Aerospace and Advanced Engineering Drawings

AN AMERICAN NATIONAL STANDARD

The American Society of
Mechanical Engineers

Aerospace and Advanced Engineering Drawings

The next edition of this Standard is scheduled for publication in 2023.

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Errata to codes and standards may be posted on the ASME website under the Committee Pages to provide corrections to incorrectly published items, or to correct typographical or grammatical errors in codes and standards. Such errata shall be used on the date posted.

The AED Committee Page can be found at http://go.asme.org/AEDcommittee. There is an option available to automatically receive an e-mail notification when errata are posted to a particular code or standard. This option can be found on the appropriate Committee Page after selecting "Errata" in the "Publication Information" section.

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FOREWORD

Since many major industries are increasingly globalized, resulting in the decentralization of design and manufacturing activities, it is imperative that design documentation more completely and accurately identify functional requirements. In 2007, a cross-functional group met in Washington, DC, to discuss the business pressures and needs of the aerospace industry. From this meeting, a consensus was reached to explore the creation of a new standard that would enable those in the aerospace industry to harmonize best practices and further standardize the creation and interpretation of engineering drawings. Additional meetings highlighted the need to expand the scope of the group's efforts beyond the aerospace industry, and the ASME Aerospace and Advanced Engineering Drawing (AED) Committee now represents all advanced manufacturing technologies.

ASME AED-1 provides a method for documenting the design requirements that are common to aerospace and other industries that use advanced manufacturing technologies. This Standard offers symbologies, terminologies, and concepts to enhance the understanding and abilities of those who create and use design documentation. It is hoped that a common documentation methodology will decrease design and manufacturing costs and improve quality.

This Standard is not intended to replace any of the ASME Y14 series of standards for engineering drawings. Rather, it functions as a supplement to the ASME Y14 series, allowing use of defined symbologies, terminologies, and concepts until the appropriate ASME Y14 subcommittee adopts the principles expressed herein.

As this Standard evolves, many of the concepts introduced herein will migrate to more appropriate ASME standards. When this occurs, the affected information will be moved into an appendix within ASME AED-1. In this manner, control and ownership of the subject matter will be placed with the proper committee, but the history and usage will remain visible within ASME AED-1.

This Standard is available for public review on a continuing basis. This provides an opportunity for additional public-review input from industry, academia, regulatory agencies, and the public-at-large.

This Standard was approved by the AED Standards Committee. It was approved as an American National Standard by the American National Standards Institute (ANSI) Board of Standards Review on September 25, 2018.

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Aerospace and Advanced Engineering Drawings

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General.   ASME Standards are developed and maintained with the intent to represent the consensus of concerned interests. As such, users of this Standard may interact with the Committee by requesting interpretations, proposing revisions or a case, and attending Committee meetings. Correspondence should be addressed to:

> Secretary, AED Standards Committee
> The American Society of Mechanical Engineers
> Two Park Avenue
> New York, NY 10016-5990
> http://go.asme.org/Inquiry

Proposing Revisions.   Revisions are made periodically to the Standard to incorporate changes that appear necessary or desirable, as demonstrated by the experience gained from the application of the Standard. Approved revisions will be published periodically.

The Committee welcomes proposals for revisions to this Standard. Such proposals should be as specific as possible, citing the paragraph number(s), the proposed wording, and a detailed description of the reasons for the proposal, including any pertinent documentation.

Proposing a Case.   Cases may be issued to provide alternative rules when justified, to permit early implementation of an approved revision when the need is urgent, or to provide rules not covered by existing provisions. Cases are effective immediately upon ASME approval and shall be posted on the ASME Committee web page.

Requests for Cases shall provide a Statement of Need and Background Information. The request should identify the Standard and the paragraph, figure, or table number(s), and be written as a Question and Reply in the same format as existing Cases. Requests for Cases should also indicate the applicable edition(s) of the Standard to which the proposed Case applies.

Attending Committee Meetings.   The AED Standards Committee regularly holds meetings and/or telephone conferences that are open to the public. Persons wishing to attend any meeting and/or telephone conference should contact the Secretary of the AED Standards Committee. Future Committee meeting dates and locations can be found on the Committee Page at http://go.asme.org/AEDcommittee.

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AEROSPACE AND ADVANCED ENGINEERING DRAWINGS

1 GENERAL

1.1 Scope

This Standard provides a method to document requirements that are common across aerospace and other industries that use advanced manufacturing technologies. This Standard offers symbologies, terminologies, and concepts to enhance the understanding and abilities of those who create and use design documentation.

1.2 Conventions

The conventions in paras. 1.2.1 through 1.2.10 are used in this Standard. With the exception of the system of units described in para. 1.2.7, these conventions are similar to the conventions used in the ASME Y14 standards.

1.2.1 Mandatory, Nonmandatory, Guidance, and Optional Words

(a) The words "shall" and "will" establish a mandatory requirement.

(b) The words "should" and "may" establish a recommended practice.

(c) The words "typical," "example," "for reference," and the Latin abbreviation "e.g." indicate suggestions given for guidance only.

(d) The word "or" used in conjunction with a mandatory requirement or a recommended practice indicates that there are two or more options on how to comply with the stated requirement.

1.2.2 Cross-Reference of Standards.
Cross-reference of standards in text with or without a date following the standard designator shall be interpreted as follows:

(a) Reference to ASME Y14 standards in the text without a date following the standard designator indicates that the issue of the standard identified in the References section (section 2) shall be used to meet the requirement.

(b) Reference to ASME Y14 standards in the text with a date following the standard designator indicates that only that issue of the standard shall be used to meet the requirement.

1.2.3 Invocation of Referenced Standards.
The following examples define the invocation of a standard when specified in the References section (section 2) and referenced in the text of this Standard:

(a) When a reference standard is cited in the text with no limitations to a specific subject or paragraph(s) of the standard, the entire standard is invoked, e.g., "Dimensioning and tolerancing shall be in accordance with ASME Y14.5" is invoking the complete standard because the subject of the standard is dimensioning and tolerancing and no specific subject or paragraph(s) within the standard are invoked.

(b) When a referenced standard is cited in the text with limitations to a specific subject or paragraph(s) of the standard, only the paragraph(s) on that subject are invoked, e.g., "Assign part or identifying numbers in accordance with ASME Y14.100" is invoking only the paragraph(s) on part or identifying numbers because the subject of the standard is engineering drawing practices, and part or identifying numbers is a specific subject within the standard.

(c) When a reference standard is cited in the text without an invoking statement such as "in accordance with," the standard is for guidance only; e.g., "For gaging principles, see ASME Y14.43" is for guidance only, and no portion of the standard is invoked.

1.2.4 Parentheses Following a Definition.
When a definition is followed by a standard referenced in parentheses, the standard referenced in parentheses is the source for the definition.

1.2.5 Notes.
Notes depicted in this Standard in ALL UPPERCASE letters are intended to reflect actual drawing entries. Notes depicted in initial uppercase or lowercase letters are to be considered supporting data to the contents of this Standard and are not intended for literal entry on drawings. A statement requiring the addition of a note with the qualifier "such as" is a requirement to add a note, and the content of the text is allowed to vary to suit the application.

1.2.6 Acronyms and Abbreviations.
Acronyms and abbreviations are spelled out the first time used in this Standard, followed by the acronym or abbreviation in parentheses. The acronym or abbreviation is used thereafter throughout the text.

1.2.7 Units.
U.S. Customary units are featured in this Standard. It should be understood that the International System of Units (SI) could equally have been used without prejudice to the principles established.

1.2.8 Figures. The figures in this Standard are intended only as illustrations to aid the user in understanding the practices described in the text. In some cases, figures show a level of detail as needed for emphasis. In other cases, figures are incomplete by intent so as to illustrate a concept or facet thereof. The absence of figure(s) has no bearing on the applicability of the stated requirements or practice. To comply with the requirements of this Standard, actual data sets shall meet the content requirements set forth in the text. To assist the user of this Standard, a list of the paragraph(s) that refer to an illustration appears in the lower right-hand corner of each figure. This list may not be all-inclusive. The absence of a list is not a reason to assume inapplicability. A lowercase "h" is used in figures for letter height or for symbol proportions (see Figure 1-1). Select the applicable letter height in accordance with ASME Y14.2.

1.2.9 Precedence of Standards. The following are ASME Y14 standards that are basic engineering drawing standards:

ASME Y14.1, Decimal Inch Drawing Sheet Size and Format
ASME Y14.1M, Metric Drawing Sheet Size and Format
ASME Y14.2, Line Conventions and Lettering
ASME Y14.3, Orthographic and Pictorial Views
ASME Y14.5, Dimensioning and Tolerancing
ASME Y14.24, Types and Applications of Engineering Drawings
ASME Y14.34, Associated Lists
ASME Y14.35, Revision of Engineering Drawings and Associated Documents
ASME Y14.36, Surface Texture Symbols
ASME Y14.38, Abbreviations and Acronyms for Use on Drawings and Related Documents
ASME Y14.41, Digital Product Definition Data Practices
ASME Y14.100, Engineering Drawing Practices

All other ASME Y14 standards are considered specialty types of standards and contain additional requirements or make exceptions to the basic standards as required to support a process or type of drawing.

1.2.10 Unless Otherwise Specified. The phrase "unless otherwise specified" (UOS) is used to indicate a default requirement. The phrase is used when the default is a generally applied requirement and an exception may be provided by another document or requirement.

1.3 Reference to This Standard

When drawings or data sets are based on this Standard, this fact shall be noted on the drawing or in the data set. A note similar to the following shall be added: "THIS DRAWING SHALL BE INTERPRETED IN ACCORDANCE WITH ASME Y14.100-2017 EXCEPT AS AMENDED BY ASME AED-1–2018."
When this Standard in its entirety is not required, specific tailoring may be accomplished in a company interpretation document, drawing note, etc.

2 REFERENCES

The following revisions of American National Standards form a part of this Standard to the extent specified herein. A more recent revision may be used, provided there is no conflict with the text of this Standard. In the event of a conflict between the text of this Standard and the references cited herein, the text of this Standard shall take precedence.

ASME Y14.2-2014, Line Conventions and Lettering
ASME Y14.5-2009, Dimensioning and Tolerancing
ASME Y14.100-2017, Engineering Drawing Practices
Publisher: The American Society of Mechanical Engineers (ASME), Two Park Avenue, New York, NY 10016-5990
   (www.asme.org)

3 DEFINITIONS

3.1 Blend

blend: a transition between adjacent surfaces intended to reduce abrupt changes in the topography.

Figure 1-1 Form and Proportions of Symbols

Aggregate Datum Feature	Blind Hole

Aggregate Datum Feature

.8 h

30.0°

1.5 h ⟨AG⟩

2.5 h

5.4.1

Blind Hole

h BH 2 h

3 h

6.1.1

Modified Blind Hole

h BH XXX 2 h

6 h

6.1.2

Dynamic Profile

△ .8 h

.6 h

I-1

I-2

Reference Plane – Symbol Method

⌀ 5.5 h

.2 h

h LBL
1024.06 h

.2 h

4.3.2

Pilot Hole Location

X

2.5 h h

2.5 h

4.6

3
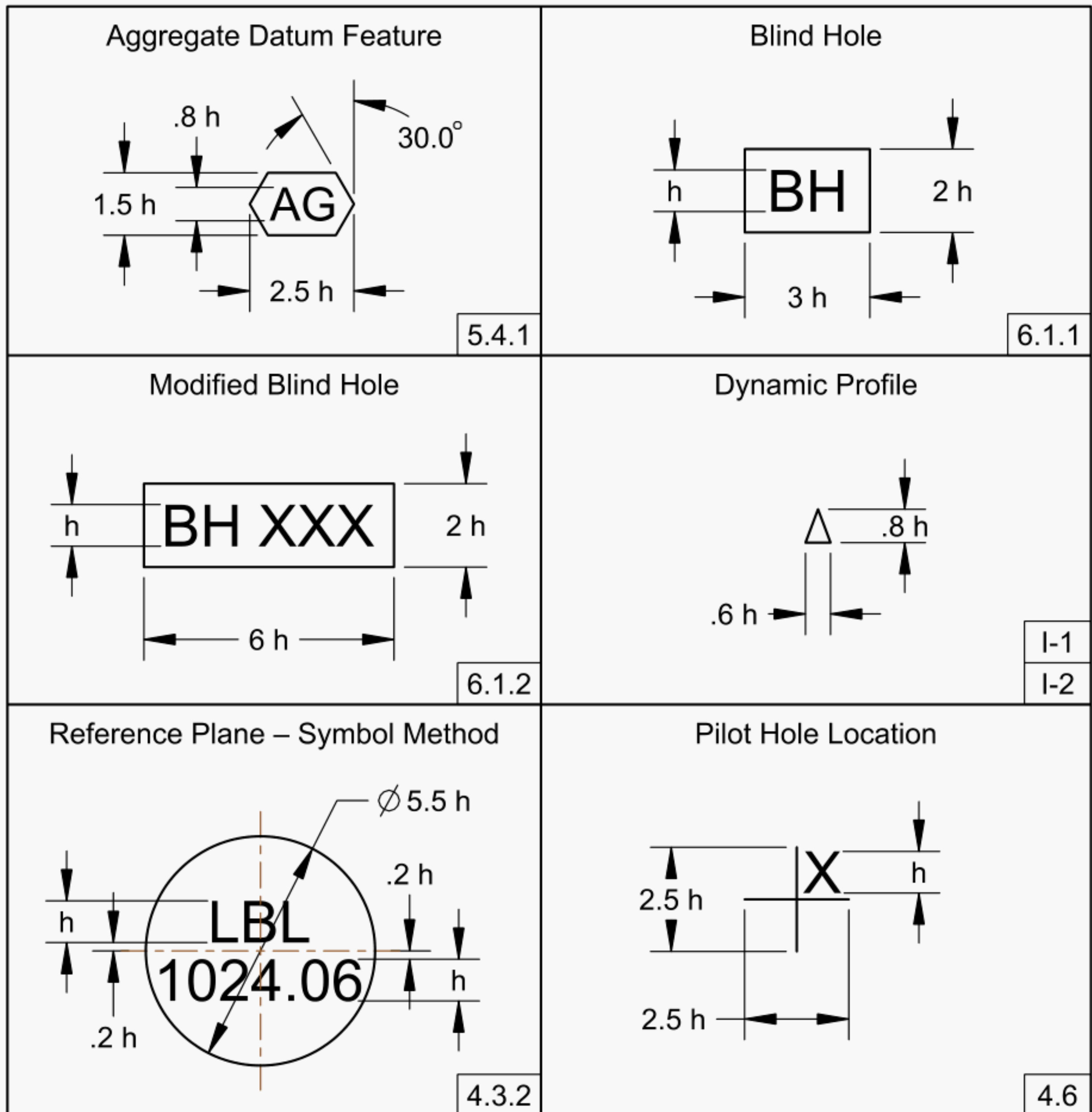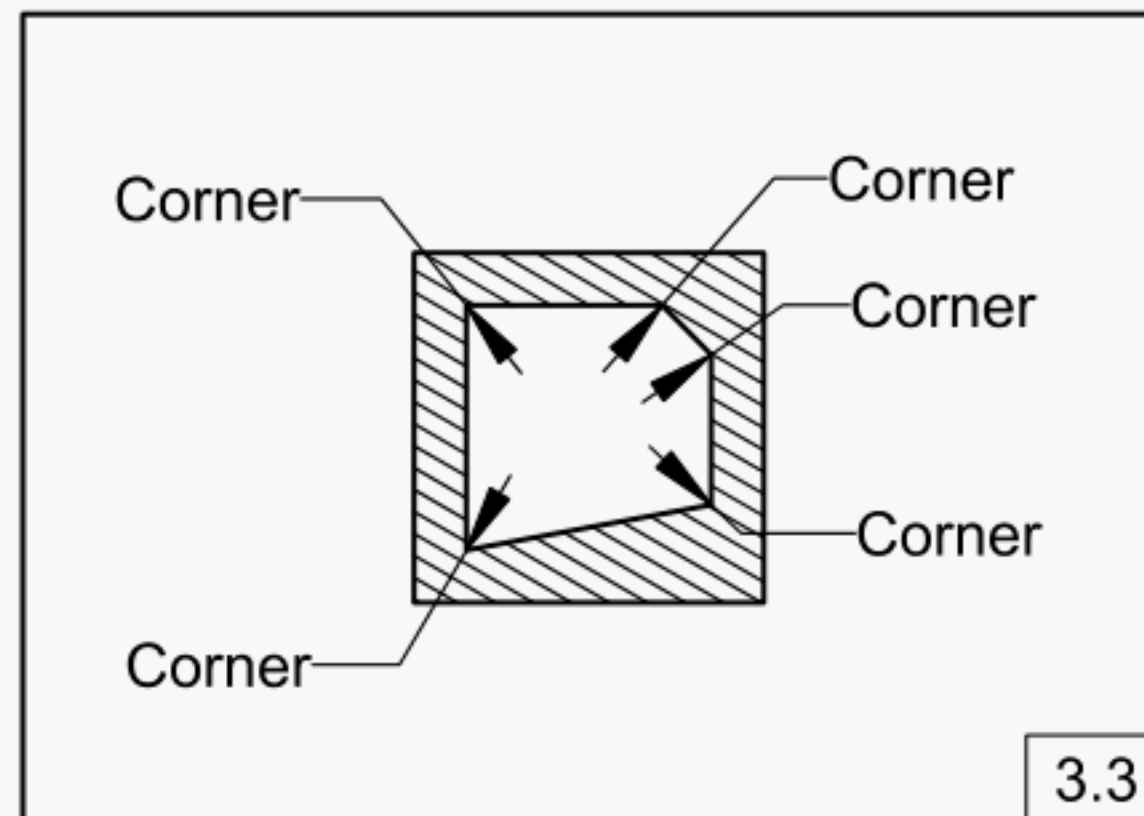
Figure 3-1 Corner

3.2 Blind Hole

blind hole: a hole with a bottom internal to the material.

3.3 Corner

corner: the projected intersection of two surfaces, flat or curved, where the angle of included space is less than 180 deg. See Figure 3-1.

3.4 Datum, Aggregate

datum, aggregate: a datum established from an assembly of parts where multiple features on multiple parts act together to create a single datum point, line, axis, or plane.

3.5 Edge

edge: the projected intersection of two surfaces, flat or curved, where the angle of included material is less than 180 deg. See Figure 3-2.

3.6 Fillet

fillet: a radius applied to a corner of a part.

3.7 Pilot Hole

pilot hole: an undersized hole used to facilitate manufacturing or other processing.

Figure 3-2 Edge
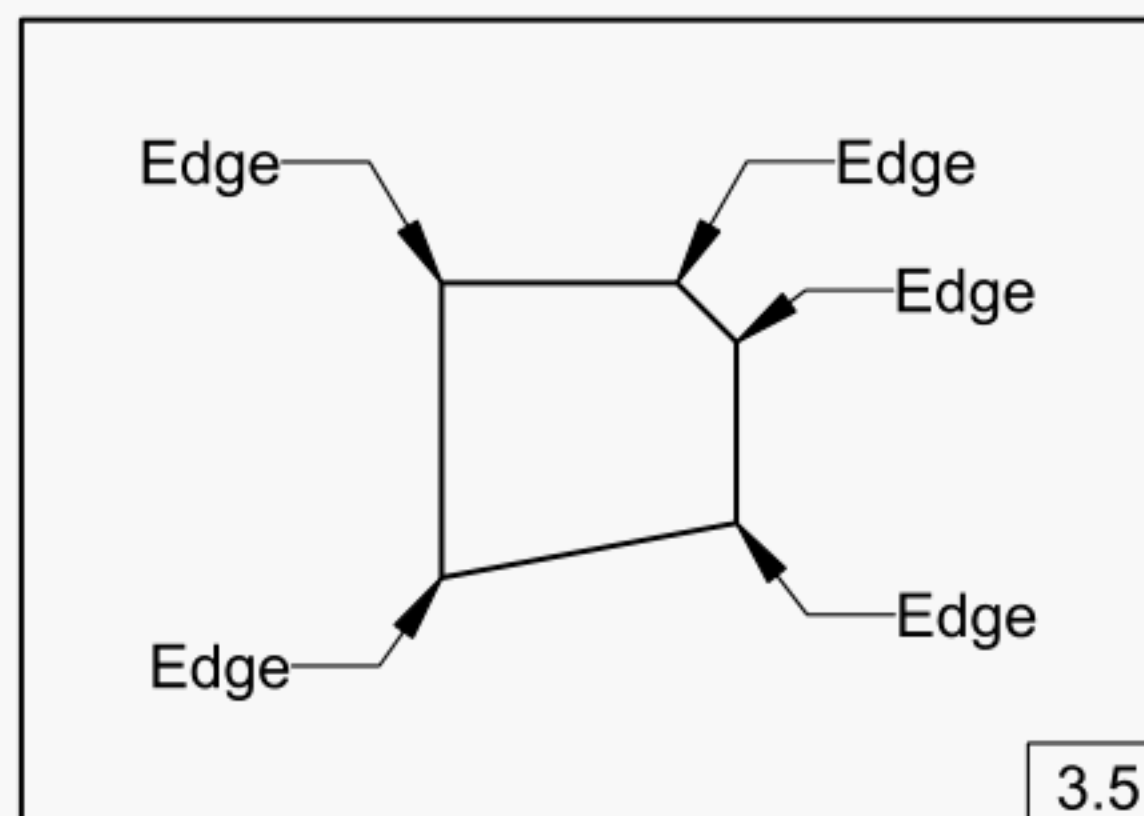
3.8 Abbreviations

Abbreviation	Definition
AED	Aerospace and advanced engineering drawing
ASME	The American Society of Mechanical Engineers
BL	Buttock line
CR	Controlled radius (ASME Y14.5-2009)
DoD	U.S. Department of Defense
EP	Elliptical fillet
FS	Fuselage station
LBL	Left buttock line
LMB	Least material boundary (ASME Y14.5-2009)
MMB	Maximum material boundary (ASME Y14.5-2009)
OPTL	Optional
PA	Parabolic fillet
R	Radius (ASME Y14.5-2009)
RBL	Right buttock line
RS	Radome station
SI	International System of Units
STA	Station
UOS	Unless otherwise specified
WG STA	Wing station
WL	Waterline

4 ENGINEERING DRAWING PRACTICES

4.1 Aircraft and Related Part(s) Views

Engineering drawings that depict complete aircraft views should be arranged using left-side, rear, and top views as the principal orthographic views. The aircraft's nose in the top and left-side views shall point to the left-hand edge of the drawing and front (forward) direction of the aircraft. See Figure 4-1 for aircraft views arrangement.

NOTE: For a piloted aircraft, the pilot's viewing direction while seated in the aircraft cockpit will be toward the front of the aircraft.

4.2 Aircraft Engines and Related Part(s) Views

UOS, the terms "FRONT," "REAR," "LEFT," and "RIGHT" may be used on engineering drawings to indicate position in an assembled aircraft engine or related parts (e.g., flange bracket mounting) as follows:

(a) Front is in the direction closest to the nose of the vehicle.

(b) Rear is the opposite of front.

(c) Left and right are based on the observer's position when facing forward.

4.3 Aircraft Geometry Reference Plane Callouts and Toleranced Reference Planes

4.3.1 Aircraft Geometry Reference Planes. Geometry reference plane callouts are used to identify location on the vehicle. Designations for geometry reference planes include, but are not limited to, station (STA), fuselage station (FS), waterline (WL), wing station (WG STA), radome station (RS), buttock line (BL), right buttock line (RBL), and left buttock line (LBL).

4.3.2 Symbol or Callout Method. When used, geometry reference plane specifications shall be shown on the field of the drawing by using the symbol method or the callout method. The symbol method uses a consistent geometric shape, e.g., circle, rectangle, or cartouche, to specify the abbreviated reference plane designation and reference plane location. The callout method uses the abbreviated reference plane designation and reference plane location. See Figures 1-1 and 4-2. When both methods are used on the same drawing, the callout method specifications shall be considered reference. A leader or extension line shall be used to designate the location of the reference plane.

Figure 4-1 Arrangement of Aircraft Views

AXONOMETRIC

TOP

RIGHT SIDE   FRONT   LEFT SIDE   REAR

BOTTOM

4.1
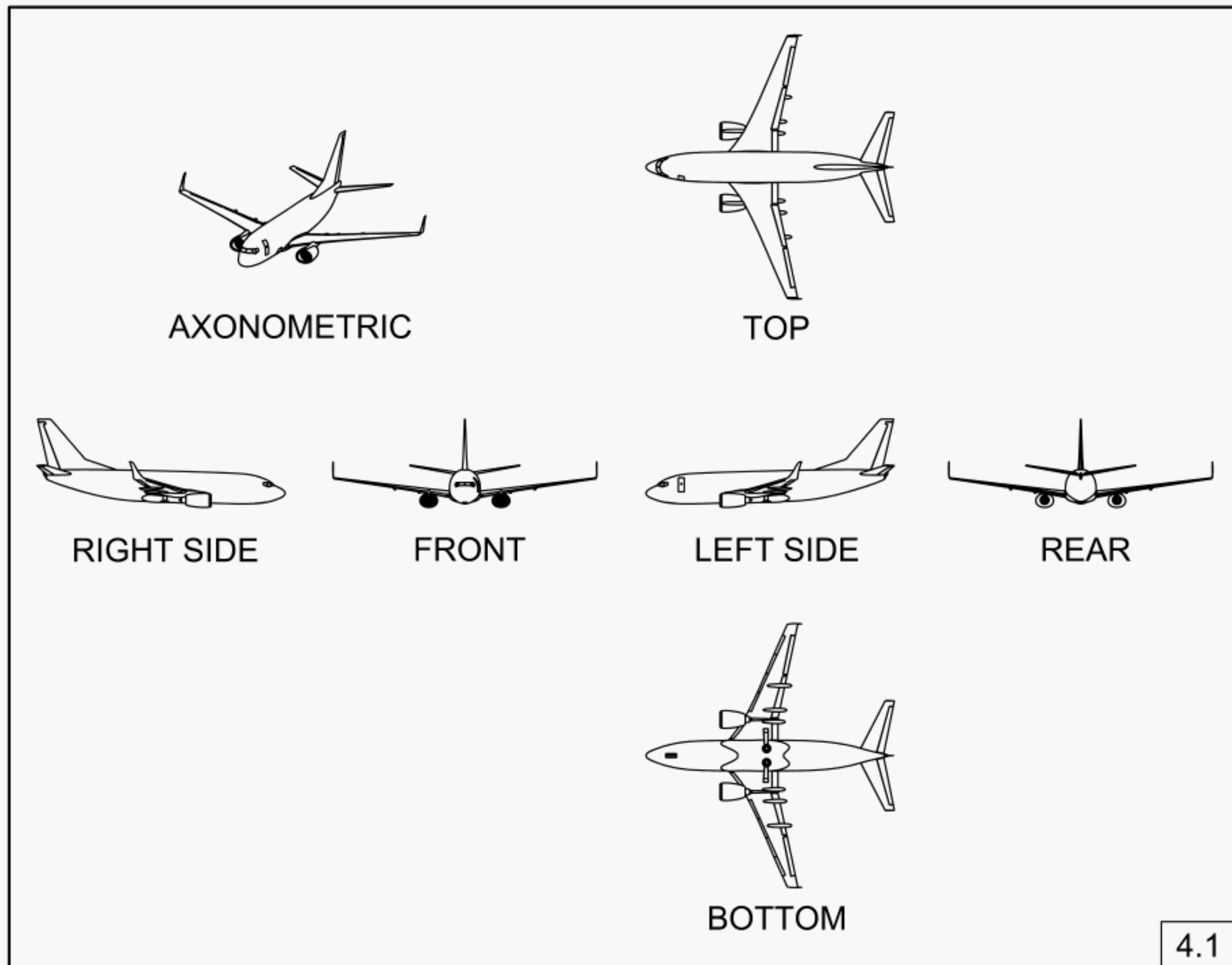
Figure 4-2 Identification of Geometry Reference Planes

Symbol Method	Callout Method
Abbreviated reference plane designation (FS = Fuselage Station)	Abbreviated reference plane designation (FS = Fuselage Station)
Reference plane location	Reference plane location
FS 844.06	FS 844.06

4.3.2
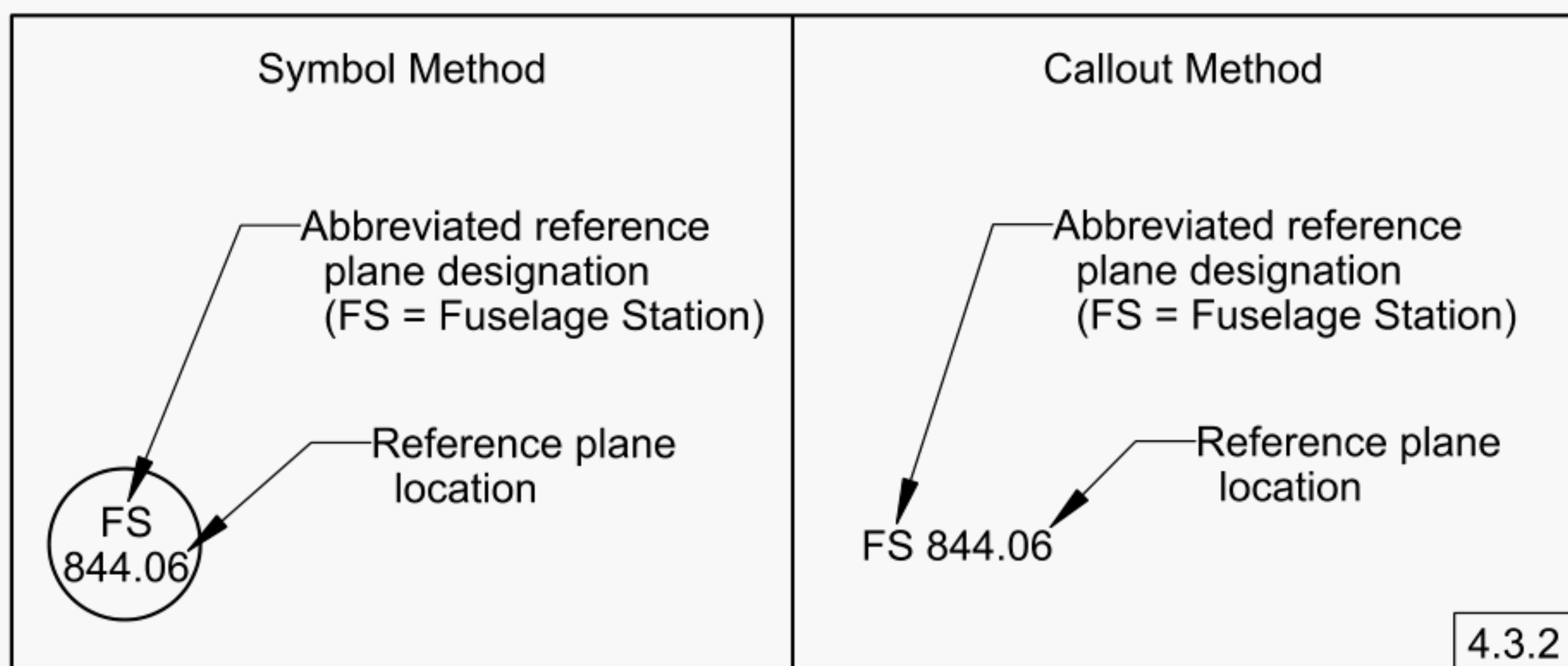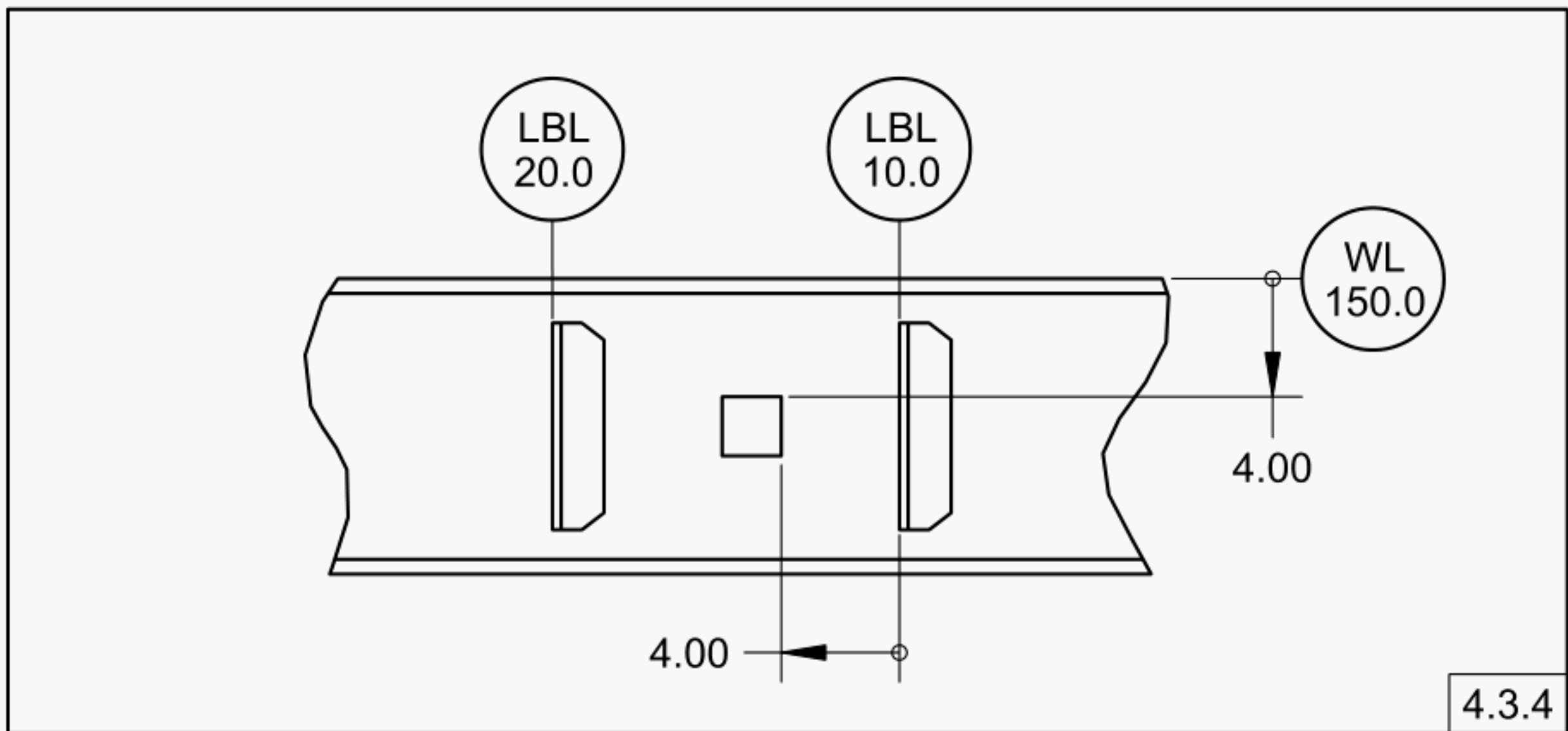
Figure 4-3 Application of the Symbol Method

4.3.3 Tolerances. UOS, reference plane dimensions have no tolerance. The reference plane is theoretical and may be coincident with a toleranced feature.

4.3.4 Use of Geometry Reference Planes. Geometry reference planes that are coincident with a part feature may be used as a dimension origin. Geometry reference planes shall not establish datums. See Figure 4-3.

4.4 Orientation and Direction of Dimensional Requirements

Where the term "AXIAL," "ANGULAR," or "RADIAL" is used to describe the orientation or direction in which a dimensional requirement (e.g., a tolerance) is to be applied, the terms are as described below. See Figure 4-4 for an example of the use of the terms "AXIAL," "ANGULAR," and "RADIAL."

(a) The term "AXIAL" indicates that the dimensional requirement applies along an axis.

(b) The term "ANGULAR" indicates that the dimensional requirement applies on either side of and parallel to an angular plane emanating from the part or feature axis.

(c) The term "RADIAL" indicates that the dimensional requirement applies along a plane or line that is constructed normal to the part or feature axis.

4.5 Use of Single Dimension Lines or Double Arrowheads on Space-Restricted or Partial Views

When space is restricted, or when only a partial view is used, features may be dimensioned using one of the methods depicted in Figure 4-5. When the opposite, unshown, surface is unclear, that surface shall be identified in another view and be considered the dimension origin UOS.

4.6 Pilot Hole Location Symbols

When indicating an area to be reserved for pilot holes, those areas shall be identified on engineering drawings using the pilot hole location symbol depicted in Figures 1-1 and 4-6 and as follows:

(a) The pilot hole location symbol includes a cross that indicates the location of the hole. The fastener diameter is indicated in the upper-right quadrant of the symbol, where X is the nominal diameter of the fastener that will be used for assembly or installation. The other three quadrants shall remain blank.

(b) The pilot hole location symbol shall be used to indicate pilot hole locations shown on an engineering drawing such as detail, assembly, or installation drawings. Where engineering documentation does not establish hole specifications at the detail or assembly level of manufacture, the manufacturing information shall control when pilot holes are included on parts and, if included, the size of the pilot holes.

Figure 4-4 Orientation and Direction of Dimensional Requirements
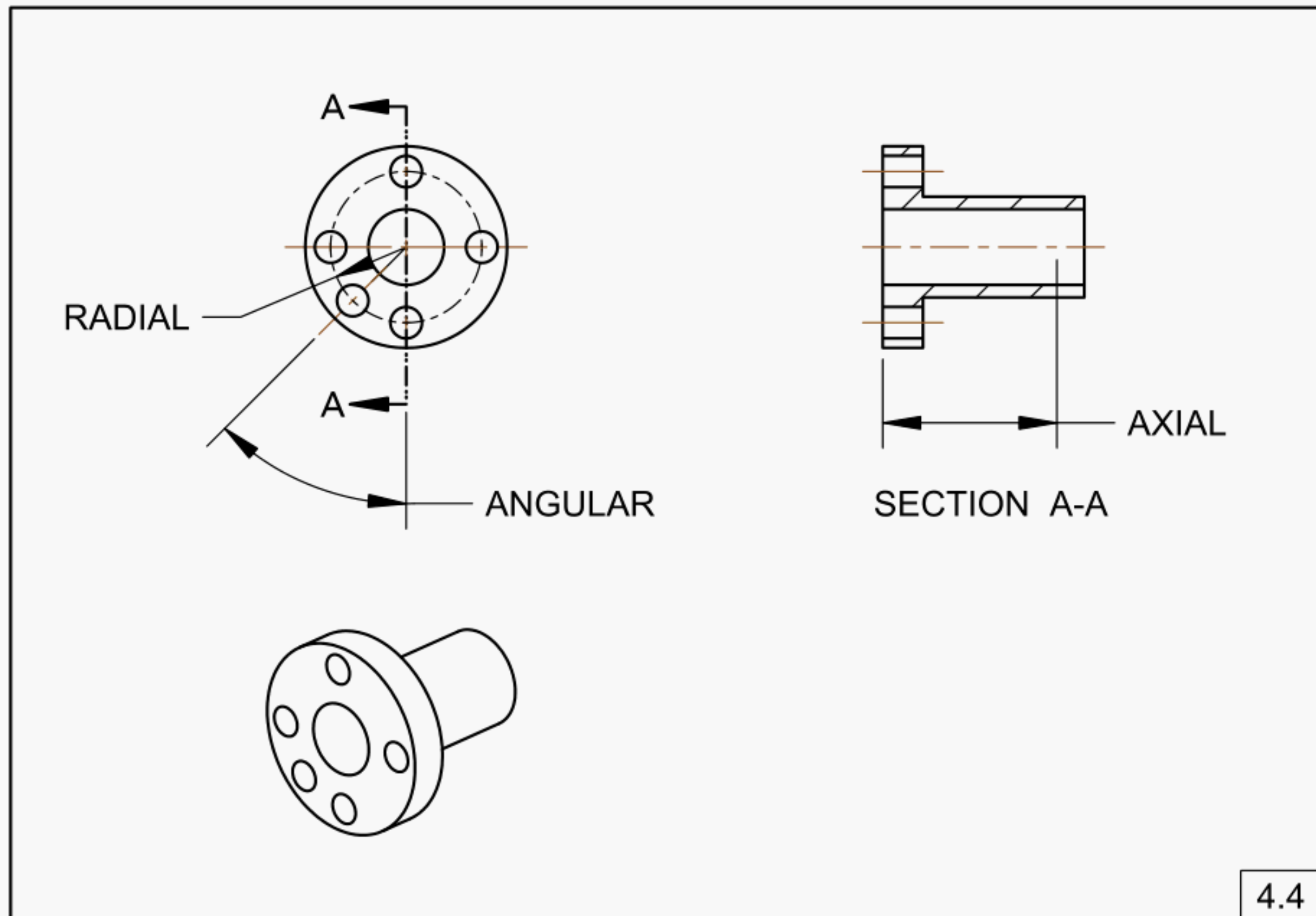
4.4

4.7 Optional Coverage

When a drawing indicates OPTIONAL or OPTL with regard to an area receiving surface treatment coverage, the indicated area(s) may have full, partial, or no surface treatment coverage applied in accordance with the surface treatment specification.

Where abrupt endings are not permissible within the optional coverage area, the drawing shall indicate it.

5 ASSEMBLY TOLERANCING

Assembly dimensioning and tolerancing pertain to the geometric control of component features that are only applicable on assembly product definitions. This section outlines the conditions necessary for such a practice to be valid.

The following two primary types of features need to be controlled with tolerance at an assembly level:

(a) Existing Features With Adjustment at Assembly. The component or subassembly containing the feature has freedom for adjustment during the assembly process to allow the specified feature to comply with the dimensional requirements after it is assembled as described in para. 5.1.

(b) Features Manufactured at Assembly. Where the feature is to be manufactured after the component or subassembly has been assembled, all feature requirements that shall be met at the assembly level shall be wholly defined on the assembly drawing as described in para. 5.2.

Tolerances applied at the assembly level for features that exist at the component or subassembly level shall be met without material removal. If material removal is allowed, it shall be noted on the drawing. For features that are defined at the assembly level, removal of material is permissible to meet the requirements. If no material removal is allowed to meet a specific requirement, it shall be noted on the drawing.

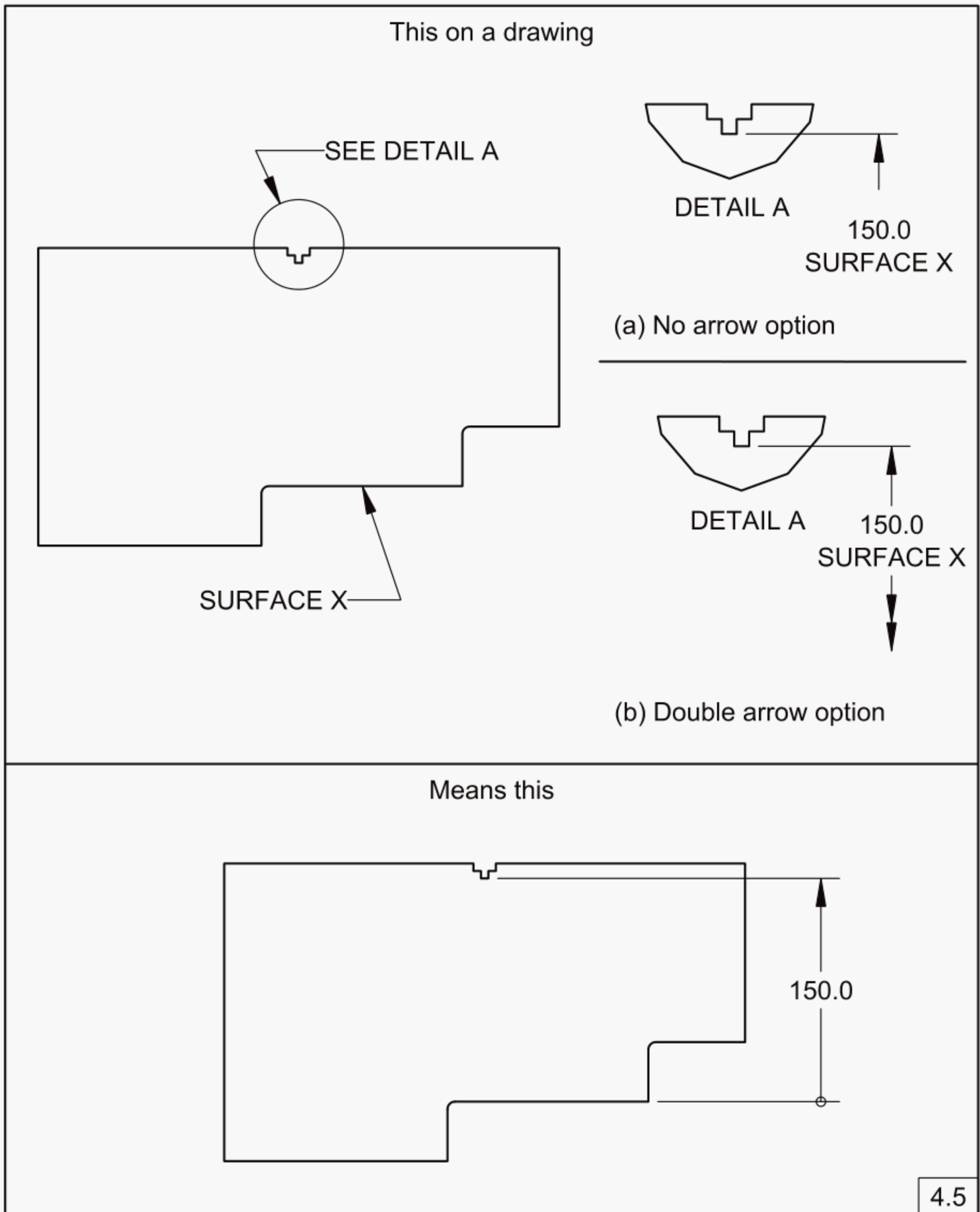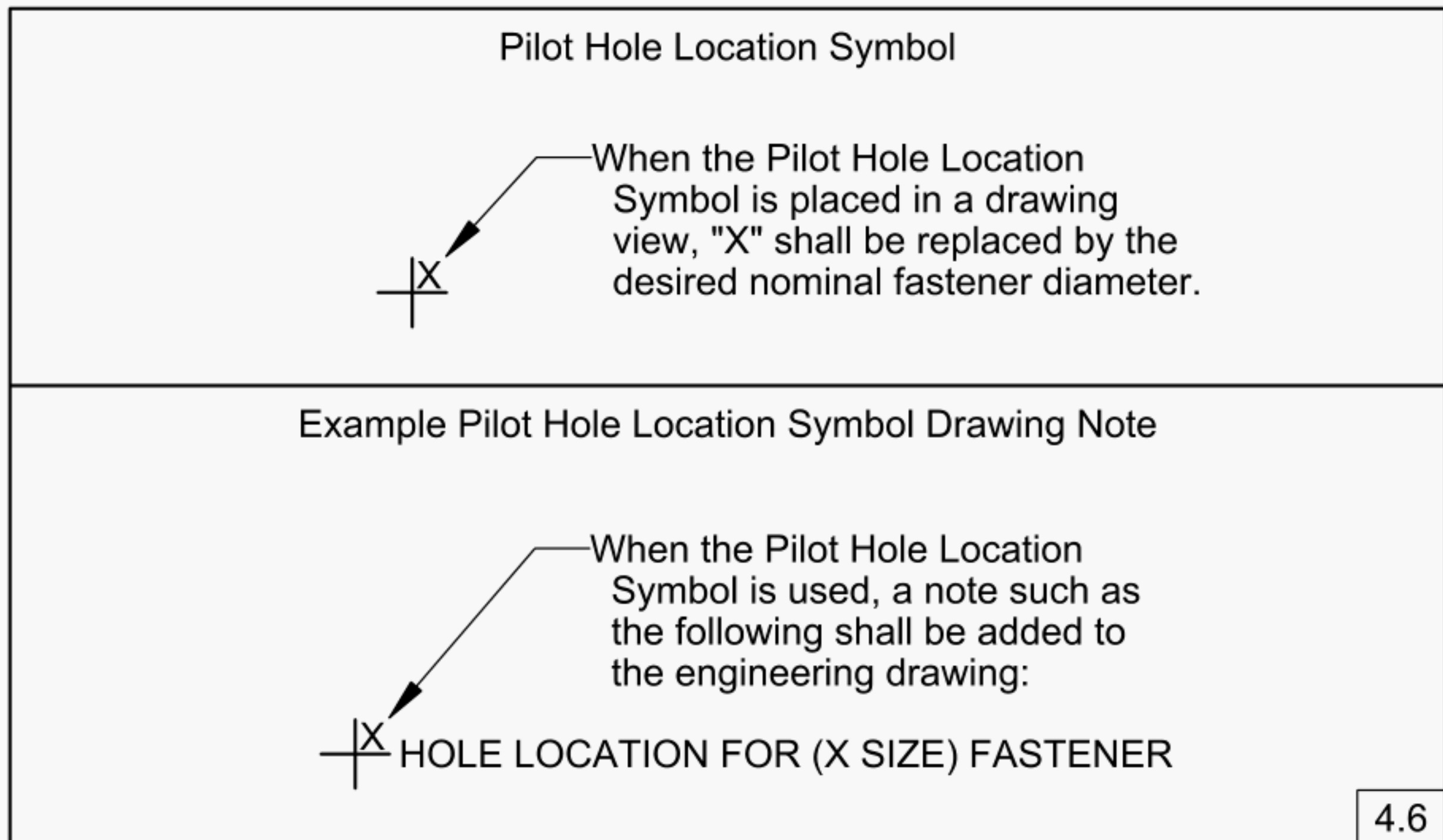**Figure 4-5 No-Arrow or Double-Arrow Option Methods**

This on a drawing

SEE DETAIL A

DETAIL A

150.0
SURFACE X

(a) No arrow option

DETAIL A   150.0
SURFACE X

SURFACE X

(b) Double arrow option

Means this

150.0

4.5

Figure 4-6 Pilot Hole Location Specification

4.6

5.1 Existing Feature Adjustment Tolerancing

Existing feature adjustment tolerancing is used when a feature is fully defined at a component or subassembly level and is also toleranced at the assembly level to meet a requirement different from that allowed by the component or subassembly dimensional requirements specified. The requirement shall be met without any material removal at the assembly. One example of this practice is to locate a feature from one part relative to a feature from a second part on a welded assembly. The features from each component are defined at a lower-level drawing and then the two components are welded as defined on the assembly drawing to meet the location requirement after welding.

In the example shown in Figure 5-1, the location of the bushings bonded to a plate is controlled by existing feature adjustment tolerances. The location of the bushing inner diameter in the plate is controlled with less variation than the hole in which it is bonded.

5.2 Manufactured at Assembly Tolerancing

Manufactured at assembly level tolerancing is used when a feature is manufactured at the assembly level. Under these conditions, the feature shall be controlled in accordance with ASME Y14.5. One example of this practice is machining mounting holes into a weldment after all the welding operations are complete. Since the holes do not exist on a lower-level drawing (or may not be at the finished size), they are to be shown in the appropriate views, when applicable, and dimensioned accordingly on the assembly level drawing.

5.3 Assembly Level Datums

Any feature or combination of features that would be a valid datum feature on a detail part can be used as a datum feature on an assembly. The datum feature shall be identified the same way as it would on a detail part. See ASME Y14.5 for requirements regarding indication and placement of datum feature symbols. Datum features on detail parts that are also used as datum features on assemblies do not need to maintain the same datum letter.

5.3.1 Surface Features as Datum Features. A feature that is not a feature of size can be designated as a datum feature on an assembly drawing using the same methods employed on detail parts. Consideration should be given to qualifying datums at the assembly level with form, orientation, or location control, as applicable, to account for the effects of any processes used to create the assembly.

Figure 5-1 Existing Feature Adjustment Tolerancing Example

Plate drawing

5.000±.030

⊥ | .006 | A | B

2X | 3.000

2X | 1.000

.400±.005

2X | 2.500

4.500±.030

⊥ | .003 | A

2X | 1.000

B

4X ⌀ .456±.001 THRU

⊕ | ⌀.008Ⓜ | A | B | C

▱ | .001

A

Bushing drawing

.105
.095

.250
.230

⌀ .257
.254

⊥ | .001 | E

E

⌀ .441
.439

⊕ | ⌀.001 | E

SECTION J-J

⌀ .700
.680

⊕ | ⌀.012 | E

5.1

Figure 5-1 Existing Feature Adjustment Tolerancing Example (Cont'd)

Assembly drawing

2X | 3.000

2X | 2.500

▶— A

4X (⌀ .257 / .254)

⊕ | ⌀ .004 Ⓜ | A

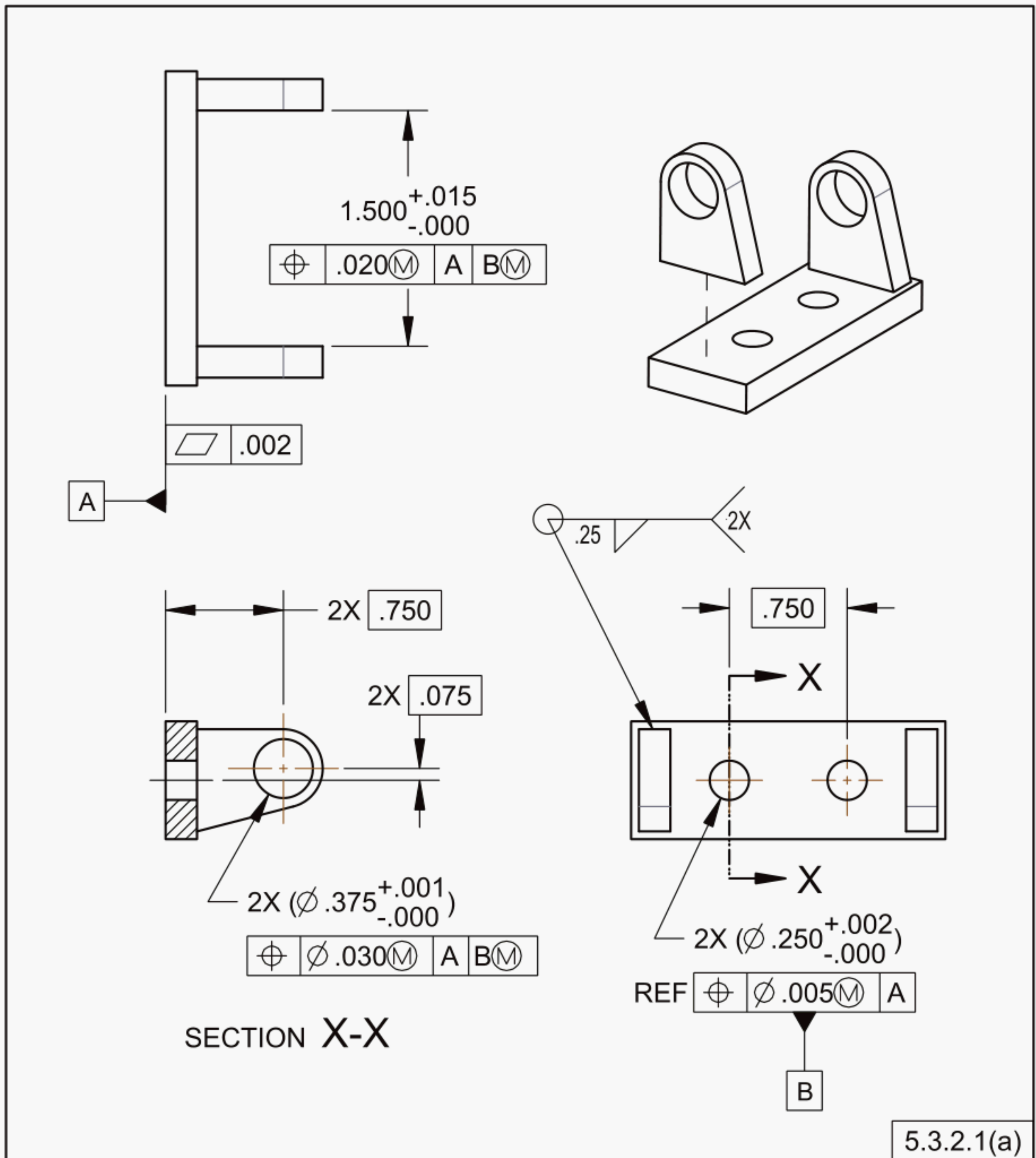4X BUSHING ADHESIVE

PLATE

4X BUSHING

5.1
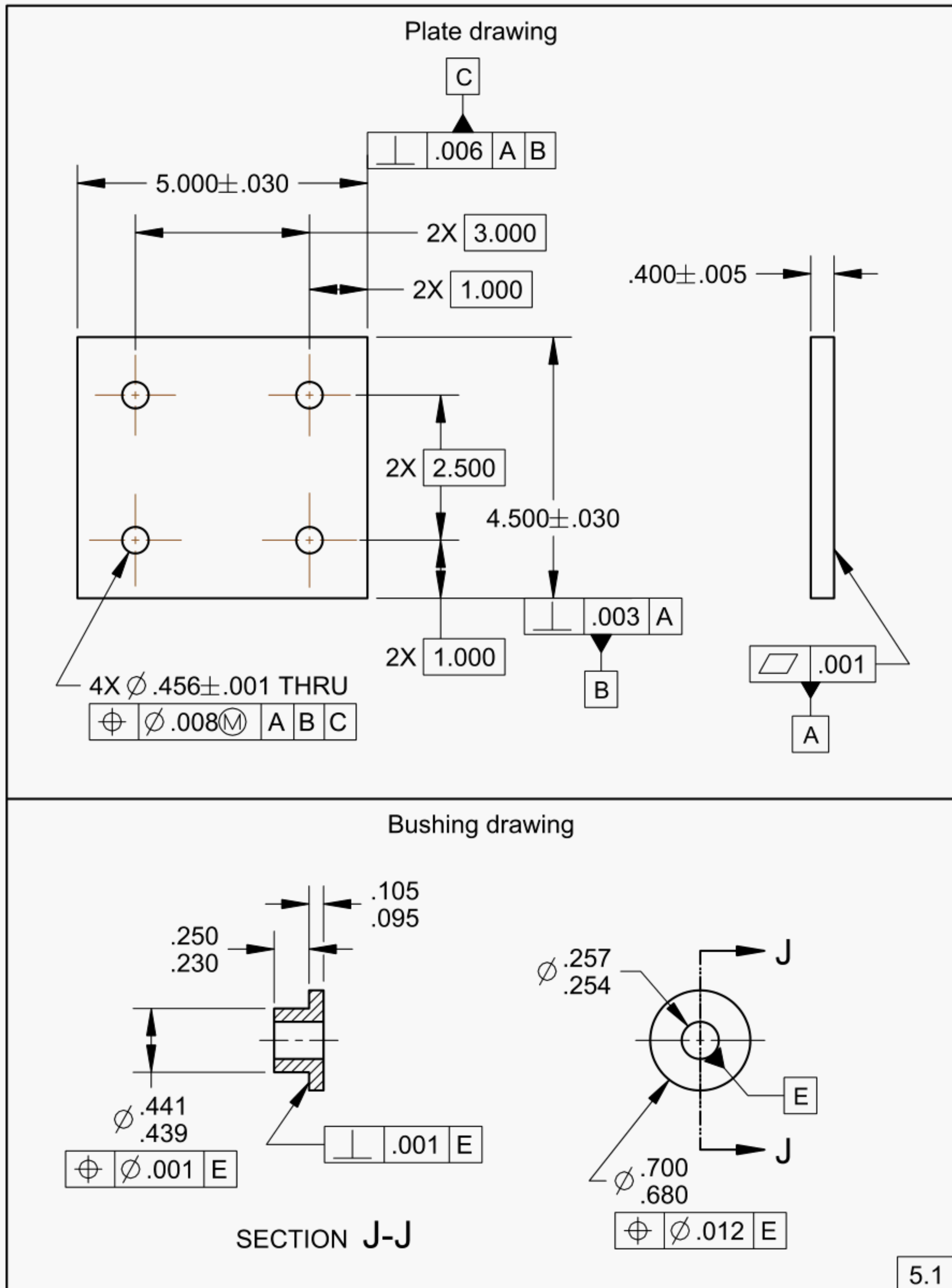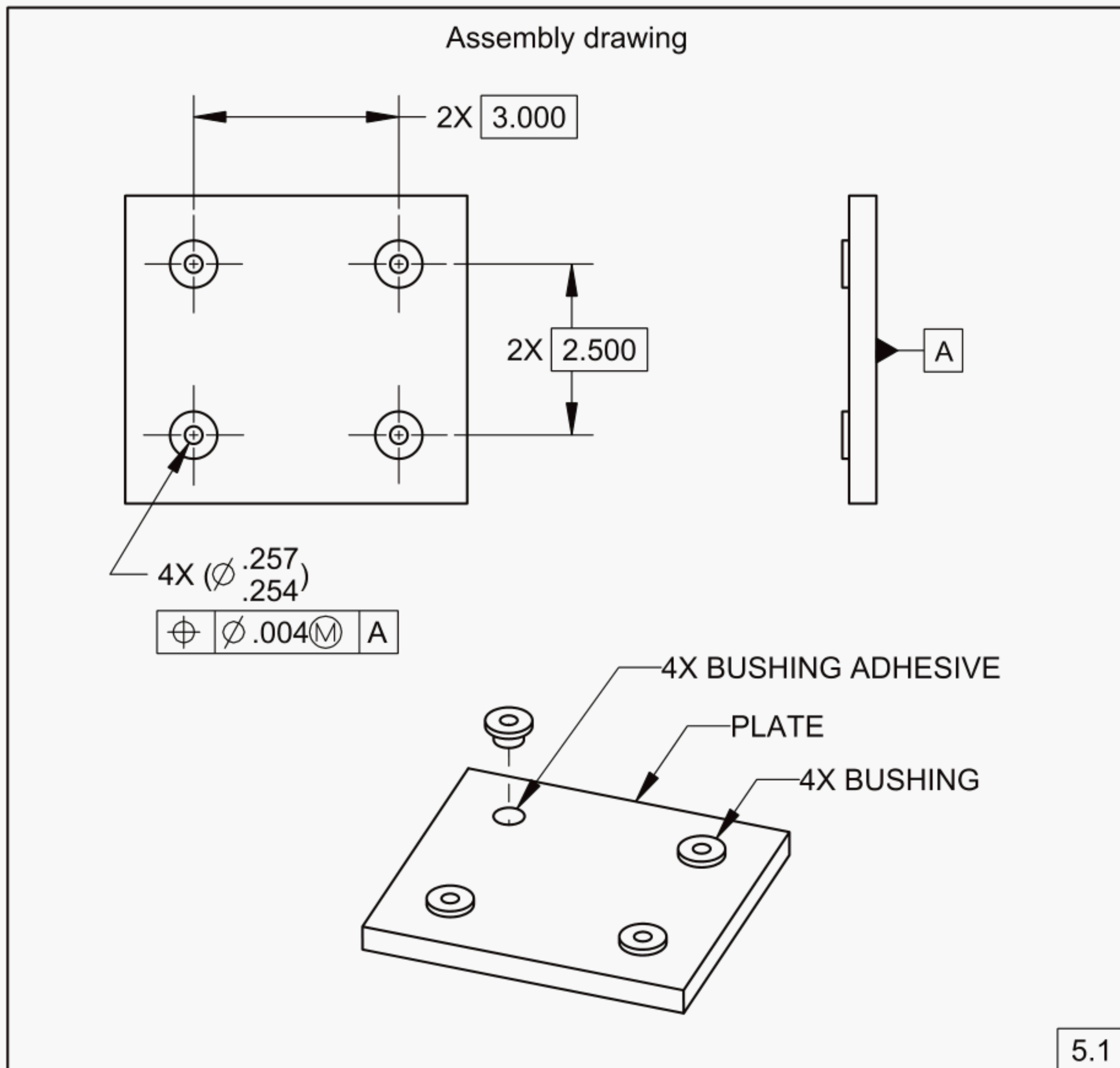
5.3.2 Features of Size as Datum Features. A feature of size can be designated as a datum feature on an assembly drawing in much the same way as it is on a detail part, subject to the considerations described in paras. 5.3.2.1 and 5.3.2.2.

5.3.2.1 Datum Features Created at a Lower Level. When a feature of size, defined at the component or subassembly level, is to be designated as a datum feature at either MMB or LMB on an assembly drawing, sufficient information shall be provided on the assembly level drawing to establish the MMB and LMB boundaries for the assembly level datum. This may be accomplished by either of the following methods:

(a) Reference Method. The size dimension and tolerance, as defined on the component or subassembly drawing, shall be shown as reference information on the assembly drawing. See Figure 5-2. When a datum feature of size is to be referenced as the secondary or tertiary datum, the geometric tolerance that establishes the inner and outer boundaries (MMB and LMB) of the feature of size is associated to the reference dimension. The geometric tolerance may be a restatement of the geometric tolerance from the component or subassembly that defined the feature. If the geometric tolerance is not a direct restatement of the original callout, it shall be consistent with the original callout on the component or subassembly. For a geometric tolerance to be considered reference information, the abbreviation "REF" shall be noted adjacent to the feature control frame. Without the "REF" designation, the feature control frame establishes a new requirement for the feature at the assembly level.

Figure 5-2 Reference Method

1.500 $^{+.015}_{-.000}$

| ⊕ | .020 Ⓜ | A | B Ⓜ |

| ▱ | .002 |

A

.25   2X

2X .750

2X .075

.750

X

X

2X (⌀.375 $^{+.001}_{-.000}$)

| ⊕ | ⌀.030 Ⓜ | A | B Ⓜ |

SECTION X-X

2X (⌀.250 $^{+.002}_{-.000}$)

REF | ⊕ | ⌀.005 Ⓜ | A |

B

5.3.2.1(a)

(b) Applicable Condition Method. For this method, the value of the boundary may be stated, enclosed in brackets, following the applicable datum reference and modifier in the feature control frame in accordance with ASME Y14.5. See Figure 5-3.

5.3.2.2 Produced at the Assembly Level. When a feature of size produced during assembly is designated as a datum feature, it shall be fully defined per ASME Y14.5.

5.4 Aggregate Datum

The aggregate features of the designated aggregate datum restrict one or more degrees of freedom without affecting any inherent motion within the assembly. An aggregate datum with no inherent motion shall be identified per para. 5.4.1 and specified in a feature control frame per ASME Y14.5. An aggregate datum with inherent motion shall be identified per para. 5.4.1 and specified in a feature control frame per para. 5.4.2. Where aggregate datum features are accessible, a physical datum feature simulator should be used. Where aggregate datum features are not accessible with a physical datum feature simulator, the combined effect of the assembly acts as the physical datum feature simulator.

5.4.1 Identifying an Aggregate Datum Feature. The aggregate datum feature symbol is the letters "AG" within a hexagon. See Figure 1-1. An aggregate datum feature symbol shall be associated with at least one of the features used to derive the aggregate datum. The aggregate datum feature symbol or the word "AGGREGATE" shall be placed next to the datum feature symbol. See Figure 5-4.

5.4.2 Identifying Inherent Motion of an Assembly. The inherent motion of the assembly shall be indicated by
(a) a datum reference frame symbol placed with at least one of the axes aligned with the inherent motion
(b) the appropriate degree of freedom symbol placed after the aggregate datum feature reference letter in the feature control frame
(c) an asterisk placed after the degree of freedom symbol to indicate motion direction or freedom rather than the motion constraint default per ASME Y14.5
A note may be used to clarify movement or method on how the assembly is to be restrained.
For an assembly with motion, at least one full cycle of the movement is required for verification, UOS. See Figure 5-4.

6 ADDITIONAL DRAWING PRACTICES

6.1 Blind Holes

6.1.1 Blind Hole Callout. The blind hole symbol may be used to indicate a blind hole. See Figure 1-1.
When the blind hole symbol is used, the acceptable boundaries of the hole bottom configuration shall be as shown in Figure 6-1 for a blind hole or Figure 6-2 for a blind hole with a through hole.

6.1.2 Drill Point Defined by Callout. The preferred drill point may be indicated with a modified blind hole symbol as shown in Figure 1-1. See Figure 6-3 for a 135-deg drill point callout example.

6.2 Edge Breaks

When the term "EDGE BREAK" or another edge break callout is specified, the edge break may take the form of a radius, a chamfer, or a combination of both. See Figure 6-4 for edge break examples.
If a blend is required between the flat surface permissible in the edge break tolerance zone and the adjacent surfaces to eliminate abrupt changes in material topography, the blend shall be specified. It could be a radius, a partial ellipse, or a combination of both.
Edge breaks may be produced in any manner that is not otherwise restricted by the design authority. Where an edge break is specified with only a high limit size, e.g., .030 MAX, the lower limit shall be interpreted as having no measurable size; however, burrs and sharp edges that may cause a cut-type injury are not permitted.

6.3 Radius

Radius tolerance and controlled radius tolerance of ASME Y14.5 are amended as described in paras. 6.3.1 through 6.3.4.

6.3.1 Radius Tolerance. A radius symbol, R, creates a zone defined by two arcs (the minimum and maximum radii). The part surface shall lie within this zone. Where the center of the radius is located via dimensions, the arcs are concentric. See Figure 6-5. Where the center of the radius is not located (tangent located), the arcs create a crescent-shaped tolerance zone whose "width" is set by placing both the maximum and minimum radii tangent to the adjacent surfaces. See Figure 6-6.

Figure 5-3 Applicable Condition Method

$1.500^{+.015}_{-.000}$

| ⊕ | .020Ⓜ | A | BⒻ [⌀ .245] |

| ▱ | .002 |

A

.25   2X

2X .750

2X .075

2X (⌀ .375$^{+.001}_{-.000}$)

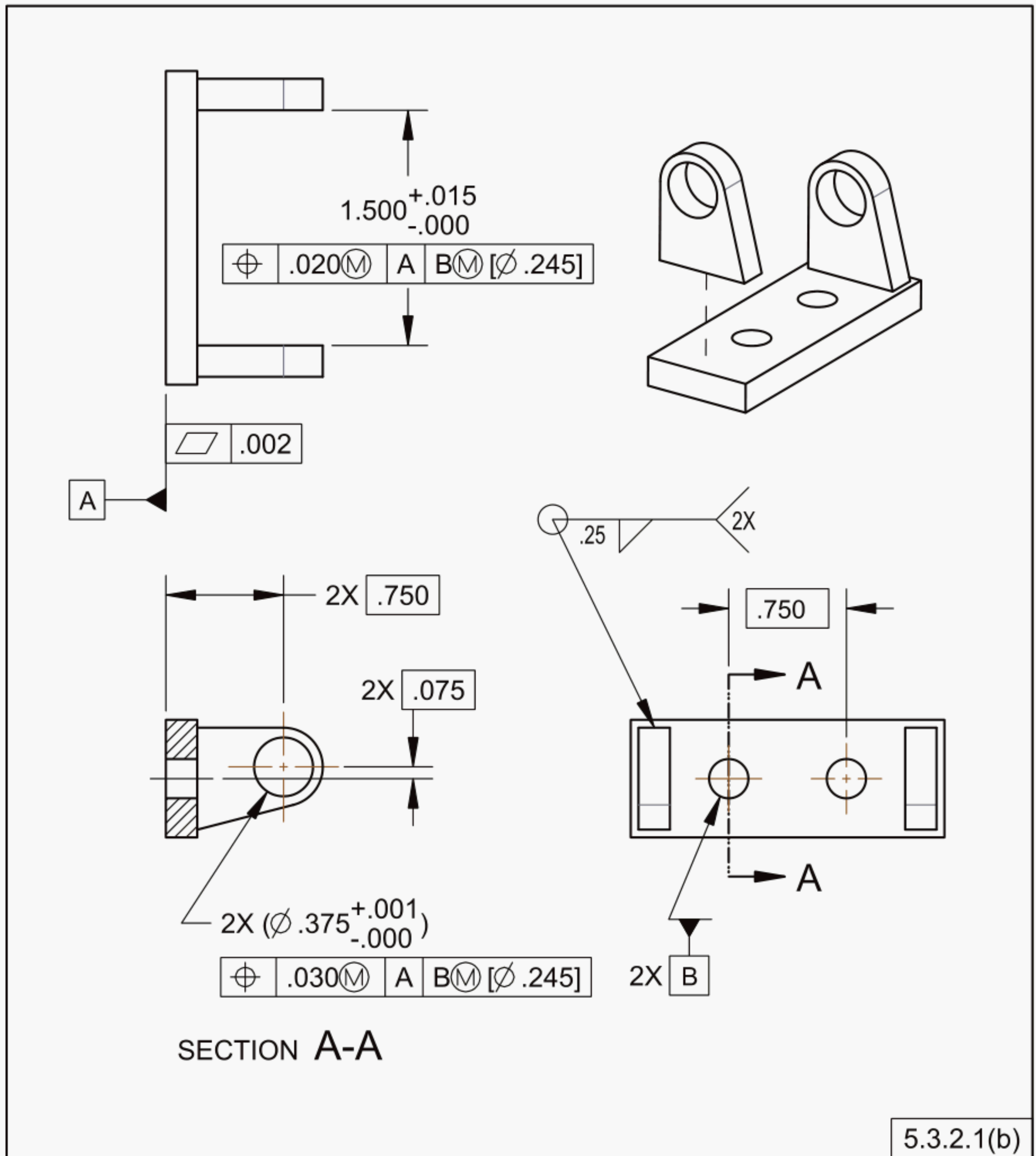| ⊕ | .030Ⓜ | A | BⒻ [⌀ .245] |

SECTION A-A

.750

A

A

2X | B |

5.3.2.1(b)

Figure 5-4 Aggregate Datum With Inherent Motion in the Assembly

Figure 6-1 Blind Hole

This on a drawing
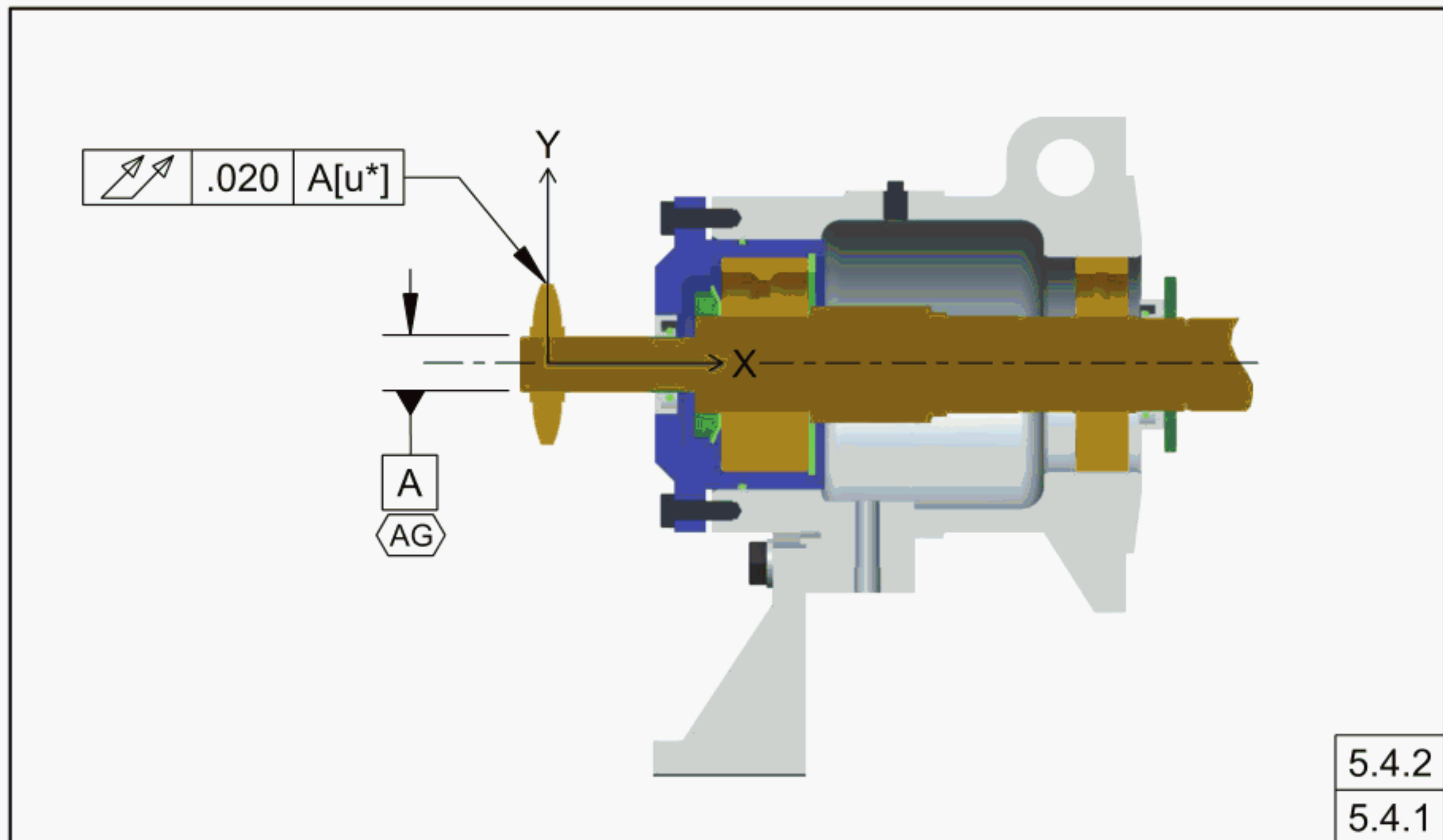
Means this

Any configuration
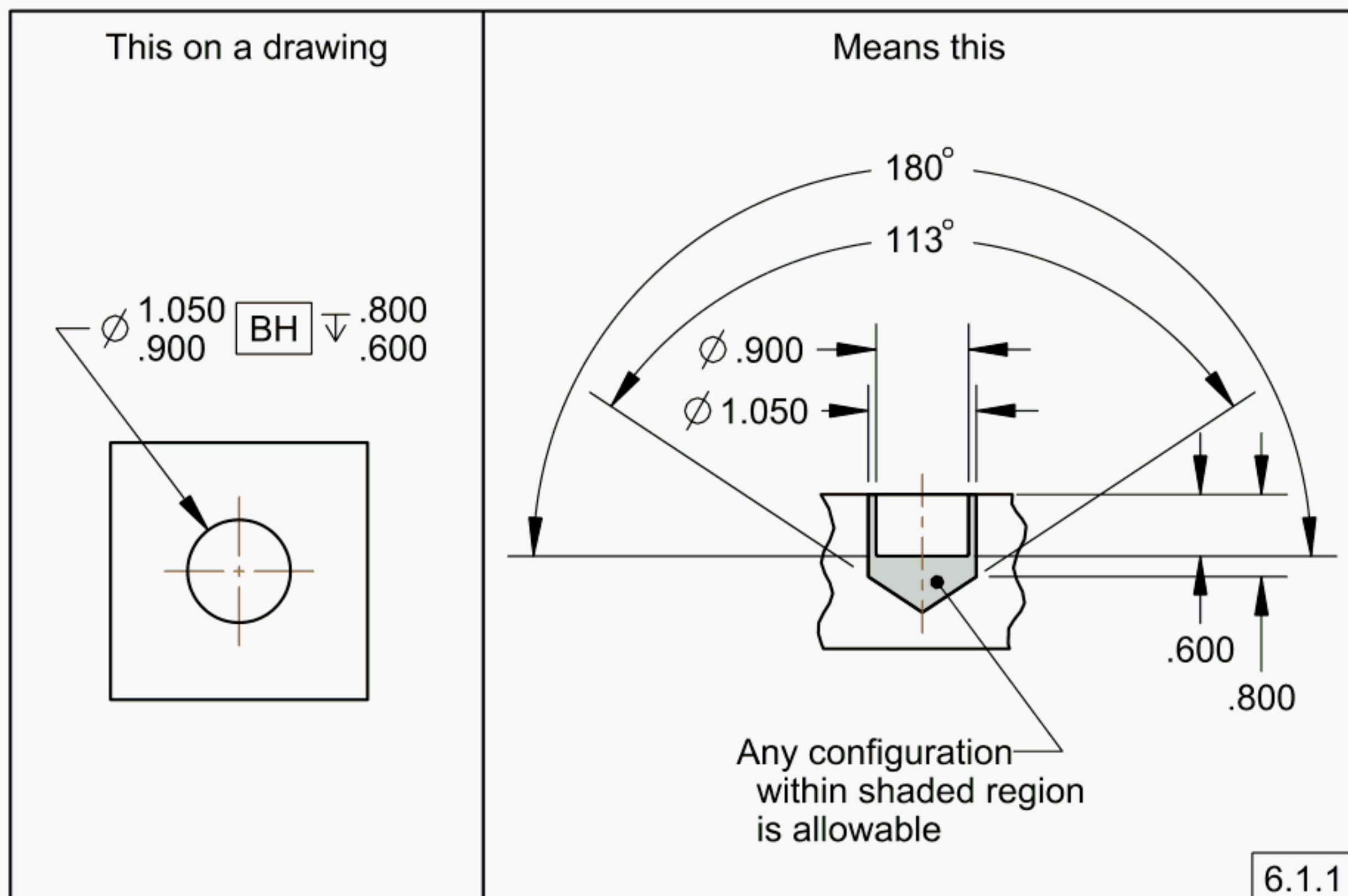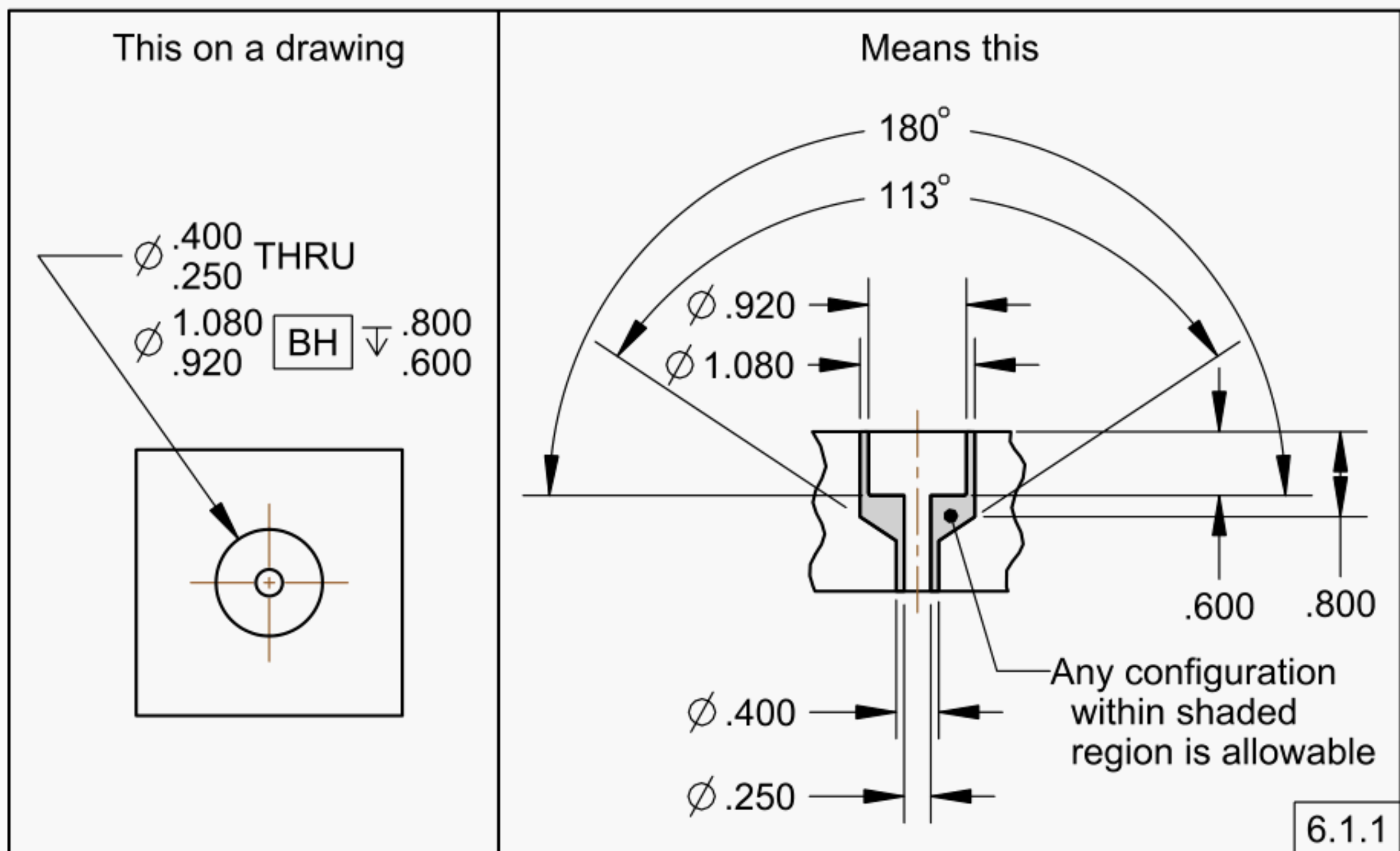within shaded region
is allowable

16

Figure 6-2 Blind Hole With a Through Hole

6.3.2 Controlled Radius Tolerance. Where a controlled radius is specified, the part contour within the tolerance zone shall be a fair curve without reversals. Controlled radii may be toleranced directly or, when specified as basic dimensions, with a profile. See Figure 6-7. When the radius is directly toleranced, all points on the part contour shall be neither smaller than the specified minimum limit nor larger than the maximum limit. Where it is necessary to apply further restrictions to the part radius, those restrictions shall be specified on the drawing or in a document referenced on the drawing.

6.3.3 Tangent Radii. The ends of a radius when shown tangent shall blend smoothly and terminate within 5 deg of tangency. See Figures 6-8 and 6-9. Because the center of the radius is not specified, the crescent zone of a tangent radius is free to rotate to contain the blended surface, which shall connect both adjacent surfaces. See Figure 6-6. Disruptions in the surface contour are not permitted. Mismatches where the radius and adjacent surface meet shall not exceed the effective form tolerance of the applicable adjacent surface.

6.3.4 Radius Tangencies. The surface of a radius near its visual tangency shall conform to the tolerance of the adjacent feature, the appropriate radius tolerance zone (crescent or concentric arcs), or both. See Figure 6-8 for an external tangency radius or Figure 6-9 for an internal tangency radius.

6.4 True Dimensioning on Nonplanar Surfaces

(a) "TRUE" following a dimension may be used to locate features or items positioned on nonplanar surfaces. See Figure 6-10.

(b) "TRUE" following a dimension means the dimension applies along the theoretically exact nonplanar surface, not in the viewing plane.

(c) "TRUE" following a dimension indicates the requirement applies along the near surface of the part.

(d) When all dimensions of a view or section are dimensioned in this way, a note such as "ALL DIMENSIONS ARE TRUE" may be added to the view callout. See Figure 6-11.

Figure 6-3 135-deg Drill Point Callout Example

This on a drawing

⌀ 1.080 / .900   BH 135   ▽ .800 / .600

Means this

180°

135°

⌀ .900

⌀ 1.080

.600

.800

Any configuration
within shaded region
is allowable

6.1.2
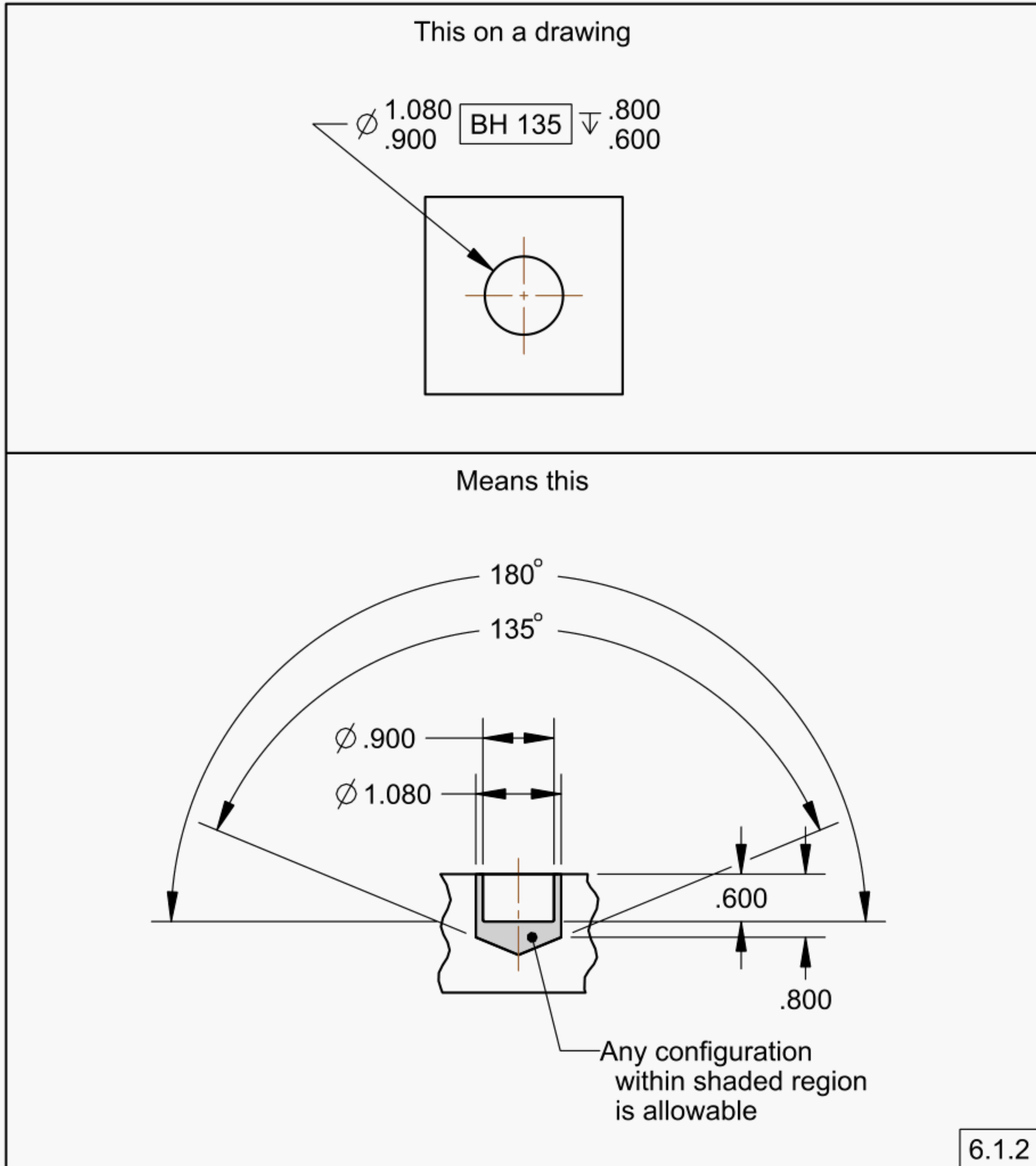
Figure 6-4 Edge Break Examples
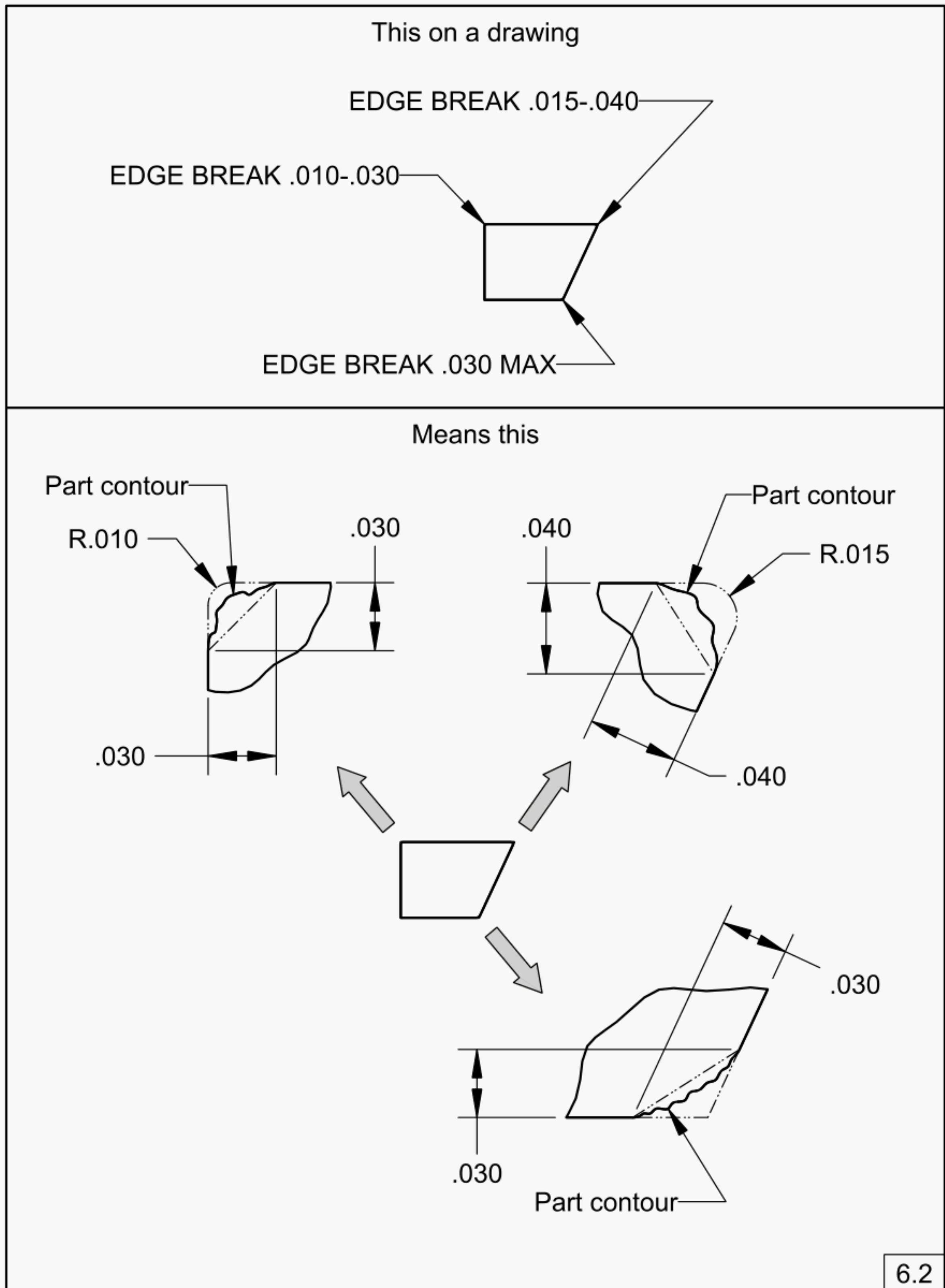
Figure 6-5 Center-Located Radius

This on a drawing

R.25±.02

.30±.01

.30±.01

At one tolerance extreme
means this

Part contour

R.23

R.27

.29

.29

At the opposite tolerance extreme
means this

Part contour

R.23

R.27

.31

.31

6.3.1
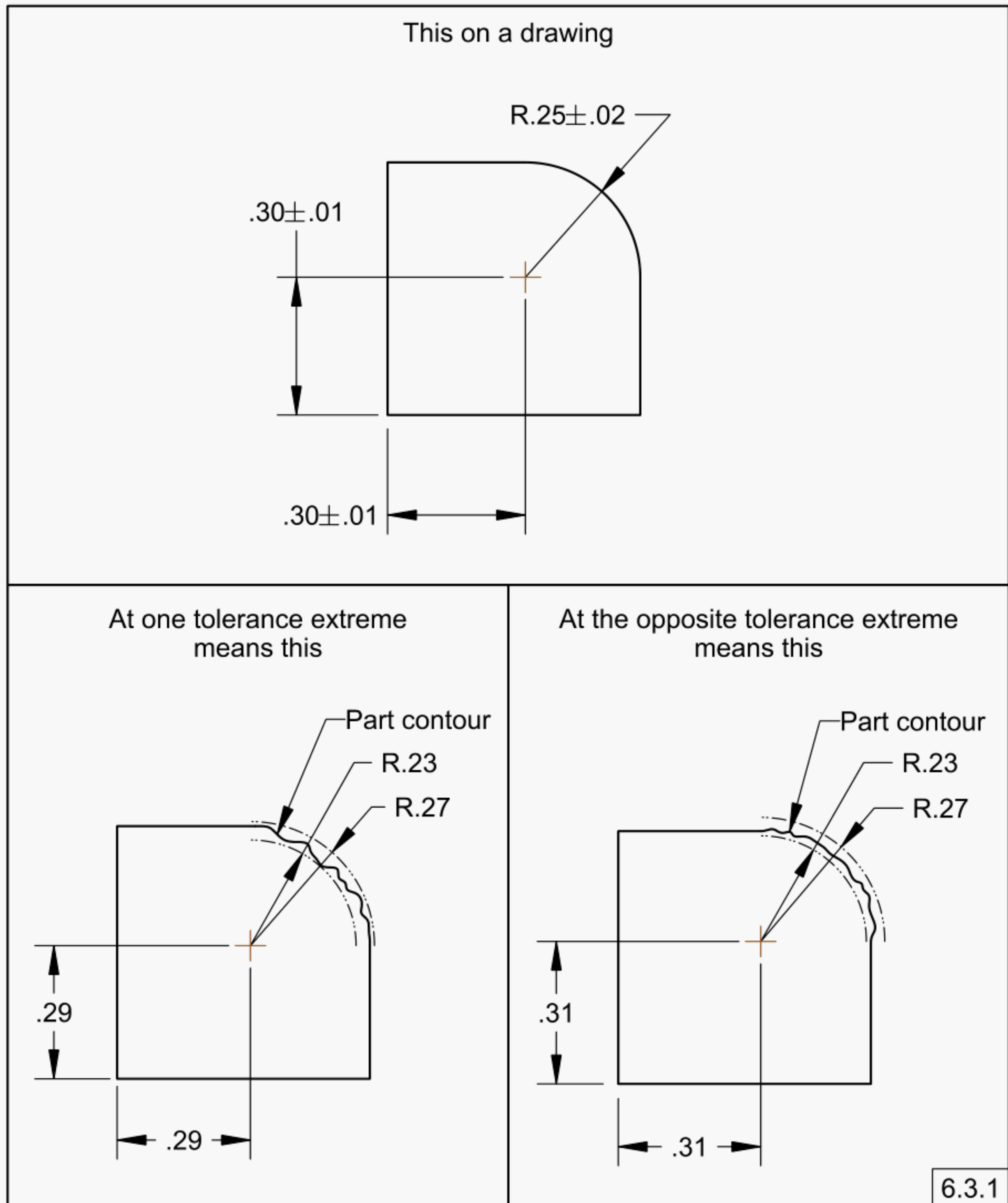
Figure 6-6 Tangent-Located Radius

This on a drawing	Means this
R.25±.05	R.20
	R.30
	Part contour
	6.3.3
	6.3.1
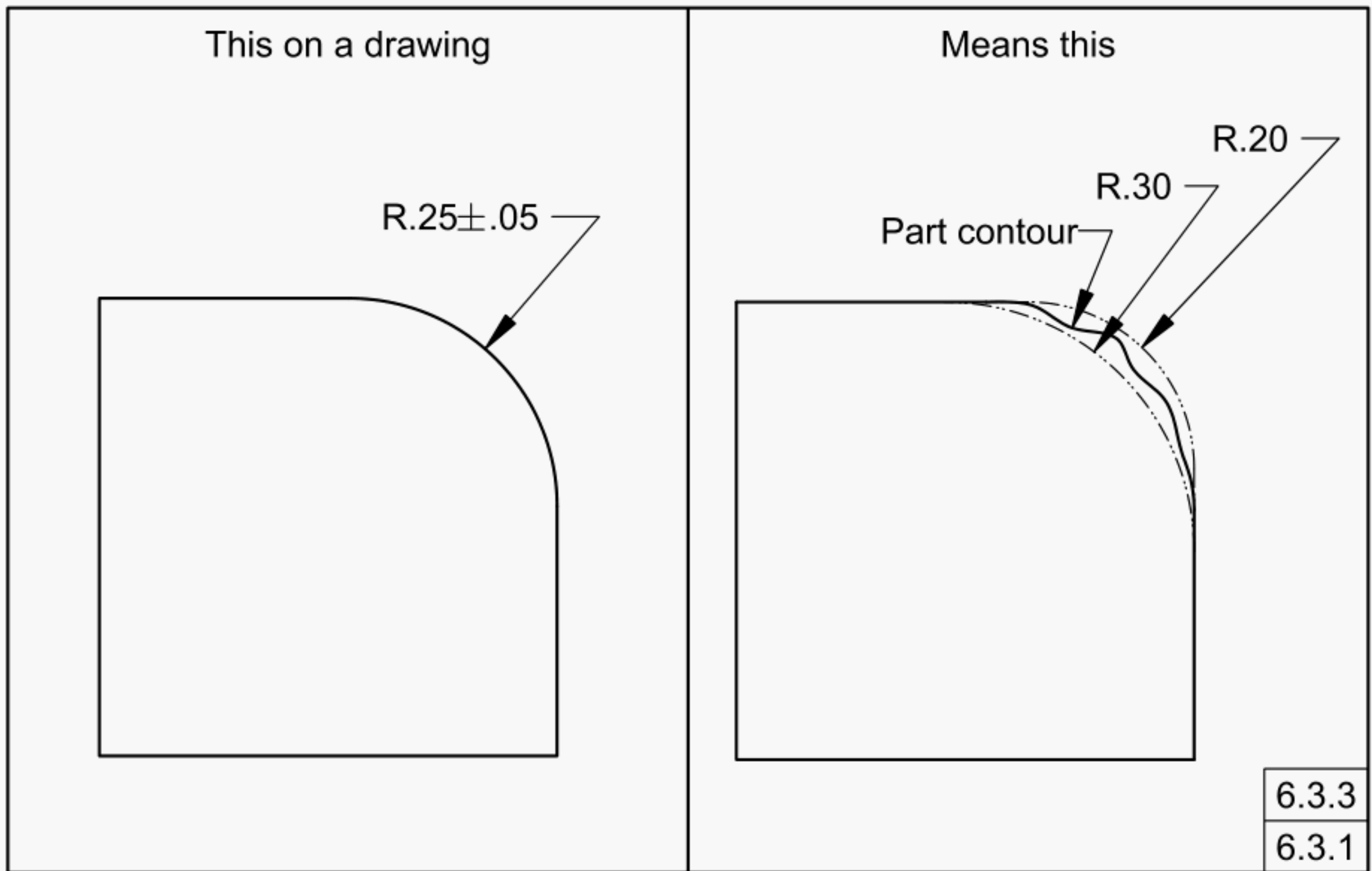
Figure 6-7 Controlled Radius

This on a drawing	Means this
⌓ .06	Part contour
CR.25	Fair curve without reversals
	Area in which fair curve exists
	.06 tolerance zone
	6.3.2
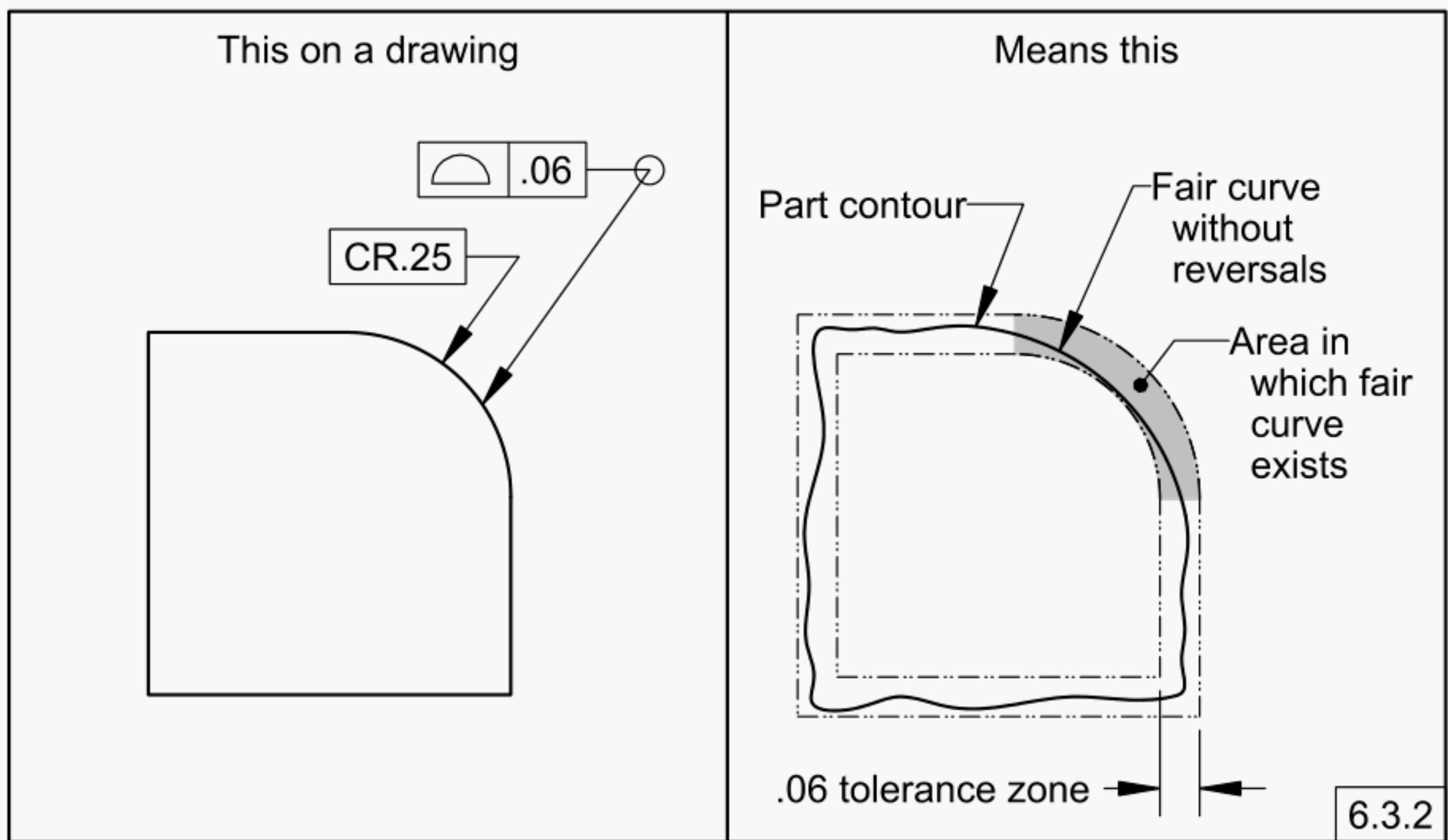
Figure 6-8 External Radius — Permissible Termination

An edge tangency radius applied to a part

R

Is allowed to be out of tangency by this amount

5° max

R

(100°)

5° max

5° max

R

(80°)

5° max

Mismatch

Mismatch

6.3.4

6.3.3
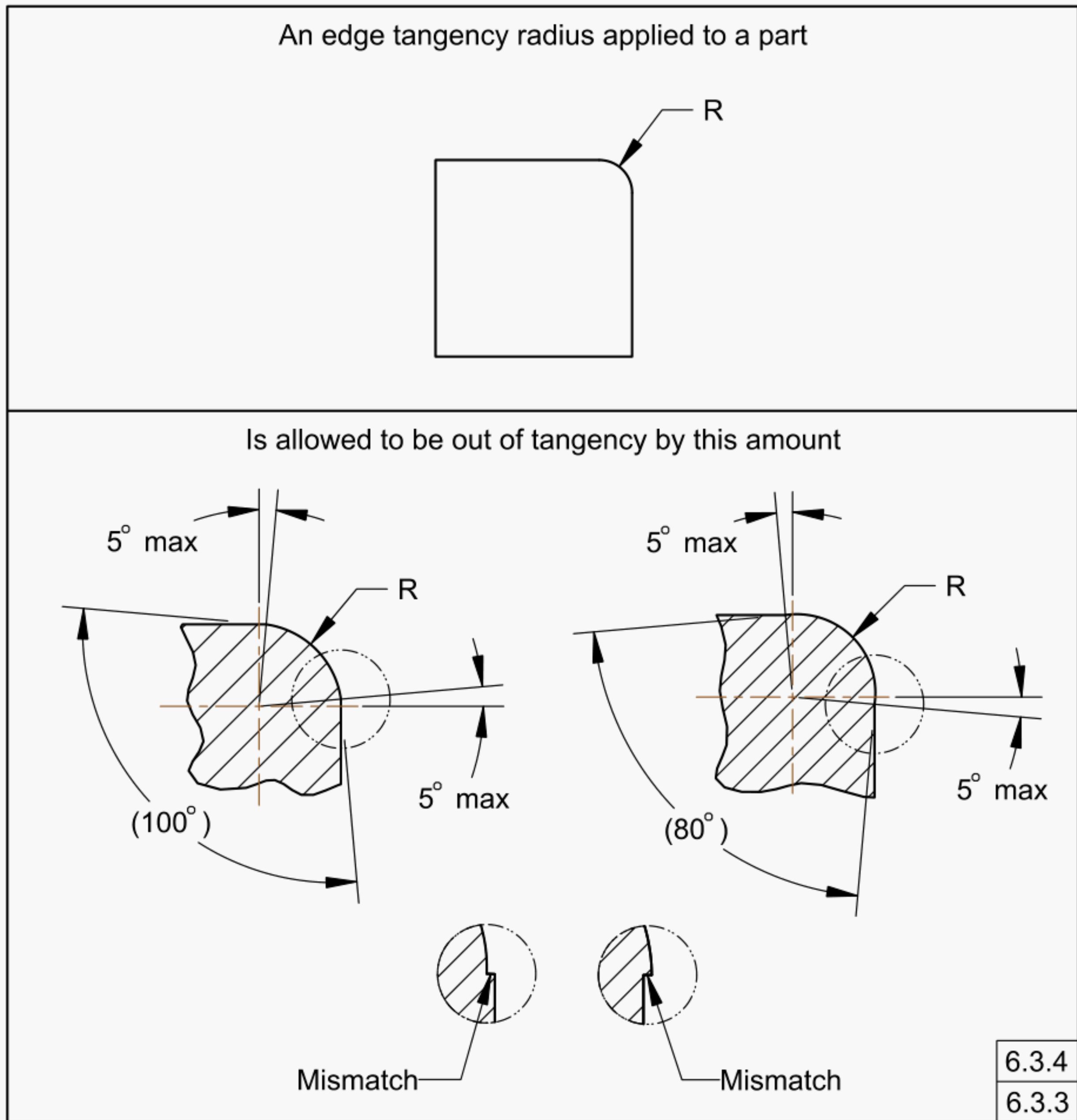
Figure 6-9 Internal Radius — Permissible Termination

A corner tangency radius applied to a part

R

Is allowed to be out of tangency by this amount

100°

5° max

R

5° max

80°

5° max

R

5° max

Mismatch

Mismatch
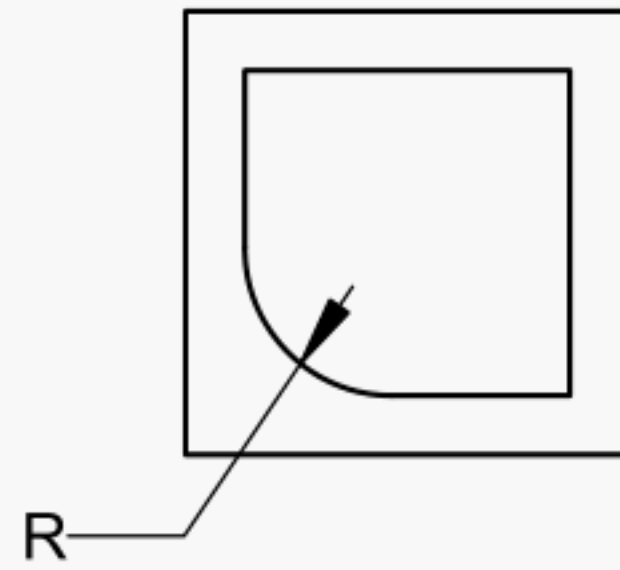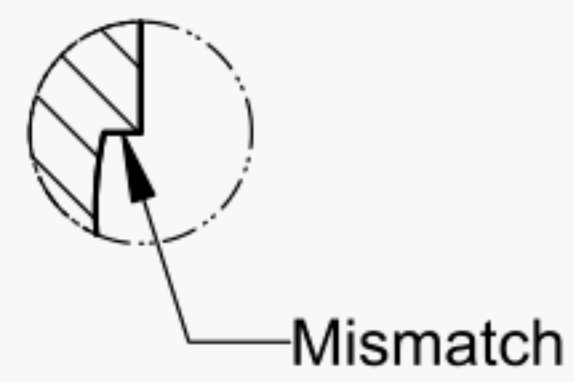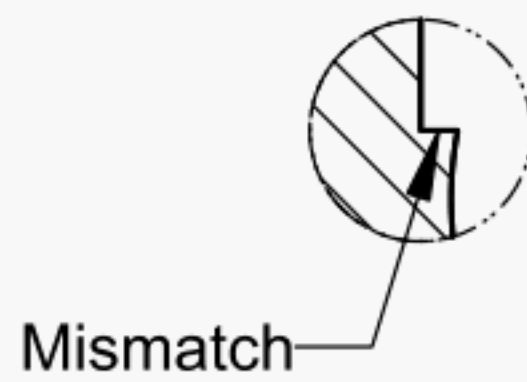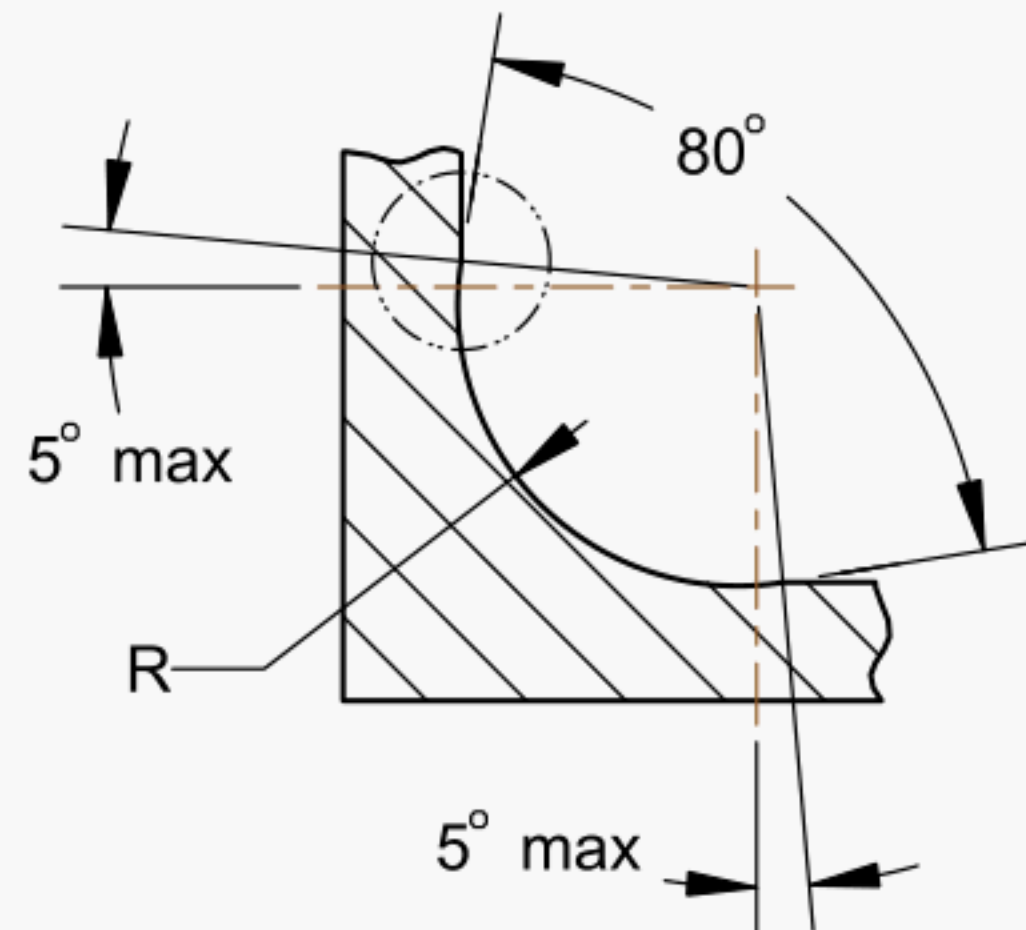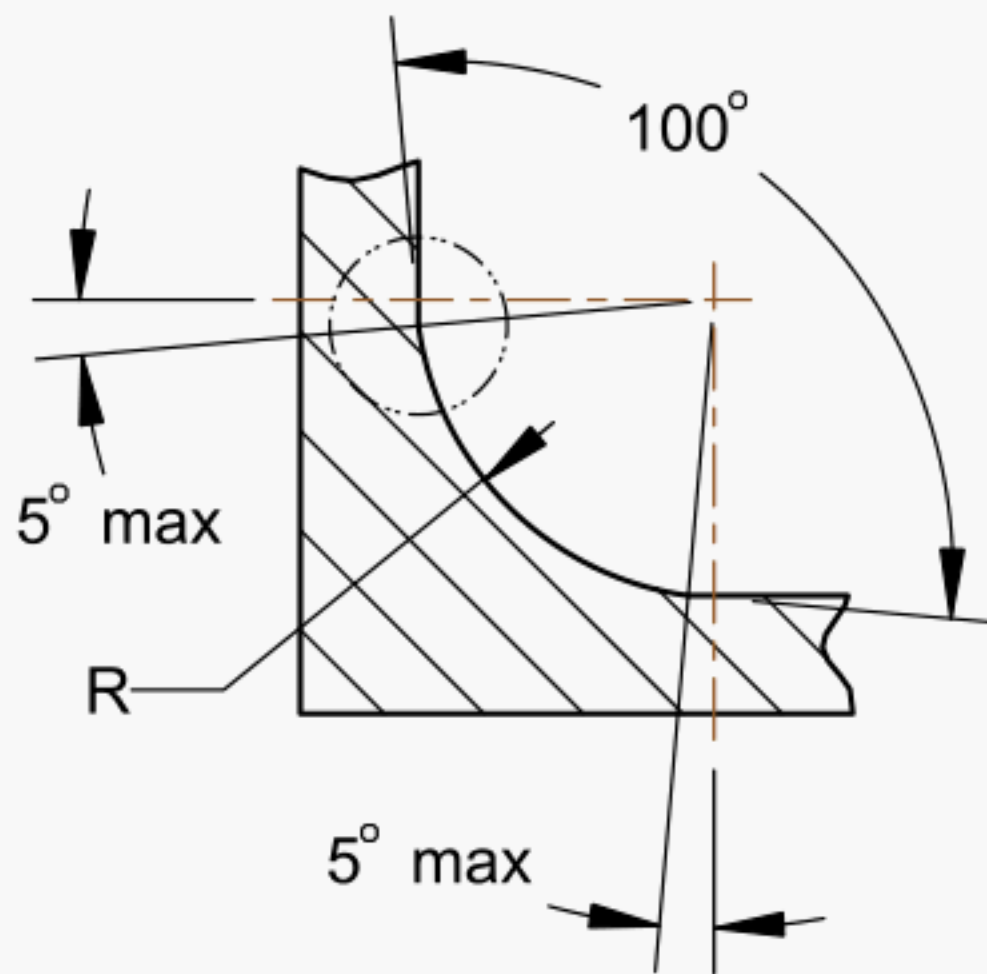
6.3.4

6.3.3

Figure 6-10 Views With True Dimensions

This on a drawing

120

176.798
TRUE

Means this

120

176.798 TRUE

6.4(a)
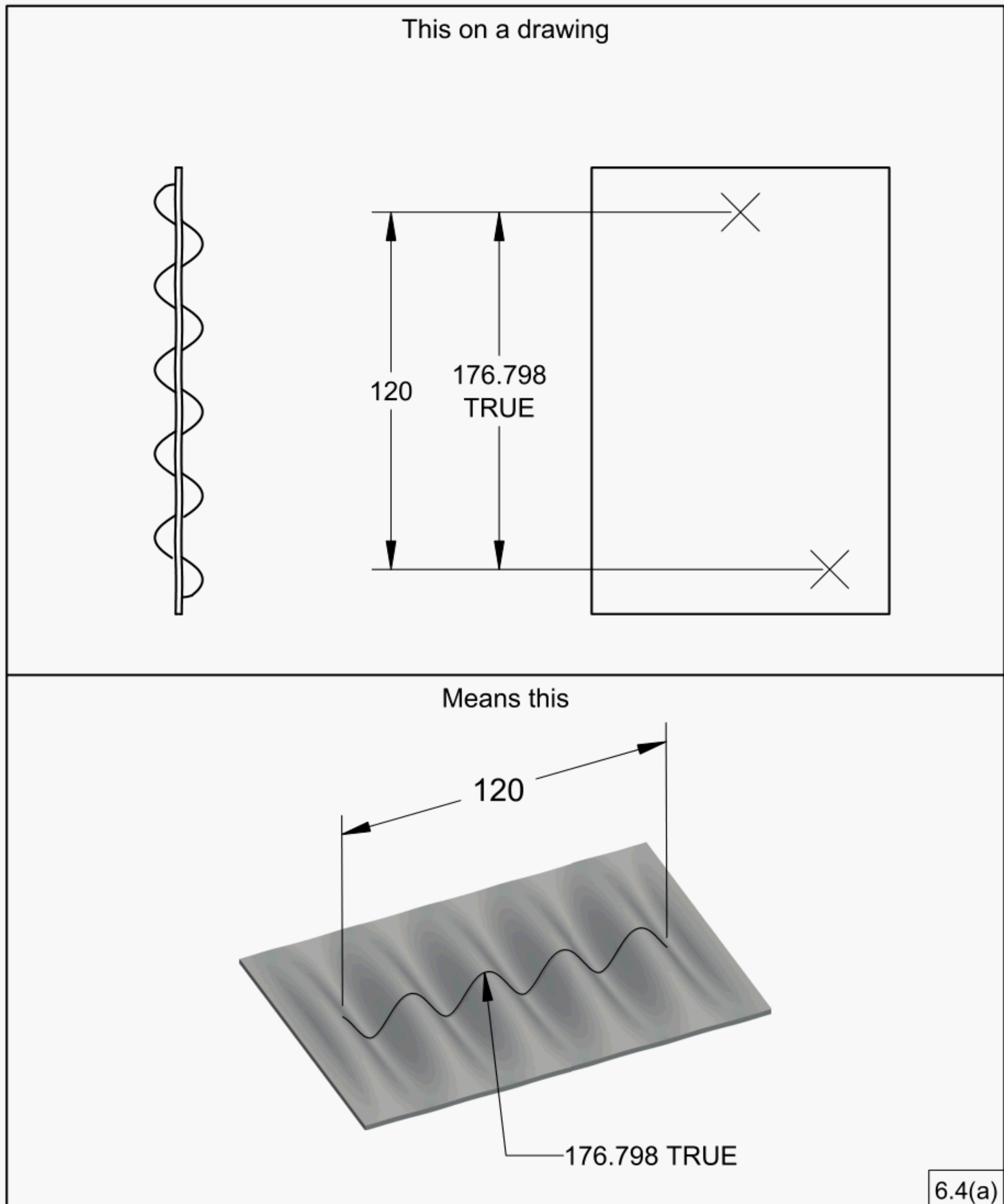
Figure 6-11 Views With All Dimensions True

This on a drawing

A

114.724
TRUE

129.806

50.099
TRUE

24°

A

SECTION A-A
ALL DIMENSIONS ARE TRUE

Means this

50.099 TRUE

114.724 TRUE

129.806 TRUE

6.4(d)

Figure 6-12 Parabola Tangency

6.5 Parabolic Fillets

(a) Parabolic fillets are a concave easing of a corner or convex easing of an edge of a part typically used to reduce stress concentration. The cross section of a parabolic fillet is a parabola instead of a constant arc. This provides greater strength characteristics than a fillet radius in a direction parallel or nearly parallel to the axis of symmetry.

(b) General applications for parabolic fillets include areas where a circular arc of sufficient size to mitigate stress reduction is not feasible, and where the strength requirement in one direction is greater than in another.

(c) Due to the geometric definition of a parabola, the axis of symmetry for a parabola cannot be perpendicular to either blending surface when the surfaces meet at a 90-deg angle. Additionally, a parabola cannot be tangent to any surface parallel to the parabola's axis of symmetry. For these reasons, the axis of symmetry shall be set to an angle other than 0 deg from the adjacent surface to the long leg of the parabolic fillet. See Figure 6-12.
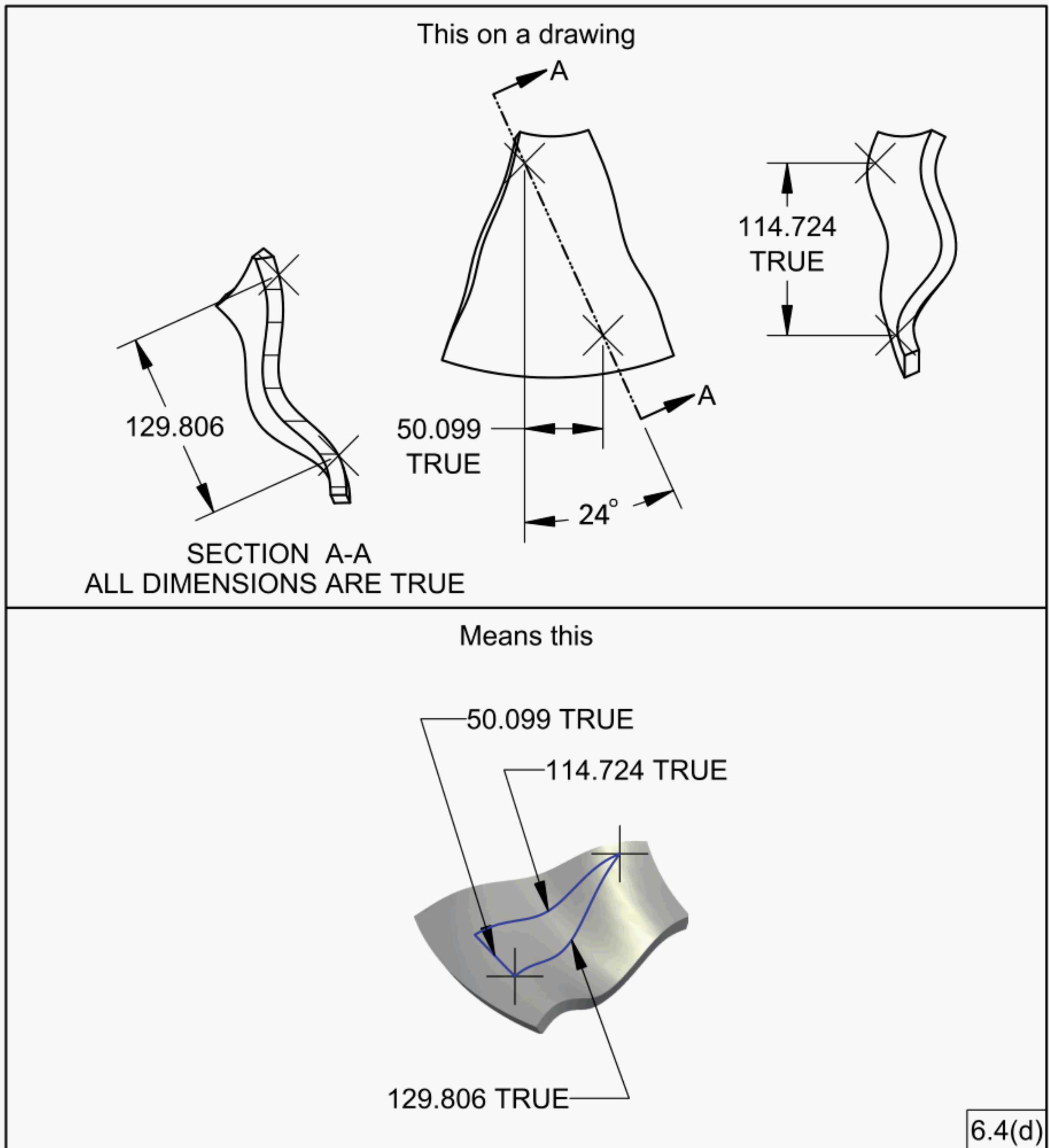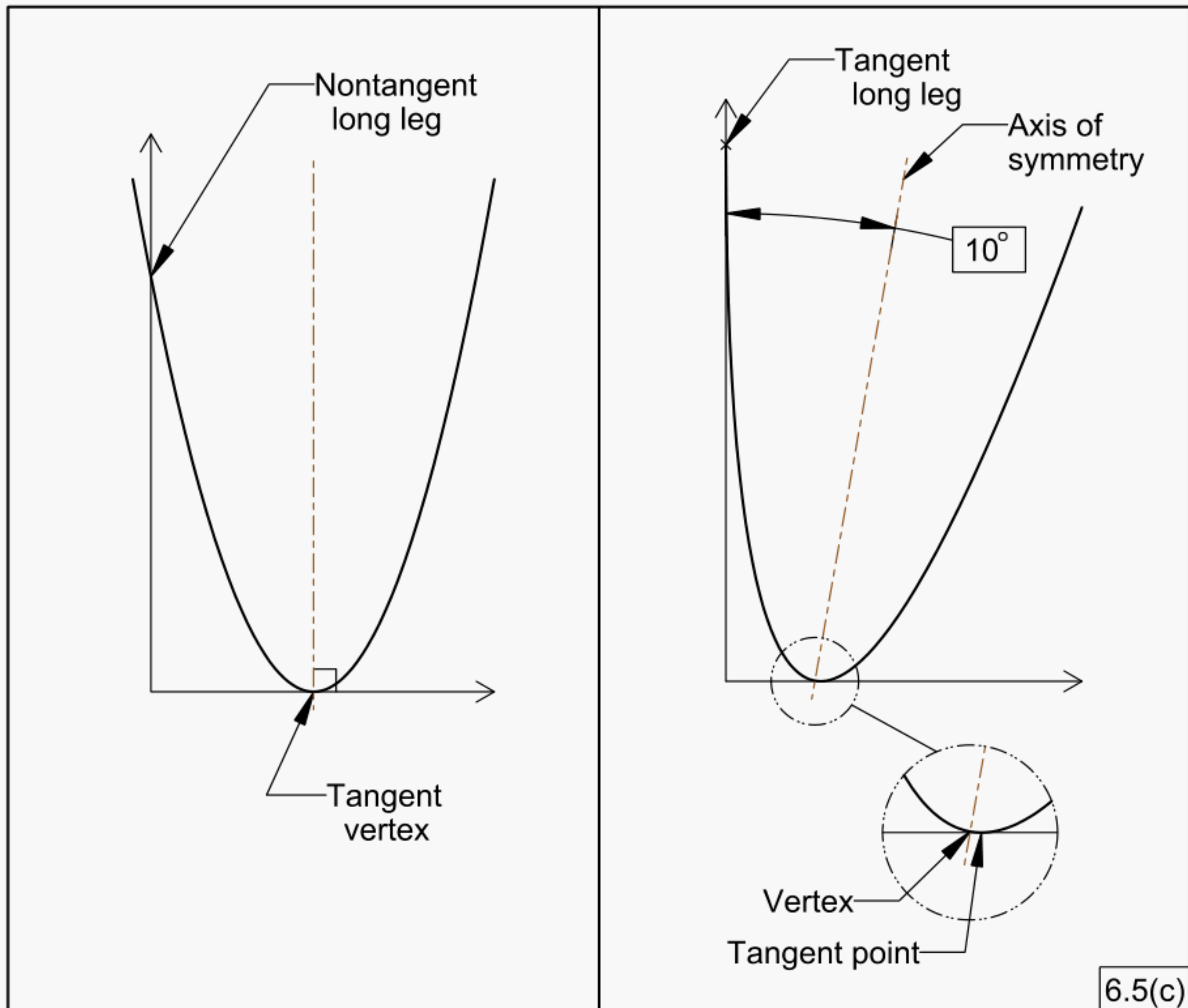
(d) A parabolic fillet is expressed using three terms in the following order: length of long leg as measured parallel to the surface adjacent to the long leg, length of short leg as measured perpendicular to the surface adjacent to the long leg, angle between the surface adjacent to the long leg and the parabola's axis of symmetry.

(1) The parabolic fillet angle is between 0 deg and 90 deg. A parabolic fillet whose axis of symmetry is parallel to the surface adjacent to the long leg of the fillet has an angle of 0 deg.

(2) Similar to a radius, the included angle of the adjacent surfaces of a parabolic fillet shall be dimensioned using the product definition.

(3) The dimension of a parabolic fillet shall be expressed as a basic dimension by being enclosed in a basic dimension symbol.

(4) At least one leg of the parabolic fillet shall be dimensioned to clarify the fillet's orientation. When used to relieve stress concentrations, the long leg of the parabolic fillet should correspond to the direction of the major fatigue load. See Figure 6-13.

(5) Parabolic fillets shall be specified using the abbreviation "PA" followed by a three-segment representation of the parabola's proportion and orientation. Each segment of the parabolic fillet symbol shall be separated with a "by" symbol (an uppercase X preceded and followed by a space) (see Figure 6-13); for example

(-a) PA 3.00 X 1.00 X 0° = parabolic fillet, 3.00 on the longer leg and 1.00 on the shorter leg, with the major axis aligned parallel to the long leg

(-b) PA .750 X .200 X 10° = parabolic fillet, .750 on the longer leg and .200 on the shorter leg, with the major axis rotated 10 deg from the long leg into the included angle of the fillet

(e) Tolerances shall be specified in the form of a profile tolerance in accordance with ASME Y14.5.

(f) The preferred fillet height-to-length ratios are 1:2, 1:3, 1:4, and 1:5.

NOTE: A 1:1 ratio is a constant arc and shall not be specified as a parabolic fillet.

6.6 Elliptical Fillets

(a) Elliptical fillets are a concave easing of a corner or convex easing of an edge of a part typically used to reduce stress concentration. The cross section of an elliptical fillet is an ellipse instead of a constant arc. This provides greater strength characteristics than a fillet radius in a direction parallel or nearly parallel to the ellipse's major axis.

(b) General applications for elliptical fillets include areas where a circular arc of sufficient size to mitigate stress reduction is not feasible, and where the strength requirement in one direction is greater than in another.

(c) Due to their geometric definition, ellipses may be tangent at their major and minor axes when blending two surfaces that meet at a 90-deg angle. For these reasons, an ellipse may be easier to define than a parabola when blending surfaces that meet at a 90-deg angle. See Figure 6-14.

(d) An elliptical fillet is expressed using three terms in the following order: length of long leg as measured parallel to the surface adjacent to the long leg, length of short leg as measured perpendicular to the surface adjacent to the long leg, angle between the surface adjacent to the long leg and the ellipse's major axis.

(1) The elliptical fillet angle is between 0 deg and 90 deg. An elliptical fillet whose major axis is parallel to the surface adjacent to the long leg of the fillet has an angle of 0 deg.

(2) Similar to a radius, the included angle of the adjacent surfaces of an elliptical fillet shall be dimensioned using the product definition.

(3) The dimension of an elliptical fillet shall be expressed as a basic dimension by being enclosed in a basic dimension symbol.

(4) At least one leg of the elliptical fillet shall be dimensioned to clarify the fillet's orientation. When used to relieve stress concentrations, the long leg of the elliptical fillet should correspond to the major fatigue load. See Figure 6-14.

(5) Elliptical fillets shall be specified using the abbreviation "EP" followed by a three-segment representation of the ellipse's proportion and orientation. Each segment of the elliptical fillet symbol shall be separated with a "by" symbol (an uppercase X preceded and followed by a space) (see Figure 6-14); for example

(-a) EP 3.00 X 1.00 X 0° = elliptical fillet, 3.00 on the longer leg and 1.00 on the shorter leg, with the major axis aligned parallel to the long leg

(-b) EP .750 X .200 X 10° = elliptical fillet, .750 on the longer leg and .200 on the shorter leg, with the major axis rotated 10 deg from the long leg into the included angle of the fillet

(e) Tolerances shall be specified in the form of a profile tolerance in accordance with ASME Y14.5.

(f) The preferred fillet ratios are shown in para. 6.5(f).

NOTE: A 1:1 ratio is a constant arc and shall not be specified as an elliptical fillet.

Figure 6-13 Parabolic Fillet — Example Specification

This on a drawing

4.00

⌒ .80 | A | B | C

PA 4.00 X 1.00 X 19.5°

Means this

19.5°

Angle between the long leg adjacent surface and the axis of symmetry

Axis of symmetry

Basic parabola

4.00
Length of long leg

.80 tolerance zone

1.00
Length of short leg

6.5(d)(4)

6.5(d)(5)

ASME AED-1–2018
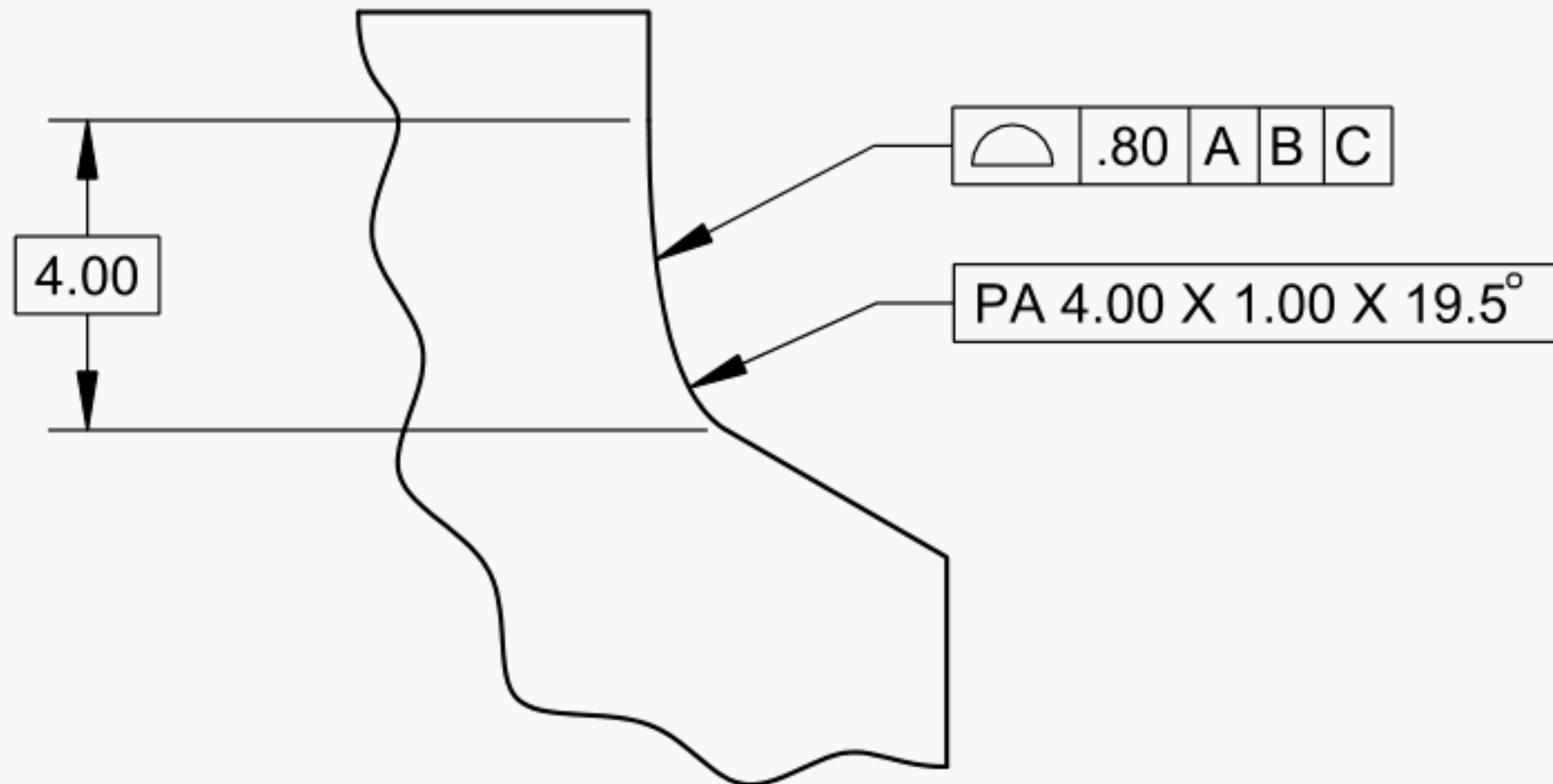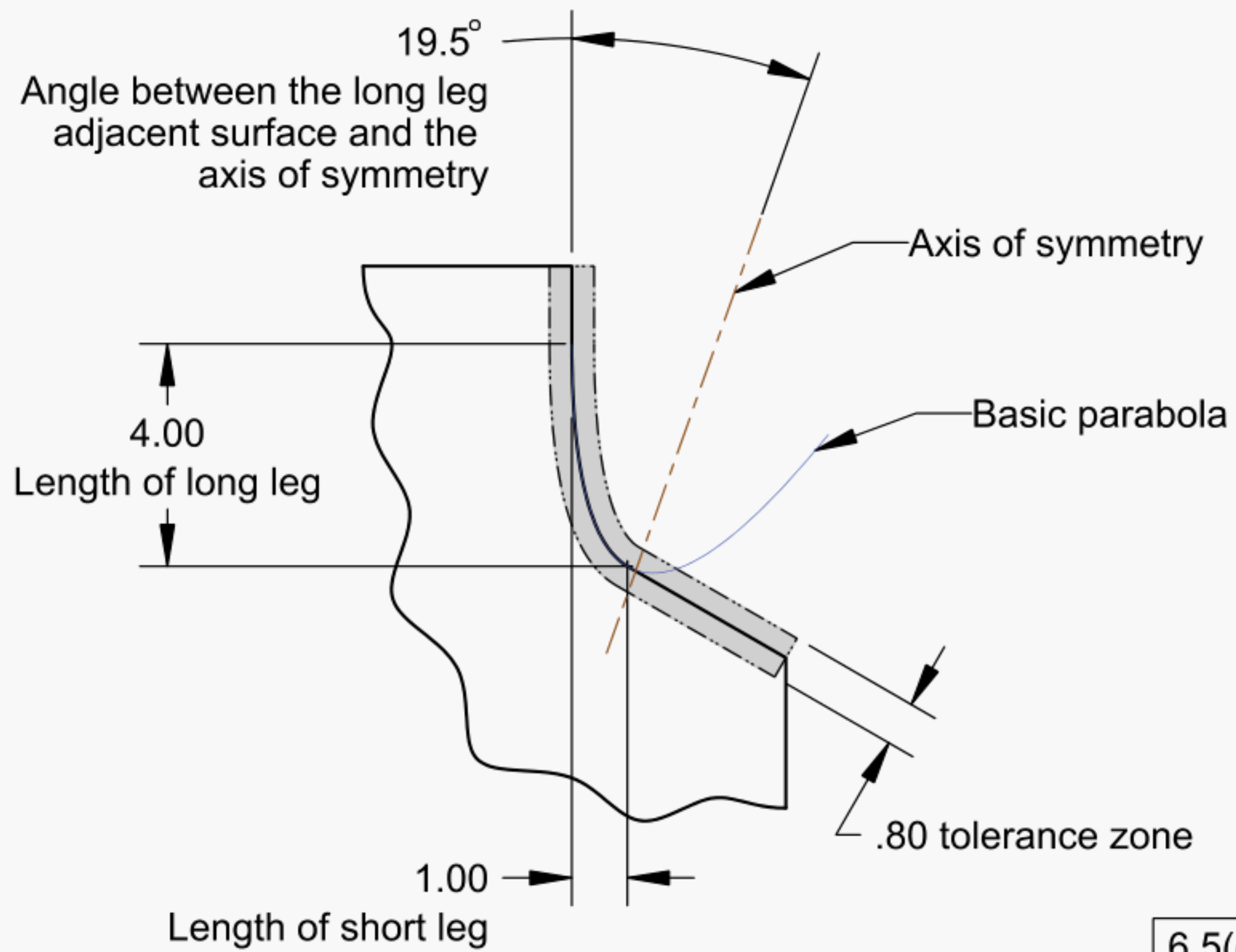
Figure 6-14 Elliptical Fillet — Example Specification

This on a drawing

⌒ | .50 | A | B | C

EP 4.00 X 1.00 X 0°

4.00

Means this

The long leg adjacent
surface and major
axis are parallel.

Major axis

Basic ellipse

4.00
Length of long leg

.50 tolerance zone

1.00
Length of short leg

6.6(c)
6.6(d)(4)
6.6(d)(5)

29
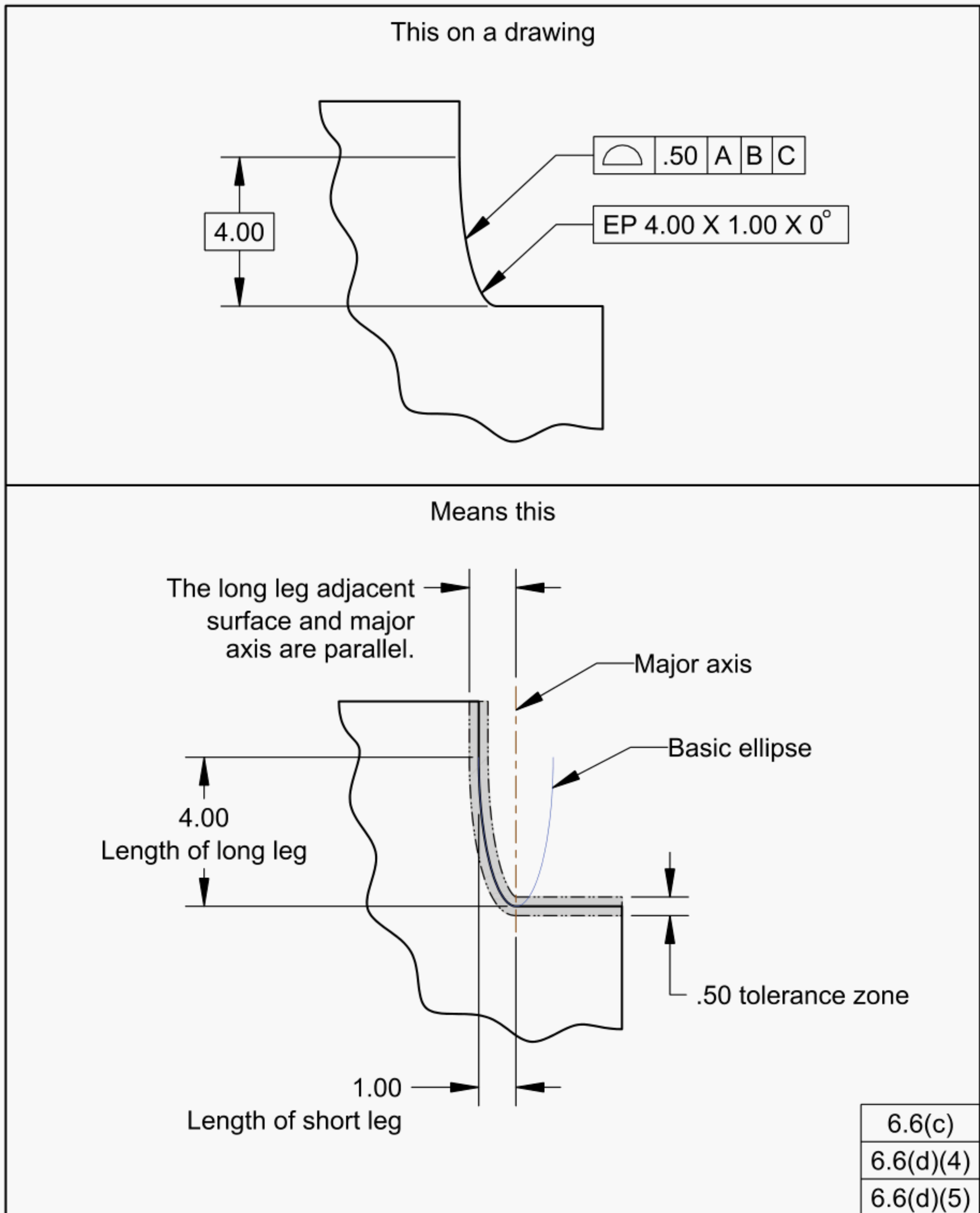
MANDATORY APPENDIX I
DYNAMIC PROFILE

I-1 DYNAMIC PROFILE TOLERANCE MODIFIER

By default, a profile tolerance zone follows the true profile of the considered feature. A profile tolerance zone is static and controls both the form and size of the considered feature unless the dynamic profile tolerance modifier is applied. Where it is desirable to refine the form but not the size of a considered feature that is controlled by a profile tolerance, the dynamic profile tolerance modifier, Δ, may be applied to a refining profile tolerance. See Figure 1-1. The function of the dynamic profile is to allow form to be controlled independent of size.

Where the dynamic profile tolerance modifier is applied, the zone is permitted to progress (expand or contract normal to the true profile) while maintaining the specified constant width (distance between the boundaries). This retains the form control while relaxing the size control. The actual feature shall simultaneously be within the dynamic profile tolerance zone and any other applicable tolerance zone.

I-2 DYNAMIC PROFILE TOLERANCE CONTROLLING FORM

When the dynamic tolerance modifier is applied to a lower segment of a composite tolerance without datum feature references, the tolerance zone controls the form but not the size of the feature, and it uniformly progresses (expands or contracts) normal to the true profile, UOS. See Figures 1-1 and I-1. The 2-mm profile tolerance zone is constrained in translation and rotation relative to the datum reference frame established by datum features A, B, and C. In Figure I-1, the 0.4 dynamic profile zone is an equal bilateral about the basic profile of the toleranced feature. The 0.4 dynamic profile tolerance zone may rotate and translate relative to the datum reference frame. In addition, the tolerance zone may expand or contract while maintaining the form of the feature within the 0.4 tolerance zone, releasing the size of the feature to uniformly progress within the upper segment of the composite profile tolerance. The actual feature shall simultaneously be within both tolerance zones.
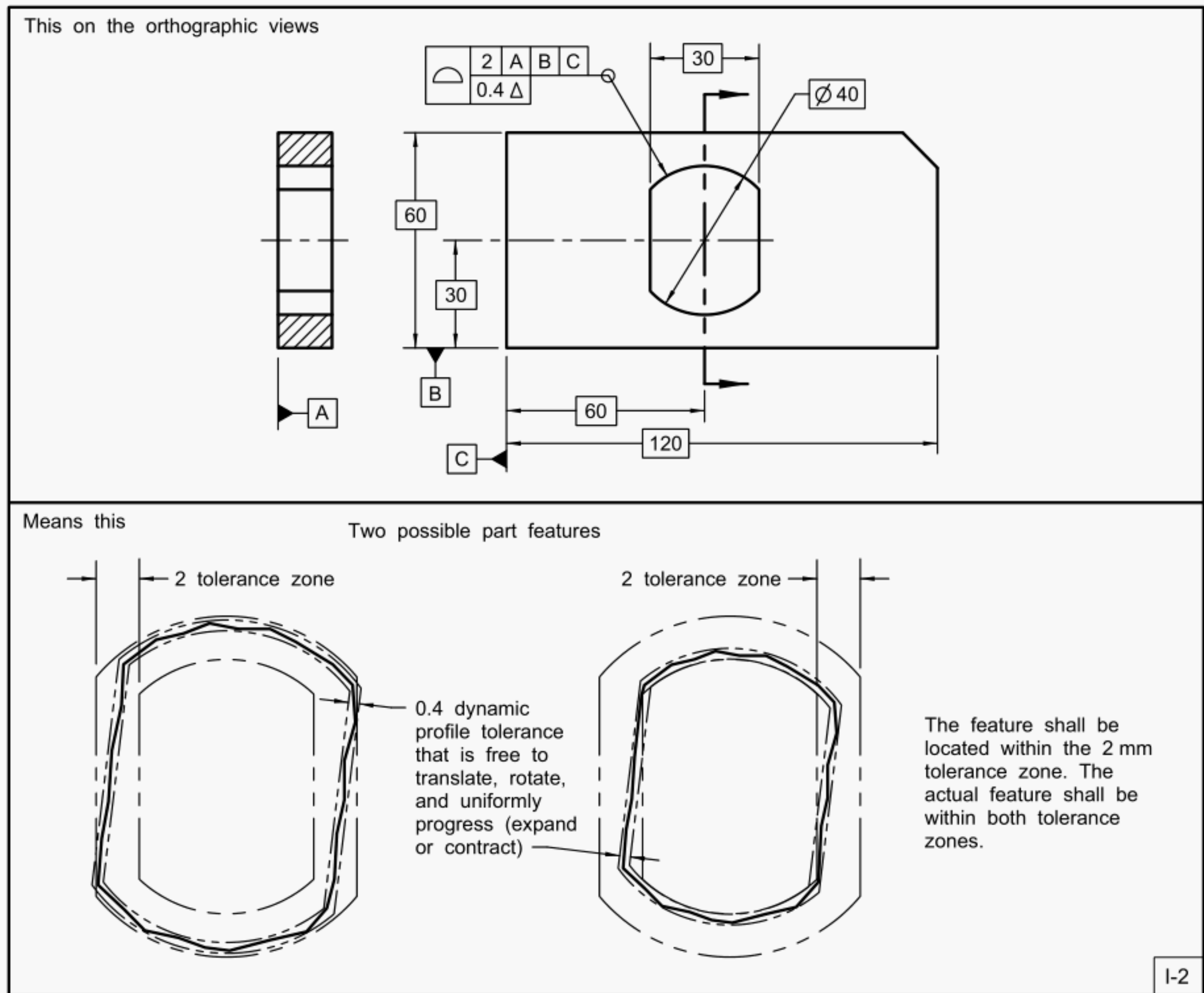
I-3 DYNAMIC PROFILE TOLERANCE CONTROLLING FORM AND ORIENTATION

When the dynamic tolerance modifier is applied to a lower segment of a composite tolerance and includes datum feature references, the tolerance zone controls the form and orientation but not the size of the feature. See Figure I-2. The 2-mm profile tolerance zone is constrained in translation and rotation relative to the datum reference frame established by datum features A, B, and C. The 0.4 dynamic profile tolerance zone in the lower segment of the composite tolerance is constrained in rotation but not translation relative to the datum reference frame. This tolerance zone may expand or contract while maintaining the form and orientation of the feature within the 0.4 tolerance zone, allowing the size of the feature to uniformly progress. The actual feature shall simultaneously be within both tolerance zones.

I-4 DYNAMIC PROFILE TOLERANCE APPLIED TO THE LOWER SEGMENT OF MULTIPLE SINGLE-SEGMENT FEATURE CONTROL FRAMES

When the dynamic profile tolerance modifier is applied in a segment of multiple single-segment feature control frames and includes datum feature references, the tolerance zone is constrained in translation and rotation, but not size, as applicable. The tolerance zone controls the form and orientation of the feature. See Figure I-3. The 2-mm profile tolerance zone is constrained in translation and rotation relative to the datum reference frame established by datum features A, B, and C. The 0.4 dynamic profile tolerance zone in the lower segment is constrained in translation and rotation relative to the datum reference frame. This tolerance zone may expand or contract while controlling the form, orientation, and location of the feature within the 0.4 tolerance zone, allowing the size of the feature to uniformly progress. The actual feature shall simultaneously be within both tolerance zones.

NOTE: Multiple features are considered to be a single feature unless "INDIVIDUALLY" follows a separate single-segment profile feature control frame.

Figure I-1 Composite Profile With Dynamic Profile to Control Form

This on the orthographic views

Means this

Two possible part features

2 tolerance zone

2 tolerance zone

0.4 dynamic profile tolerance that is free to translate, rotate, and uniformly progress (expand or contract)

The feature shall be located within the 2 mm tolerance zone. The actual feature shall be within both tolerance zones.

I-2

I-5 DYNAMIC PROFILE TOLERANCE APPLIED TO A SURFACE OF REVOLUTION

The dynamic profile tolerance may be applied to surfaces of revolution to maintain the shape of the considered feature while allowing its size to vary. See Figure I-4. The entire surface between A and B shall be within the profile tolerance of 0.25, which is equally disposed about the true profile. The 0.15 dynamic profile tolerance zone in the lower segment is constrained in translation and rotation relative to the datum reference frame. This tolerance zone may expand or contract while controlling the form, orientation, and location of the feature within the 0.15 tolerance zone, allowing the size of the feature to uniformly progress. The actual feature shall simultaneously be within both tolerance zones.

Figure I-2 Composite Profile With Dynamic Profile to Control Form and Constrain Rotational Degrees of Freedom

This on the orthographic views

⌒ | 2 | A | B | C
0.4 △ | A | B

30

Ø 40

60

30

B

A

60

120

C

Means this

Two possible part features

2 tolerance zone

2 tolerance zone

0.4 dynamic profile tolerance that is free to translate and uniformly progress (expand or contract)

The feature shall be located within the 2 mm tolerance zone. The actual feature shall be within both tolerance zones.

I-3
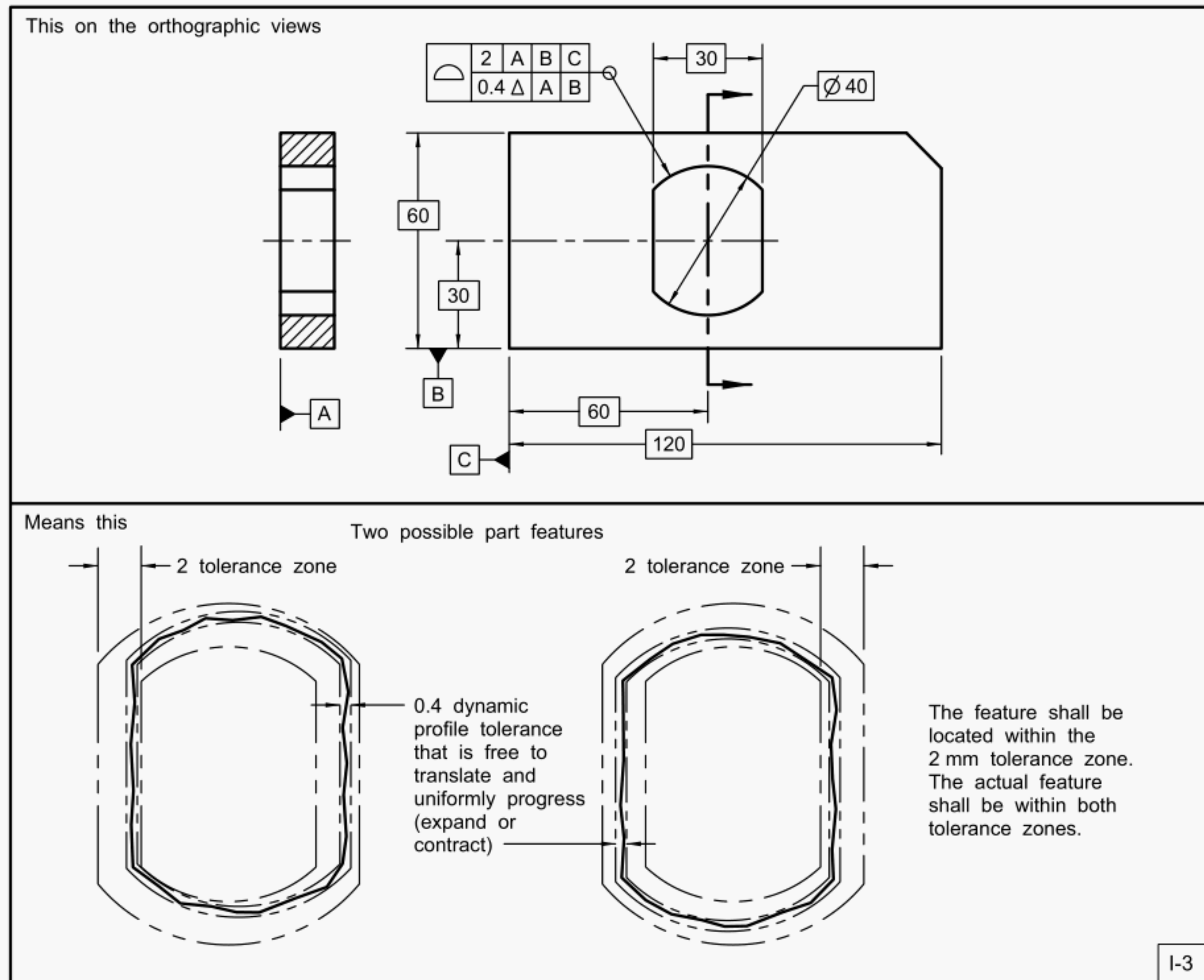
Figure I-3 Use of Dynamic Profile in a Two-Single-Segment Profile Tolerance

This on the orthographic views

2 | A | B | C
0.4 △ | A | B

30

Ø 40

60

30

B

A

C

60

120

Means this

Two possible part features

2 tolerance zone

2 tolerance zone

0.4 dynamic profile tolerance zone that is free to translate along one axis and uniformly progress (expand or contract)

The feature shall be located within the 2 mm tolerance zone. The actual feature shall be within both tolerance zones.

I-4

Figure I-4 Dynamic Profile of a Surface of Revolution

This on the orthographic view

⌒ 0.4 Ⓜ

Ø20

B

R10

⌒ 0.25 Ⓜ N
⌒ 0.15 △ Ⓜ N

A ◄─► B

R127

78

45

Ø38

25

Ø50

4

A

⊥ Ø0Ⓜ M

N

M

Ø45 ±0.2

Ø62

Means this

0.15 Profile zone

0.25 Profile zone

B

Datum axis established
by datum features
M and N

45

Ø38

25

Ø50

Datum plane M

A
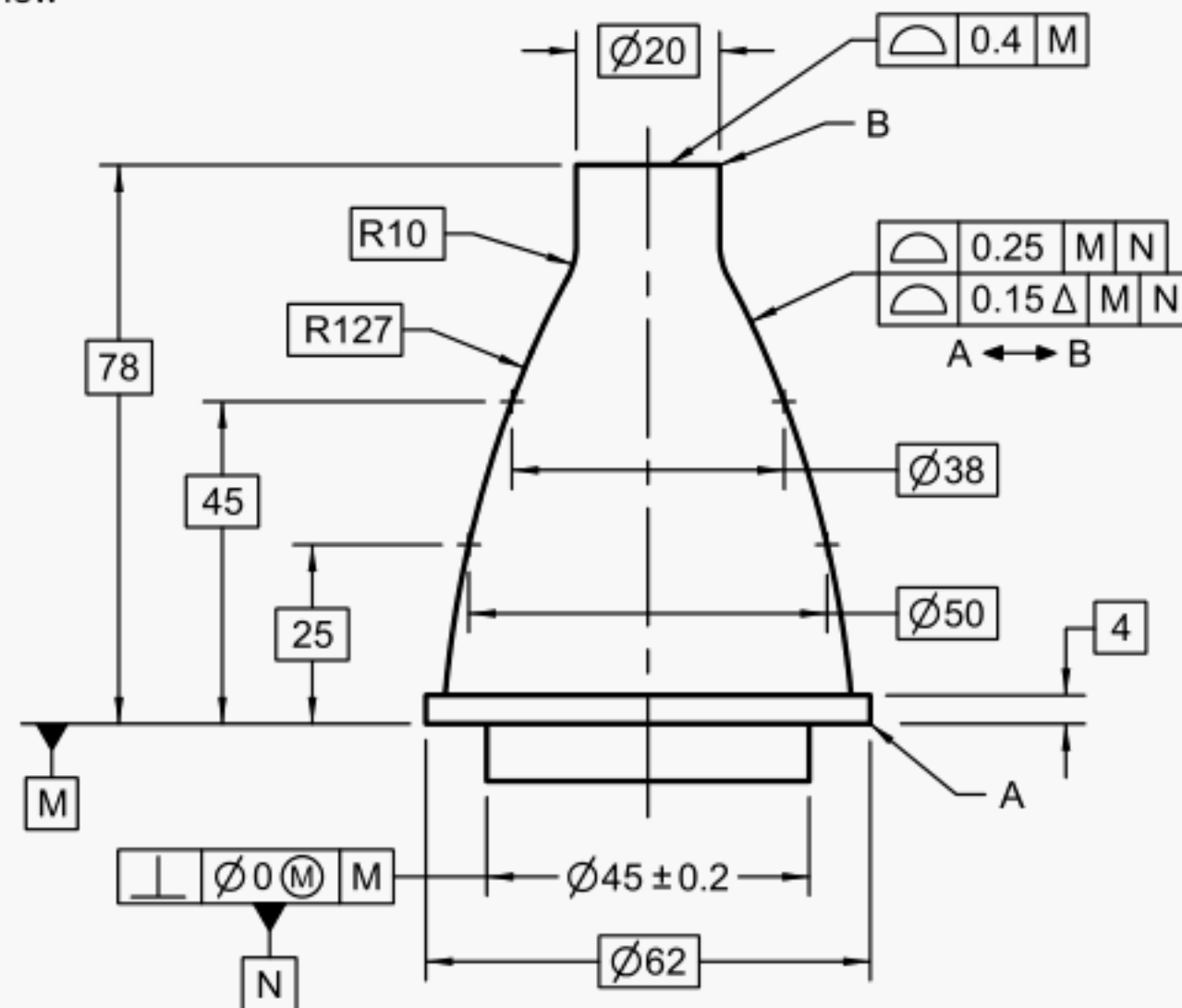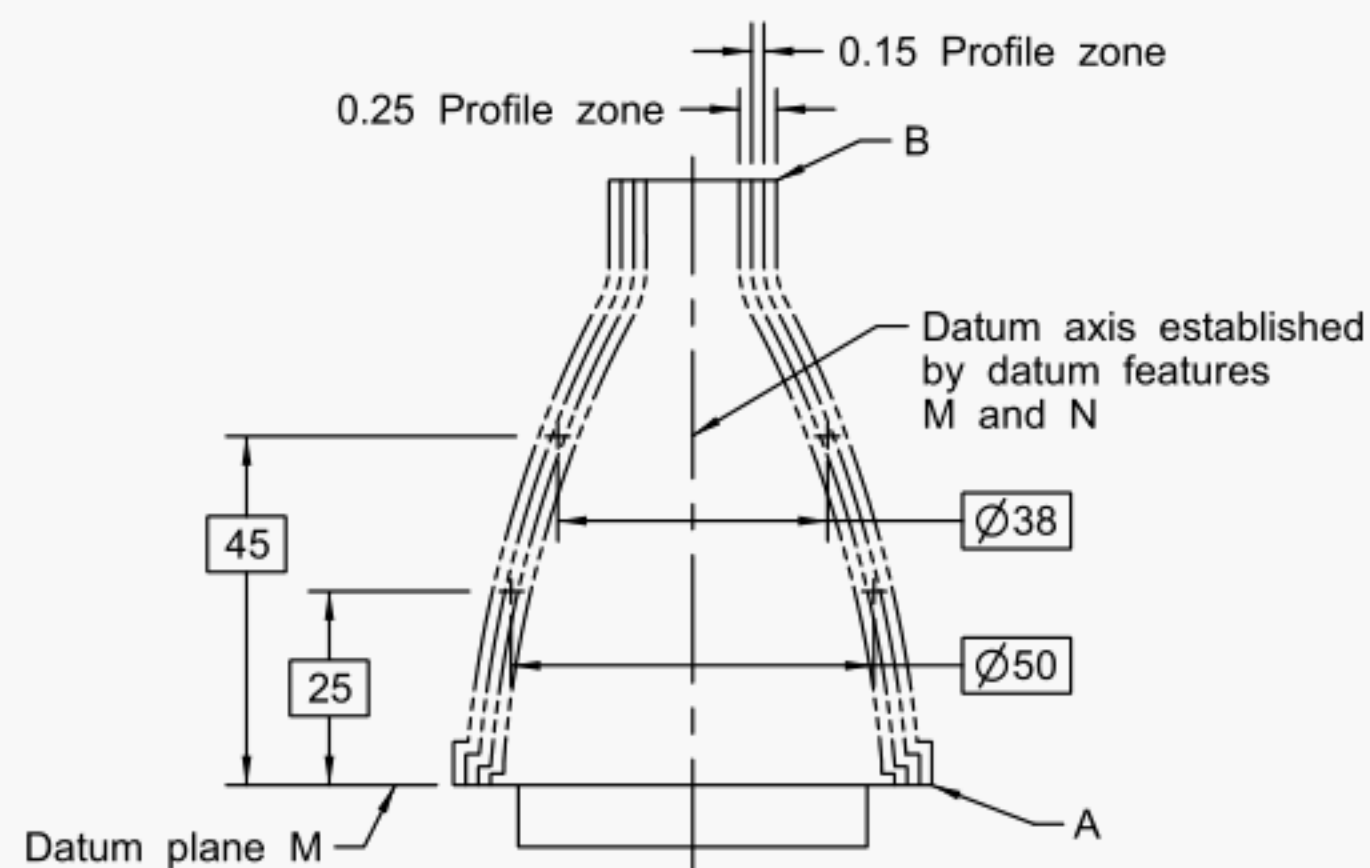
The surface between A and B shall be within two dynamic profile zones 0.25 apart, equally disposed about the true profile and positioned with respect to the primary plane M and secondary datum axis N. Additionally, the surface shall be within the 0.15 dynamic profile tolerance zone. These boundaries also are constrained in translation and rotation with respect to primary plane M and secondary datum axis N but may progress (expand and contract) within the 0.25 profile zone.

I-5

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