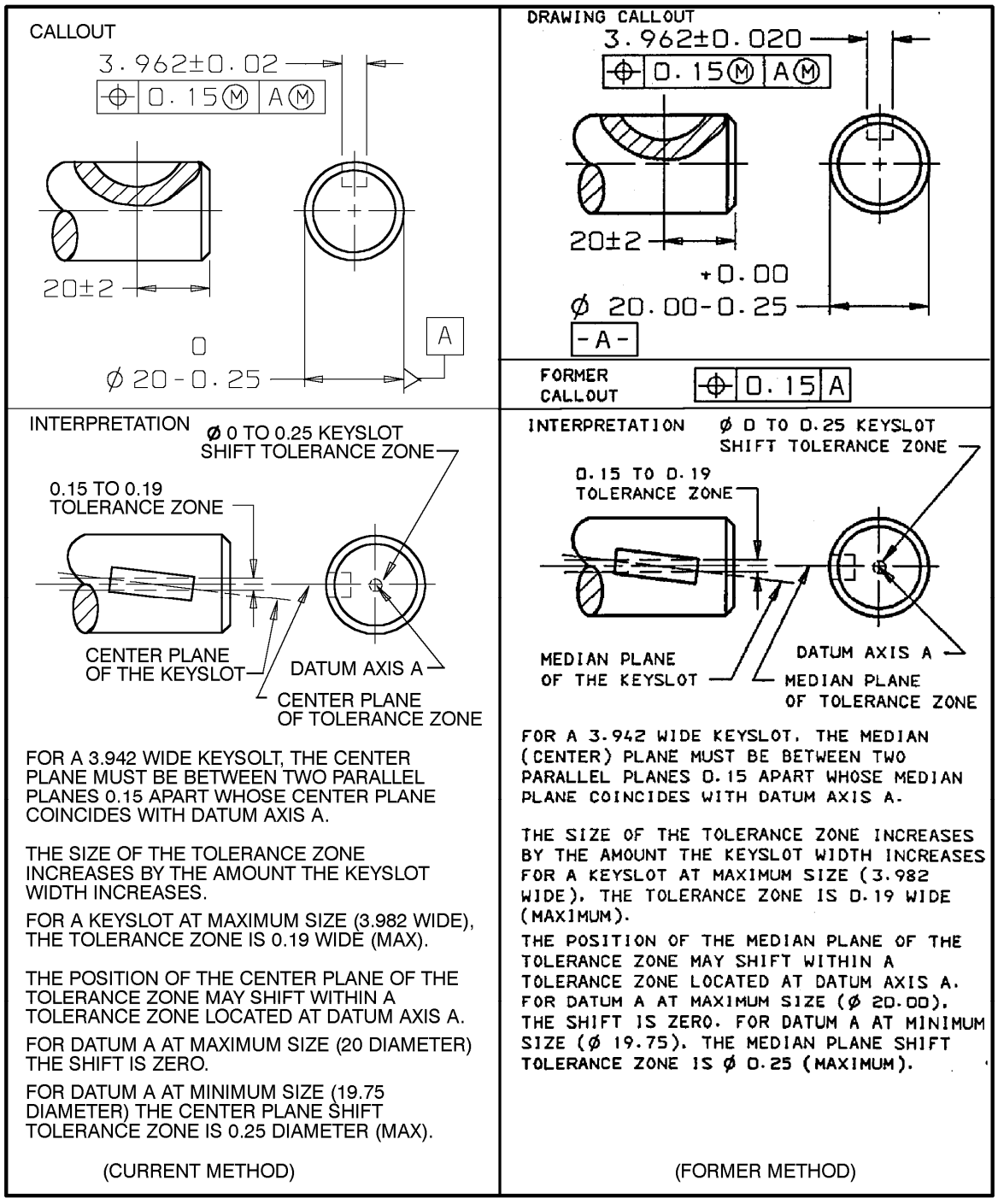


Figure 82 - Positional Tolerance Non-Circular Feature

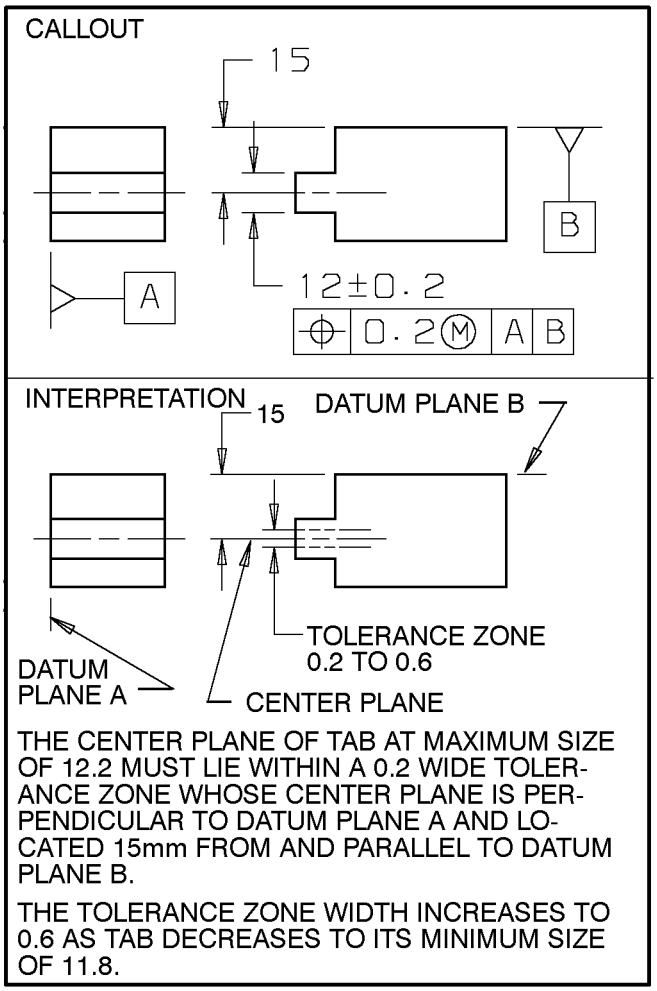


Figure 83 - Positional Tolerance Non-Circular Feature

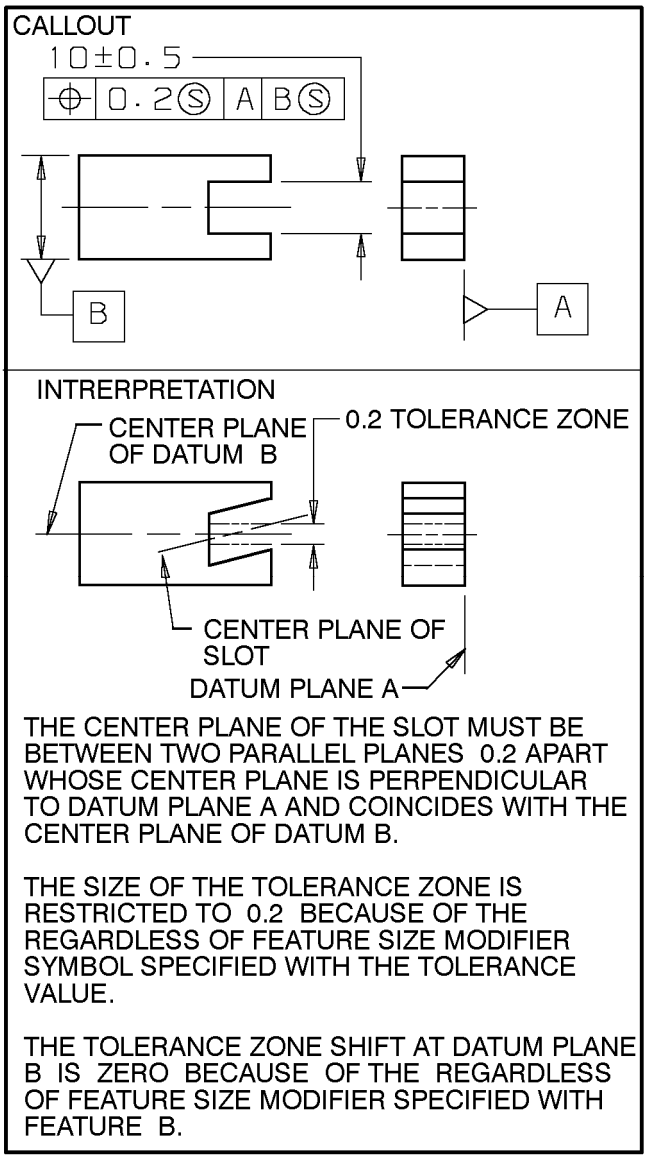


Figure 84 - Positional Tolerance Symmetrical Feature

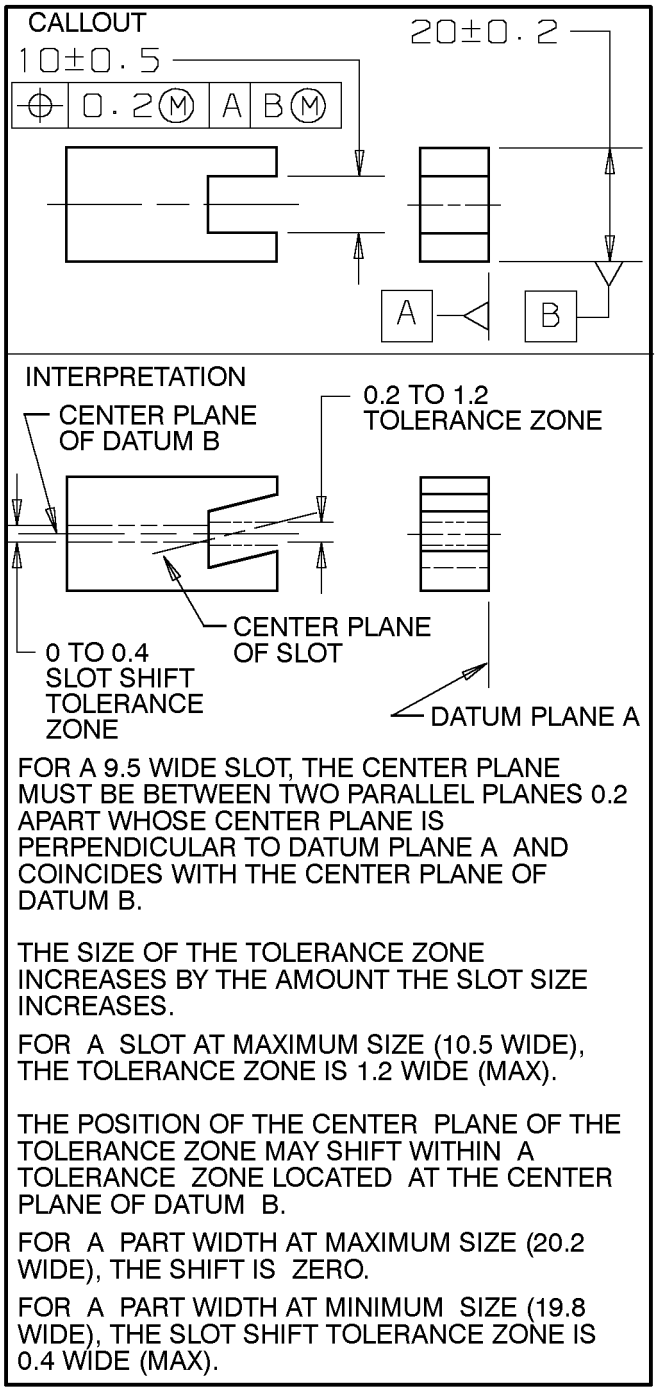


Figure 85 - Positional Tolerance Symmetrical Feature (Variable Tolerance)

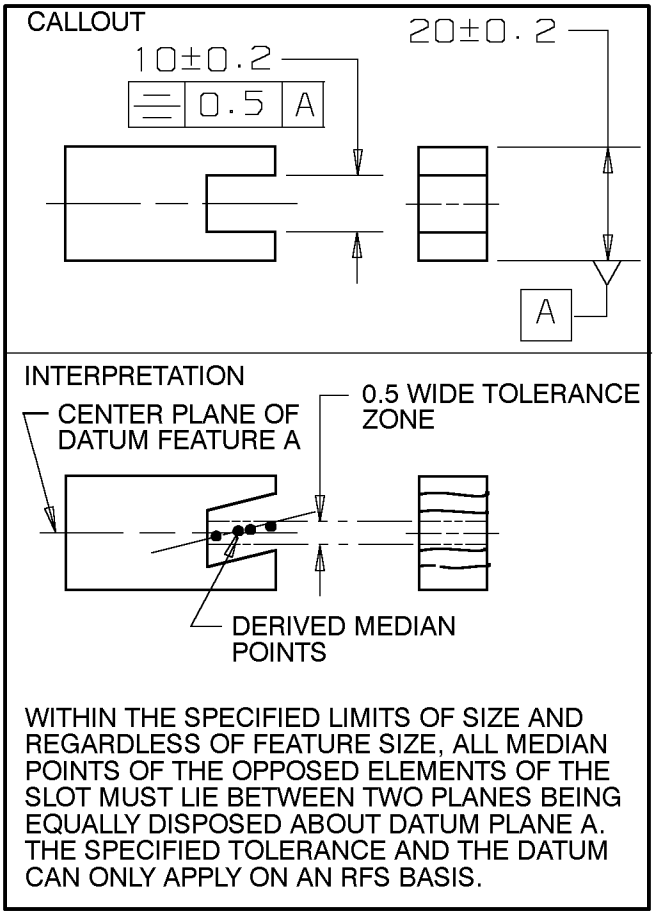


Figure 86 - Symmetry

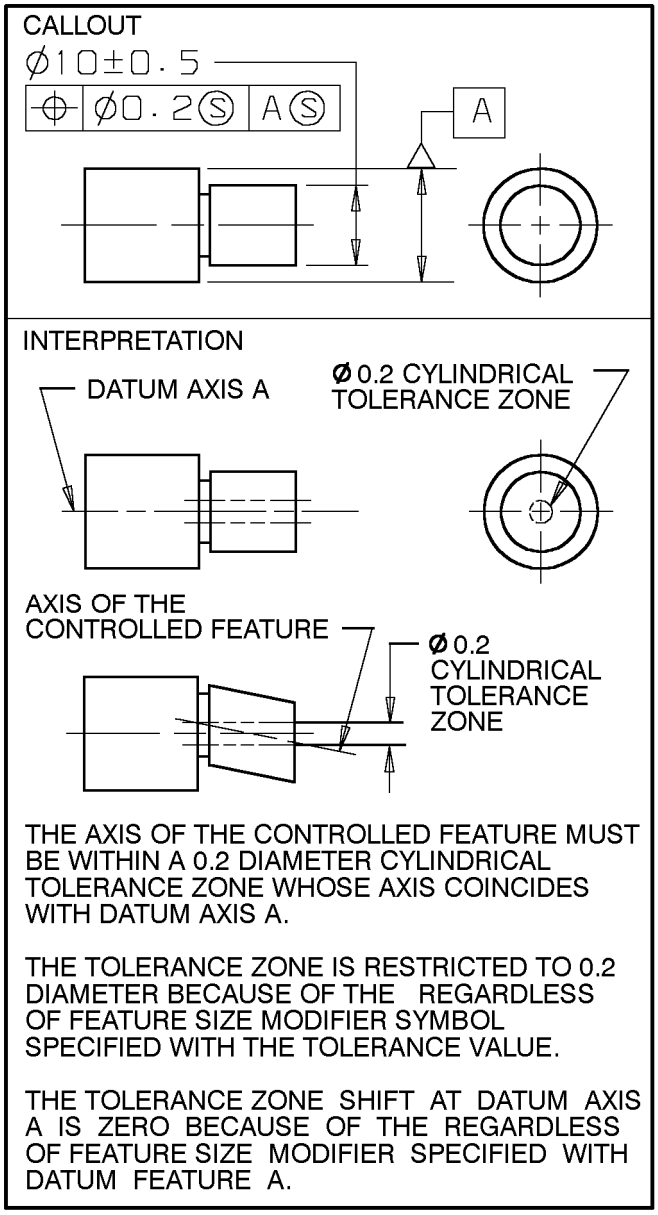


Figure 87 - Positional Tolerance Concentric Features

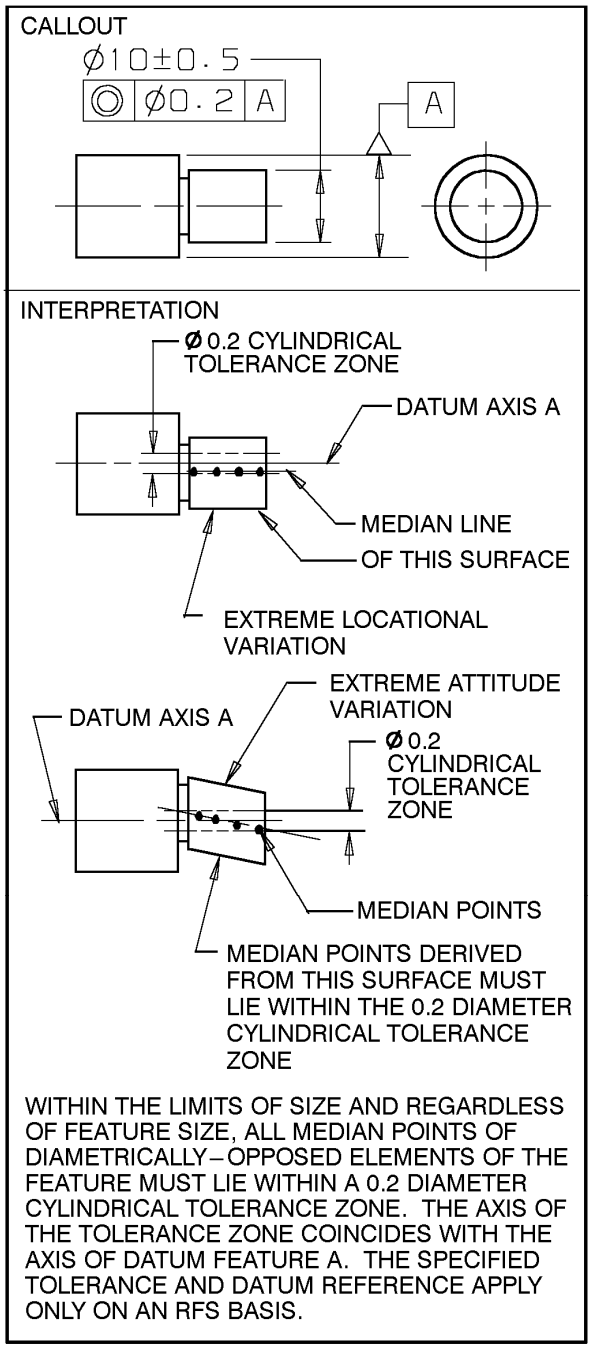


Figure 88 - Concentricity

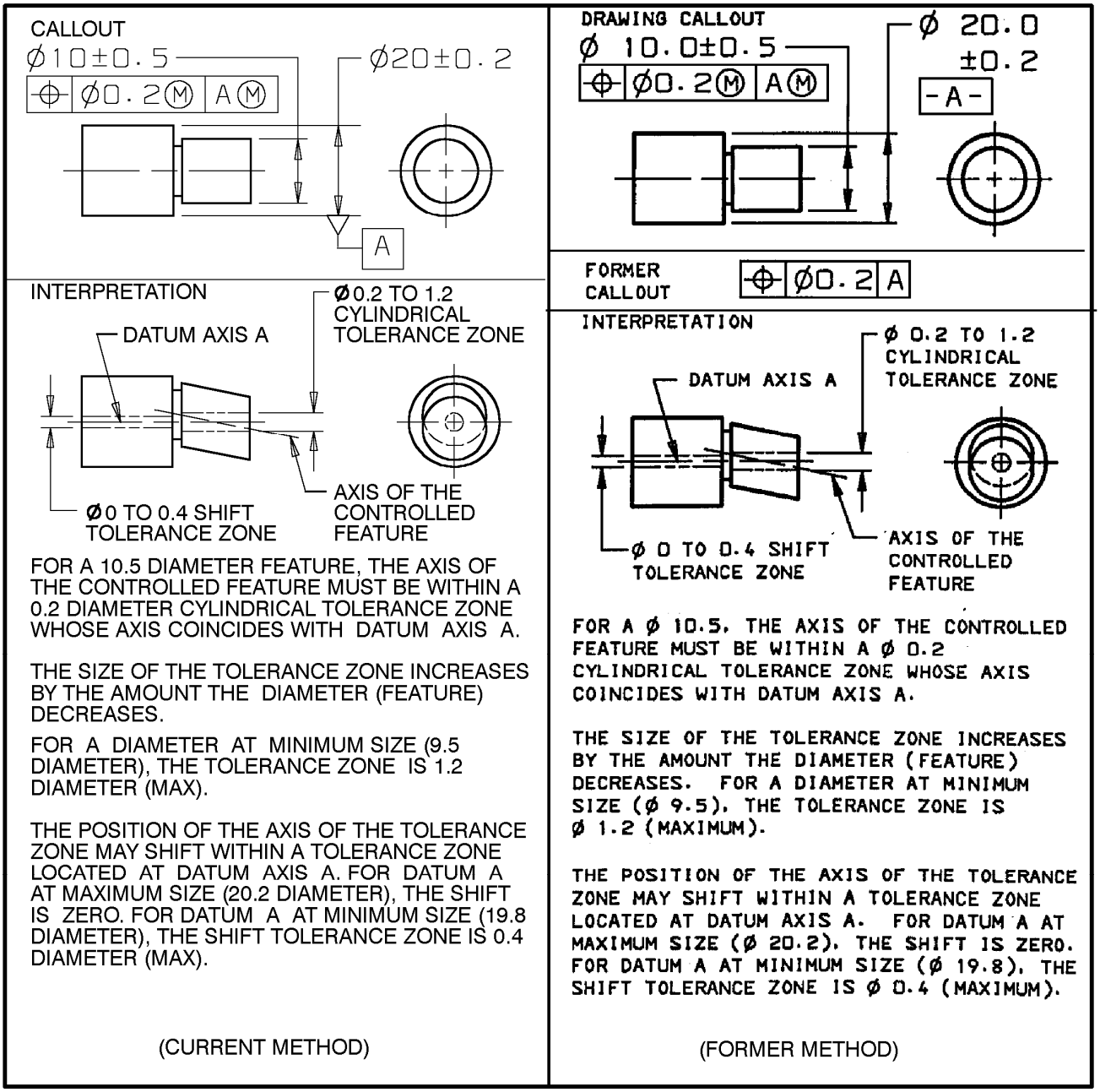
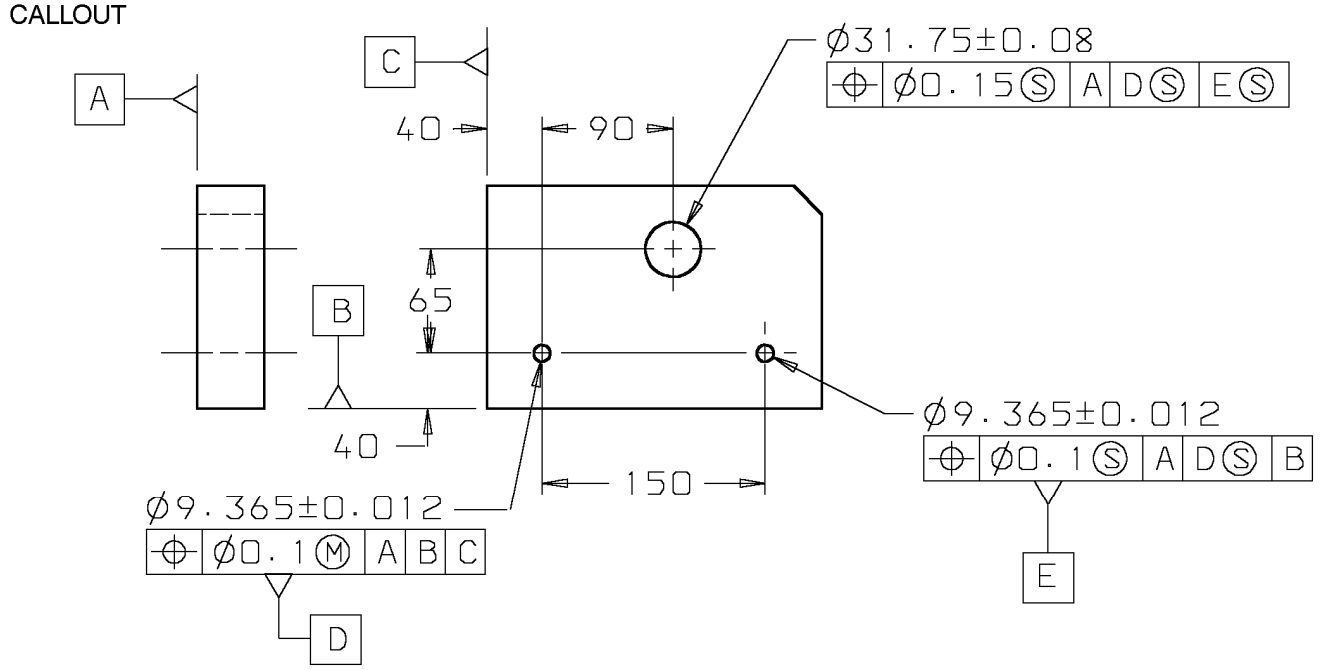


Figure 89 - Positional Tolerance Concentric Features (Increased Tolerance)

CALLOUT



INTERPRETATION

FOR A 9.353 DIAMETER HOLE (DATUM FEATURE D), THE AXIS MUST BE WITHIN A 0.1 DIAMETER CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A, LOCATED AT TRUE POSITION FROM DATUM FEATURES B AND C BY THE 40mm DIMENSIONS.

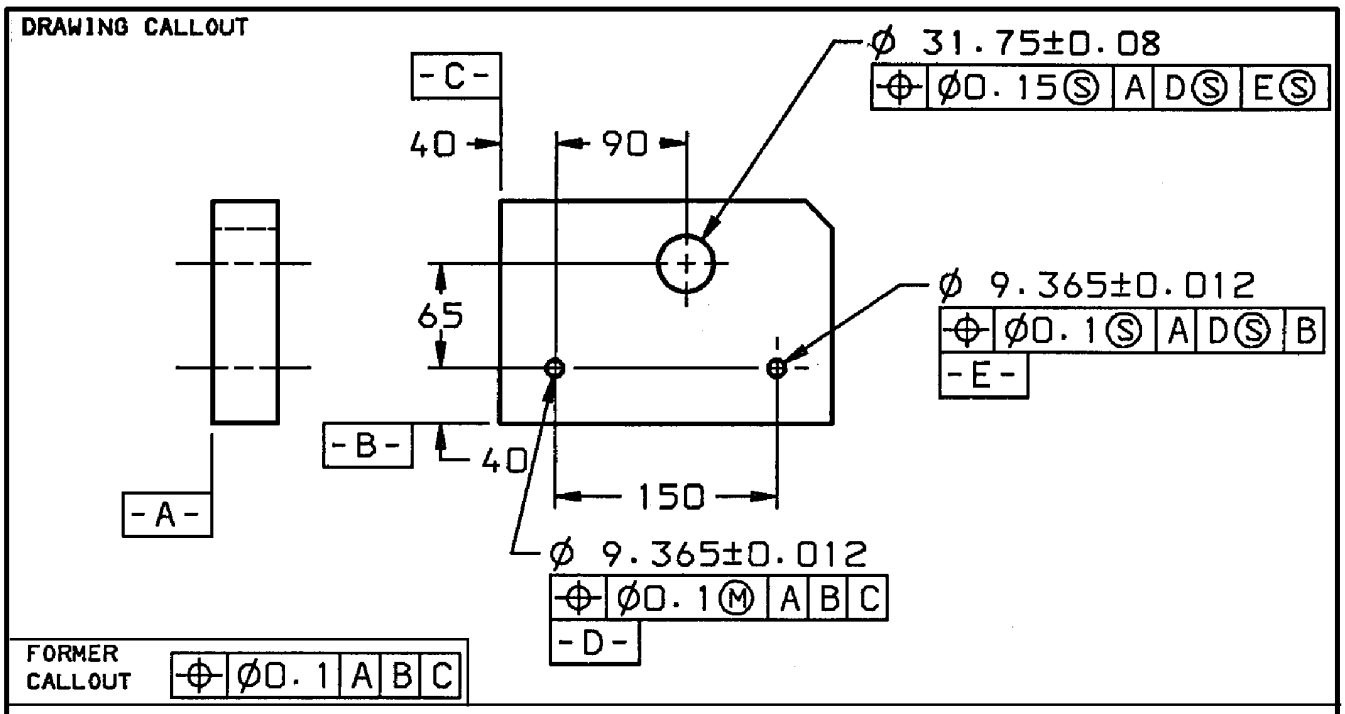
THE SIZE OF THE TOLERANCE ZONE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES. FOR A HOLE AT MAXIMUM SIZE (9.377 DIAMETER), THE TOLERANCE ZONE IS 0.124 DIAMETER (MAX).

FOR THE 9.365 DIAMETER HOLE (DATUM FEATURE E), THE AXIS MUST BE WITHIN A 0.1 DIAMETER CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A, LOCATED BY TRUE POSITION FROM DATUM AXIS D (150mm DIMENSION), AND ORIENTED BY DATUM PLANE B. THE TOLERANCE ZONE SHIFT AT DATUM FEATURE D IS RESTRICTED TO ZERO BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL SPECIFIED WITH DATUM FEATURE D.

FOR A 31.67 DIAMETER HOLE, THE AXIS MUST BE WITHIN A 0.15 DIAMETER CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A, LOCATED AT TRUE POSITION FROM DATUM FEATURE D BY THE 65 AND 90mm DIMENSIONS, AND ORIENTED FROM DATUM FEATURE E. THE TOLERANCE ZONE DIAMETER IS RESTRICTED TO 0.15 DIAMETER BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL SPECIFIED WITH THE TOLERANCE VALUE. THE TOLERANCE ZONE SHIFT AT DATUM AXIS D (AND E) IS ZERO BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL SPECIFIED IN CONJUNCTION WITH DATUM FEATURE D (AND E).

(CURRENT METHOD)

Figure 90 - Dowel Holes As Datum Features (Continued on Next Page)



INTERPRETATION

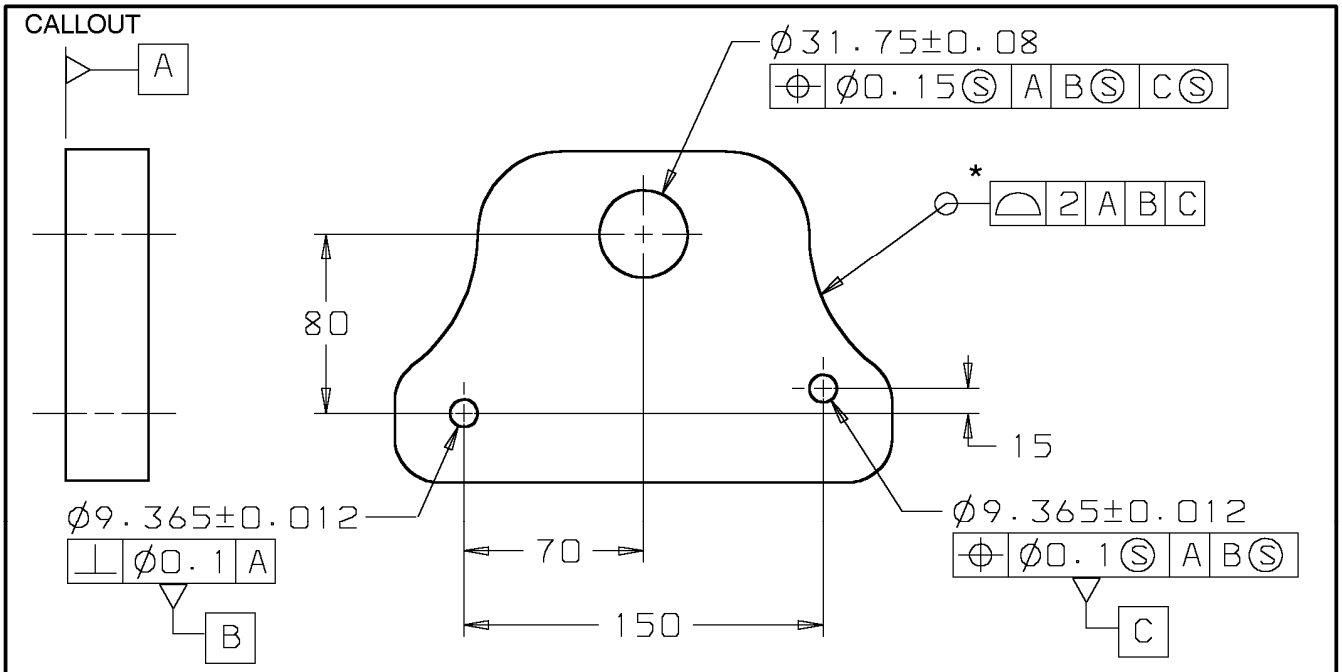
FOR A Ø 9.353 HOLE (DATUM FEATURE D). THE AXIS MUST BE WITHIN A Ø 0.1 CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A. LOCATED AT TRUE POSITION FROM DATUM FEATURES B AND C BY THE 40mm DIMENSIONS. THE SIZE OF THE TOLERANCE ZONE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES. FOR A HOLE AT MAXIMUM SIZE (Ø 9.377). THE TOLERANCE ZONE IS Ø 0.124 (MAXIMUM).

FOR THE Ø 9.365 HOLE (DATUM FEATURE E). THE AXIS MUST BE WITHIN A Ø 0.1 CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A. LOCATED AT TRUE POSITION FROM DATUM AXIS D (150mm DIMENSION), AND ORIENTED BY DATUM PLANE B. THE TOLERANCE ZONE SHIFT AT DATUM FEATURE D IS RESTRICTED TO ZERO BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL SPECIFIED WITH DATUM FEATURE D.

FOR A Ø 31.67 HOLE. THE AXIS MUST BE WITHIN A Ø 0.15 CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A. LOCATED AT TRUE POSITION FROM DATUM FEATURE D BY THE 65 AND 90mm DIMENSIONS. AND ORIENTED FROM DATUM FEATURE E. THE TOLERANCE ZONE DIAMETER IS RESTRICTED TO Ø 0.15 BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL SPECIFIED WITH THE TOLERANCE VALUE. THE TOLERANCE ZONE SHIFT AT DATUM AXIS D (AND E) IS ZERO BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL SPECIFIED IN CONJUNCTION WITH DATUM FEATURE D (AND E).

(FORMER METHOD)

Figure 90 - Dowel Holes As Datum Features (Continued)



INTERPRETATION

FOR A 9.365 DIAMETER HOLE (DATUM FEATURE B), THE AXIS IS CONTROLLED BY A PERPENDICULARITY REQUIREMENT FROM DATUM PLANE A.

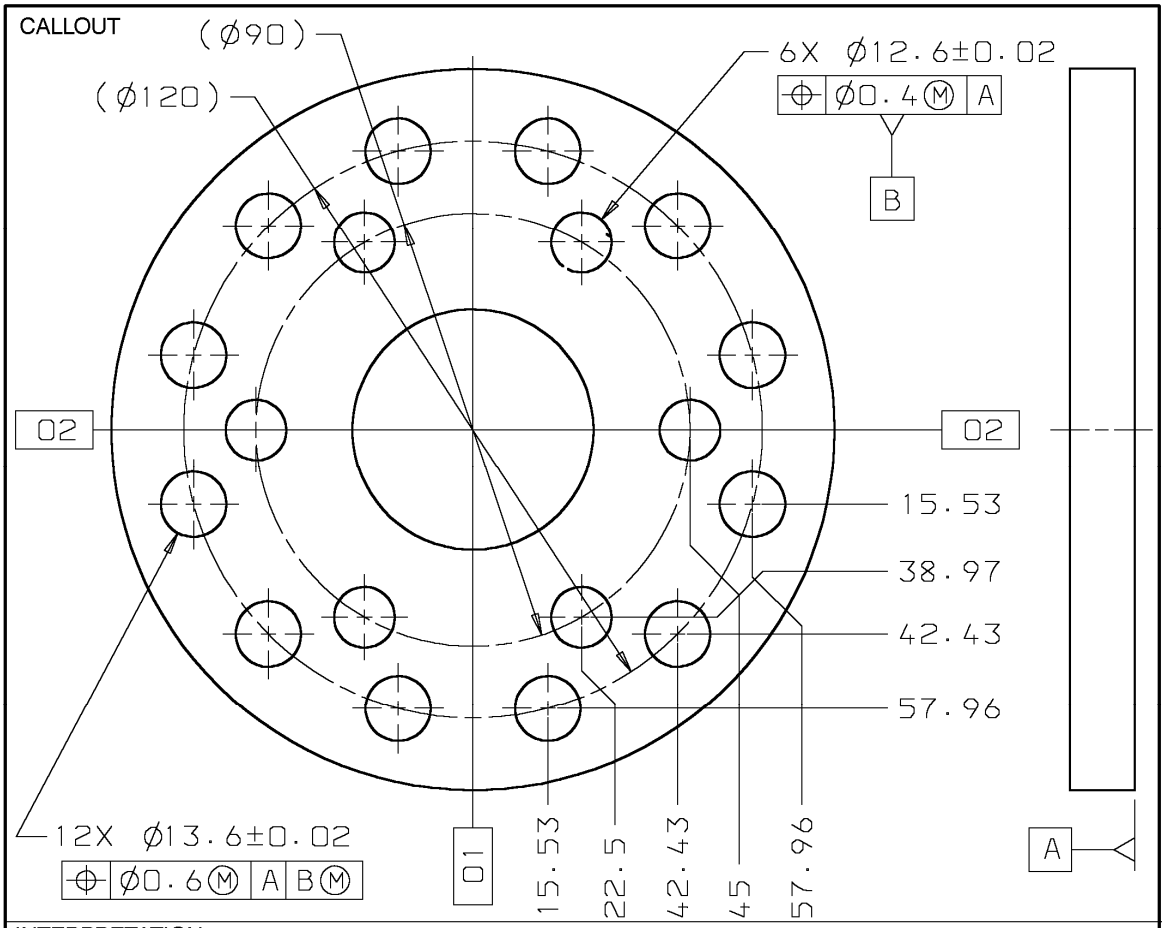
FOR A 9.365 DIAMETER HOLE (DATUM FEATURE C), THE AXIS MUST BE WITHIN A 0.1 DIAMETER CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A, LOCATED AT TRUE POSITION FROM DATUM AXIS B BY THE 15 AND 150mm DIMENSIONS.

THE SIZE OF THE TOLERANCE ZONE IS RESTRICTED TO 0.1 DIAMETER BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL SPECIFIED WITH THE TOLERANCE VALUE. THE TOLERANCE ZONE SHIFT AT DATUM AXIS B IS ZERO BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL SPECIFIED WITH DATUM FEATURE B.

FOR THE 31.75 DIAMETER HOLE, THE AXIS MUST BE WITHIN A 0.15 DIAMETER CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A, LOCATED AT TRUE POSITION FROM DATUM FEATURE B BY THE 70 AND 80mm DIMENSIONS AND ORIENTED FROM DATUM AXIS C. THE SIZE OF THE TOLERANCE ZONE IS RESTRICTED TO 0.15 DIAMETER BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL SPECIFIED WITH THE TOLERANCE VALUE. THE TOLERANCE ZONE SHIFT AT DATUM AXIS B (AND C) IS ZERO BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL SPECIFIED WITH DATUM FEATURES B (AND C).

*THE PROFILE OF A SURFACE TOLERANCE SYMBOL SPECIFIES THE TOLERANCE ZONE FOR ALL EXTERNAL EDGE SURFACES LOCATED FROM DATUM FEATURES A, B, AND C. DIMENSIONS DEFINING THE SURFACE HAVE BEEN OMITTED. REFER TO ARTICLE 10.0.

Figure 91 - Dowel Holes As Datum Features



INTERPRETATION

FOR THE SIX 12.6±0.02 DIAMETER HOLES (DATUM FEATURES B), THE AXIS OF EACH HOLE MUST BE WITHIN A 0.4 DIAMETER CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A AND LOCATED BY BASIC DIMENSIONS RELATIVE TO EACH OTHER. THE SIZE OF THE TOLERANCE ZONE FOR EACH HOLE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES. FOR A HOLE AT MAXIMUM SIZE (12.62 DIAMETER), THE TOLERANCE ZONE IS 0.44 DIAMETER (MAX). DATUM B IS ESTABLISHED AS THE CENTRAL DATUM AXIS FOR THE PATTERN OF HOLES BY FIRST ESTABLISHING INDIVIDUAL DATUM AXES PERPENDICULAR TO DATUM A AT THE TRUE POSITIONS OF EACH HOLE FROM TRUE CYLINDERS WHICH SIMULATE THE VIRTUAL CONDITION OF THE HOLES (MMC SIZE – POSITION TOLERANCE). WHEN THE PART IS MOUNTED ON THE PRIMARY DATUM SURFACE (DATUM A), THE PATTERN OF HOLES EATABLISHES THE SECOND AND THIRD DATUM PLANES OF THE DATUM REFERENCE FRAME. WHERE THE PATTERN OF FEATURES IS THE SECONDARY DATUM IN THE FEATURE CONTROL FRAME AND REFERENCED AT MMC, THE AXIS OF THE FEATURE PATTERN ESTABLISHED BY ALL OF THE HOLES MAY DEPART FROM THE AXIS OF THE DATUM REFERENCE FRAME AS THE DATUM FEATURE DEPARTS FROM MMC.

FOR THE TWELVE 13.6±0.02 DIAMETER HOLES, THE AXIS OF EACH HOLE MUST BE WITHIN A 0.6 DIAMETER CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM A AND LOCATED BY BASIC DIMENSIONS FROM DATUM B. THE SIZE OF THE TOLERANCE ZONE FOR EACH HOLE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES. FOR A HOLE AT MAXIMUM SIZE (13.62 DIAMETER), THE TOLERANCE ZONE IS 0.64 DIAMETER (MAX). (CURRENT METHOD)

Figure 92 - Pattern of Holes As Single Datum (MMC) (Continued on Next Page)

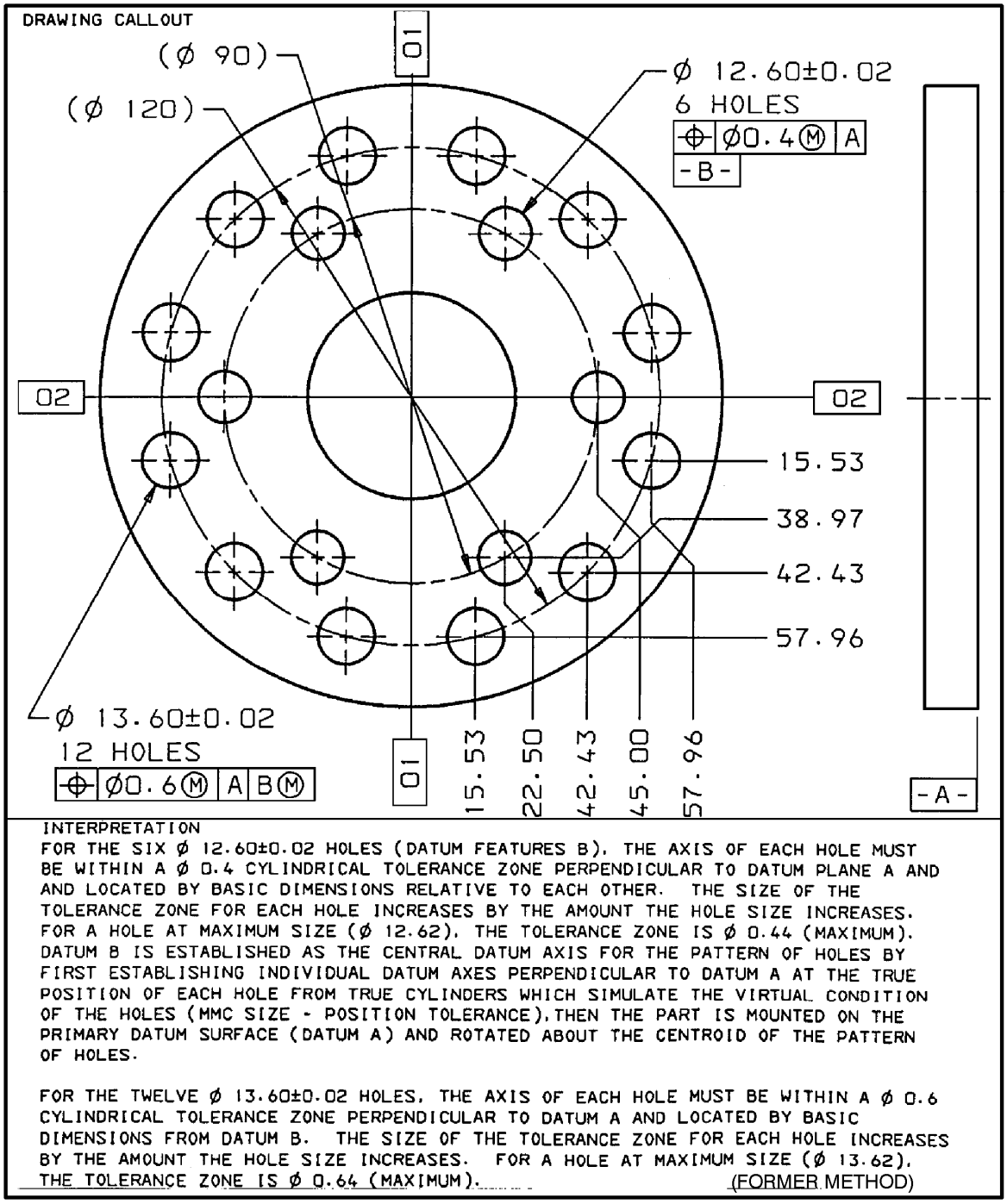
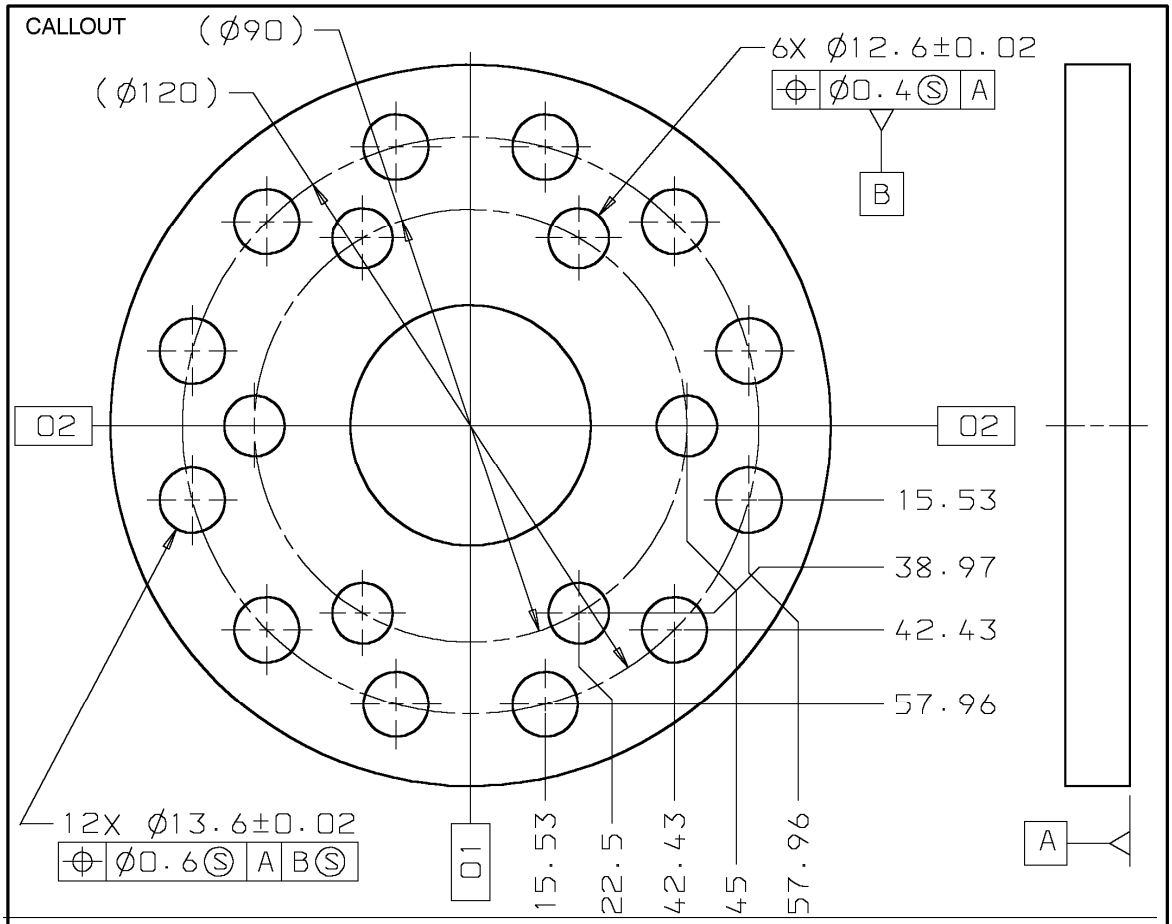


Figure 92 - Pattern Of Holes As Single Datum (MMC) (Continued)



INTERPRETATION

FOR THE SIX 12.6±0.02 DIAMETER HOLES (DATUM FEATURES B), THE AXIS OF EACH HOLE MUST BE WITHIN A 0.4 DIAMETER CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM PLANE A AND LOCATED BY DIMENSIONS RELATIVE TO EACH OTHER. THE TOLERANCE ZONE IS RESTRICTED TO 0.4 DIAMETER BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL. DATUM B IS ESTABLISHED AS THE CENTRAL DATUM AXIS FOR THE PATTERN OF HOLES BY FIRST ESTABLISHING INDIVIDUAL DATUM AXES PERPENDICULAR TO DATUM A FOR EACH HOLE REGARDLESS OF FEATURE SIZE, THEN THE PART IS MOUNTED ON THE PRIMARY DATUM SURFACE (DATUM A) AND ROTATED ABOUT THE CENTROID OF THE PATTERN OF HOLES (CENTROID DETERMINED BY BEST FIT METHOD FROM DATUM AXES OF HOLES). THE PATERN OF HOLES ESTABLISH THE SECOND AND THIRD PLANES OF THE DATUM REFERENCE PLANE.

WHERE THE PATTERN OF FEATURES IS THE SECONDARY DATUM IN THE FEATURE CONTROL FRAME AND REFERENCED AT RFS, THE AXIS OF THE FEATURE PATTERN ESTABLISHED BY ALL OF THE HOLES IS NOT PERMITTED TO DEPART FROM THE AXIS OF THE DATUM REFERENCE FRAME AS THE FEATURE DEPARTS FROM MMC.

FOR THE TWELVE 13.6±0.02 DIAMETER HOLES, THE AXIS OF EACH HOLE MUST BE WITHIN A 0.6 DIAMETER CYLINDRICAL TOLERANCE ZONE PERPENDICULAR TO DATUM A AND LOCATED BY BASIC DIMENSIONS FROM DATUM B. THE TOLERANCE ZONE IS RESTRICTED TO 0.6 DIAMETER BECAUSE OF THE REGARDLESS OF FEATURE SIZE MODIFIER SYMBOL.

(CURRENT METHOD)

Figure 93 - Pattern Of Holes As Single Datum (RFS) (Continued on Next Page)

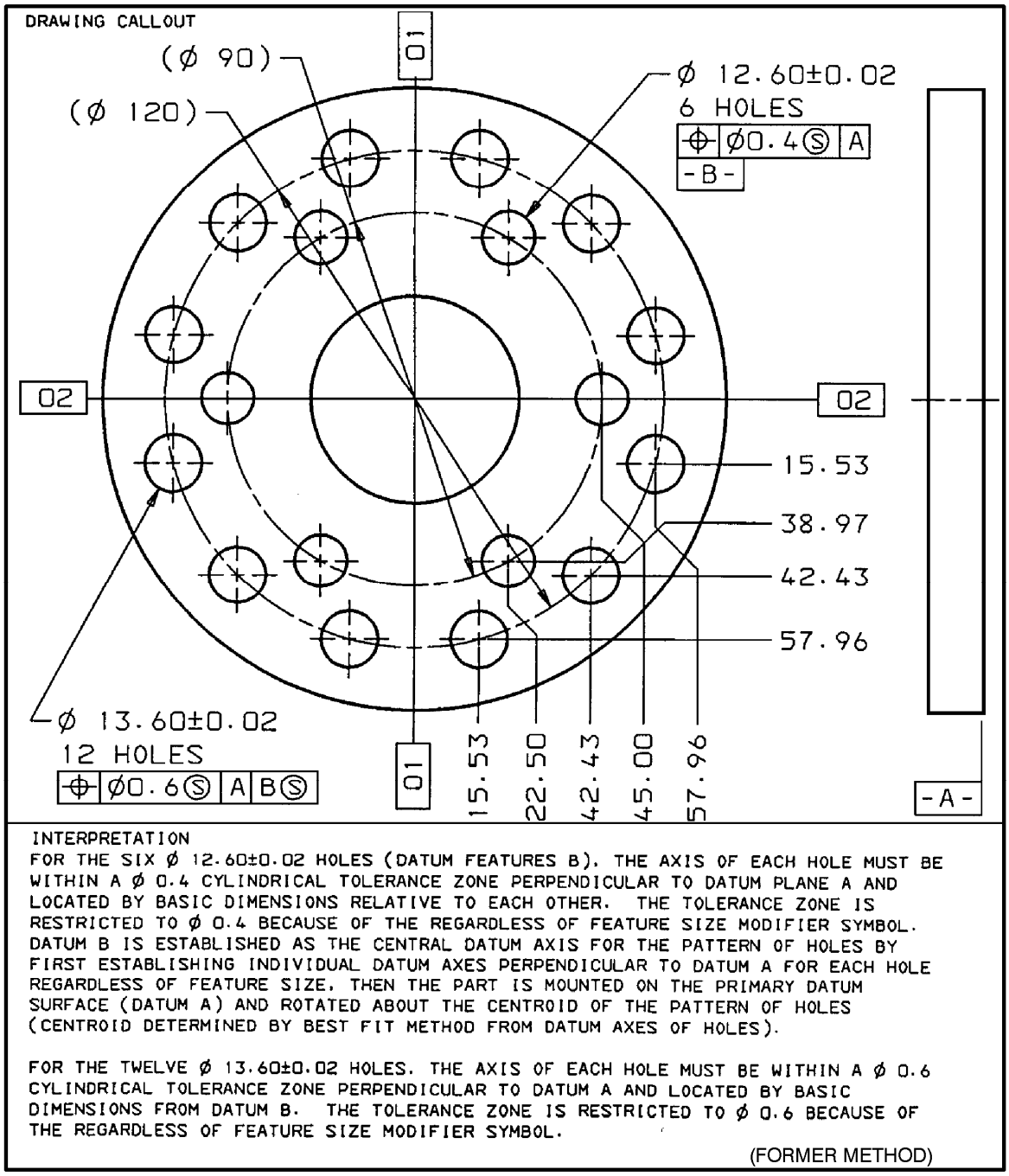
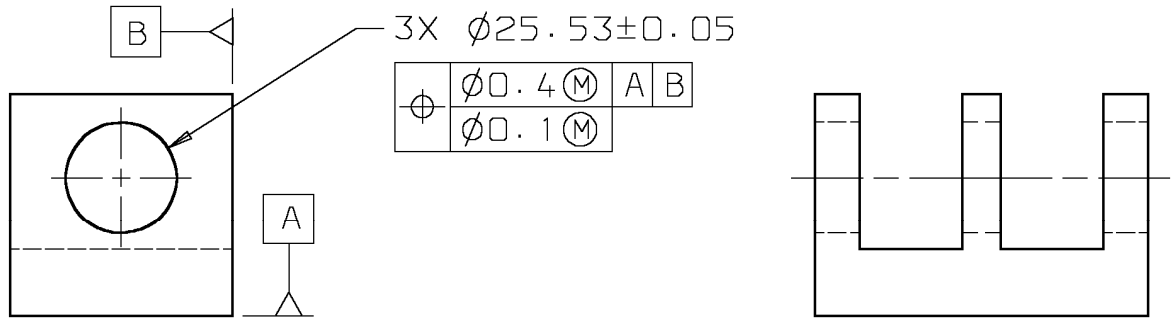
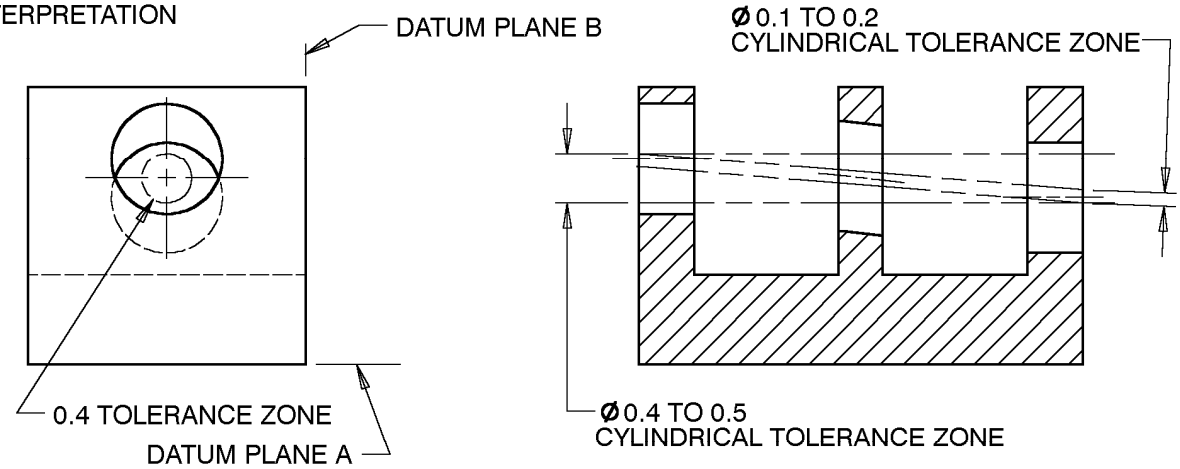


Figure 93 - Pattern Of Holes As Single Datum (RFS) (Continued)

CALLOUT



INTERPRETATION



FOR A 25.48 DIAMETER (3 COAXIAL HOLES), THE AXES MUST SIMULTANEOUSLY MEET THE FOLLOWING CONDITIONS:

a. THE AXES MUST BE WITHIN THE 0.4 DIAMETER CYLINDRICAL TOLERANCE ZONE LOCATED AT TRUE POSITION FROM DATUM PLANES A AND B AND FROM EACH OTHER.

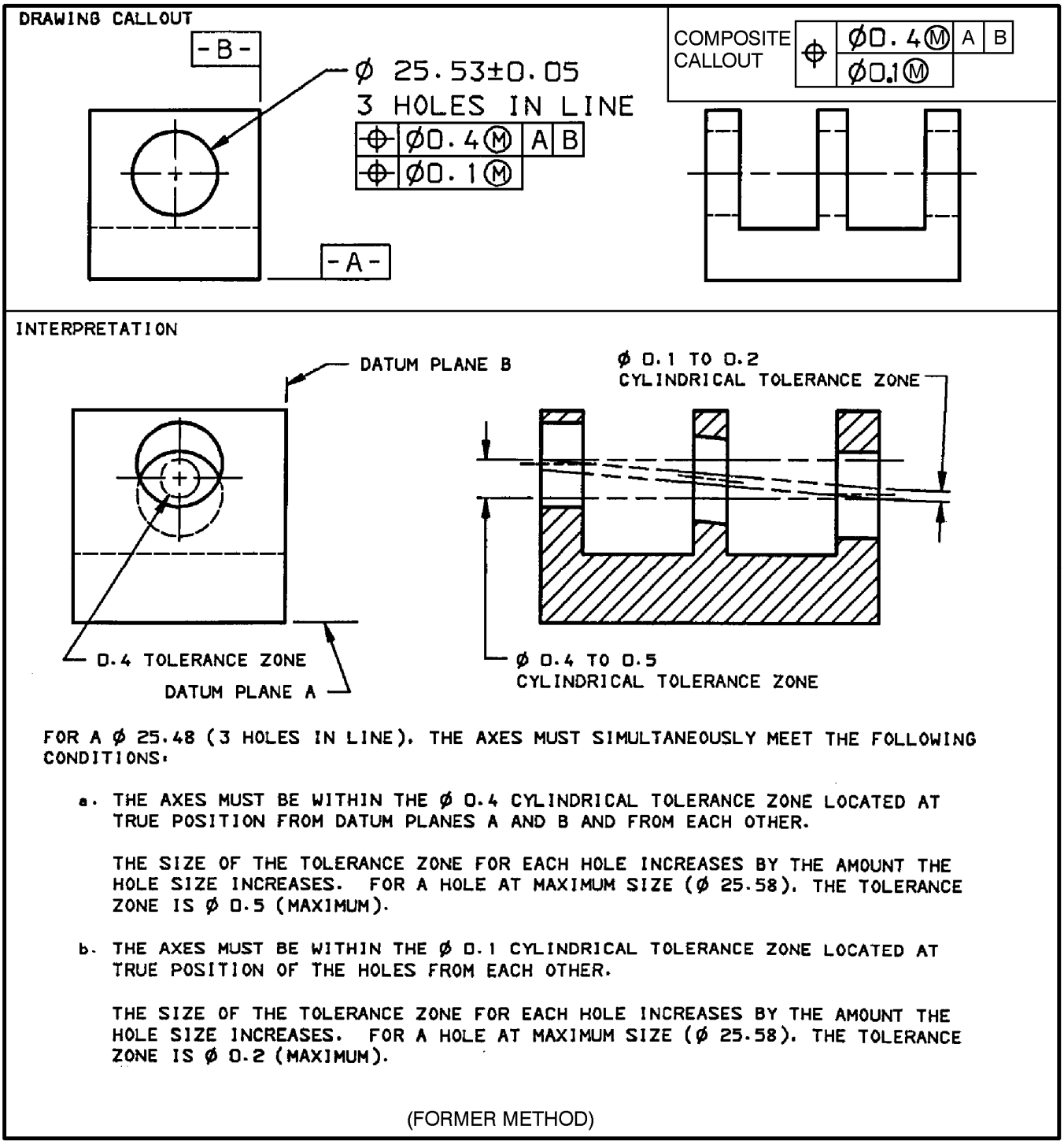
   THE SIZE OF THE TOLERANCE ZONE FOR EACH HOLE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES. FOR A HOLE AT MAXIMUM SIZE (25.58 DIAMETER), THE TOLERANCE ZONE IS 0.5 DIAMETER (MAX).

b. THE AXES MUST BE WITHIN THE 0.1 DIAMETER CYLINCRICAL TOLERANCE ZONE LOCATED AT TRUE POSITION OF THE HOLES FROM EACH OTHER.

   THE SIZE OF THE TOLERANCE ZONE FOR EACH HOLE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES. FOR A HOLE AT MAXIMUM SIZE (25.58 DIAMETER), THE TOLERANCE ZONE IS 0.2 DIAMETER (MAX).

(CURRENT METHOD)

Figure 94 - Coaxial Holes (Same Size) (Continued on Next Page)



FOR A Ø 25.48 (3 HOLES IN LINE), THE AXES MUST SIMULTANEOUSLY MEET THE FOLLOWING CONDITIONS:

a. THE AXES MUST BE WITHIN THE Ø 0.4 CYLINDRICAL TOLERANCE ZONE LOCATED AT TRUE POSITION FROM DATUM PLANES A AND B AND FROM EACH OTHER.

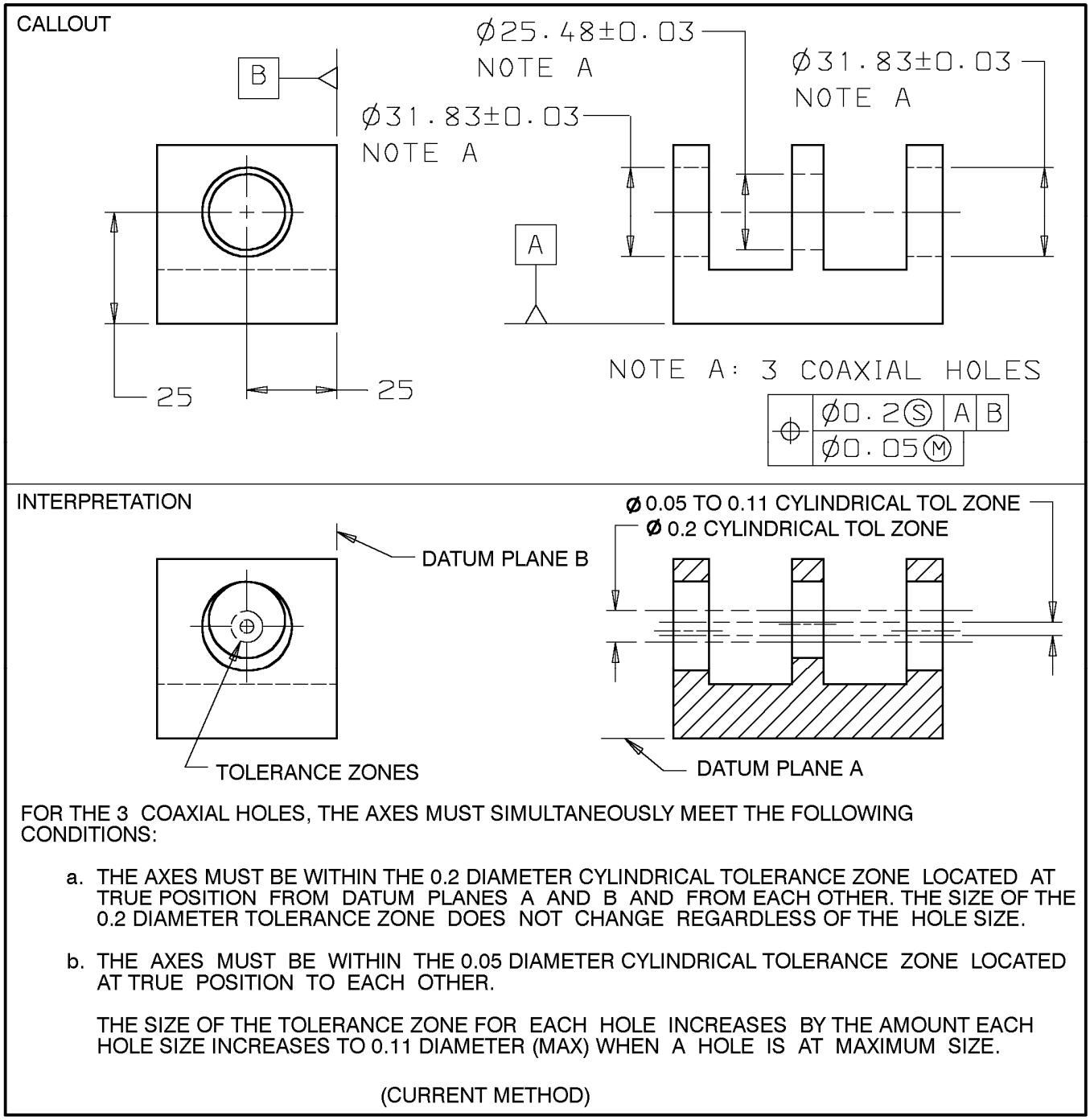
   THE SIZE OF THE TOLERANCE ZONE FOR EACH HOLE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES. FOR A HOLE AT MAXIMUM SIZE (Ø 25.58), THE TOLERANCE ZONE IS Ø 0.5 (MAXIMUM).

b. THE AXES MUST BE WITHIN THE Ø 0.1 CYLINDRICAL TOLERANCE ZONE LOCATED AT TRUE POSITION OF THE HOLES FROM EACH OTHER.

   THE SIZE OF THE TOLERANCE ZONE FOR EACH HOLE INCREASES BY THE AMOUNT THE HOLE SIZE INCREASES. FOR A HOLE AT MAXIMUM SIZE (Ø 25.58), THE TOLERANCE ZONE IS Ø 0.2 (MAXIMUM).

(FORMER METHOD)

Figure 94 - Coaxial Holes (Same Size) (Continued)



FOR THE 3 COAXIAL HOLES, THE AXES MUST SIMULTANEOUSLY MEET THE FOLLOWING CONDITIONS:

a. THE AXES MUST BE WITHIN THE 0.2 DIAMETER CYLINDRICAL TOLERANCE ZONE LOCATED AT TRUE POSITION FROM DATUM PLANES A AND B AND FROM EACH OTHER. THE SIZE OF THE 0.2 DIAMETER TOLERANCE ZONE DOES NOT CHANGE REGARDLESS OF THE HOLE SIZE.

b. THE AXES MUST BE WITHIN THE 0.05 DIAMETER CYLINDRICAL TOLERANCE ZONE LOCATED AT TRUE POSITION TO EACH OTHER.

THE SIZE OF THE TOLERANCE ZONE FOR EACH HOLE INCREASES BY THE AMOUNT EACH HOLE SIZE INCREASES TO 0.11 DIAMETER (MAX) WHEN A HOLE IS AT MAXIMUM SIZE.

(CURRENT METHOD)

Figure 95 - Coaxial Holes (Different Sizes) (Continued on Next Page)

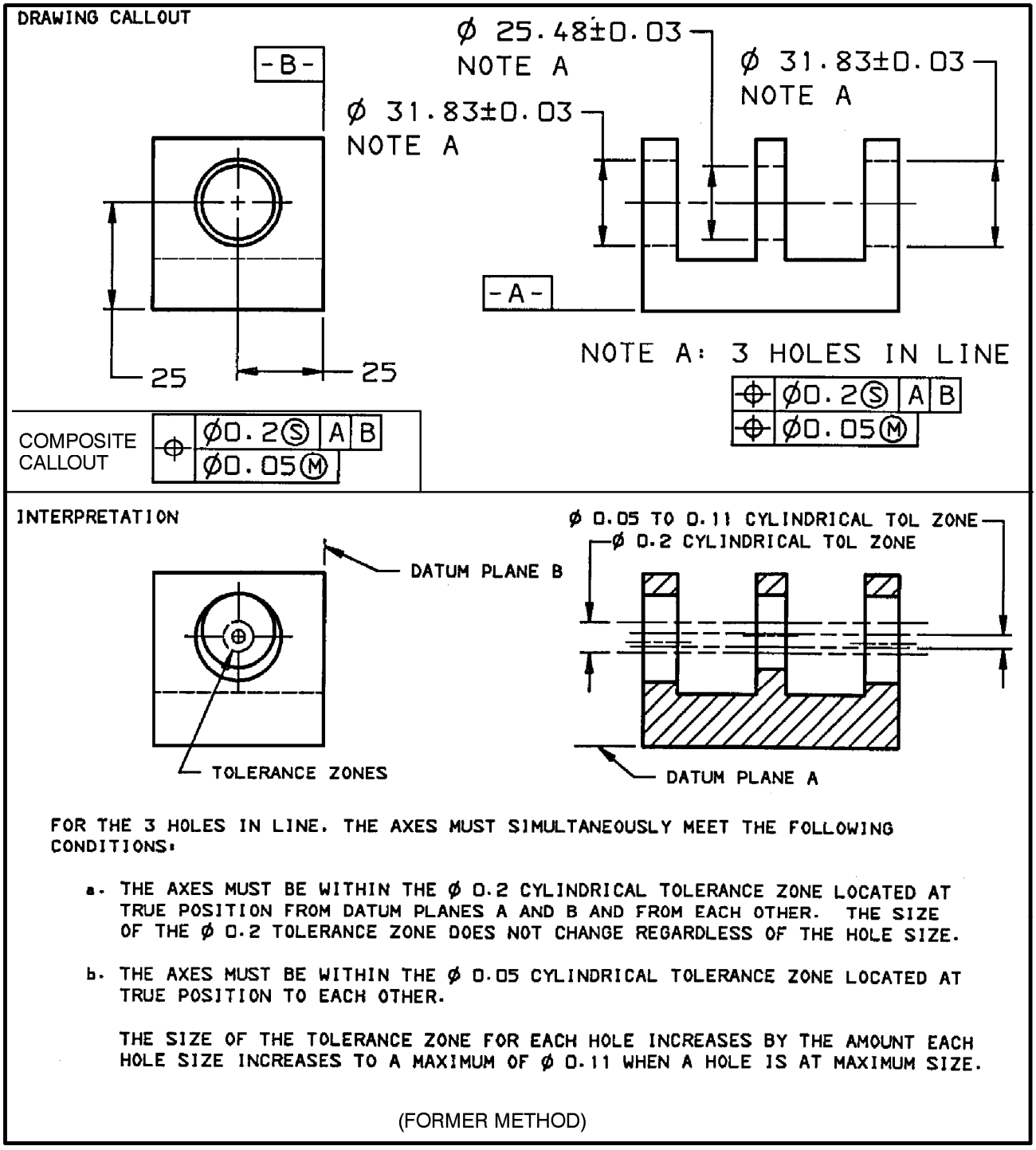
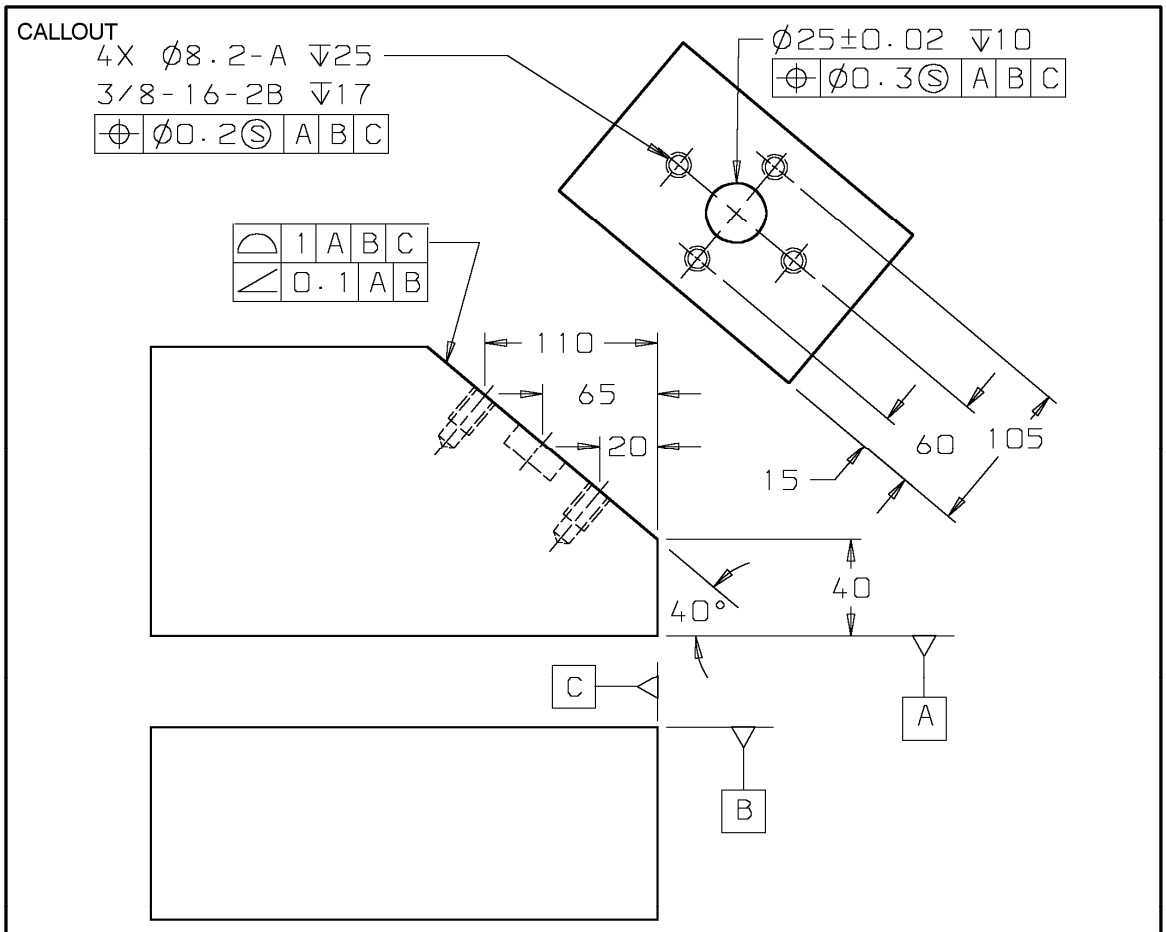


Figure 95 - Coaxial Holes (Different Sizes) (Continued)



INTERPRETATION

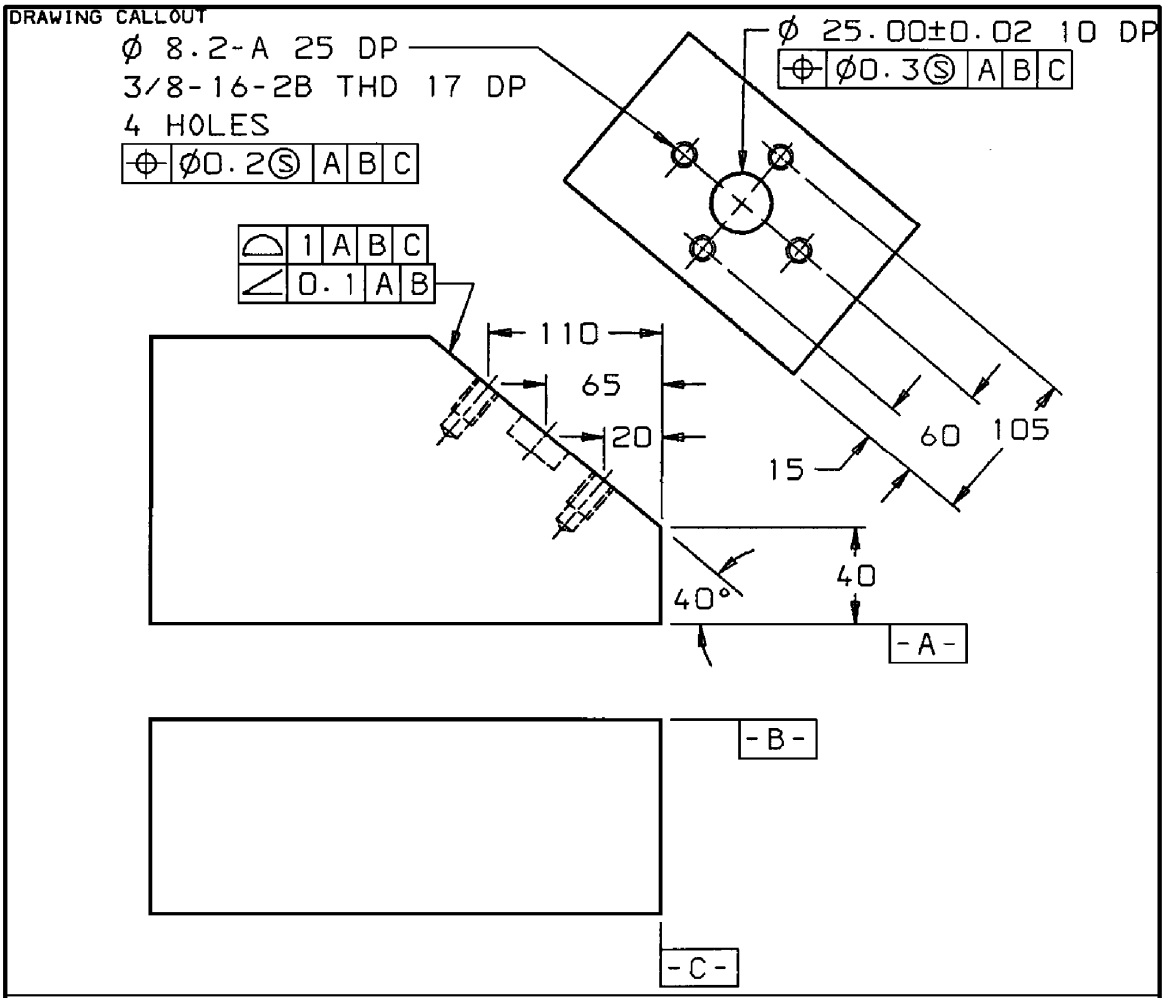
THE AXIS OF THE 25±0.02 DIAMETER HOLE MUST LIE WITHIN A 0.3 DIAMETER CYLINDRICAL TOLERANCE ZONE THAT IS LOCATED AND ORIENTED WITH RESPECT TO DATUMS A, B, AND C BY BASIC DIMENSIONS 60, 65, 40, AND 40 DEGREES.

THE AXES OF THE FOUR 3/8–16 THREADED HOLES MUST LIE WITHIN 0.2 DIAMETER CYLINDRICAL TOLERANCE ZONES THAT ARE LOCATED AND ORIENTED WITH RESPECT TO DATUMS A, B, AND C BY BASIC DIMENSIONS 105, 60, 15, 110, 65, 20, 40, AND 40 DEGREES.

THE ANGLED SURFACE MUST BE WITHIN THE 1 PROFILE OF A SURFACE TOLERANCE ZONE RELATIVE TO DATUMS A, B, AND C AND THE 40 BASIC DIMENSION. ADDITIONALLY THE ANGLED SURFACE MUST BE ORIENTED WITH RESPECT TO DATUMS A AND B BY THE 40 DEGREES AND THE 0.1 ANGULARITY TOLERANCE ZONE.

(CURRENT METHOD)

Figure 96 - Holes In Angled Surface (Controlled Relative To Datums A, B, C)
(Continued on Next Page)



INTERPRETATION

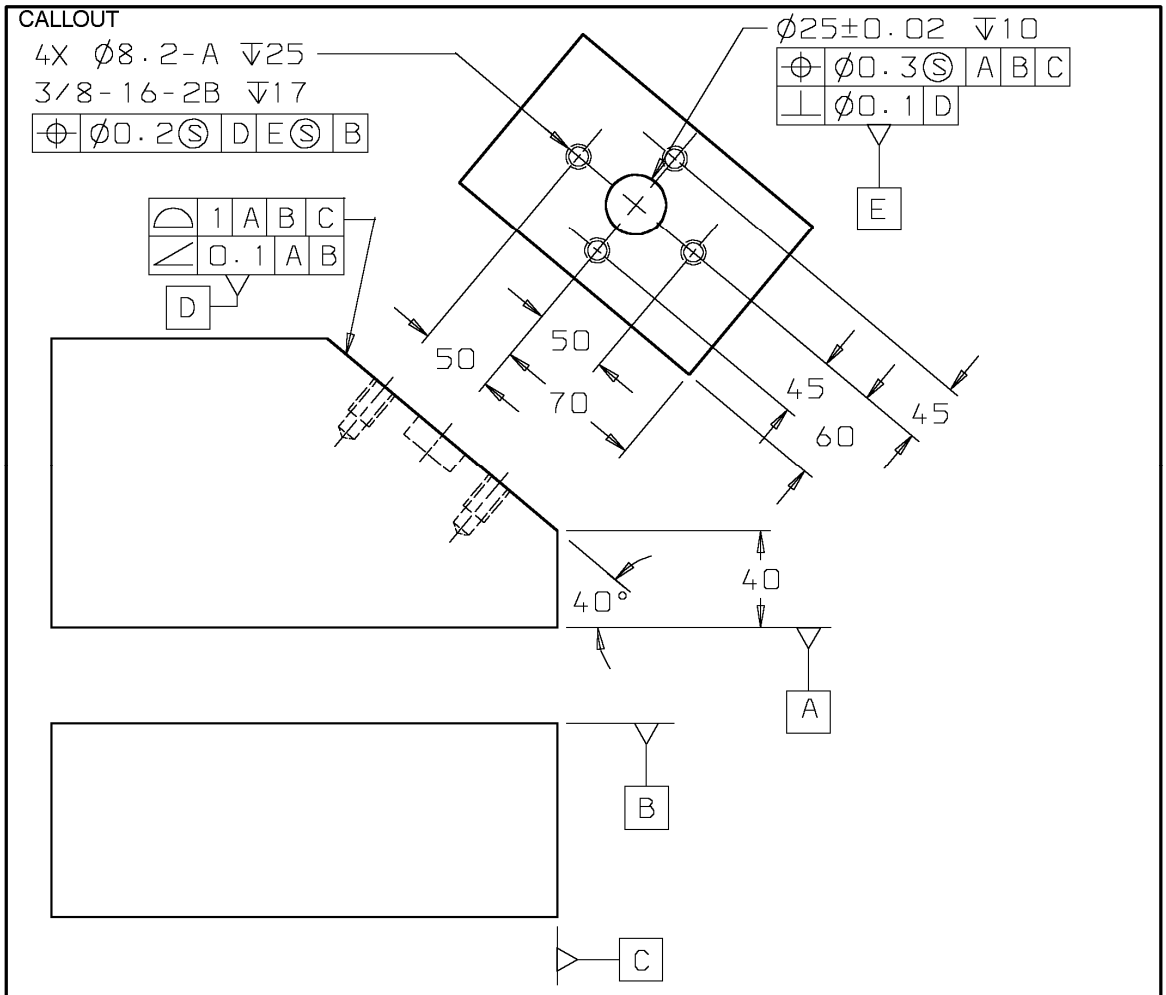
THE AXIS OF THE Ø 25.00±0.02 HOLE MUST LIE WITHIN A Ø 0.3 CYLINDRICAL TOLERANCE ZONE THAT IS LOCATED AND ORIENTED WITH RESPECT TO DATUMS A, B, AND C BY BASIC DIMENSIONS 60, 65, 40, AND 40°.

THE AXES OF THE FOUR 3/8-16 THREADED HOLES MUST LIE WITHIN Ø 0.2 CYLINDRICAL TOLERANCE ZONES THAT ARE LOCATED AND ORIENTED WITH RESPECT TO DATUMS A, B, AND C BY BASIC DIMENSIONS 105, 60, 15, 110, 65, 20, 40, AND 40°.

THE ANGLED SURFACE MUST BE WITHIN THE 1 PROFILE OF A SURFACE TOLERANCE ZONE RELATIVE TO DATUMS A, B, AND C AND THE 40 BASIC DIMENSION. ADDITIONALLY, THE ANGLED SURFACE MUST BE ORIENTED WITH RESPECT TO DATUMS A AND B BY THE 40° AND THE 0.1 ANGULARITY TOLERANCE ZONE.

(FORMER METHOD)

Figure 96 - Holes In Angled Surface (Controlled Relative To Datums A, B, C) (Continued)



INTREPERTATION

THE ANGLED SURFACE MUST BE WITHIN THE 1 PROFILE OF A SURFACE TOLERANCE ZONE RELATIVE TO DATUMS A, B, AND C AND THE 40 BASIC DIMENSION. ADDITIONALLY, THE ANGLED SURFACE MUST BE ORIENTED WITH RESPECT TO DATUMS A AND B BY THE 40 DEGREES AND THE 0.1 ANGULARITY TOLERANCE ZONE.

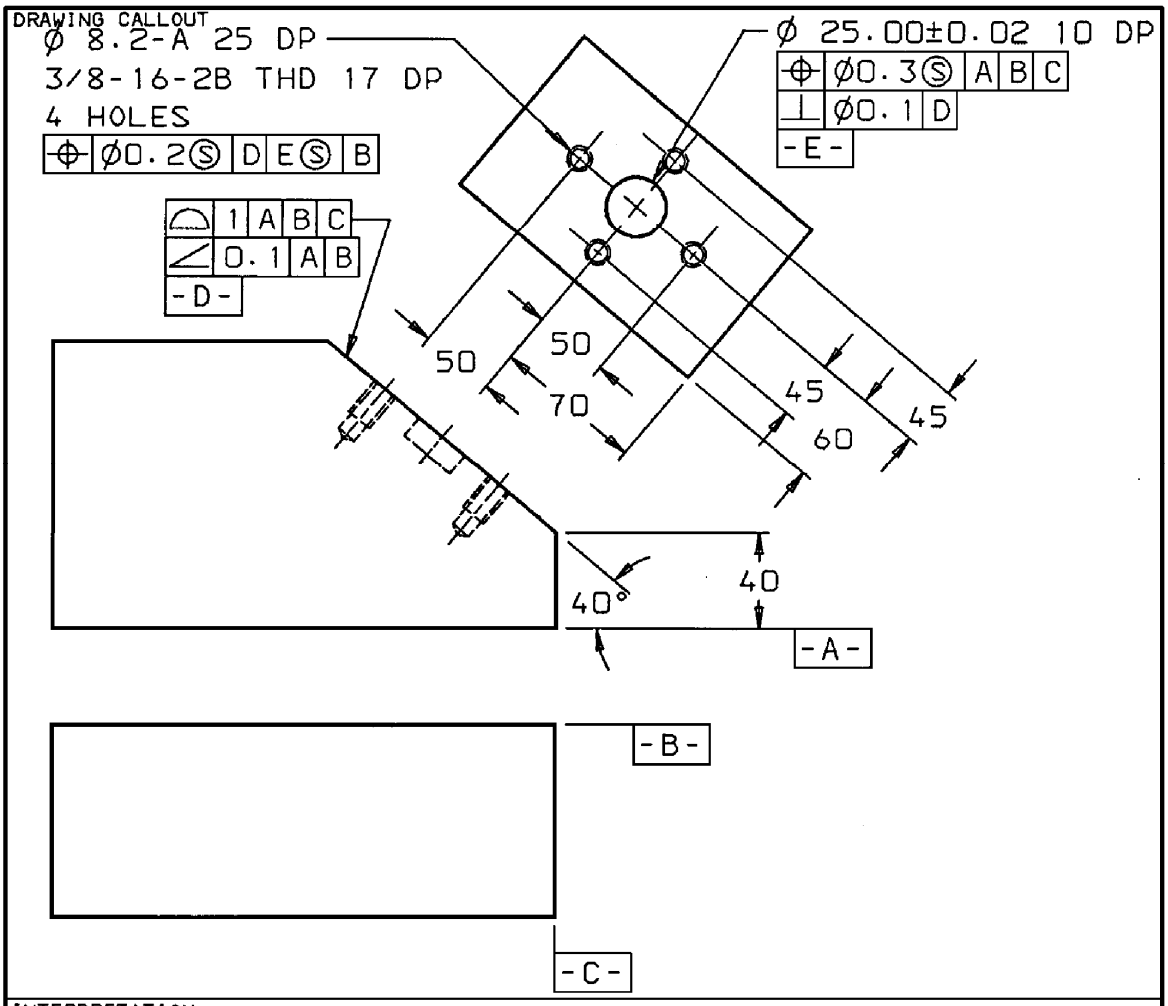
THE AXIS OF THE 25±0.02 DIAMETER HOLE MUST LIE WITHIN A 0.3 DIAMETER CYLINDRICAL TOLERANCE ZONE THAT IS LOCATED AND ORIENTED WITH RESPECT TO DATUMS A, B, AND C BY BASIC DIMENSIONS 70, 60, 40, AND 40 DEGREES. THE AXIS OF THE 25±0.02 DIAMETER HOLE MUST ALSO LIE WITHIN A 0.1 DIAMETER PERPENDICULAR TOLERANCE ZONE RELATIVE TO DATUM SURFACE D. THE 0.3 DIAMETER AND 0.1 DIAMETER TOLERANCE ZONES SIZES ARE CONSTANT REGARDLESS OF THE HOLE SIZE. THE HOLE IS DESIGNATED AS DATUM E.

THE AXIS OF THE FOUR 3/8–16 THREADED HOLES MUST LIE WITHIN 0.2 DIAMETER CYLINDRICAL TOLERANCE ZONES THAT ARE LOCATED AND ORIENTED WITH RESPECT TO DATUMS D, E, AND B BY BASIC DIMENSIONS 60, 45, 70, AND 50. THE TOLERANCE ZONE IS 0.2 DIAMETER REGARDLESS OF THE HOLE SIZE.

(CURRENT METHOD)

Figure 97 - Holes In Angled Surface (Controlled To Angled Surface)

(Continued on Next Page)



INTERPRETATION

THE ANGLED SURFACE MUST BE WITHIN THE 1 PROFILE OF A SURFACE TOLERANCE ZONE RELATIVE TO DATUMS A, B, AND C AND THE 40 BASIC DIMENSION. ADDITIONALLY, THE ANGLED SURFACE MUST BE ORIENTED WITH RESPECT TO DATUMS A AND B BY THE 40° AND THE 0.1 ANGULARITY TOLERANCE ZONE.

THE AXIS OF THE Ø 25.00±0.02 HOLE MUST LIE WITHIN A Ø 0.3 CYLINDRICAL TOLERANCE ZONE THAT IS LOCATED AND ORIENTED WITH RESPECT TO DATUMS A, B, AND C BY BASIC DIMENSIONS 70, 60, 40, AND 40°. THE AXIS OF THE Ø 25.00±0.02 HOLE MUST ALSO LIE WITHIN A Ø 0.1 PERPENDICULAR TOLERANCE ZONE RELATIVE TO DATUM SURFACE D. THE Ø 0.3 AND Ø 0.1 TOLERANCE ZONES SIZES ARE CONSTANT REGARDLESS OF THE HOLE SIZE. THE HOLE IS DESIGNATED AS DATUM E.

THE AXES OF THE FOUR 3/8-16 THREADED HOLES MUST LIE WITHIN Ø 0.2 CYLINDRICAL TOLERANCE ZONES THAT ARE LOCATED AND ORIENTED WITH RESPECT TO DATUMS D, E, AND B BY BASIC DIMENSIONS 60, 45, 70, AND 50. THE TOLERANCE ZONE IS Ø 0.2 REGARDLESS OF THE HOLE SIZE.

(FORMER METHOD)

Figure 97 - Holes In Angled Surface (Controlled Relative To Angled Surface)
(Continued from Previous Page)

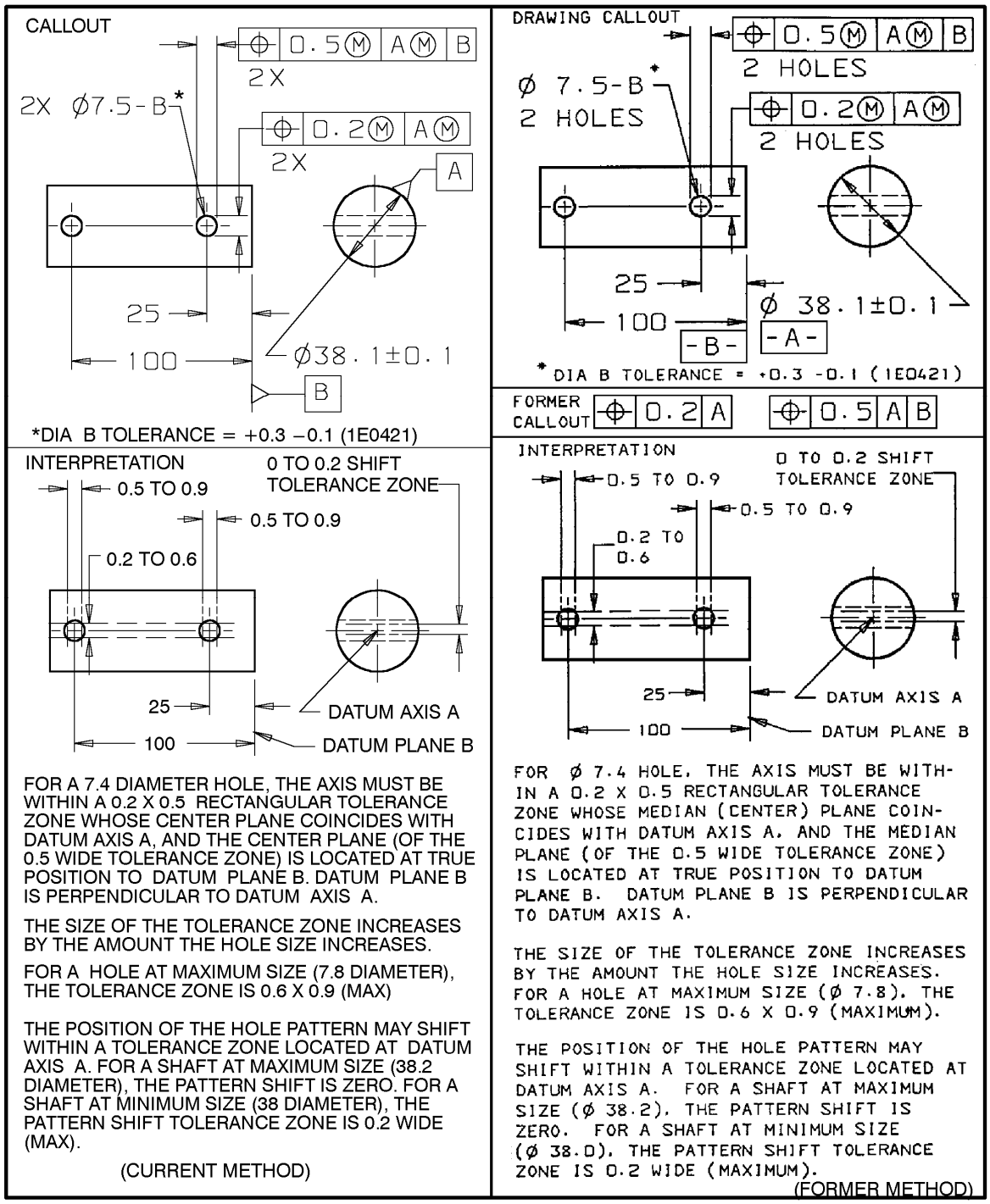


Figure 98 - Rectangular Positional Tolerance Zone

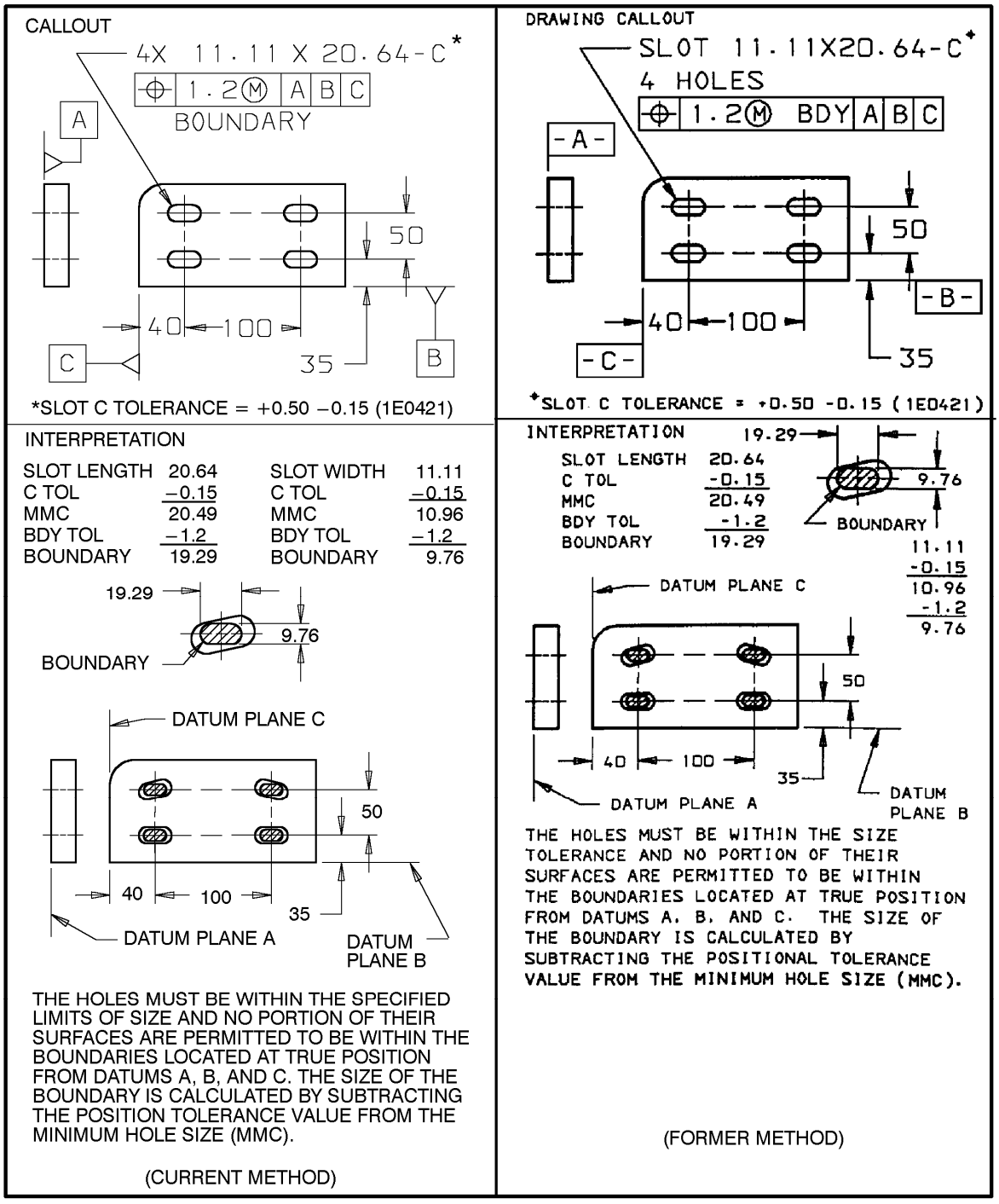
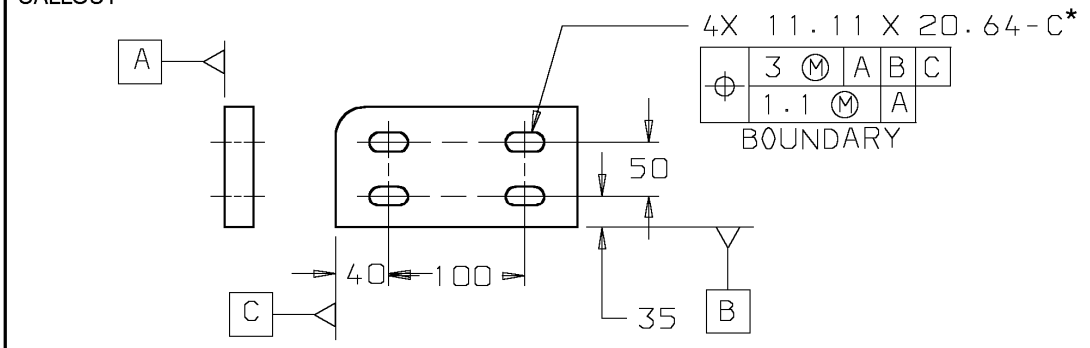


Figure 99 - Boundary Positional Tolerance

CALLOUT



*SLOT C TOLERANCE = +0.50 −0.15 (1E0421)

INTERPRETATION

⌖ 3 Ⓜ A B C
BOUNDARY

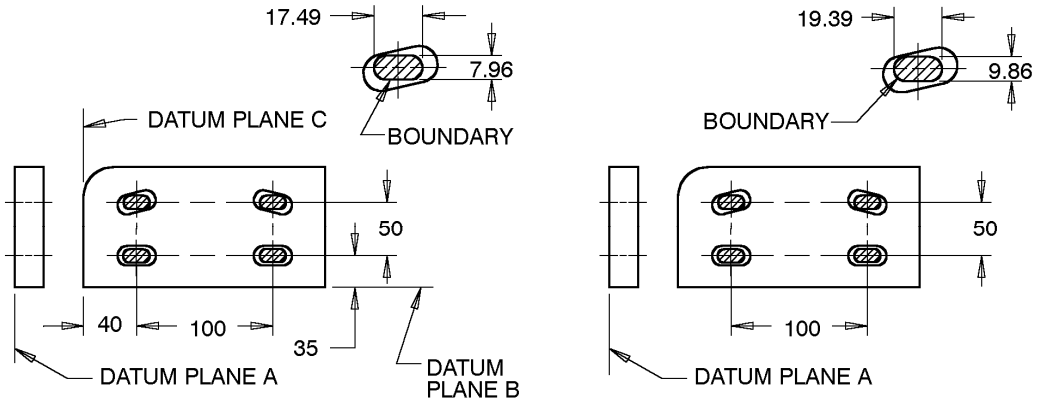
⌖ 1.1 Ⓜ A
BOUNDARY

PATTERN LOCATING TOLERANCE

SLOT LENGTH	20.64	SLOT WIDTH	11.11
C TOL	−0.15	C TOL	−0.15
MMC	20.49	MMC	10.96
BDY TOL	−3	BDY TOL	−3
BOUNDARY	17.49	BOUNDARY	7.96

FEATURE TO FEATURE TOLERANCE

SLOT LENGTH	20.64	SLOT WIDTH	11.11
C TOL	−0.15	C TOL	−0.15
MMC	20.49	MMC	10.96
BDY TOL	−1.1	BDY TOL	−1.1
BOUNDARY	19.39	BOUNDARY	9.86



THE 4 SLOTS MUST BE WITHIN THE SPECIFIED LIMITS OF SIZE AND NO PORTION OF THEIR SURFACES ARE PERMITTED TO BE WITHIN EITHER THE PATTERN LOCATING TOLERANCE BOUNDARIES OR THE FEATURE TO FEATURE TOLERANCE BOUNDARIES. THE BOUNDARIES FOR THE 3 (PATTERN LOCATING) POSITION TOLERANCE ARE LOCATED AT TRUE POSITION FROM DATUMS A, B, AND C AND THE SIZE OF EACH BOUNDARY IS CALCULATED BY SUBTRACTING THE POSITION TOLERANCE VALUE FROM THE MINIMUM SLOT SIZE (MMC). THE BOUNDARIES FOR THE 1.1 (FEATURE TO FEATURE) POSITION TOLERANCE ARE ESTABLISHED RELATIVE TO EACH OTHER BY THE 50 AND 100 BASIC DIMENSIONS AND THE SIZE OF EACH BOUNDARY IS CALCULATED BY SUBTRACTING THE POSITION TOLERANCE VALUE FROM THE MINIMUM SLOT SIZE (MMC). THE FEATURE TO FEATURE TOLERANCE BOUNDARIES ARE FREE TO MOVE AS A GROUP WITHIN THE CONSTRAINTS OF THE PATTERN LOCATING TOLERANCE BOUNDARIES AS LONG AS NO PORTION OF THE SLOT SURFACES ARE WITHIN EITHER THE PATTERN LOCATING TOLERANCE BOUNDARIES OR THE FEATURE TO FEATURE TOLERANCE BOUNDARIES.

Figure 100 - Boundary Positional Tolerance

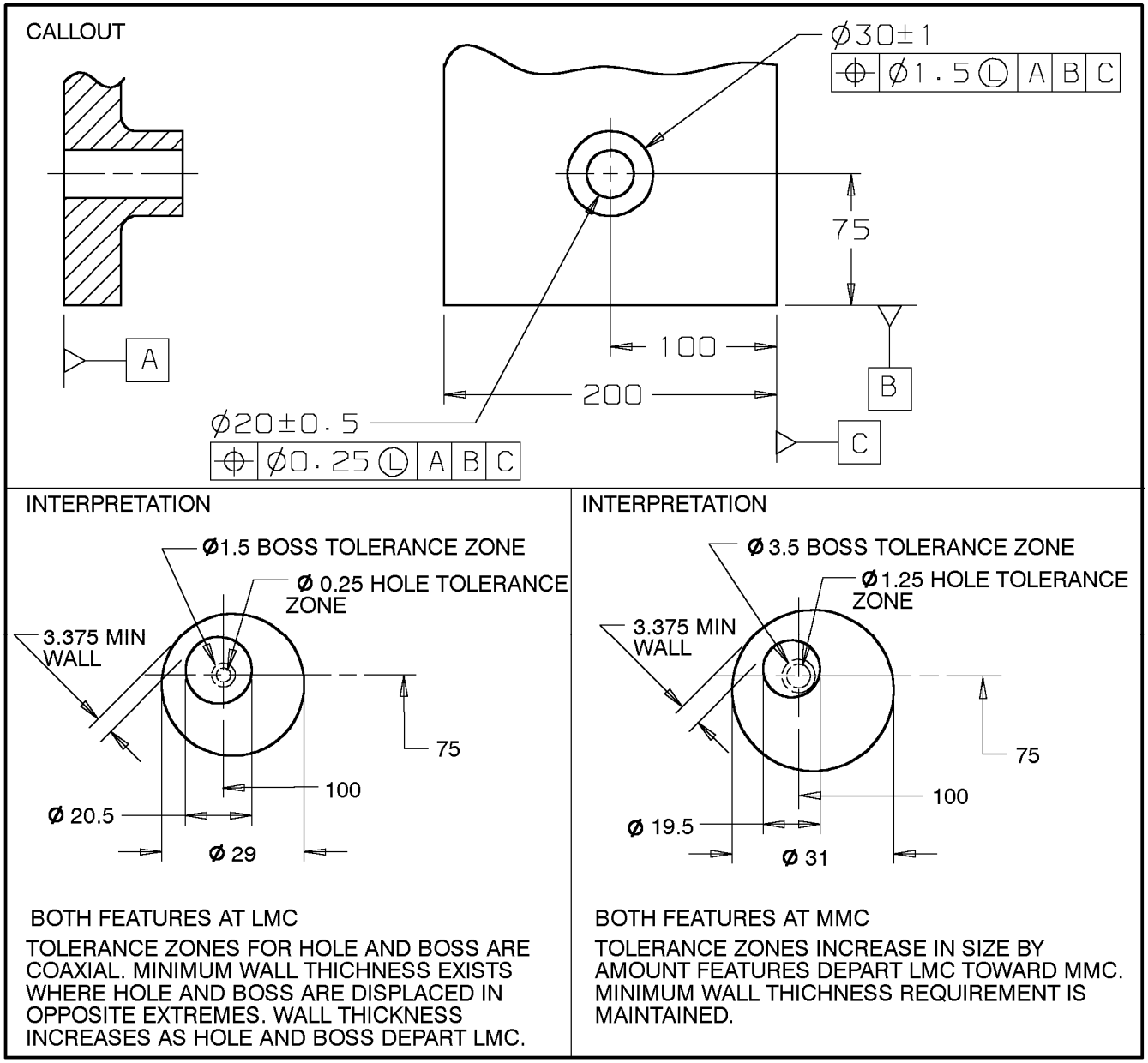


Figure 101 - LMC Applied To Boss And Hole

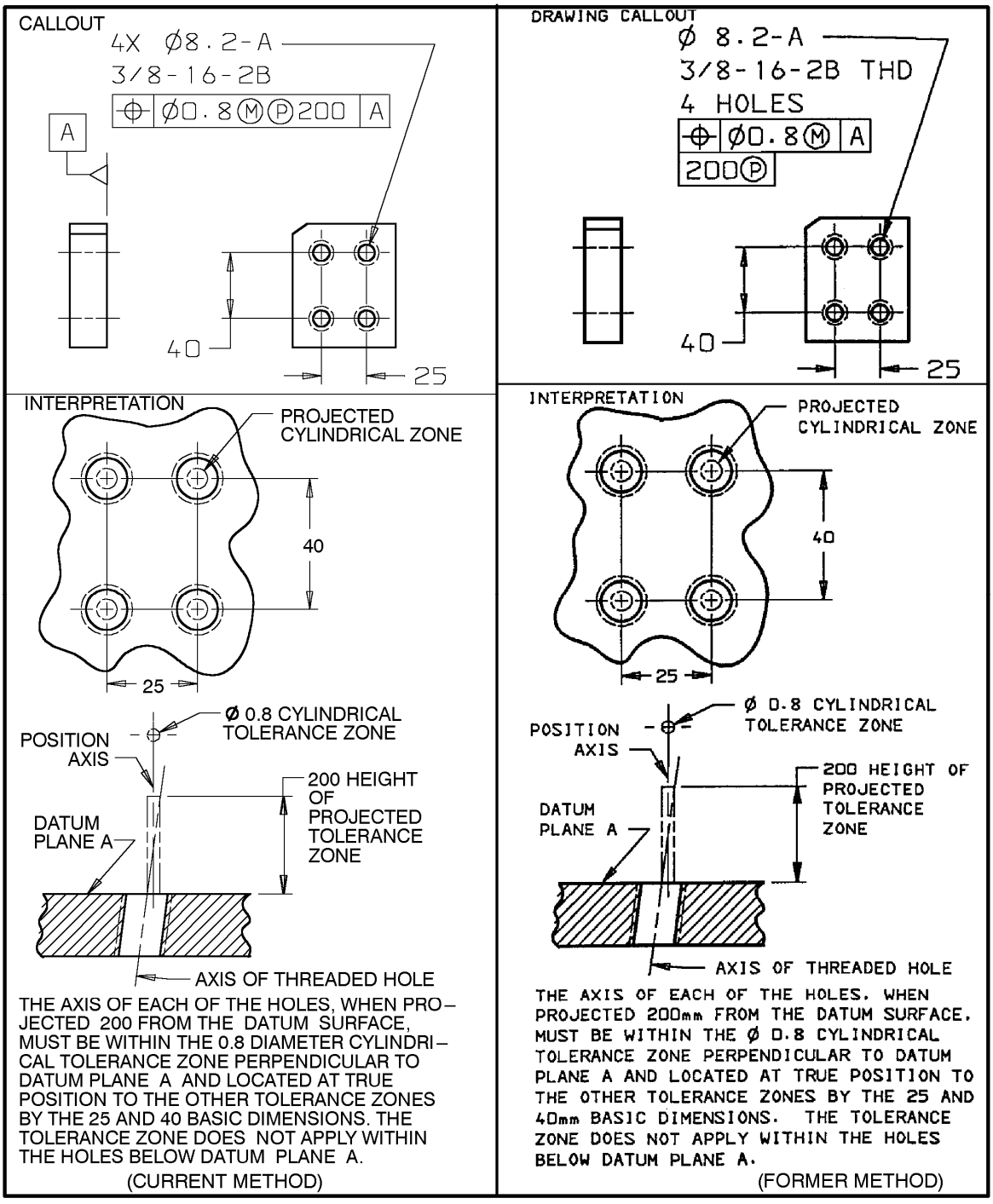


Figure 102 - Projected Tolerance Zone

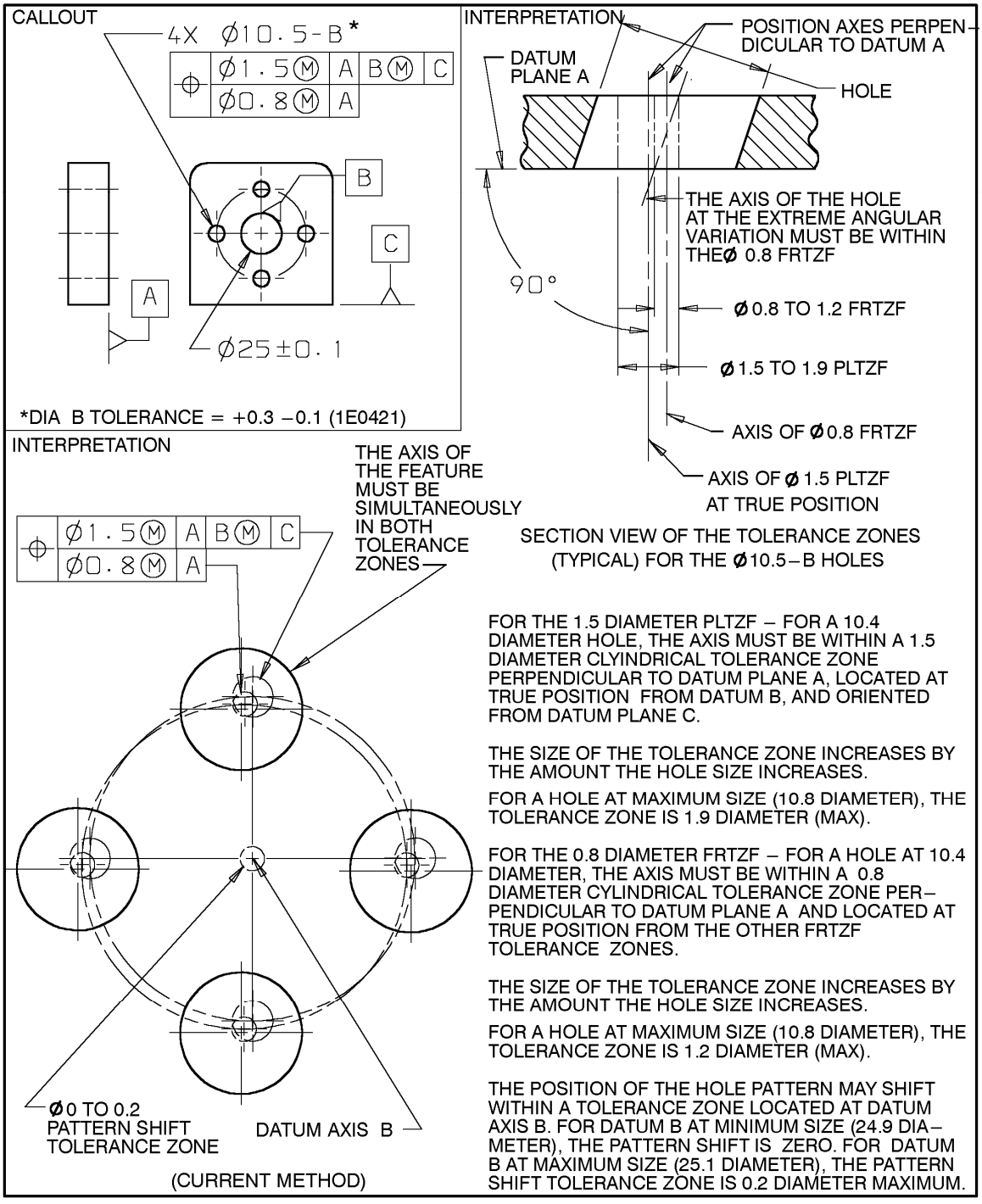


Figure 103 - Composite Positional Tolerance (Continued on Next Page)

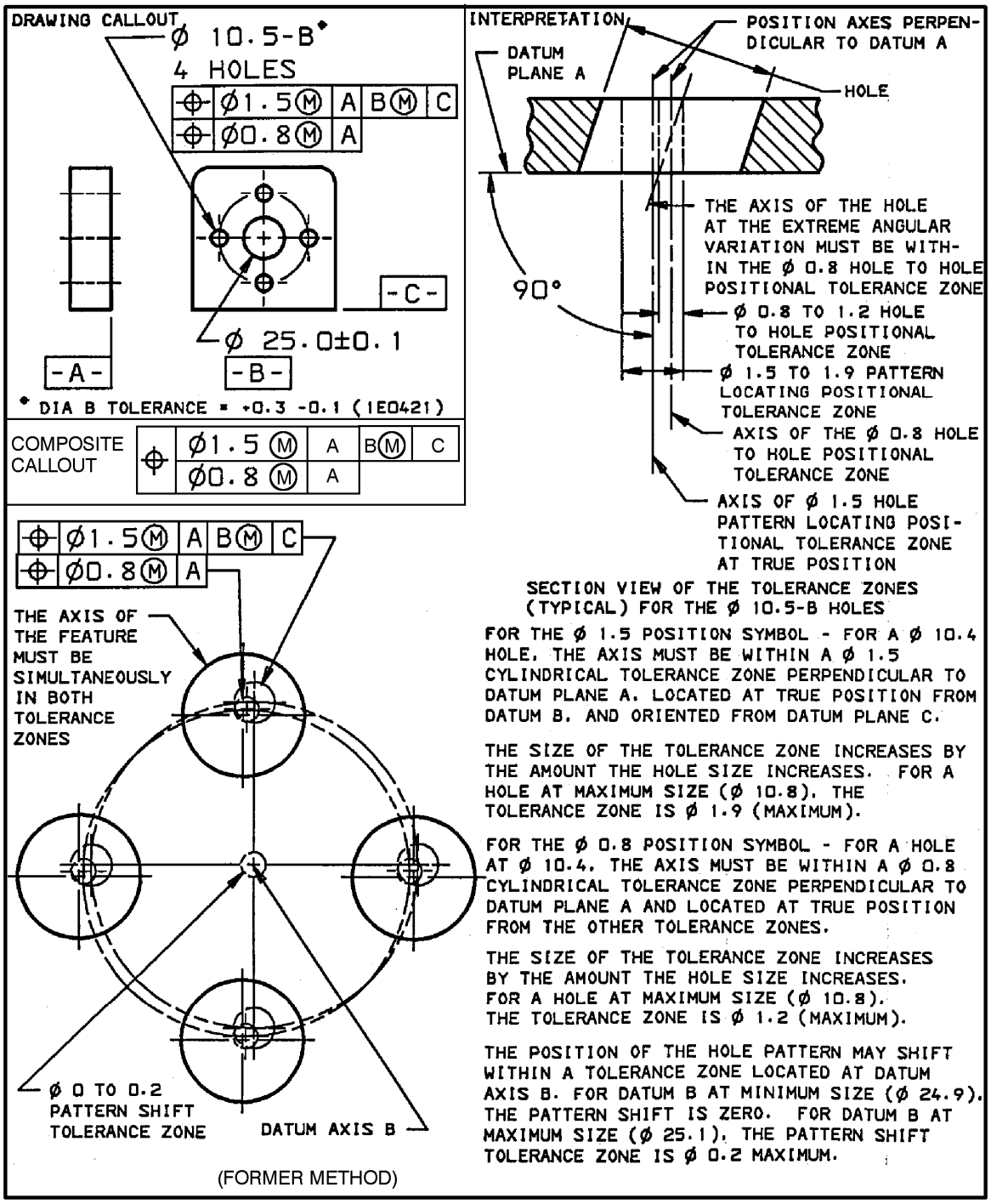


Figure 103 - Composite Positional Tolerance (Continued)

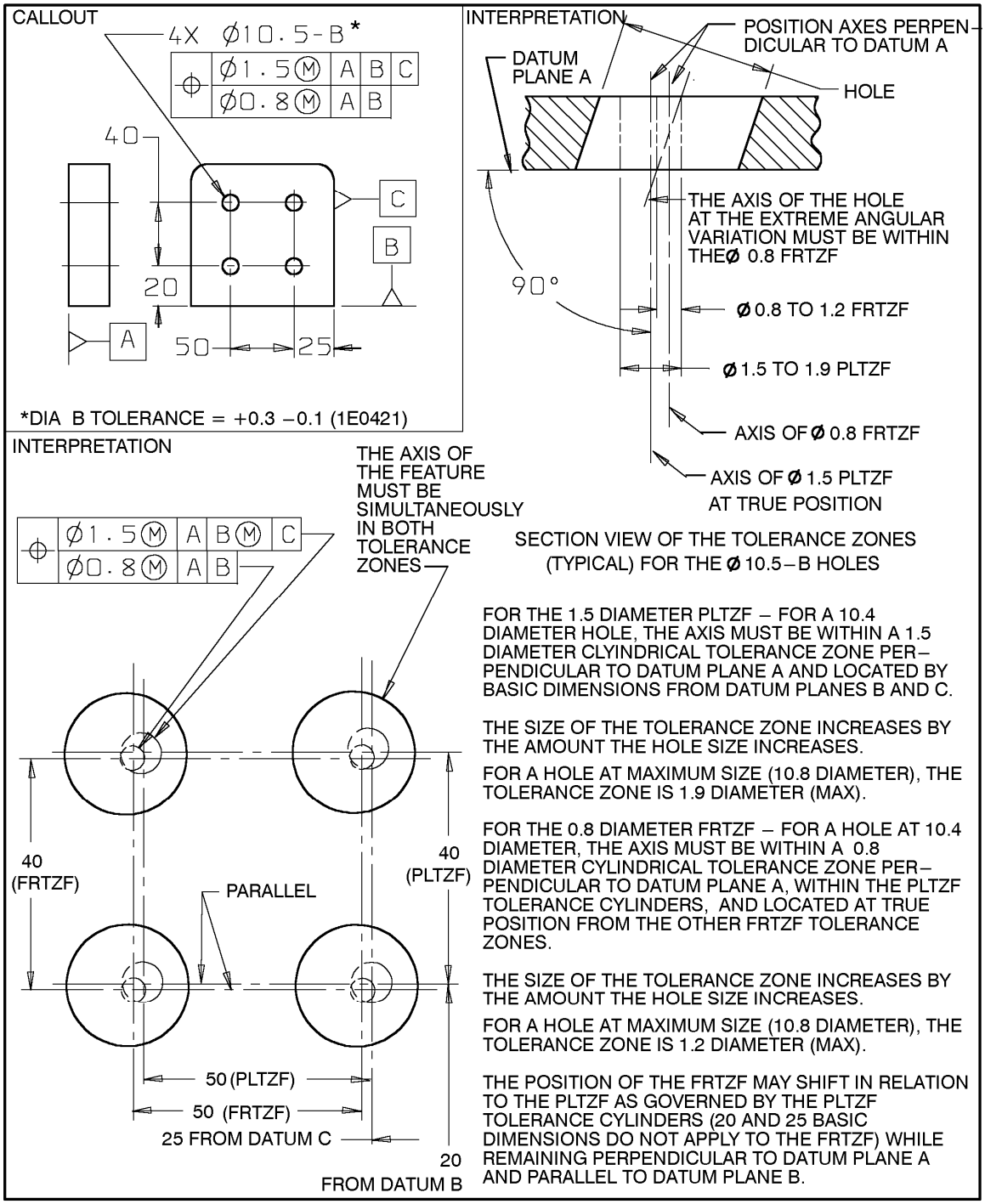


Figure 104 - Composite Positional Tolerance

INTERPRETATION - GEOMETRIC TOLERANCE	DATE	CHG NO	NUMBER
	26 MAR 2013	10	1E0012

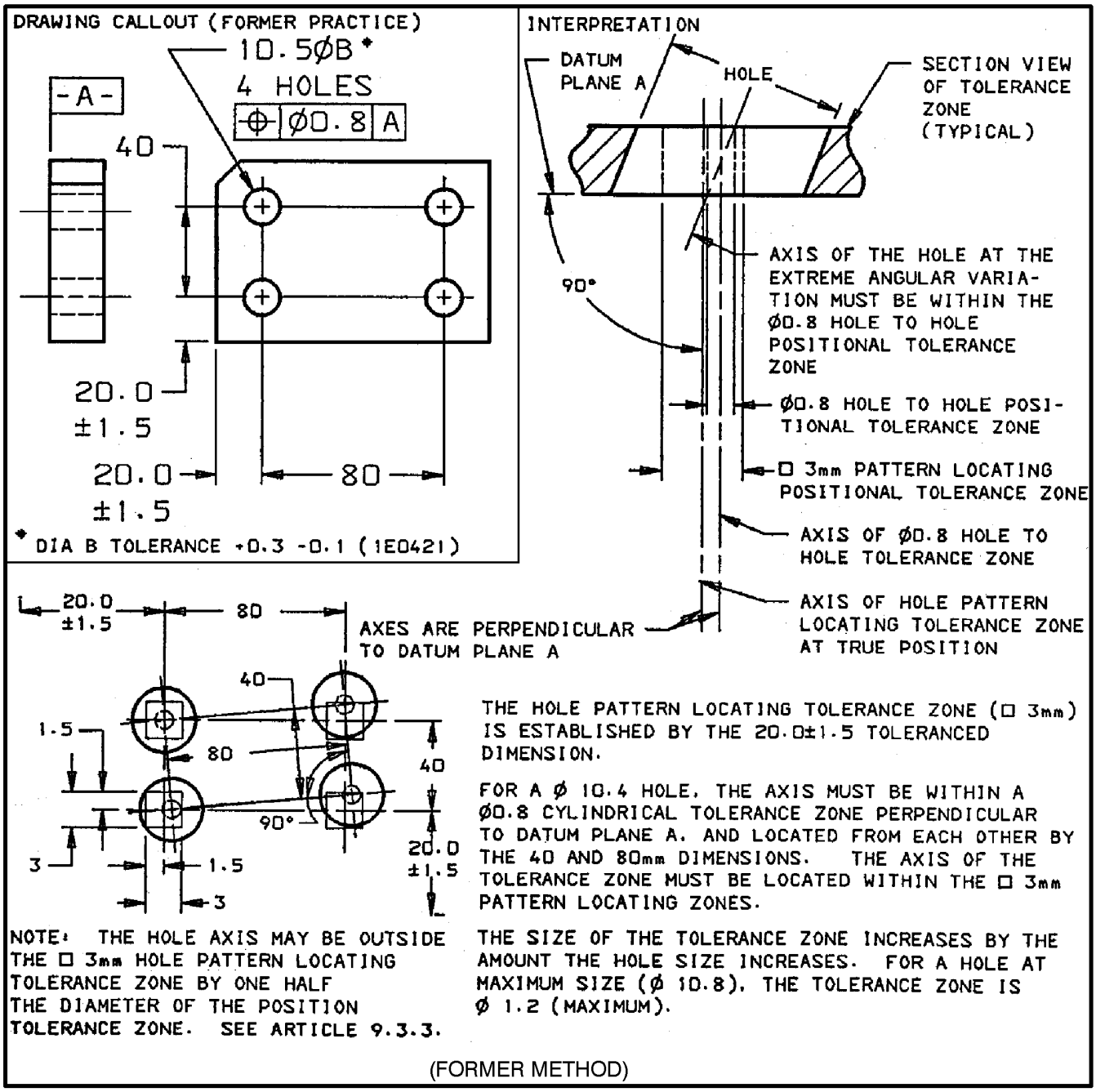
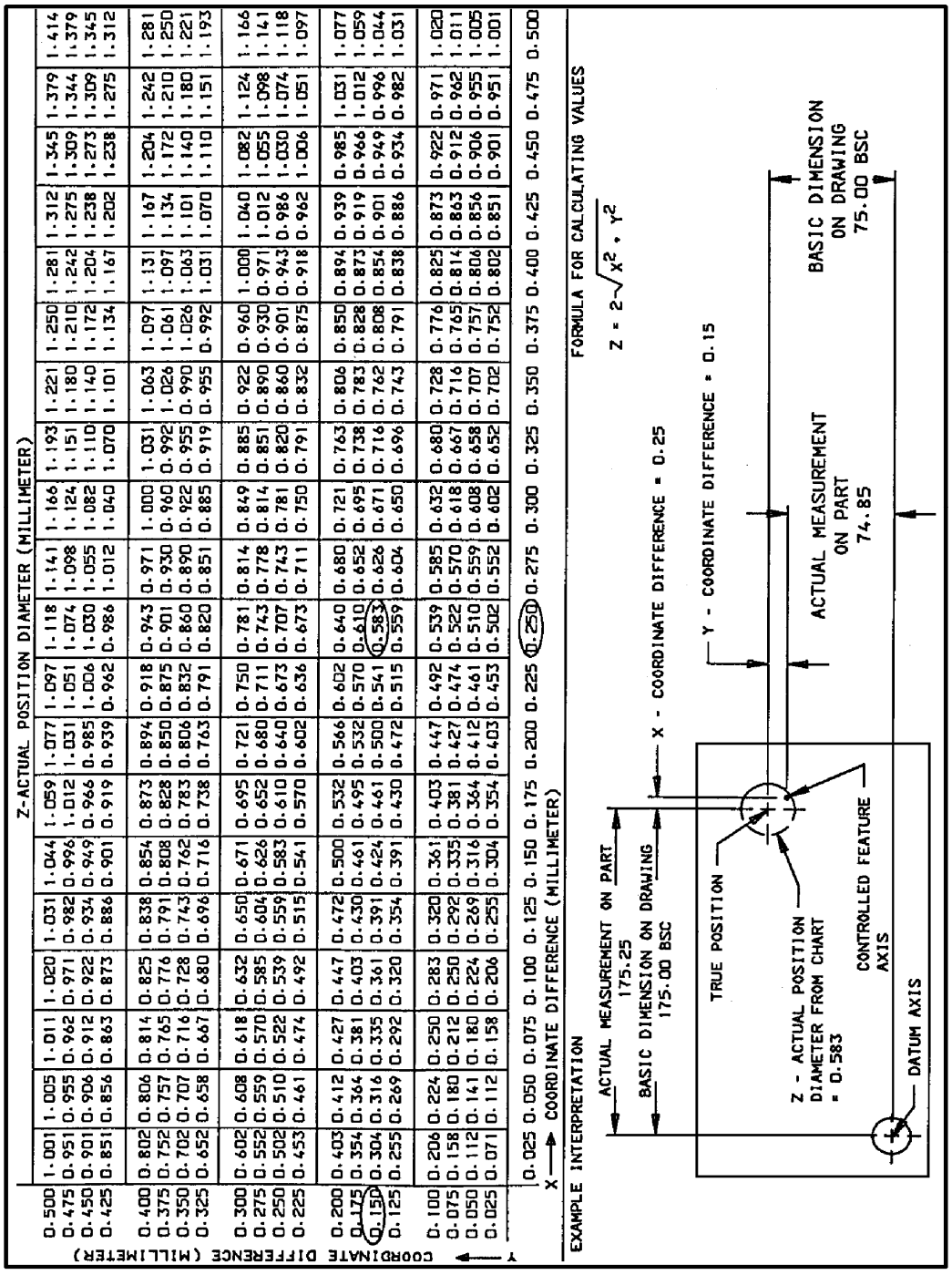


Figure 105 - Combination Positional Tolerancing And Toleranced Locating Dimensions



COORDINATE DIFFERENCE (MILLIMETER) Y	Z-ACTUAL POSITION DIAMETER (MILLIMETER)																				
0.500	1.001	1.005	1.011	1.020	1.031	1.044	1.059	1.077	1.097	1.118	1.141	1.166	1.193	1.221	1.250	1.281	1.312	1.345	1.379	1.414
0.475	0.951	0.955	0.962	0.971	0.982	0.996	1.012	1.031	1.051	1.074	1.098	1.124	1.151	1.180	1.210	1.242	1.275	1.309	1.344	1.379
0.450	0.901	0.906	0.912	0.922	0.934	0.949	0.966	0.985	1.006	1.030	1.055	1.082	1.110	1.140	1.172	1.204	1.238	1.273	1.309	1.345
0.425	0.851	0.856	0.863	0.873	0.886	0.901	0.919	0.939	0.962	0.986	1.012	1.040	1.070	1.101	1.134	1.167	1.202	1.238	1.275	1.312
0.400	0.802	0.806	0.814	0.825	0.838	0.854	0.873	0.894	0.918	0.943	0.971	1.000	1.031	1.063	1.097	1.131	1.167	1.204	1.242	1.281
0.375	0.752	0.757	0.765	0.776	0.791	0.808	0.828	0.850	0.875	0.901	0.930	0.960	0.992	1.026	1.061	1.097	1.134	1.172	1.210	1.250
0.350	0.702	0.707	0.716	0.728	0.743	0.762	0.783	0.806	0.832	0.860	0.890	0.922	0.955	0.990	1.026	1.063	1.101	1.140	1.180	1.221
0.325	0.652	0.658	0.667	0.680	0.696	0.716	0.738	0.763	0.791	0.820	0.851	0.885	0.919	0.955	0.992	1.031	1.070	1.110	1.151	1.193
0.300	0.602	0.608	0.618	0.632	0.650	0.671	0.695	0.721	0.750	0.781	0.814	0.849	0.885	0.922	0.960	1.000	1.040	1.082	1.124	1.166
0.275	0.552	0.559	0.570	0.585	0.604	0.626	0.652	0.680	0.711	0.743	0.778	0.814	0.851	0.890	0.930	0.971	1.012	1.055	1.098	1.141
0.250	0.502	0.510	0.522	0.539	0.559	0.583	0.610	0.640	0.673	0.707	0.743	0.781	0.820	0.860	0.901	0.943	0.986	1.030	1.074	1.118
0.225	0.453	0.461	0.474	0.492	0.515	0.541	0.570	0.602	0.636	0.673	0.711	0.750	0.791	0.832	0.875	0.918	0.962	1.006	1.051	1.097
0.200	0.403	0.412	0.427	0.447	0.472	0.500	0.532	0.566	0.602	0.640	0.680	0.721	0.763	0.806	0.850	0.894	0.939	0.985	1.031	1.077
0.175	0.354	0.364	0.381	0.403	0.430	0.461	0.495	0.532	0.570	0.610	0.652	0.695	0.738	0.783	0.828	0.873	0.919	0.966	1.012	1.059
0.150	0.304	0.316	0.335	0.361	0.391	0.424	0.461	0.500	0.541	0.583	0.626	0.671	0.716	0.762	0.808	0.854	0.901	0.949	0.996	1.044
0.125	0.255	0.269	0.292	0.320	0.354	0.391	0.430	0.472	0.515	0.559	0.604	0.650	0.696	0.743	0.791	0.838	0.886	0.934	0.982	1.031
0.100	0.206	0.224	0.250	0.283	0.320	0.361	0.403	0.447	0.492	0.539	0.585	0.632	0.680	0.728	0.776	0.825	0.873	0.922	0.971	1.020
0.075	0.158	0.180	0.212	0.250	0.292	0.335	0.381	0.427	0.474	0.522	0.570	0.618	0.667	0.716	0.765	0.814	0.863	0.912	0.962	1.011
0.050	0.112	0.141	0.180	0.224	0.269	0.316	0.364	0.412	0.461	0.510	0.559	0.608	0.658	0.707	0.757	0.806	0.856	0.906	0.955	1.005
0.025	0.071	0.112	0.158	0.206	0.255	0.304	0.354	0.403	0.453	0.502	0.552	0.602	0.652	0.702	0.752	0.802	0.851	0.901	0.951	1.001
X →	0.025	0.050	0.075	0.100	0.125	0.150	0.175	0.200	0.225	0.250	0.275	0.300	0.325	0.350	0.375	0.400	0.425	0.450	0.475	0.500

COORDINATE DIFFERENCE (MILLIMETER)

Figure 106 - Positional Tolerance Chart (mm)

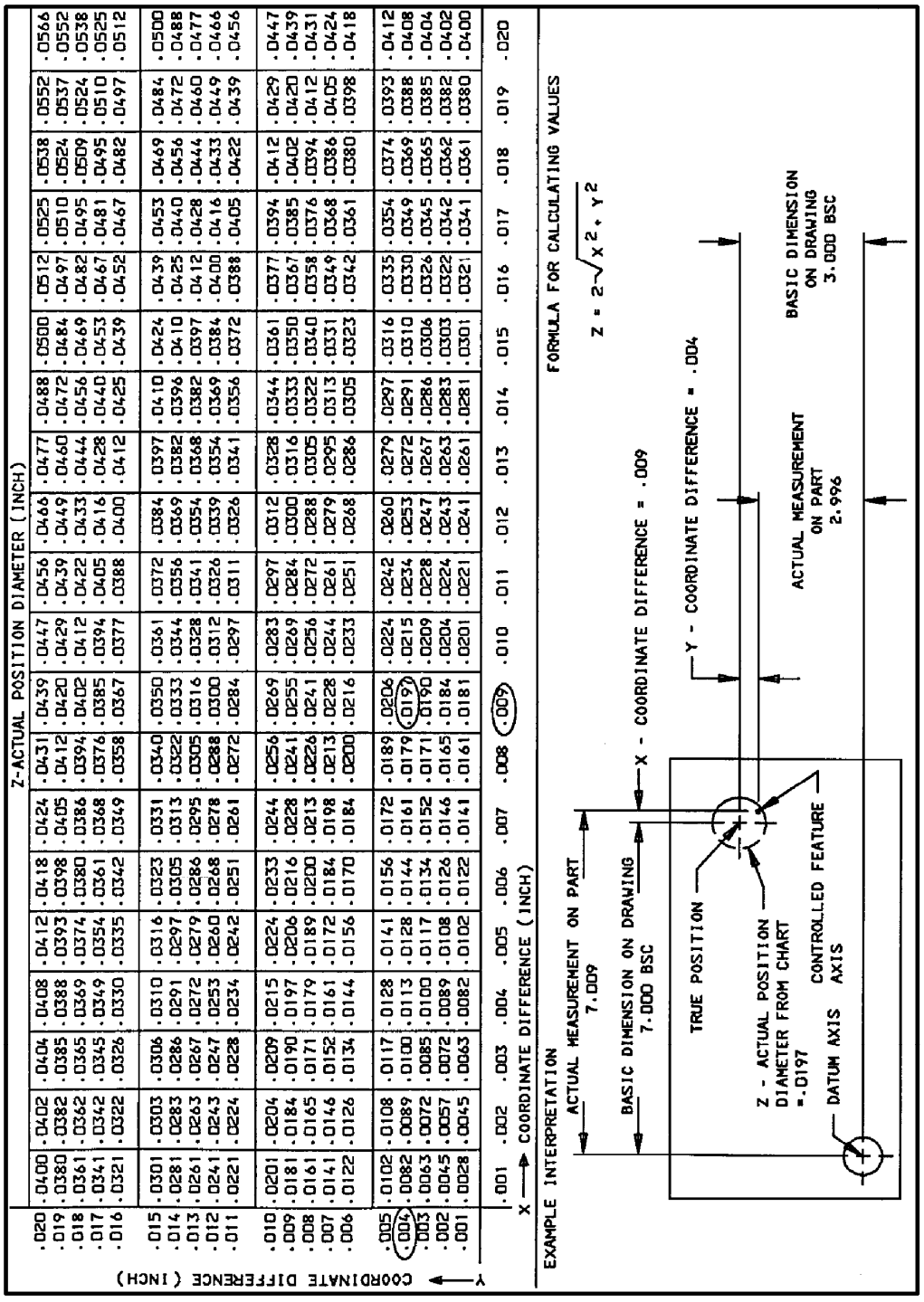


Figure 107 - Positional Tolerance Chart (Inch)

11.0 PROFILE TOLERANCES

11.1 Definition - A surface has basic (true) profile when all points of the surface are at the exact specified location. A line has basic (true) profile when all points of the line are at the exact specified location.

11.2 Profile Tolerance - Profile tolerance is the width of a zone disposed relative to the basic profile within which the controlled surface or line element shall lie. The tolerance zone may be equally or unequally disposed bilaterally (on both sides) relative to the basic profile or applied unilaterally to either side of the basic profile.

11.2.1 Where the tolerance zone is to be equally disposed bilaterally relative to the basic profile, the feature control frame is shown with a leader directed to the surface or line element that is to be controlled. (See Figure 108).

11.2.2 Where the tolerance zone is to be disposed unilaterally on one side of the basic profile, a phantom line is drawn parallel to the basic profile to show whether the tolerance zone is inside or outside of the basic profile. The feature control frame is shown attached to the extension of a dimension line that has arrowheads at the phantom line and the basic profile line. (See Figure 108).

11.2.3 Where the tolerance zone is to be disposed unequally on both sides of the basic profile, phantom lines are drawn parallel to the basic profile and dimensioned to show the offset from the basic profile. The feature control frame is shown attached to the extension of a dimension line that has arrowheads at both phantom lines. (See Figure 108).

11.3 Profile of a Surface - Profile of a surface tolerance applies to one of the following:

11.3.1 To the entire profile shown in a view where the all around symbol or **ALL AROUND** is specified. (See Figures 109, 111, and 116).

11.3.2 To a circular or flat feature defined by a single dimension by connecting the feature control frame to the surface by a leader. (Equal bilateral tolerance may be attached to an extension line). (See Figure 113).

11.3.3 To a circular feature by applying the feature control frame to the basic dimension defining the size of the feature (equal bilateral tolerance only). (See Figures 115, 118, and119).

11.3.4 To a portion of a feature surface by specifying the end points of the controlled surface. (See Figures 112 and 113).

11.3.5 To the edges of an entire view of a component with a note such as shown in Figure 115 (equal bilateral tolerance only).

11.3.6 Where applicable, a general three-dimensional profile tolerance requirement has been specified in a note on the drawing/model. Specific surfaces that have other tolerance requirements are toleranced separately. The general profile tolerance specifies the datum system from which the basic profile of the toleranced surfaces is located (equal bilateral tolerance only). (See Figures 111 and 119).

11.3.7 Current Method - New or redrawn casting, forging, and molded part drawings released after 1974, normally will contain a dimension and tolerance block. These drawings are interpreted according to Figure 120. Refer to 1E0100 (Forgings), 1E0576 (Castings), or 1E2600 (Interpretation - Profile Tolerance) for detailed interpretations of the rough and finished component.

The profile tolerance related to datums A, B, and C in the dimension and tolerance block applies to all unmachined surfaces that remain on the completed component.

11.3.8 Former Method (For Castings) - When the term **PLUS DRAFT** is specified, interpret according to Figure 121. If the term **PLUS DRAFT** is not specified, the surface, including draft if present, must be within the specified tolerance zone. Maximum draft angles are specified in 1E0576.

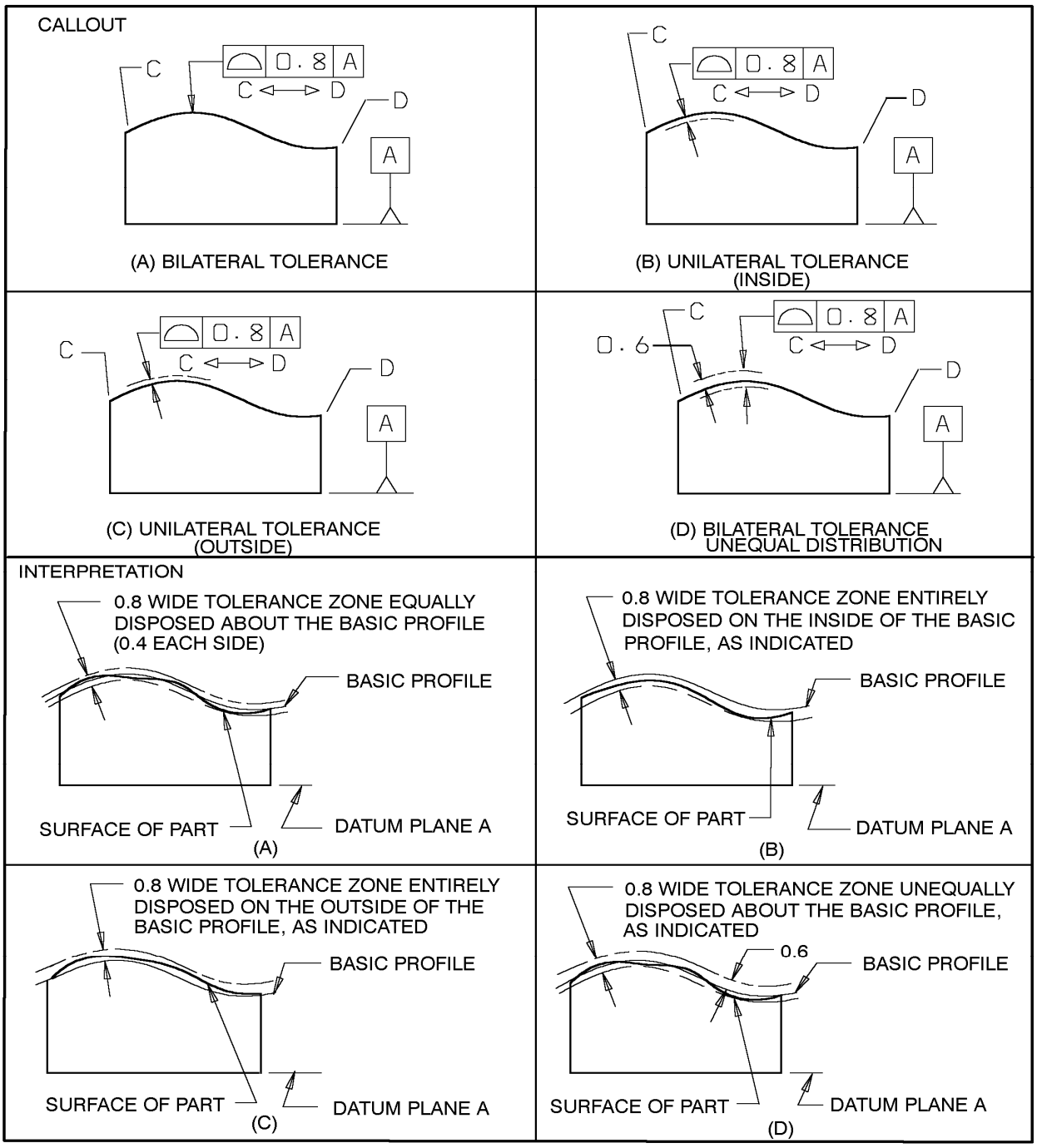
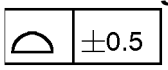
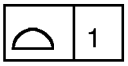
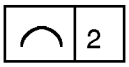


Figure 108 - Profile Of A Surface Callout

11.4 Basic Dimensions - Dimensions locating the profile controlled by profile of a surface tolerances are basic (Untoleranced). Usually these dimensions are designated basic by the term **BSC** or by a general note **DIMENSIONS W/O TOL ARE BSC** specified in the title block. The draft angle specified in the dimension and tolerance block of casting, forging, and molded part drawings/models is also designated as basic for determining the basic profile of drafted unmachined surfaces which remain on the completed component and are controlled by profile of a surface tolerances. Some dimensions are not designated **BSC** because they serve a dual purpose. For example, a dimension between machined faces usually includes a thickness tolerance, but the dimension is also basic for determining the basic profile of surfaces controlled by profile of a surface tolerance.

11.5 Former Designation - Drawings which have the former designation for profile tolerance such as 

shall be interpreted as 

11.6 Profile Of A Line - Profile of a line tolerance 

is applied to restrict line elements within profile of a surface.

11.6.1 All points of each line element shall be within a zone having the specified width. The line profile zone shall be parallel to the basic profile but the entire zone may float within the specified profile of a surface tolerance zone. (See Figure 126).

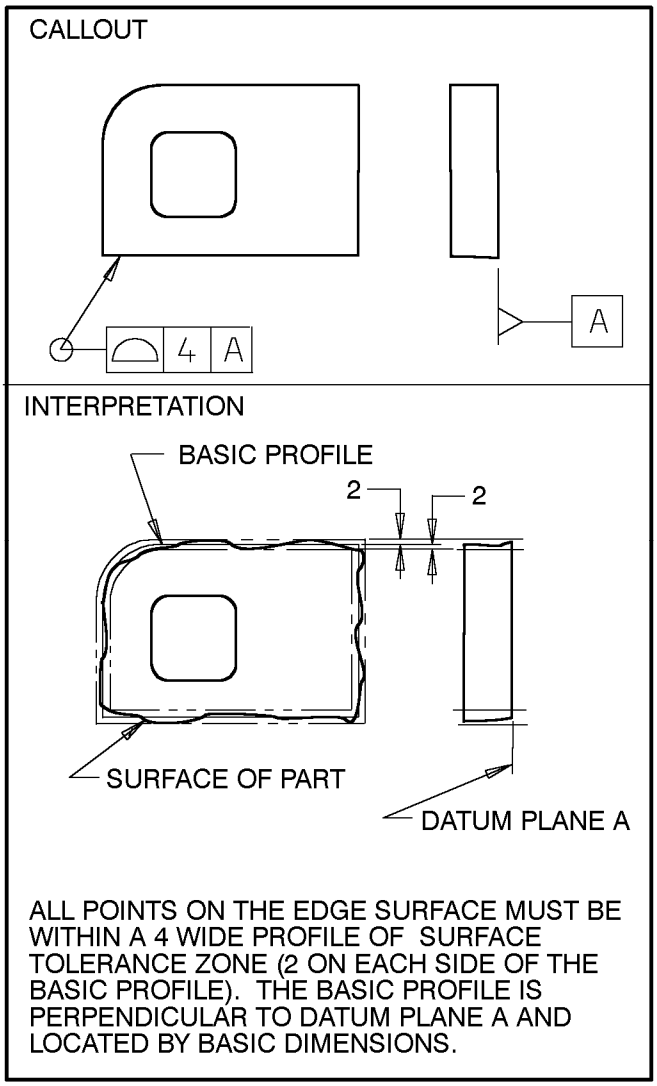


Figure 109 - Profile Of A Surface Tolerance All Around With No Locating Datums

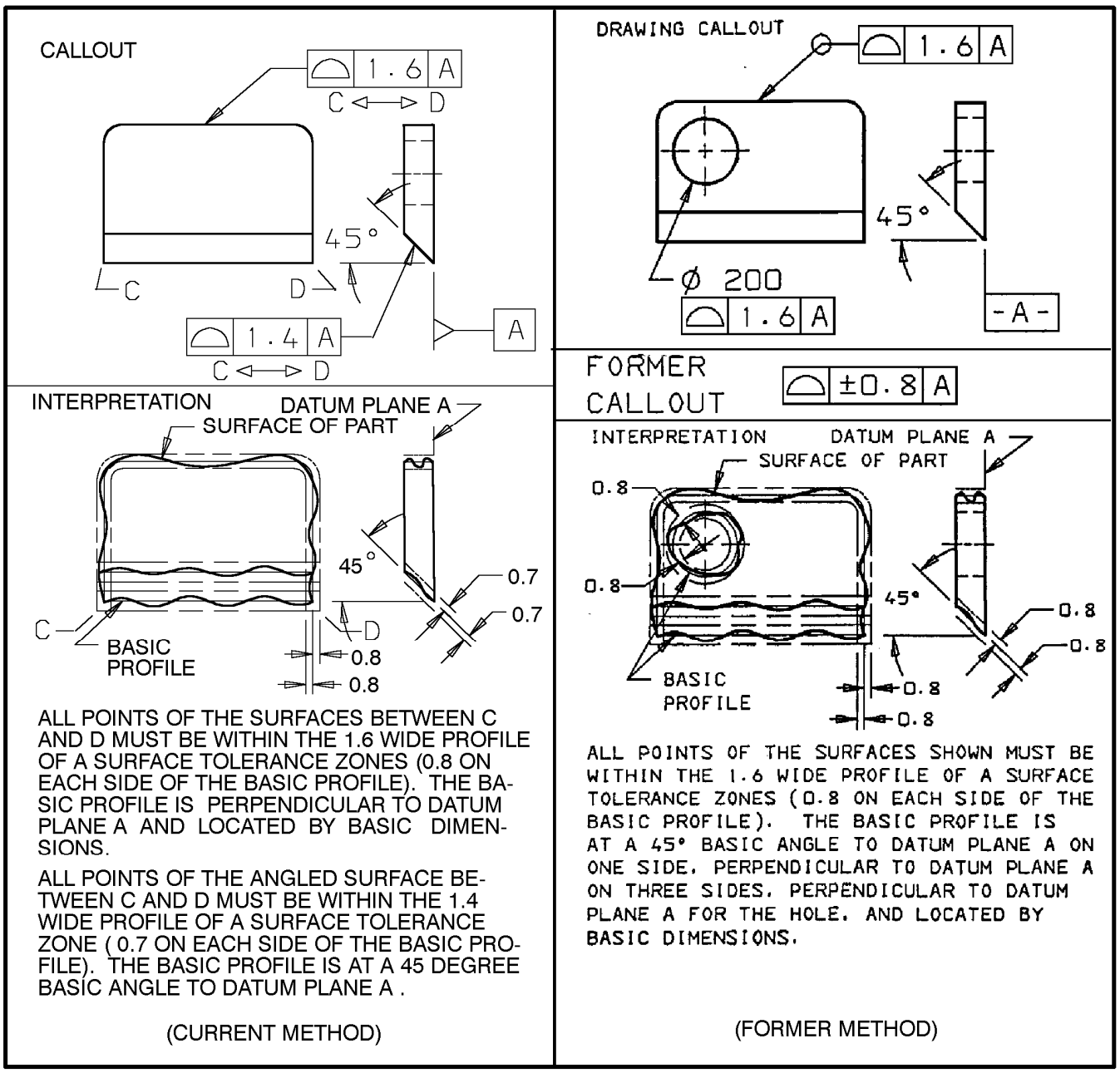


Figure 110 - Profile Of A Surface Tolerance With No Locating Datums

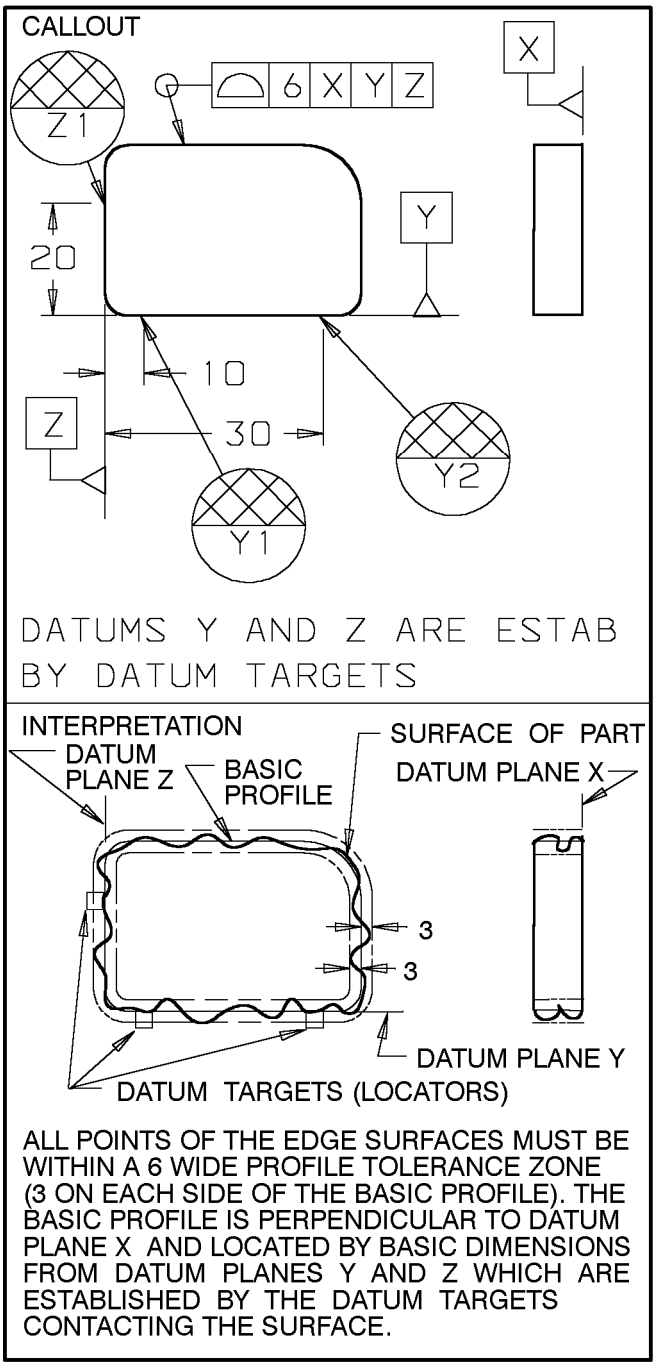


Figure 111 - Profile Of A Surface Tolerance Located From Datums Established By Datum Targets On The Controlled Profile

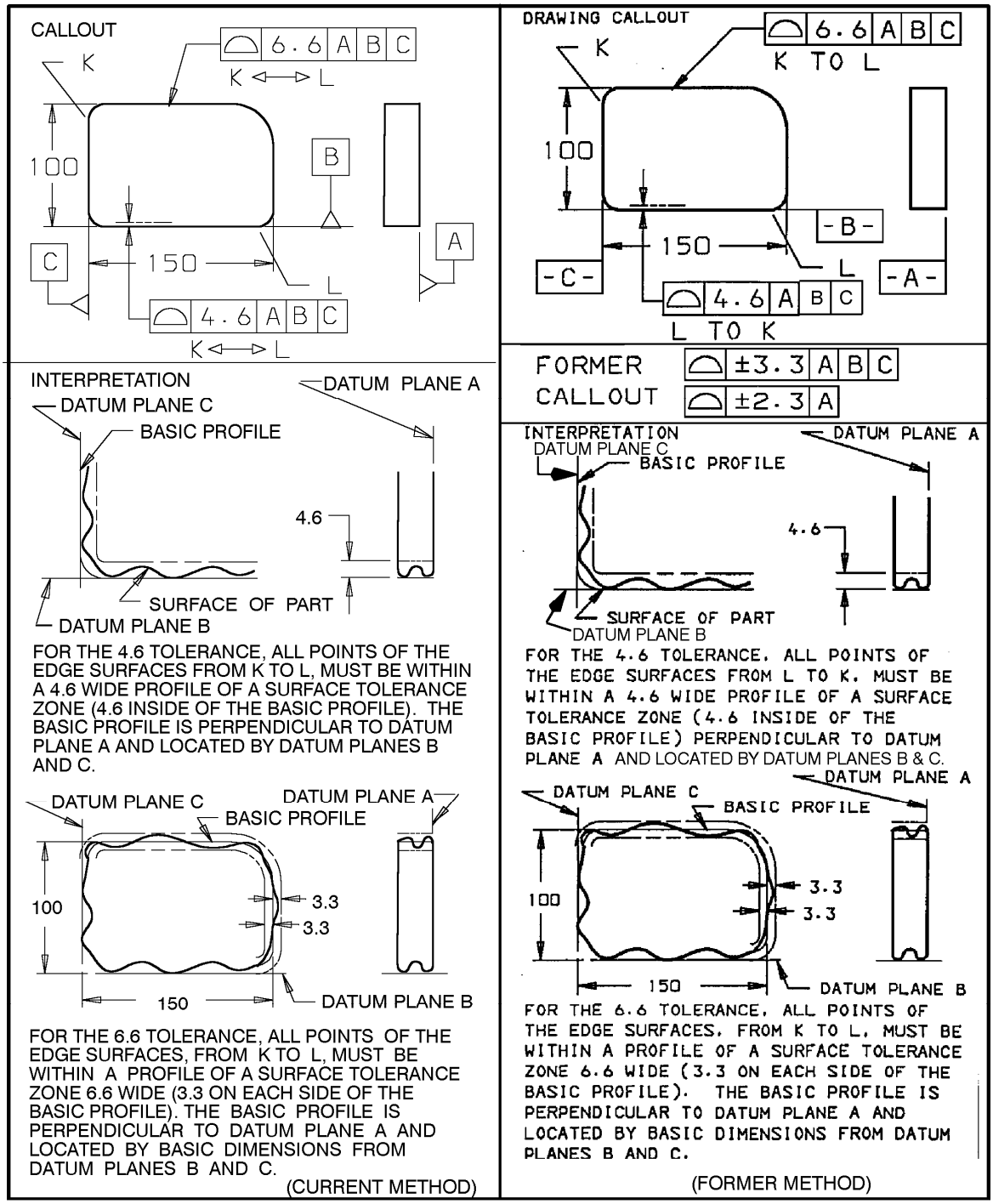


Figure 112 - Profile Of A Surface Tolerance Located From Datum Planes On Controlled Profile

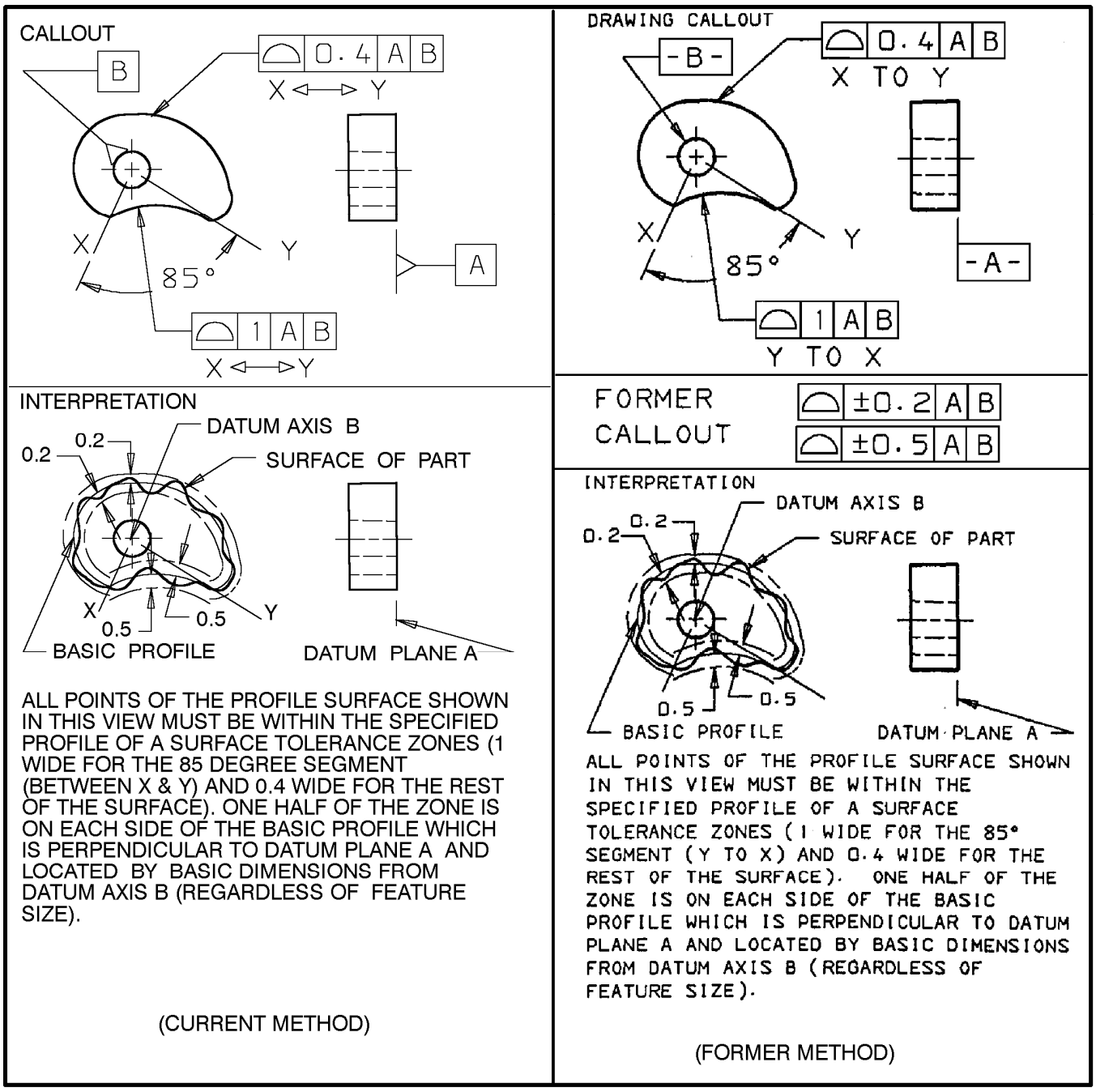


Figure 113 - Profile Of A Surface Tolerance Applied To Portions Of A Feature

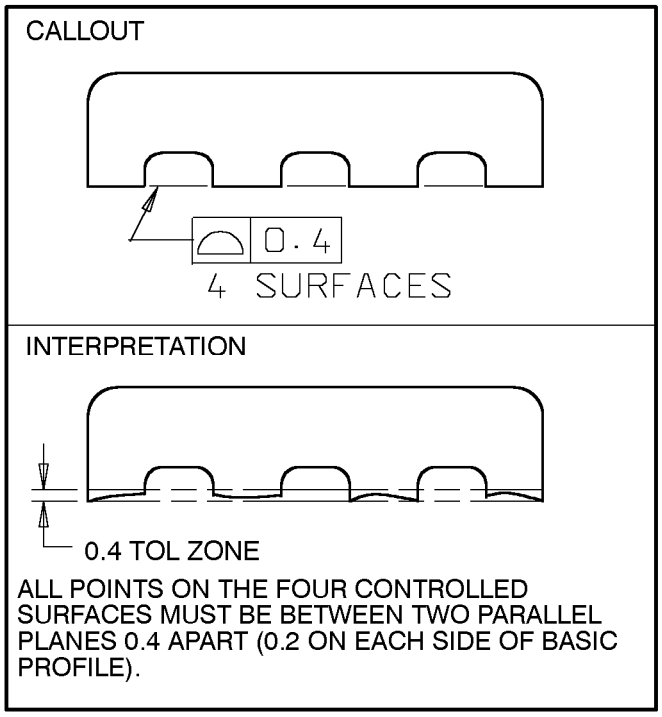


Figure 114 - Profile Of A Surface For Coplanar Surfaces

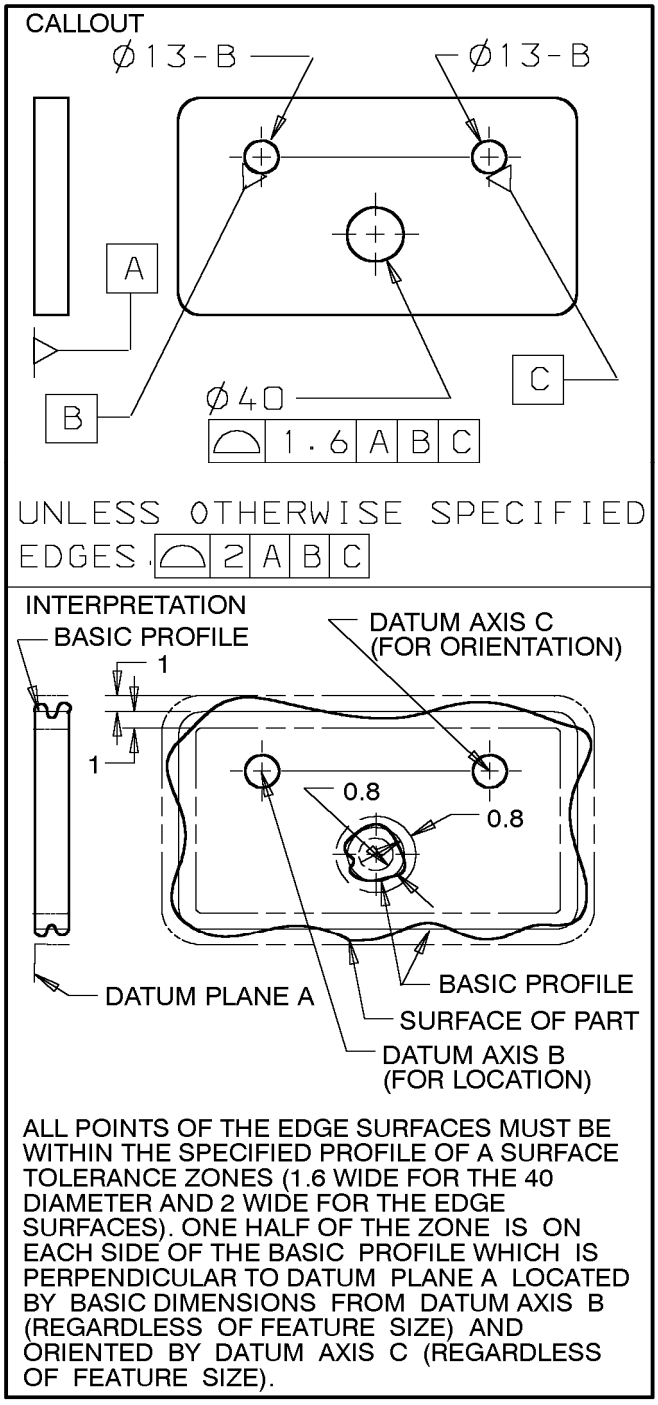


Figure 115 - Profile Of A Surface Tolerance For Edges Of A View

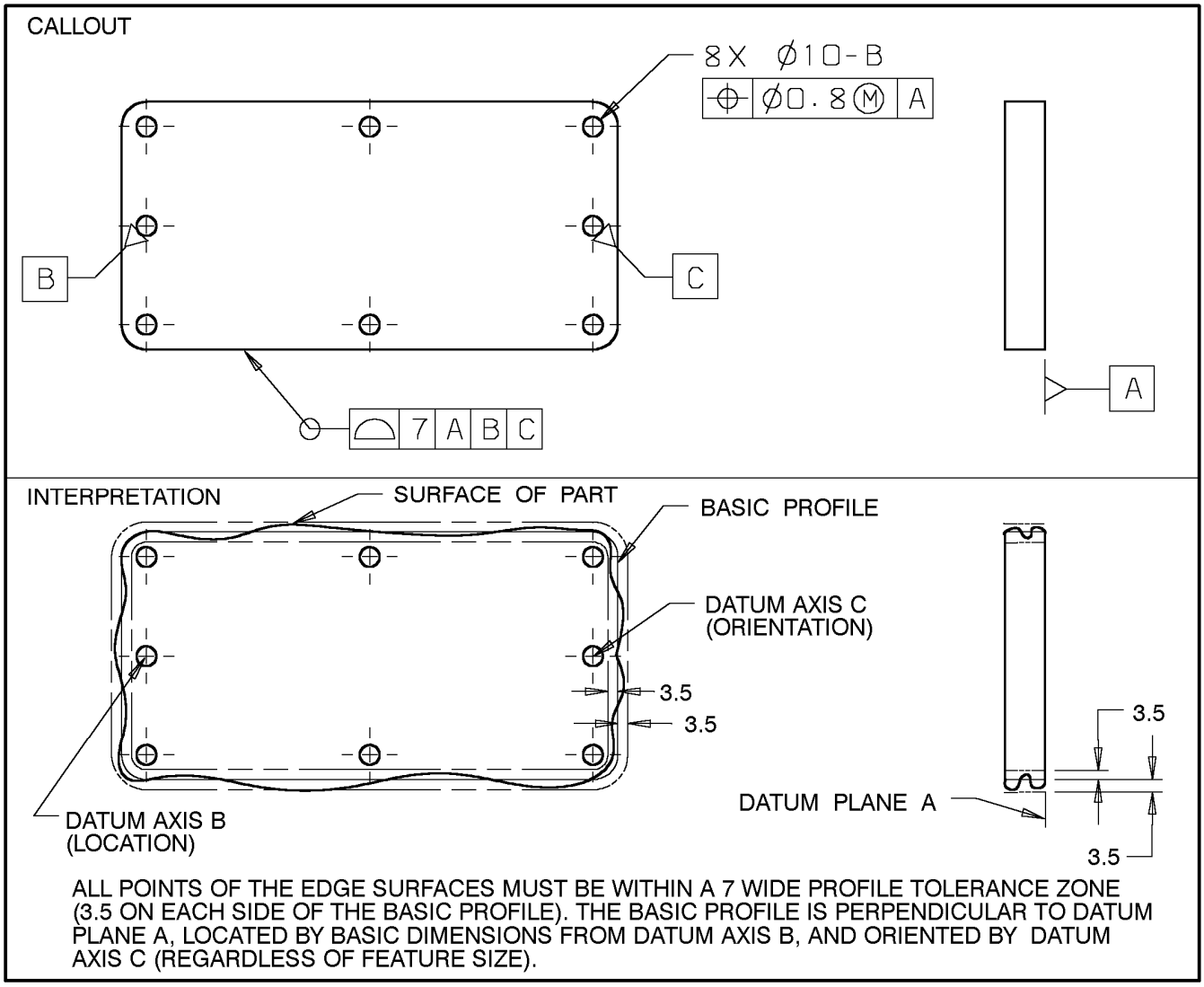
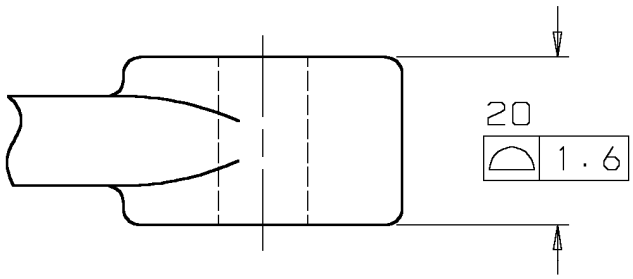
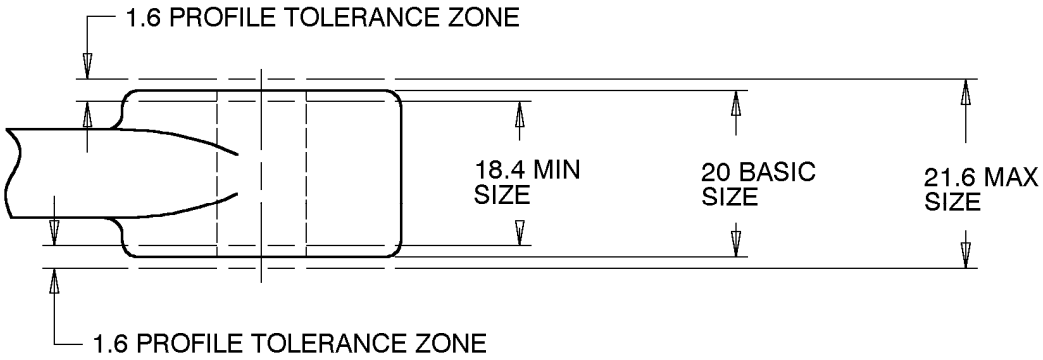


Figure 116 - Profile Of A Surface Tolerance Located From Datums Established By Features Not Controlled By The Profile Tolerance

CALLOUT



INTERPRETATION



ALL POINTS OF THE TWO SURFACES DEFINED BY THE 20 BASIC DIMENSION MUST BE WITHIN THE 1.6 WIDE PROFILE OF A SURFACE TOLERANCE ZONE. ONE HALF OF THE TOLERANCE ZONE IS ON EACH SIDE OF THE BASIC PROFILE FOR EACH SURFACE. MINIMUM SIZE FOR THE FEATURE IS 18.4 AND MAXIMUM SIZE IS 21.6.

Figure 117 - Profile Of A Surface Tolerance To Control Feature Size

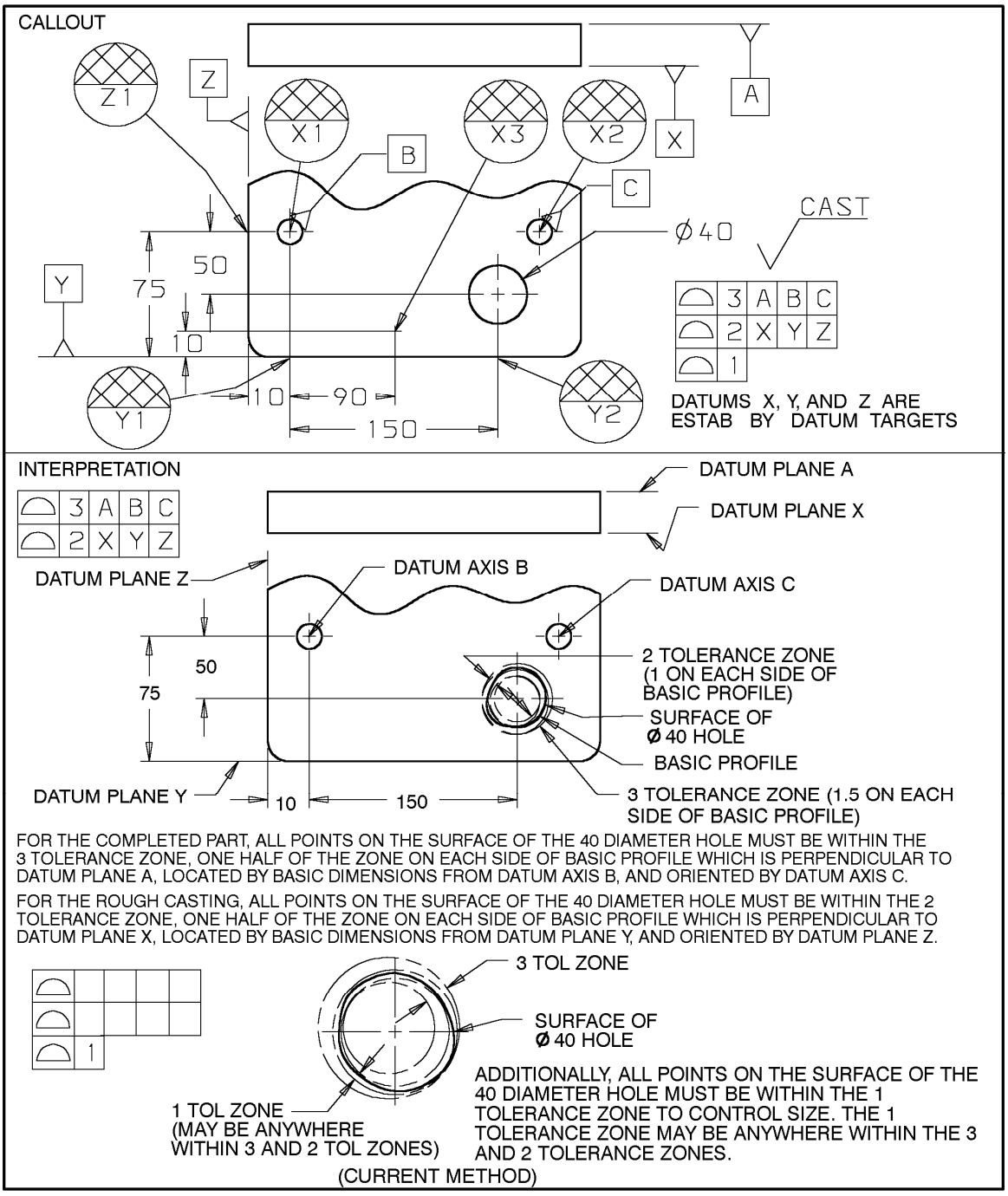


Figure 118 - Profile Of A Surface Tolerance To Control Both Location And Size Of A Feature (Continued on Next Page)

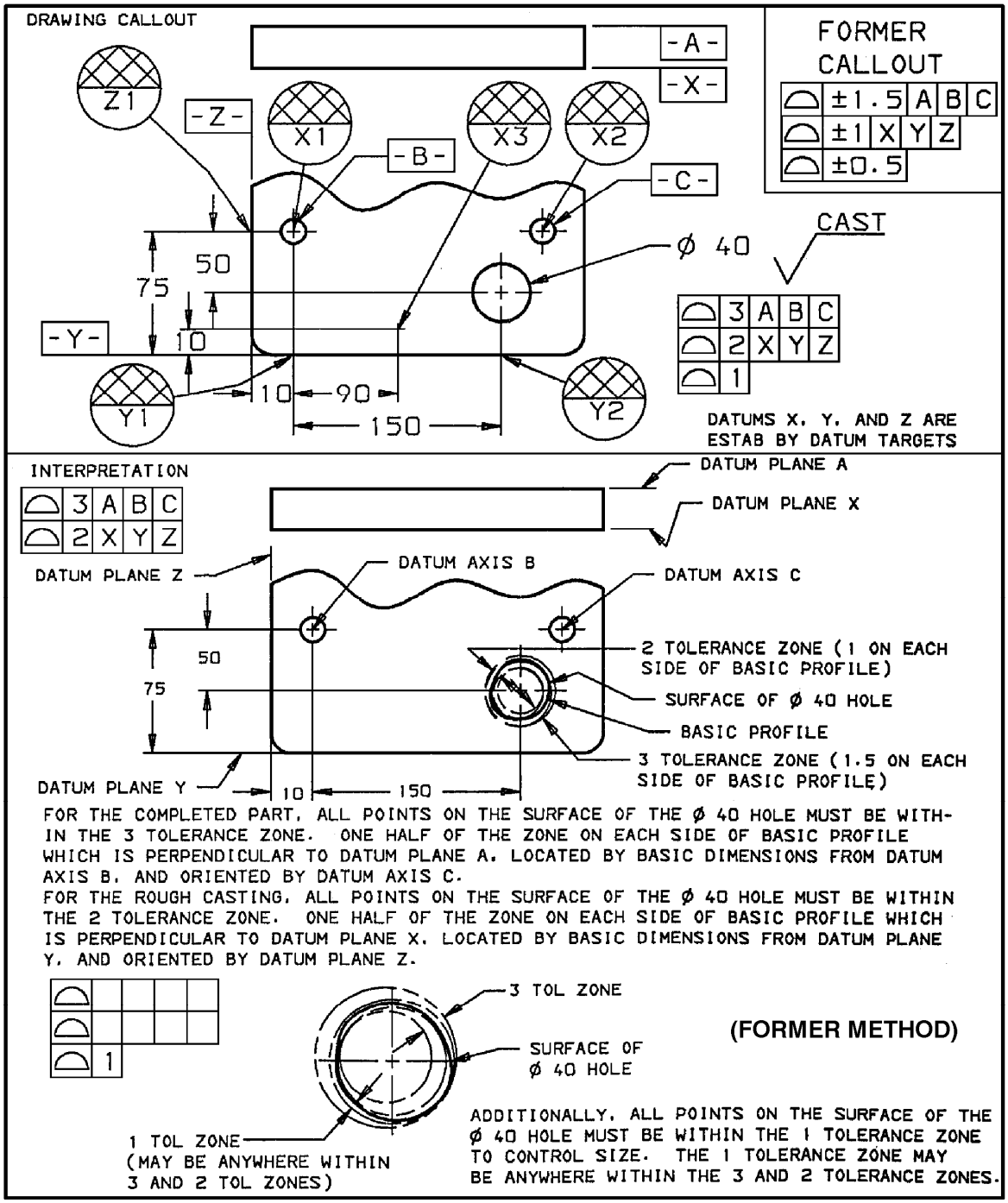


Figure 118 - Profile Of A Surface Tolerance To Control Both Location And Size Of A Feature (Continued)

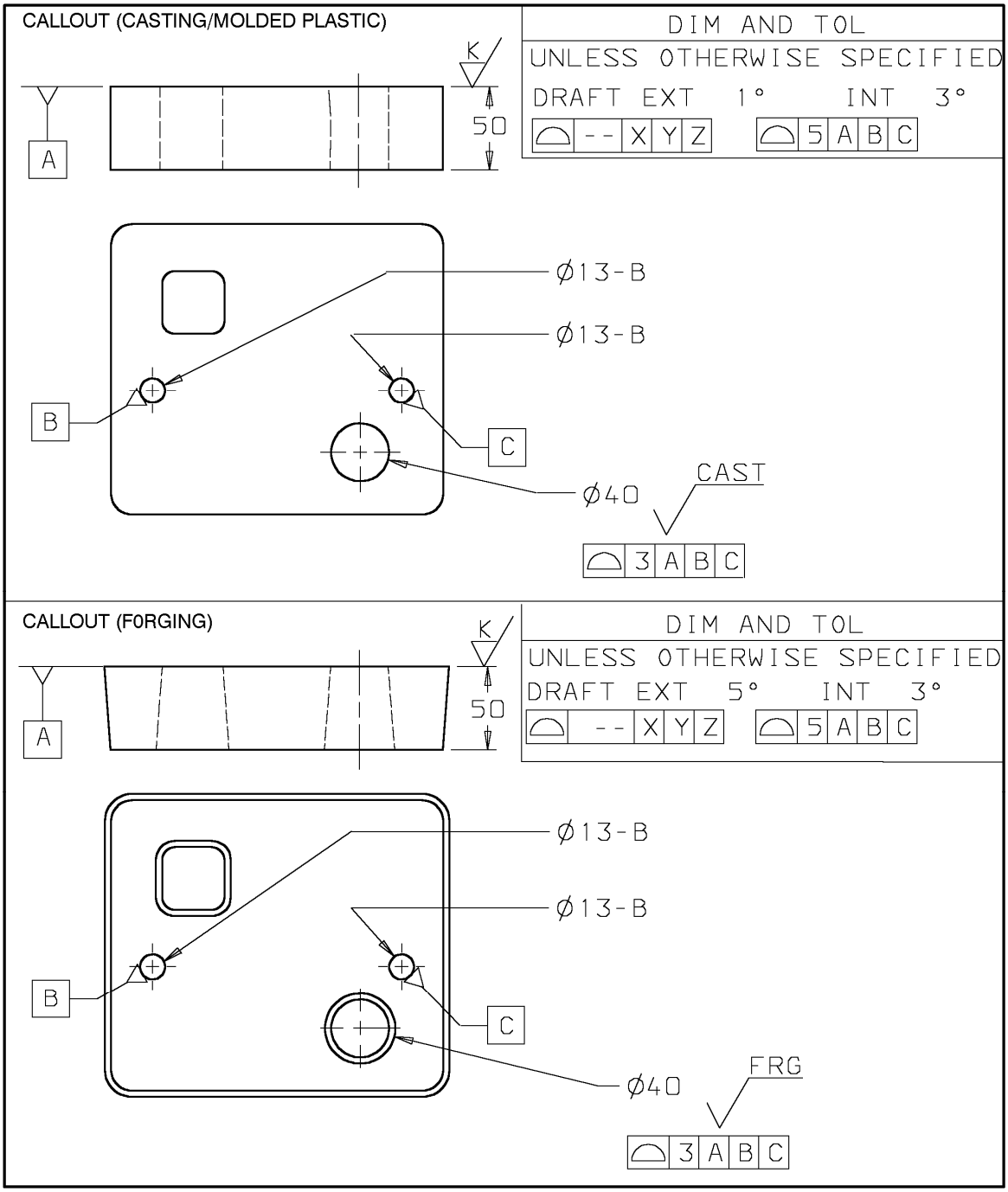
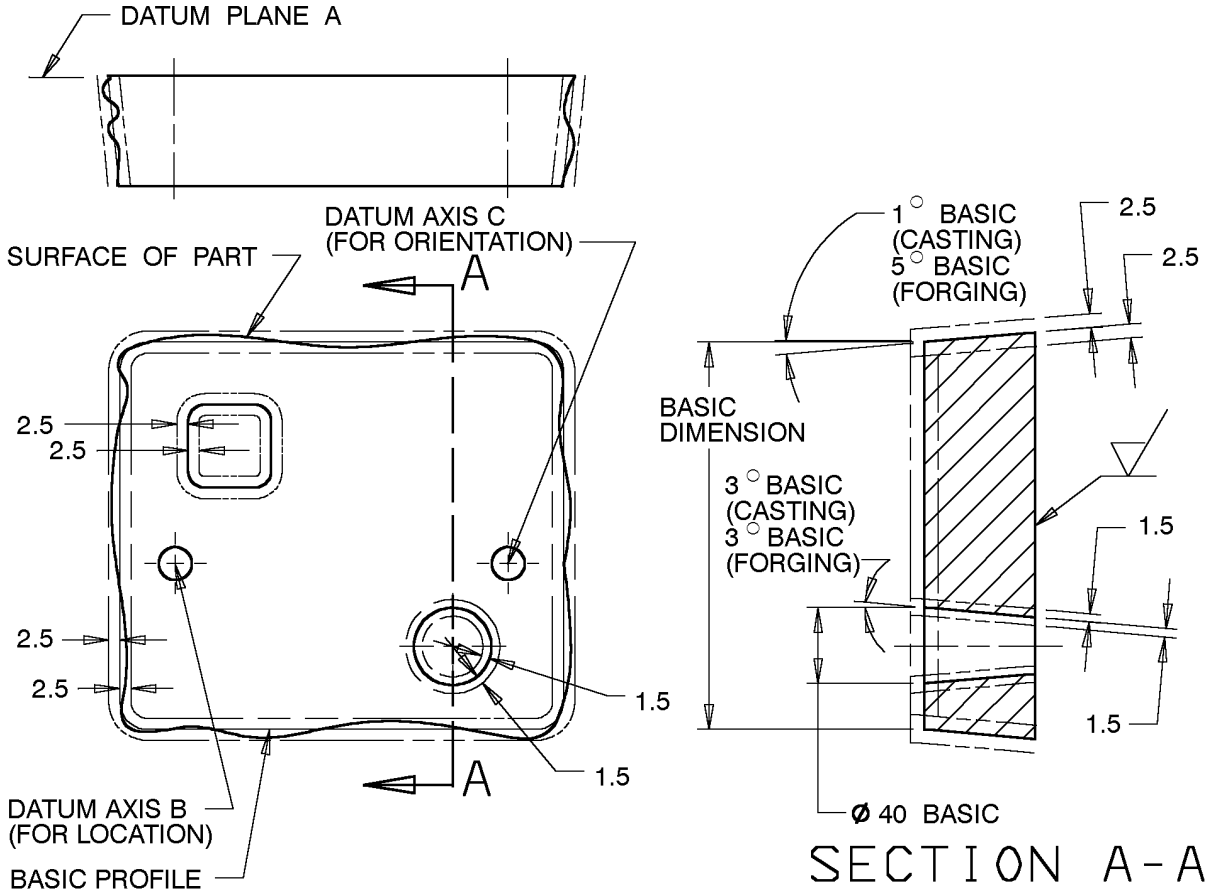


Figure 119 - Profile Of A Surface Control Of Cast, Forged, Or Molded Plastic Surfaces On The Completed Component (Callouts)

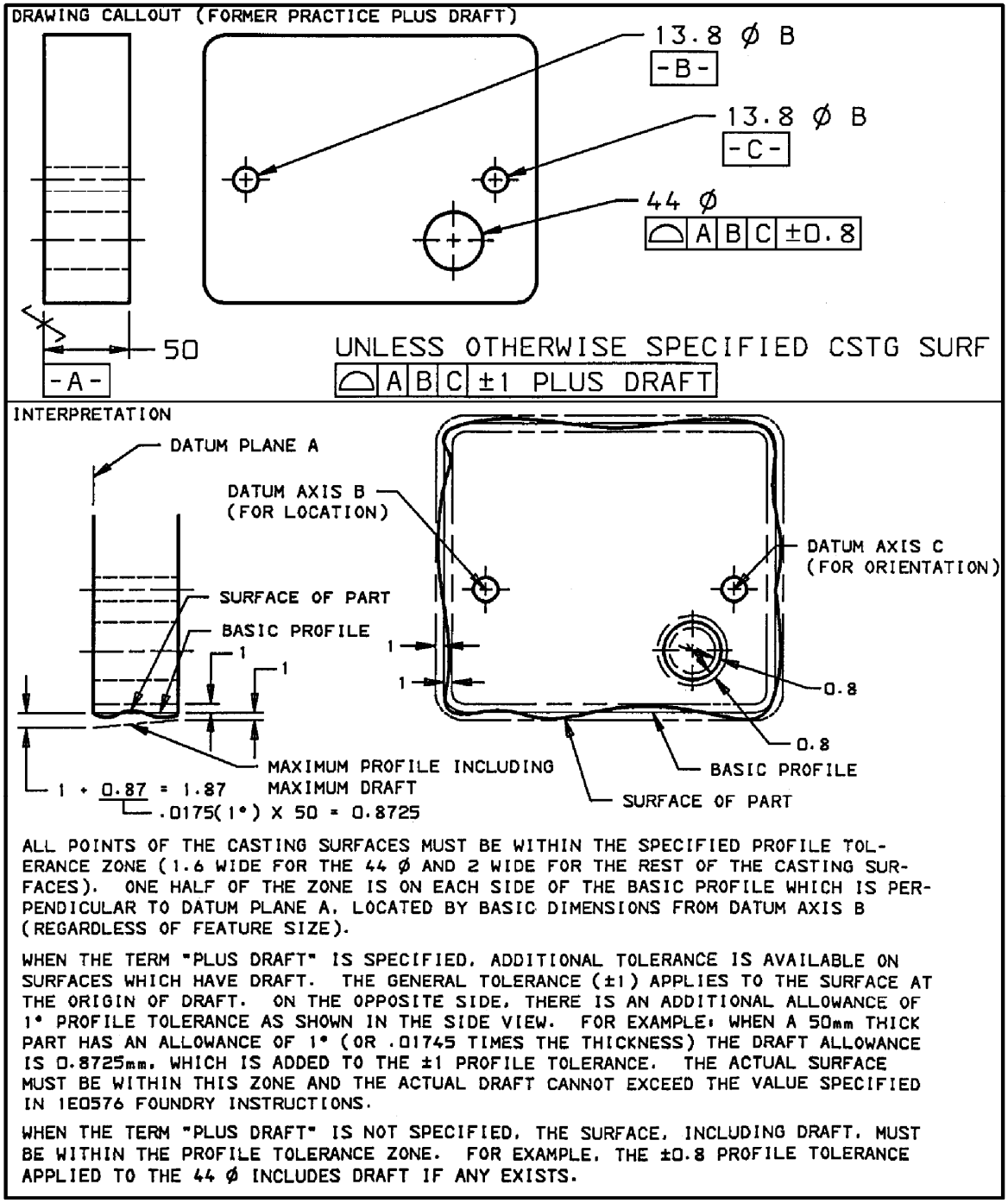
INTERPRETATION



ALL POINTS OF THE NONMACHINED SURFACES MUST BE WITHIN THE SPECIFIED PROFILE TOLERANCE ZONES (3 WIDE FOR THE 40 DIAMETER AND 5 WIDE FOR THE REST OF THE NON-MACHINED SURFACES). ONE HALF OF THE ZONE IS ON EACH SIDE OF THE BASIC PROFILE WHICH IS POSITIONED RELATIVE TO DATUM PLANE A, LOCATED BY BASIC DIMENSIONS FROM DATUM AXIS B (REGARDLESS OF FEATURE SIZE), AND ORIENTATED BY DATUM AXIS C (REGARDLESS OF FEATURE SIZE). REFER TO SPECIFICATIONS OF ROUGH AND FINISHED CASTINGS AND FORGINGS FOR DRAFT REQUIREMENTS.

THE DRAFT ANGLE IS APPLIED AS A BASIC DIMENSION WHEREVER DRAFT EXISTS ON CASTINGS, AND WHEREVER DRAFT IS SHOWN ON FORGINGS.

Figure 120 - Profile Of A Surface Control Of Cast, Forged, Or Molded Plastic Surfaces On The Completed Component (Interpretation)



ALL POINTS OF THE CASTING SURFACES MUST BE WITHIN THE SPECIFIED PROFILE TOLERANCE ZONE (1.6 WIDE FOR THE 44 Ø AND 2 WIDE FOR THE REST OF THE CASTING SURFACES). ONE HALF OF THE ZONE IS ON EACH SIDE OF THE BASIC PROFILE WHICH IS PERPENDICULAR TO DATUM PLANE A, LOCATED BY BASIC DIMENSIONS FROM DATUM AXIS B (REGARDLESS OF FEATURE SIZE).

WHEN THE TERM "PLUS DRAFT" IS SPECIFIED, ADDITIONAL TOLERANCE IS AVAILABLE ON SURFACES WHICH HAVE DRAFT. THE GENERAL TOLERANCE (±1) APPLIES TO THE SURFACE AT THE ORIGIN OF DRAFT. ON THE OPPOSITE SIDE, THERE IS AN ADDITIONAL ALLOWANCE OF 1° PROFILE TOLERANCE AS SHOWN IN THE SIDE VIEW. FOR EXAMPLE: WHEN A 50mm THICK PART HAS AN ALLOWANCE OF 1° (OR .01745 TIMES THE THICKNESS) THE DRAFT ALLOWANCE IS 0.8725mm, WHICH IS ADDED TO THE ±1 PROFILE TOLERANCE. THE ACTUAL SURFACE MUST BE WITHIN THIS ZONE AND THE ACTUAL DRAFT CANNOT EXCEED THE VALUE SPECIFIED IN 1E0576 FOUNDRY INSTRUCTIONS.

WHEN THE TERM "PLUS DRAFT" IS NOT SPECIFIED, THE SURFACE, INCLUDING DRAFT, MUST BE WITHIN THE PROFILE TOLERANCE ZONE. FOR EXAMPLE, THE ±0.8 PROFILE TOLERANCE APPLIED TO THE 44 Ø INCLUDES DRAFT IF ANY EXISTS.

Figure 121 - Profile Of A Surface Tolerance On Casting (Former Method)

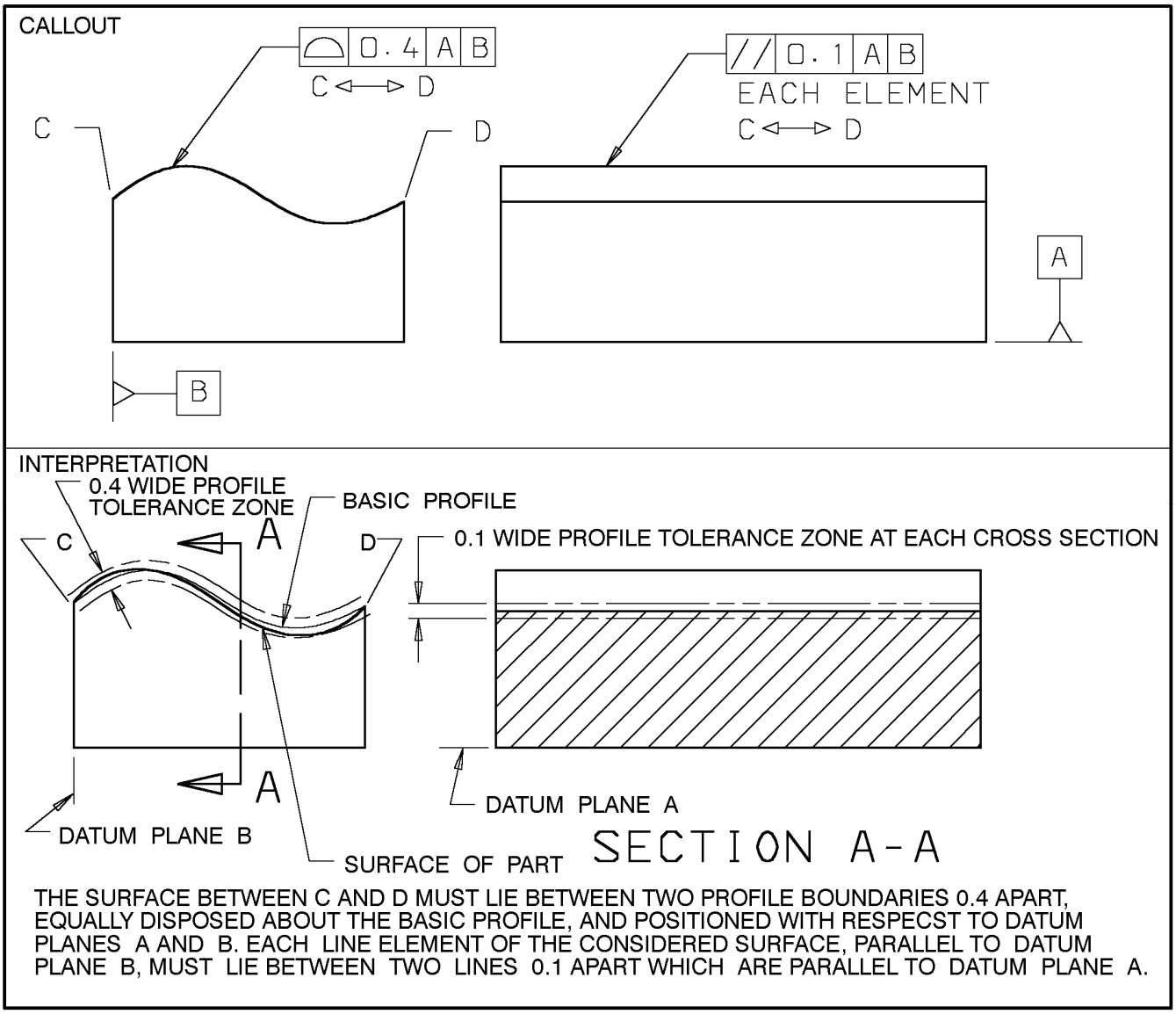


Figure 122 - Combined Profile Of A Surface And Parallelism Tolerance

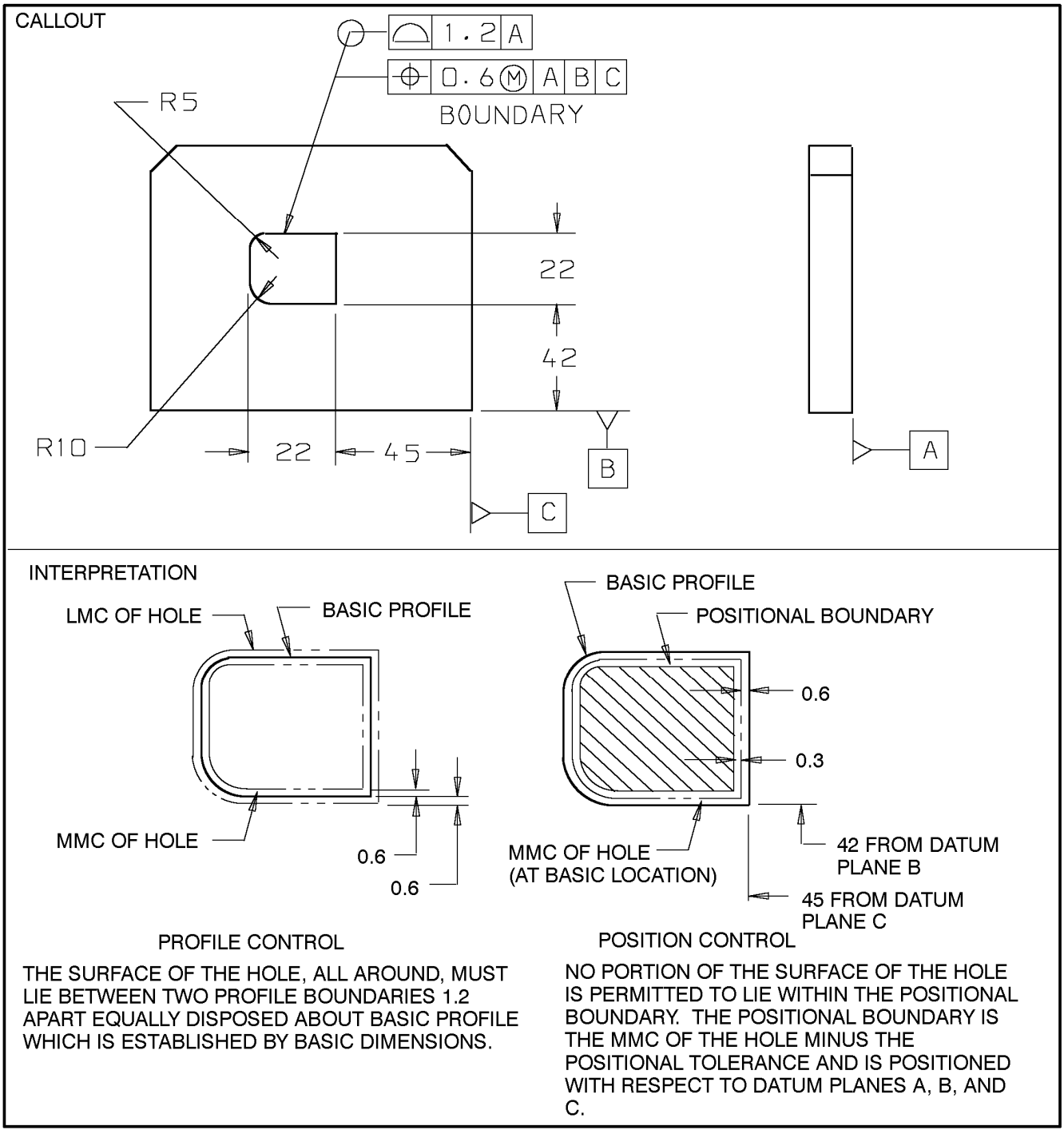


Figure 123 - Profile Of A Surface Tolerance Used With Positional Tolerance

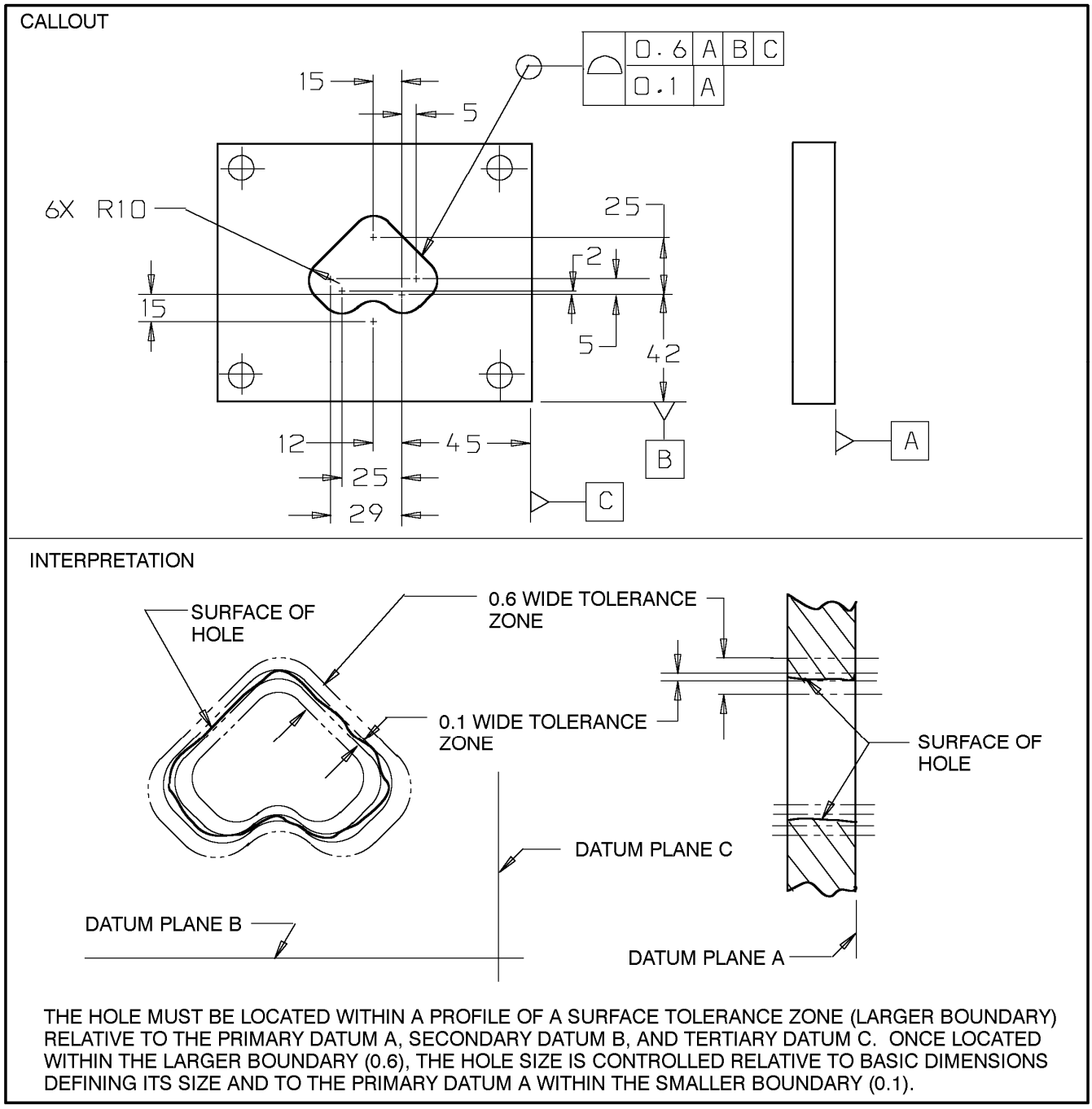


Figure 124 - Composite Profile Tolerancing Of An Irregular Surface

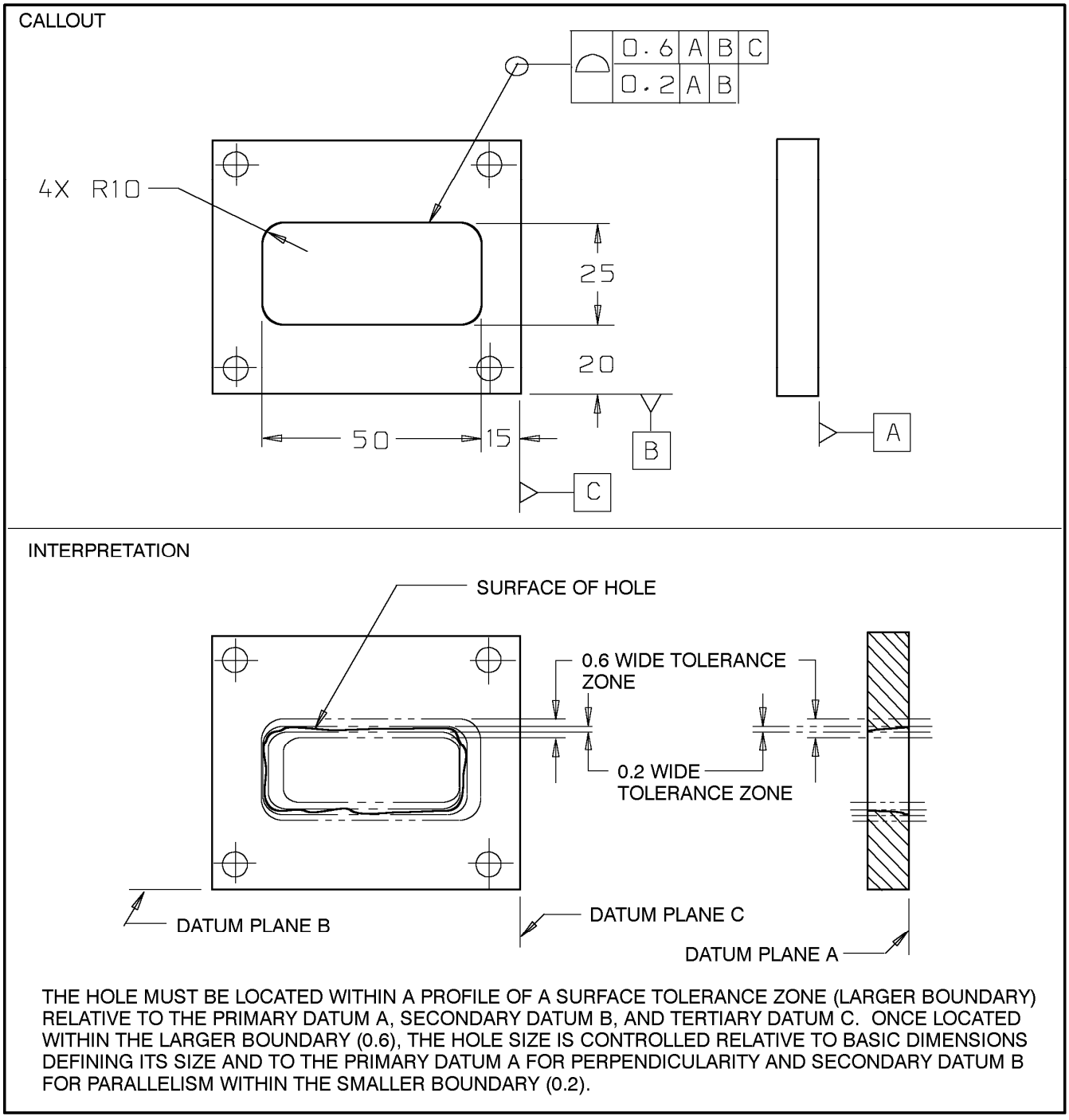


Figure 125 - Composite Profile Tolerancing Of A Feature (Orientation)

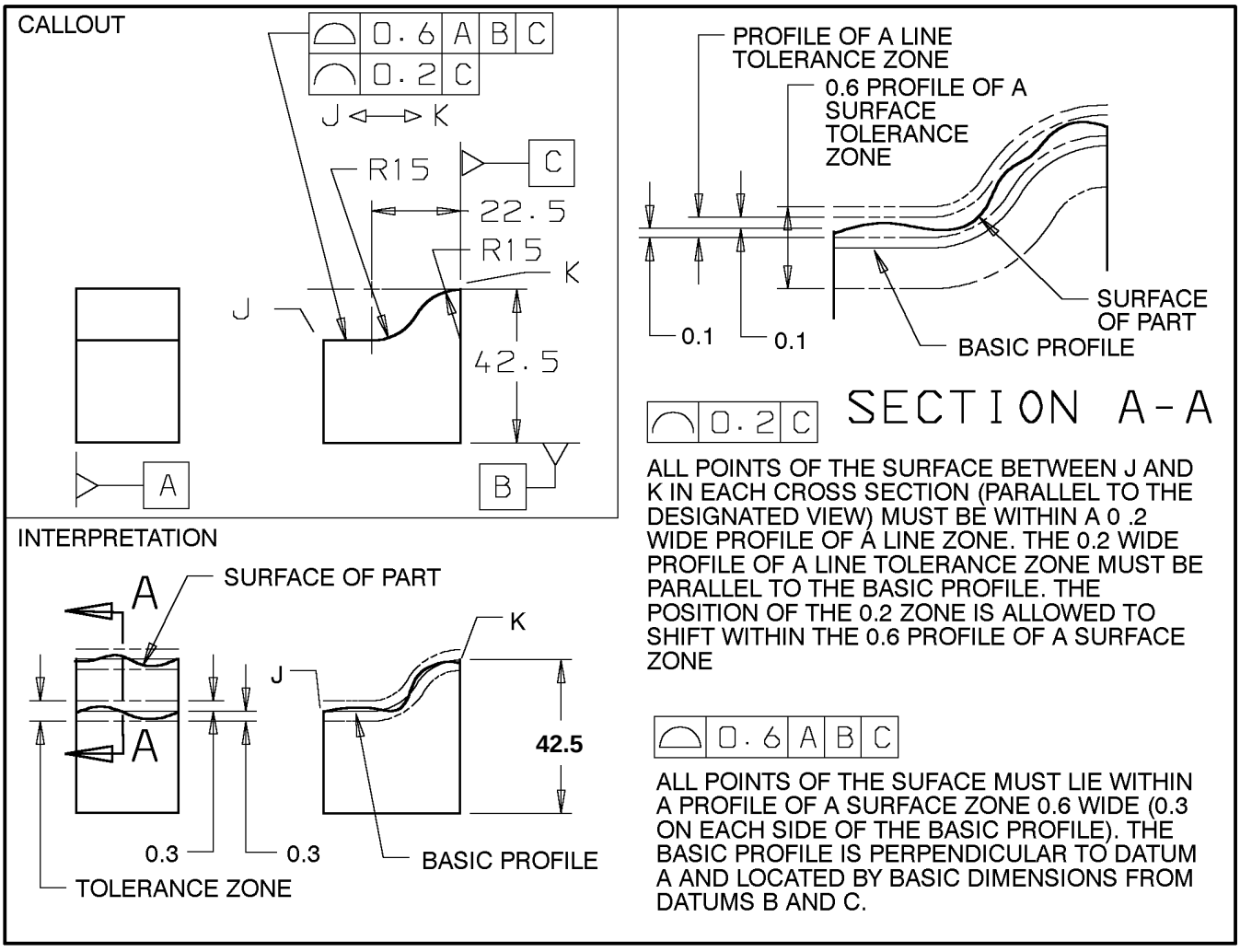


Figure 126 - Restriction Of A Line Element Within Profile Of A Surface Tolerance Zone

12.0 REFERENCES

Caterpillar Specifications	1E0011, 1E0100, 1E0392, 1E0576, 1E2028, 1E2177, 1E2315, 1E2324, 1E2325, 1E2600
Caterpillar Manufacturing Practices	MG1010
ASME	Y14.5
ISO	R1101, 1660