ASME AED-1–2023
(Revision of ASME AED-1–2018)

Aerospace and Advanced Engineering Product Definition

AN AMERICAN NATIONAL STANDARD

The American Society of Mechanical Engineers

ASME AED-1–2023
(Revision of ASME AED-1–2018)

Aerospace and Advanced Engineering Product Definition

AN AMERICAN NATIONAL STANDARD

The American Society of Mechanical Engineers
Two Park Avenue • New York, NY • 10016 USA

Date of Issuance: October 20, 2023

The next edition of this Standard is scheduled for publication in 2028.

This code or standard was developed under procedures accredited as meeting the criteria for American National Standards. The standards committee that approved the code or standard was balanced to ensure that individuals from competent and concerned interests had an opportunity to participate. The proposed code or standard was made available for public review and comment, which provided an opportunity for additional public input from industry, academia, regulatory agencies, and the public-at-large.

ASME does not "approve," "certify," "rate," or "endorse" any item, construction, proprietary device, or activity. ASME does not take any position with respect to the validity of any patent rights asserted in connection with any items mentioned in this document, and does not undertake to insure anyone utilizing a standard against liability for infringement of any applicable letters patent, nor does ASME assume any such liability. Users of a code or standard are expressly advised that determination of the validity of any such patent rights, and the risk of infringement of such rights, is entirely their own responsibility.

Participation by federal agency representatives or persons affiliated with industry is not to be interpreted as government or industry endorsement of this code or standard.

ASME accepts responsibility for only those interpretations of this document issued in accordance with the established ASME procedures and policies, which precludes the issuance of interpretations by individuals.

The endnotes and preamble in this document (if any) are part of this American National Standard.

ASME Collective Membership Mark

The American Society of Mechanical Engineers
Two Park Avenue, New York, NY 10016-5990

CONTENTS

FOREWORD

ASME AED-1–2023 is a revision of ASME AED-1–2018, Aerospace and Advanced Engineering Drawings. The objectives for this edition are to correct errors or inconsistencies, resolve deferred comments made during the review of the previous edition's draft, and introduce new concepts submitted by members of the ASME Aerospace and Advanced Engineering Product Definition (AED) Standards Committee.

It remains the intention of this Standard to be a supplement to the ASME Y14 series of standards for engineering product definition and not a replacement. To better support that aim, this Standard has been reorganized to group concepts based on the ASME Y14 standard where that concept is most likely to reside if adopted. The new section 4, Engineering Drawing Practices, covers similar concepts as ASME Y14.100. The new section 5, Dimensions and Tolerancing Practices, covers similar concepts as ASME Y14.5. ASME AED-1 will continue to add sections as it adopts additional concepts similar to those covered in other ASME Y14 standards.

In addition to regrouping existing concepts, ASME AED-1–2023 includes the following changes:

–Direction indicators for projected tolerance zones have been added.

–A symbol has replaced the "INDIVIDUALLY" notation for repetitive datum features.

–Existing concepts, such as the pilot hole location symbol, have been clarified or refined.

The ASME AED Committee collaborates with the ASME Y14 Standards Committee and its various subcommittees with the goal of migrating concepts to the appropriate ASME Y14 standards. For example, ASME Y14.5-2018 adopted the AED concept of dynamic profile. Other Y14 subcommittees are considering adopting ASME AED-1 concepts as well. When such adoptions occur, the AED Committee will move the affected information to an appendix within ASME AED-1. In this manner, control and ownership of the subject matter will be placed with the proper Y14 subcommittee, but the history and usage will remain visible within ASME AED-1. However, the AED Committee has maintained in ASME AED-1–2023 a paragraph on dynamic profile rather than moving the information to an appendix. The paragraph directs the user to ASME Y14.5-2018 for rules and methodology. Organizations that have not yet adopted ASME Y14.5-2018 can therefore access the dynamic profile control by adopting ASME AED-1–2023.

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

ASME AED-1–2023 was approved by the AED Standards Committee. It was approved as an American National Standard by the American National Standards Institute on August 18, 2023.

ASME AED COMMITTEE
Aerospace and Advanced Engineering Product Definition

(The following is the roster of the committee at the time of approval of this Standard.)

STANDARDS COMMITTEE OFFICERS

T. Bowers, *Chair*
E. F. Zwettler, *Vice Chair*
R. Ryan, *Secretary*

STANDARDS COMMITTEE PERSONNEL

T. Bowers, Lockheed Martin Space Systems Co.
J. B. Hoskins, Boeing
R. Leonovicz, Pratt & Whitney
J. Michalowicz, Stryker Corp.
D. Newman, Collins Aerospace
B. Nielsen, Boeing
M. Orthman, Jr., Honda Aircraft Co., Inc.
B. Romero, Raytheon Technologies
R. Ryan, The American Society of Mechanical Engineers
P. Vaughan, BELL Flight
S. Villyard, Lockheed Martin Space Systems Co.
W. Weiss, IBW — Ingenieurbuero Wolfgang Weiss
E. F. Zwettler, Sigmetrix, LLC
M. W. Holcumbrink, *Alternate*, Raytheon
W. Vermillion, *Alternate*, Boeing
J. V. Burleigh, *Contributing Member*, Consultant
H. Oakes, *Contributing Member*, University of Dayton Research Institute
R. A. Wheeler, *Contributing Member*, RW-OptDesign

SUBCOMMITTEE 1 — AED PROJECT TEAM ASSEMBLY LEVEL TOLERANCING

M. Orthman, Jr., *Chair*, Honda Aircraft Co., Inc.
M. P. Wright, *Vice Chair*, Lockheed Martin Aeronautics Co.
N. Guerra, GEIQ
X. Guo Deng, Bendix Commercial Vehicle Systems, LLC
M. W. Holcumbrink, Raytheon
H. Kramer, Consultant
J. Michalowicz, Stryker Corp.
S. Neumann, Geotol, Inc.
D. Newman, Collins Aerospace
B. Nielsen, Boeing
W. Vermillion, Boeing
S. Villyard, Lockheed Martin Space Systems Co.
W. Weiss, IBW — Ingenieurbuero Wolfgang Weiss
R. A. Wheeler, RW-OptDesign
E. F. Zwettler, Sigmetrix, LLC
B. A. Davis, *Contributing Member*, Ford Motor Co.
D. Klaege, *Contributing Member*, Consultant
R. Verma, *Contributing Member*, Spirit Aerosystems

CORRESPONDENCE WITH THE AED COMMITTEE

General. ASME codes and standards are developed and maintained by committees with the intent to represent the consensus of concerned interests. Users of ASME codes and standards may correspond with the committees to propose revisions or cases, report errata, or request interpretations. Correspondence for this Standard should be sent to the staff secretary noted on the committee's web page, accessible at AED https://go.asme.org/AEDcommittee.

Revisions and Errata. The committee processes revisions to this Standard on a continuous basis 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 in the next edition of the Standard.

In addition, the committee may post errata on the committee web page. Errata become effective on the date posted. Users can register on the committee web page to receive e-mail notifications of posted errata.

This Standard is always open for comment, and the committee welcomes proposals for revisions. 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 background information and supporting documentation.

Cases

(a) The most common applications for cases are

(1) to permit early implementation of a revision based on an urgent need

(2) to provide alternative requirements

(3) to allow users to gain experience with alternative or potential additional requirements prior to incorporation directly into the Standard

(4) to permit the use of a new material or process

(b) Users are cautioned that not all jurisdictions or owners automatically accept cases. Cases are not to be considered as approving, recommending, certifying, or endorsing any proprietary or specific design, or as limiting in any way the freedom of manufacturers, constructors, or owners to choose any method of design or any form of construction that conforms to the Standard.

(c) A proposed case shall be written as a question and reply in the same format as existing cases. The proposal shall also include the following information:

(1) a statement of need and background information

(2) the urgency of the case (e.g., the case concerns a project that is underway or imminent)

(3) the Standard and the paragraph, figure, or table number(s)

(4) the edition(s) of the Standard to which the proposed case applies

(d) A case is effective for use when the public review process has been completed and it is approved by the cognizant supervisory board. Approved cases are posted on the committee web page.

Interpretations. The committee does not issue interpretations for this Standard.

Committee Meetings. The AED Standards Committee regularly holds meetings that are open to the public. Persons wishing to attend any meeting should contact the secretary of the committee. Information on future committee meetings can be found on the committee web page at AED https://go.asme.org/AEDcommittee.

INTENTIONALLY LEFT BLANK

AEROSPACE AND ADVANCED ENGINEERING PRODUCT DEFINITION

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 word "shall" establishes a mandatory requirement.
(b) The word "will" establishes a declaration of purpose on the part of the design activity.
(c) The word "should" establishes a recommended practice.
(d) The word "may" establishes an allowed practice.
(e) The words "typical," "example," "for reference," and the Latin abbreviation "e.g." indicate suggestions given for guidance only.
(f) The word "or" used in conjunction with a requirement or a recommended practice indicates that there are two or more options for complying with the stated requirement or practice.
(g) The phrase "unless otherwise specified" or the abbreviation "UOS" shall be 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.

1.2.2 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 edition 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 edition 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 section 2 and referenced in the text of this Standard:
(a) When a referenced standard is cited in the text with no limitations to a specific subject or paragraphs of the standard, the entire standard is invoked. For example, "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 paragraphs within the standard are invoked.
(b) When a referenced standard is cited in the text with limitations to a specific subject or paragraphs of the standard, only the paragraphs on that subject are invoked. For example, "Assign part or identifying numbers in accordance with ASME Y14.100" is invoking only the paragraphs 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 referenced standard is cited in the text without an invoking statement such as "in accordance with," the standard is for guidance only. For example, "For gaging principles, see ASME Y14.43" is for guidance only, and no portion of the standard is invoked.

1.2.4 Definitions. Section 3 provides definitions specific to this Standard. For definitions of words used in but not defined in this Standard, see Merriam-Webster's Unabridged Dictionary at https://www.merriam-webster.com/.

1.2.5 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.

1.2.6 Notes. Notes depicted in this Standard in ALL UPPERCASE letters are intended to reflect actual production definition 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 the product definition. 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 note is allowed to vary to suit the application.

1.2.7 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.

1.2.8 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.

1.2.9 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 figures 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 paragraphs 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 paragraph reference is not a reason to assume inapplicability. When the letter "h" is used in figures for letter heights or for symbol proportions, select the applicable letter height in accordance with ASME Y14.2.

1.2.10 Precedence of Standards. The following are ASME Y14 standards that are basic engineering product definition standards:

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

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.

1.3 Reference to This Standard

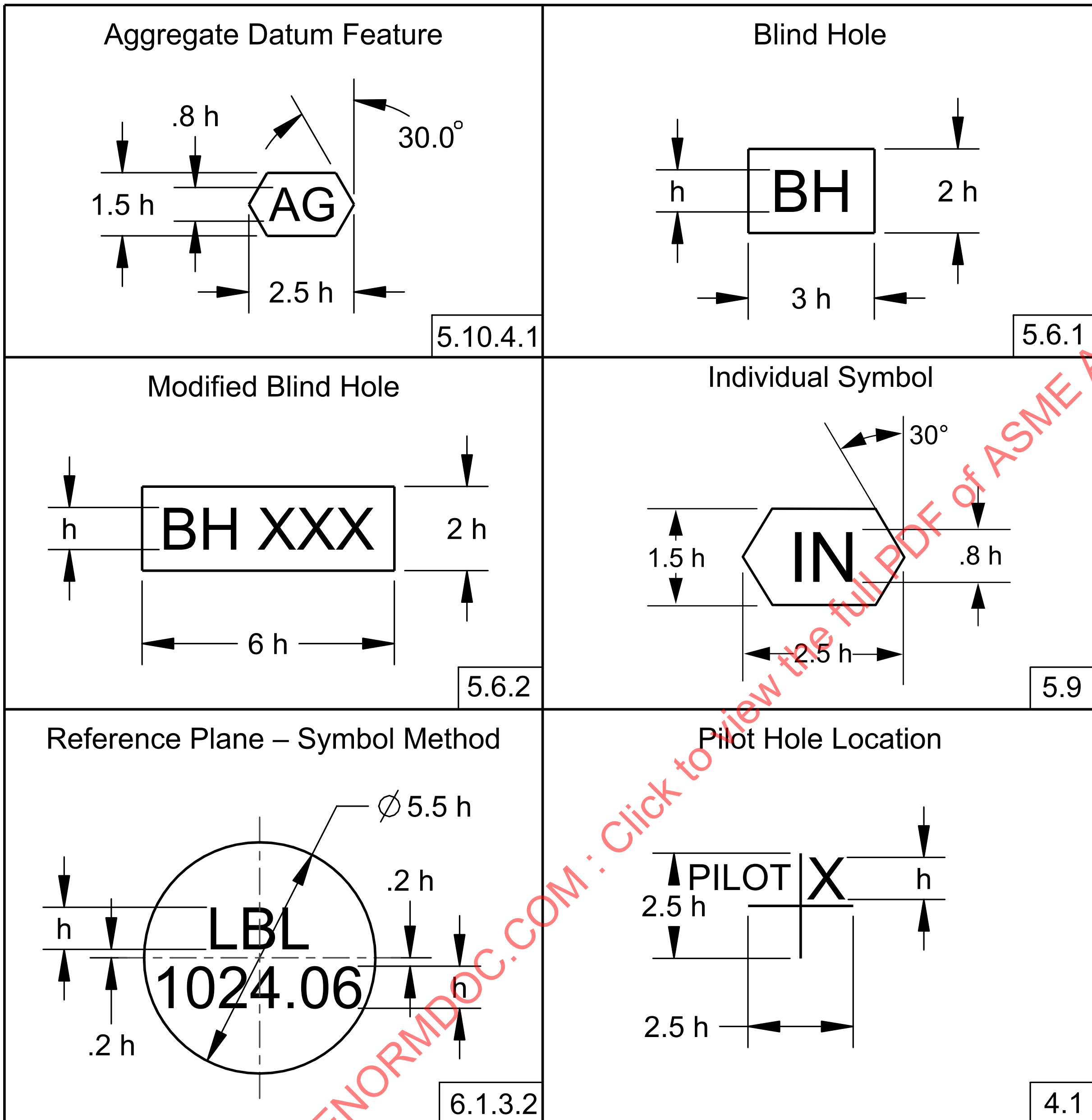
When drawings or data sets are based on this Standard, this fact shall be noted on the drawing or in the data set. A note similar to the following shall be added: "THIS DRAWING SHALL BE INTERPRETED IN ACCORDANCE WITH ASME Y14.100-2017 EXCEPT AS AMENDED BY ASME AED-1–2023."

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.

Figure 1-1
Form and Proportions of Symbols



ASME Y14.2-2014. Line Conventions and Lettering. The American Society of Mechanical Engineers.
ASME Y14.5-2009. Dimensioning and Tolerancing. The American Society of Mechanical Engineers.
ASME Y14.100-2017. Engineering Drawing Practices. The American Society of Mechanical Engineers.

3 DEFINITIONS

3.1 Blend

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

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.

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

Figure 3-1
Corner

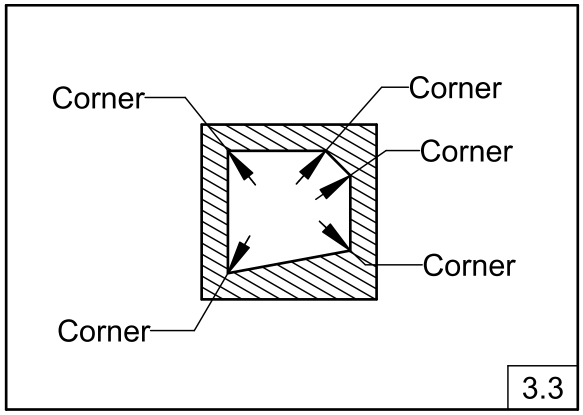
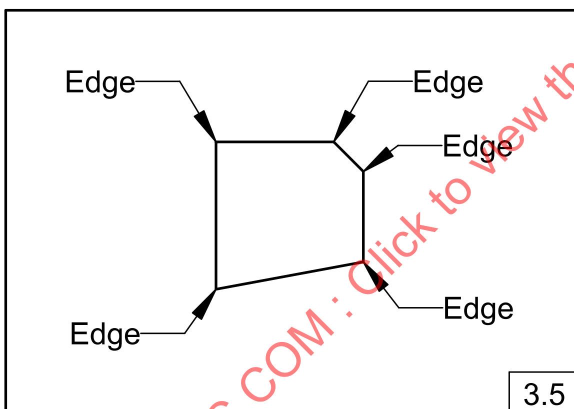


Figure 3-2
Edge



4 ENGINEERING PRODUCT DEFINITION PRACTICES

4.1 Pilot Hole Location Symbols

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

(a) The pilot hole location symbol includes a cross that indicates the location of the hole. The notation "PILOT" shall be placed in the upper-left quadrant of the symbol, and the hole size required for the final fastener shall be in the upper-right quadrant of the symbol. The hole size is a reference dimension. The other two quadrants shall remain blank.

(b) The pilot hole location symbol may be used to indicate pilot hole locations shown on an engineering drawing, such as detail, assembly, or installation drawings.

4.2 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. Either the allowable transition or the disallowed abrupt ending shall be defined on the drawing (see Figure 4-2).

4.3 Dynamic Profile Tolerance Modifier

Where it is desirable to refine the form independent of 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 ASME Y14.5-2018, para. 11.10 for rules and methodology regarding the application and interpretation of the dynamic profile tolerance modifier.

Figure 4-1
Pilot Hole Location Specification

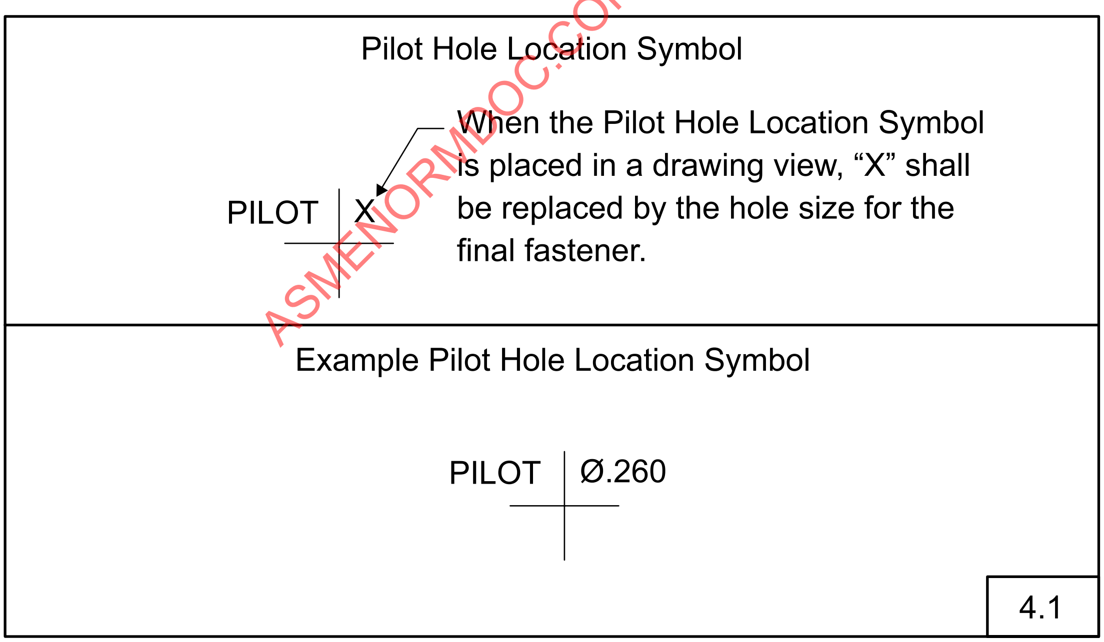
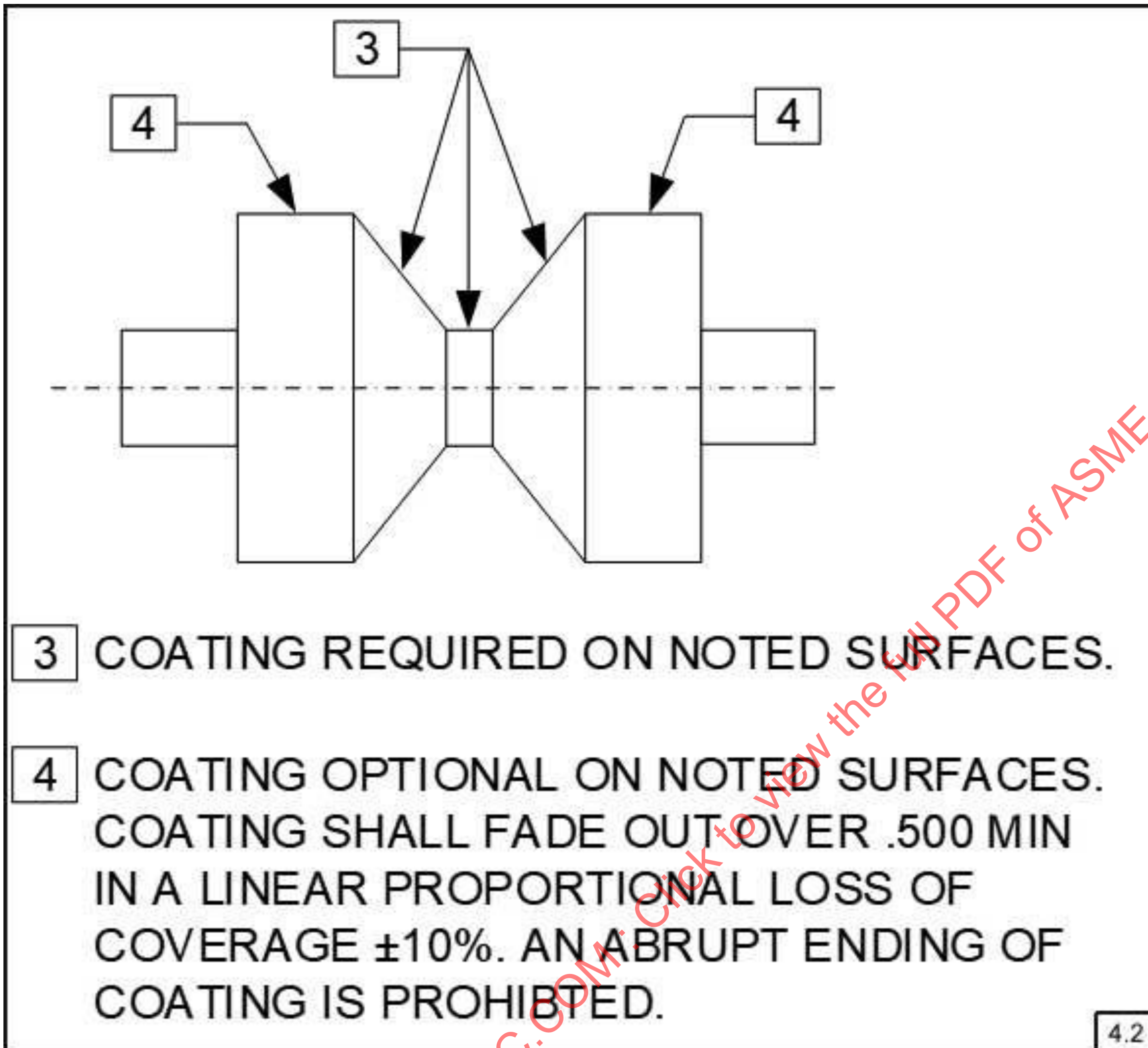


Figure 4-2
Optional Coverage



5 DIMENSIONING AND TOLERANCING PRACTICES

5.1 Orientation and Direction of Dimensional Requirements

The terms "AXIAL," "ANGULAR," or "RADIAL" may be used to describe a direction in which a dimensional requirement (e.g., a tolerance) is applied.

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

(b) The term "ANGULAR" indicates that the dimensional requirement applies in a rotational direction from a plane passing through the 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.

See Figure 5-1 for an example of the use of the terms "AXIAL," "ANGULAR," and "RADIAL."

5.2 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 5-2. When the opposite, unshown surface is unclear, that surface shall be identified in another view and be considered the dimension origin UOS.

5.3 True Dimensioning on Nonplanar Surfaces

The notation "TRUE" may be used to locate features or items on nonplanar surfaces (see Figure 5-3).

The notation "TRUE" shall be placed following a dimension, below a dimension, or in a note indicating that it applies to multiple dimensions.

(a) "TRUE" associated with a dimension means the dimension applies along the theoretically exact nonplanar surface, not in the viewing plane. Where it is unclear which path the true dimension should take, it shall be specified.

(b) When applied in a view that does not show the surface in its true shape, "TRUE" associated with a dimension indicates the requirement applies along the near (visible) surface of the part in the view.

(c) 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 5-4).

5.4 Radius

Radius tolerance and controlled radius tolerance of ASME Y14.5 are amended as described in paras. 5.4.1 through 5.4.4.

5.4.1 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 5-5). 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 the minimum radii tangent to the adjacent surfaces (see Figure 5-6).

5.4.2 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 5-7). 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.

5.4.3 Tangent Radii. The ends of a radius when shown tangent shall blend smoothly and terminate within 5 deg of tangency (see Figures 5-8 and 5-9). 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 5-6). 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.

5.4.4 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 5-8 for an external tangency radius or Figure 5-9 for an internal tangency radius.

5.5 Noncircular Fillets

Noncircular fillets are a concave easing of a corner or convex easing of an edge of a part typically used to reduce stress concentration. This provides greater strength characteristics than a fillet radius in a direction parallel or nearly parallel to the axis of symmetry, for parabolic fillets, or the major axis, for elliptical fillets. General applications for noncircular 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.

Similar to a fillet radius, the included angle of the adjacent surfaces of a noncircular fillet shall be dimensioned using the product definition. The dimensions of a noncircular fillet shall be expressed as a basic dimension. Tolerances for noncircular fillets shall be specified in the form of a profile tolerance in accordance with ASME Y14.5. The preferred noncircular 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 noncircular fillet.

5.5.1 Parabolic Fillets. The cross section of a parabolic fillet is a parabola instead of a constant arc.

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 5-10).

A parabolic fillet is expressed using three terms in the following order: length of long leg parallel to the surface adjacent to the long leg, length of short leg perpendicular to the surface adjacent to the long leg, and angle between the surface adjacent to the long leg and the parabola's axis of symmetry.

(a) 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.

(b) 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 5-11).

(c) 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 5-11), for example

(1) 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

(2) 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

5.5.2 Elliptical Fillets. The cross section of an elliptical fillet is an ellipse instead of a constant arc.

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 5-12).

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

(a) 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.

(b) 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 5-12).

(c) 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 5-12), for example

(1) 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

(2) 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

5.6 Blind Holes

5.6.1 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 5-13 for a blind hole or Figure 5-14 for a blind hole with a through hole.

5.6.2 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 5-15 for a 135-deg drill point callout example.

5.7 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 5-16 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.

5.8 Projected Tolerance Zone Direction Indicators

5.8.1 Projected Tolerance Zone Direction Specification. As an alternate practice to indicate the direction of a projected tolerance zone as defined in ASME Y14.5, the symbol + or −, or both, may be used. The + or − symbol shall be placed in the feature control frame following the projected tolerance symbol. The value of the projected tolerance zone length shall follow the + or − symbol as appropriate. The + symbol specifies a tolerance zone that projects from the near-side surface, the − symbol specifies a tolerance zone that projects from the far-side surface, and the + and − symbols used together specify a tolerance zone that projects from both the near-side surface and the far-side surface. When both a near-side tolerance zone and a far-side tolerance zone are specified, the values shall be separated by a slash. See Figures 5-17 through 5-21.

The near-side surface is

(a) the surface to which a solid line style leader from a feature control frame, or size dimension associated with a feature control frame, is attached in an edge view

(b) the visible surface in a direct view, or

(c) the hidden surface when a hidden line style leader is used in a direct view

The far-side surface is the surface opposite of the near-side surface that bounds the considered feature.

5.8.2 Tolerance Zone Definition With Projected Tolerance Zone Indicators. A tolerance zone specified with the + or − symbol shall use the axis interpretation per ASME Y14.5. A tolerance zone projected in only one direction, with either the + or the − symbol, shall be defined as specified in ASME Y14.5, with the tolerance zone length applying from the near-side or far-side surface to the projected length outside the part (see Figures 5-17 and 5-18). A tolerance zone projected in both directions for the same feature using the + and − symbols specifies a tolerance zone of the minimum projected length from the near-side surface, through the part, to the minimum projected length from the far-side surface (see Figures 5-19 and 5-20). A tolerance zone projected from only one surface and including the length of the considered feature shall have a 0 value specified for the projection length from the opposite surface (see Figure 5-21).

5.9 Individual Symbol

As an alternate practice for the specification of repetitive datums, as defined in ASME Y14.5, a repetitive feature used as a datum feature may be indicated using the individual symbol. The individual symbol is shown in Figure 1-1. The symbol shall be placed inside the feature control frame of the considered feature following the datum reference and any material boundary modifiers. The individual symbol indicates that the datum feature is being referenced on an individual basis to establish multiple, individual datum reference frames (see Figure 5-22).

When a datum feature is referenced with and without the individual symbol within the same data set, the reference without the individual symbol establishes a datum from the datum feature pattern and not on an individual basis. In Figure 5-22, the reference to datum feature D without the individual symbol in the feature control frame for the pattern of two holes establishes a single datum plane across both planar surfaces. In instances where the repetitive feature to be used on an individual basis is unclear, additional notation or datum feature identification may be required.

5.10 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. 5.8.1.

(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. 5.8.2.

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.

5.10.1 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 5-23, 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.

5.10.2 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.

5.10.3 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.

5.10.3.1 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.

5.10.3.2 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. 5.10.3.2.1 and 5.10.3.2.2.

5.10.3.2.1 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 5-24). 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.

(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 5-25).

5.10.3.2.2 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.

5.10.4 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. 5.10.4.1 and specified in a feature control frame per ASME Y14.5. An aggregate datum with inherent motion shall be identified per para. 5.10.4.1 and specified in a feature control frame per para. 5.10.4.2. 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.

5.10.4.1 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 5-26).

5.10.4.2 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 5-26).

Figure 5-1
Orientation and Direction of Dimensional Requirements

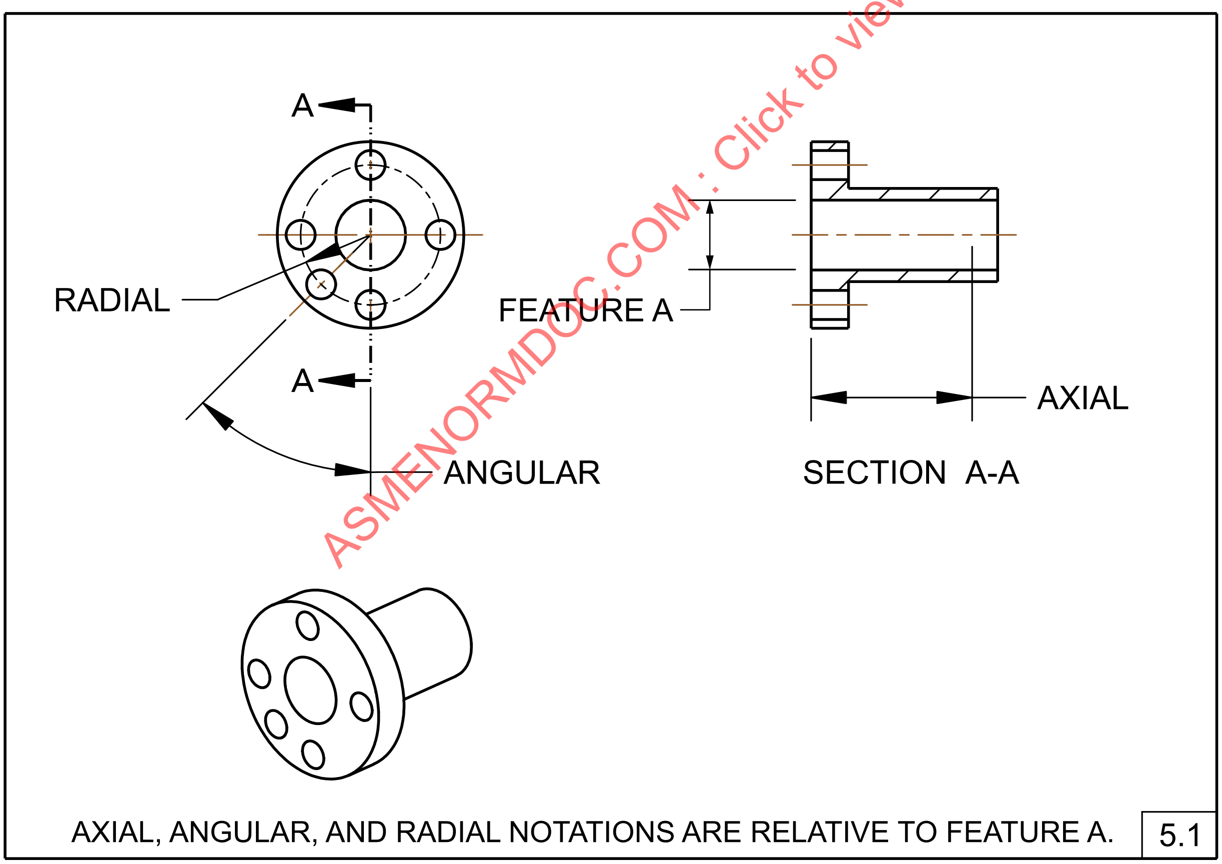


Figure 5-2
Single Dimension Line or Double-Arrowhead Methods

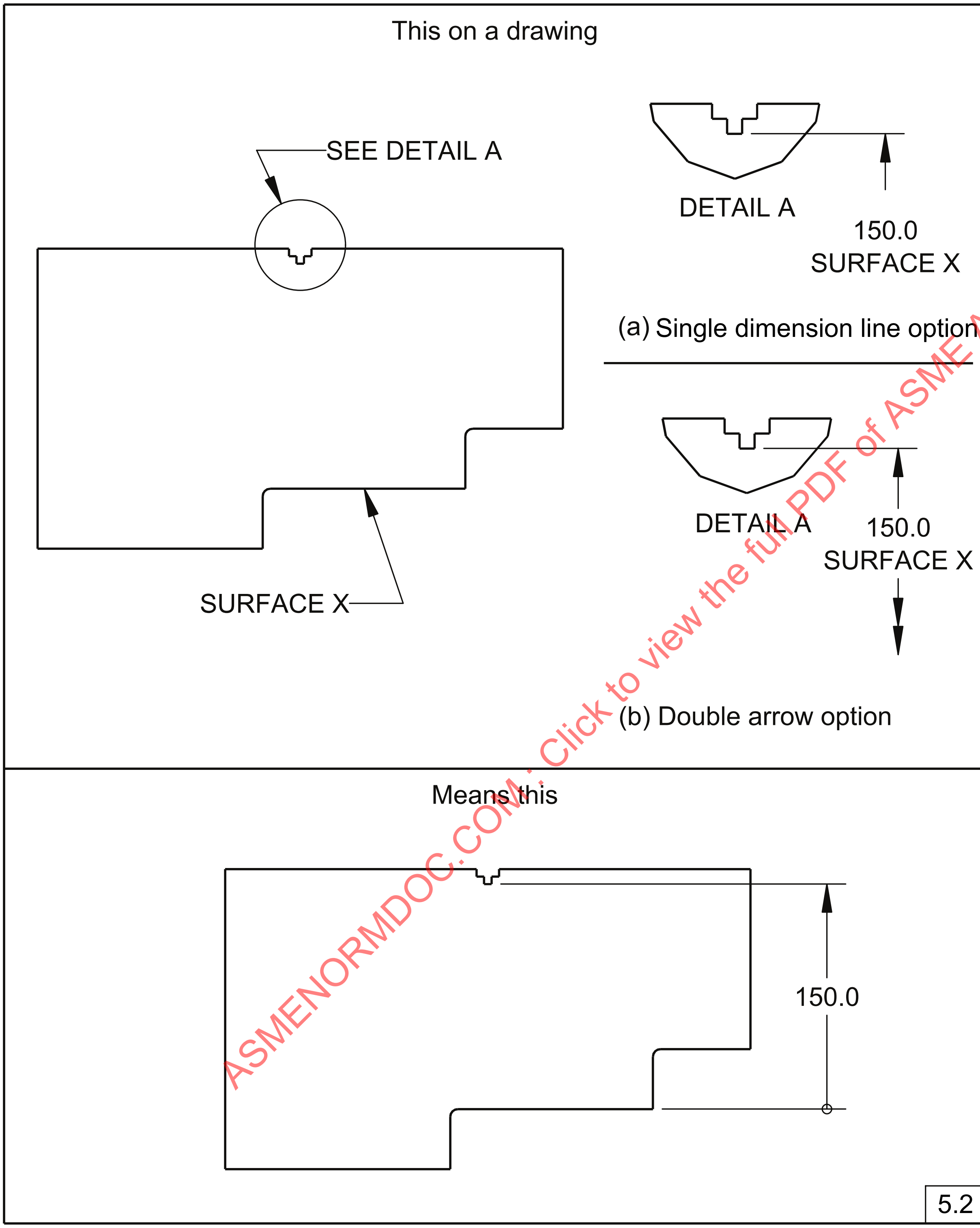


Figure 5-3
Views With True Dimensions

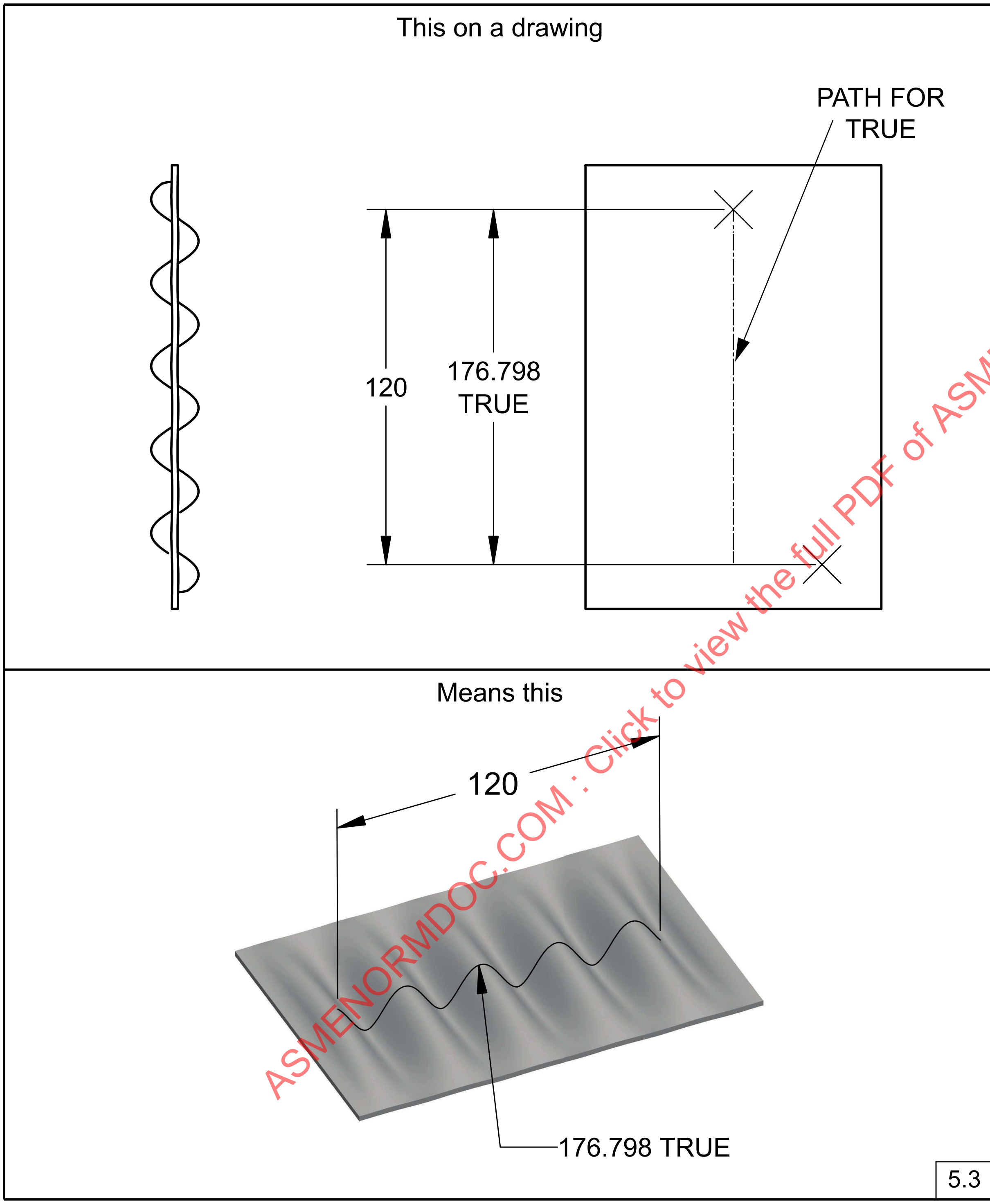


Figure 5-4
Views With All Dimensions True

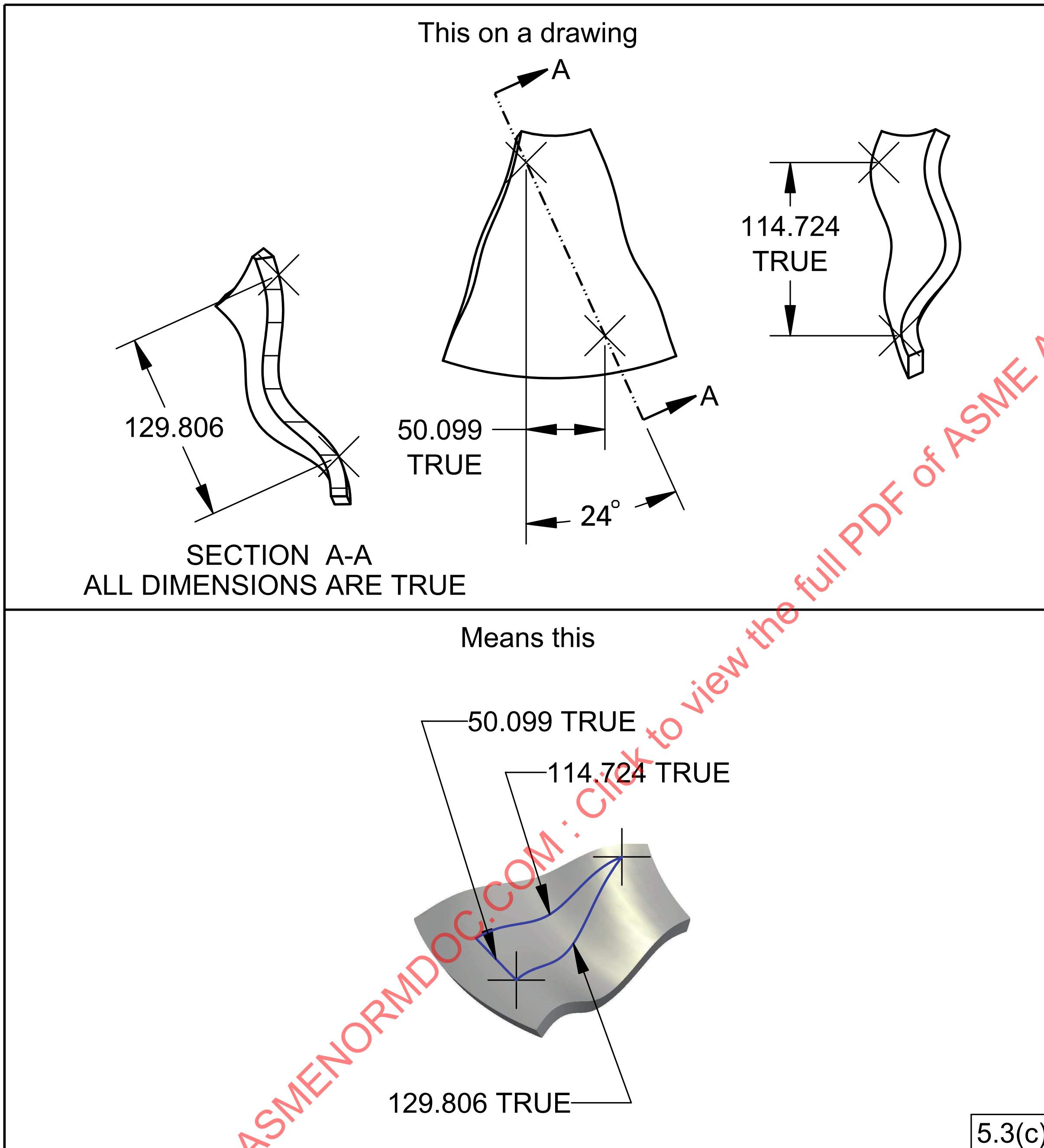


Figure 5-5
Center-Located Radius

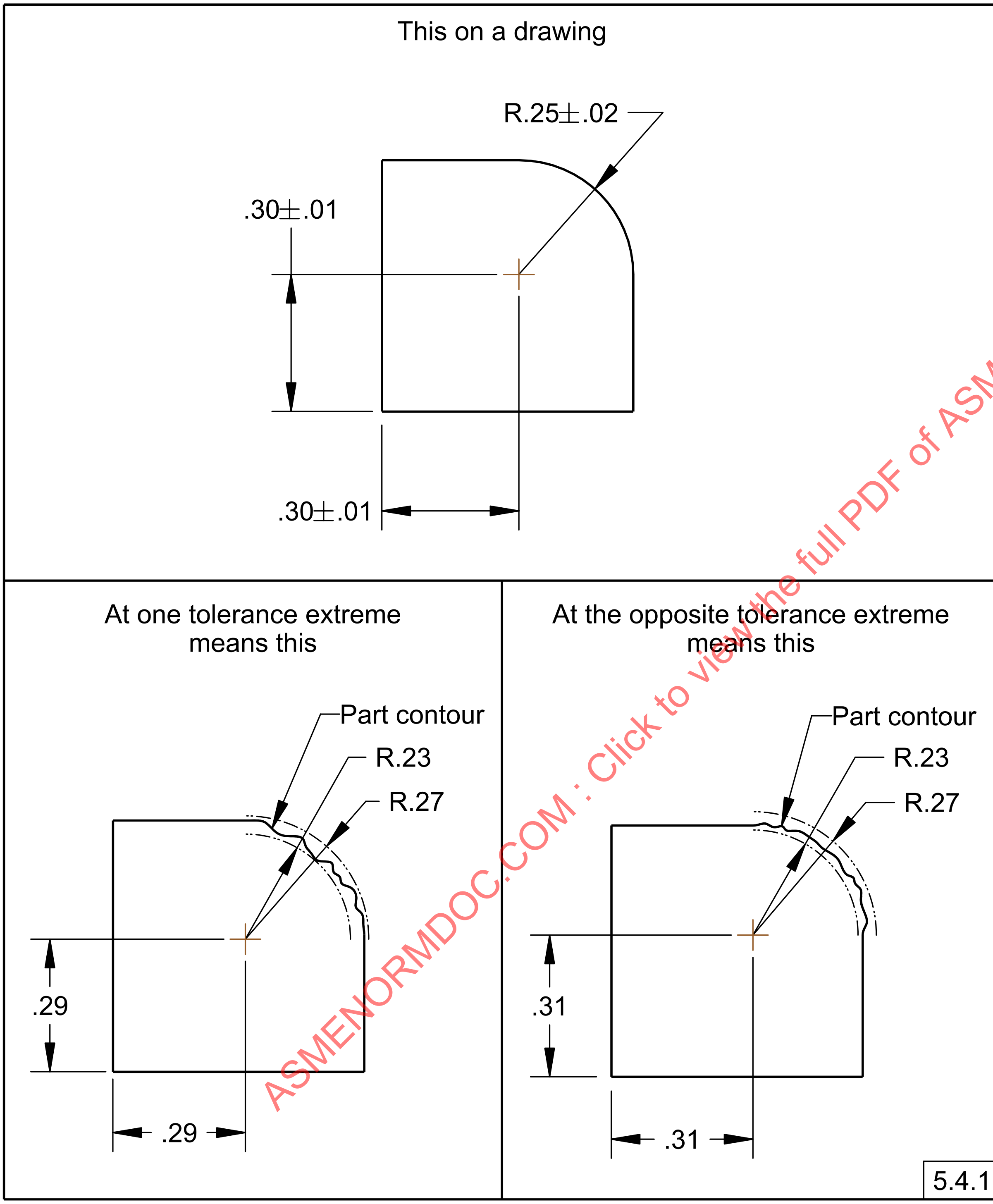


Figure 5-6
Tangent-Located Radius

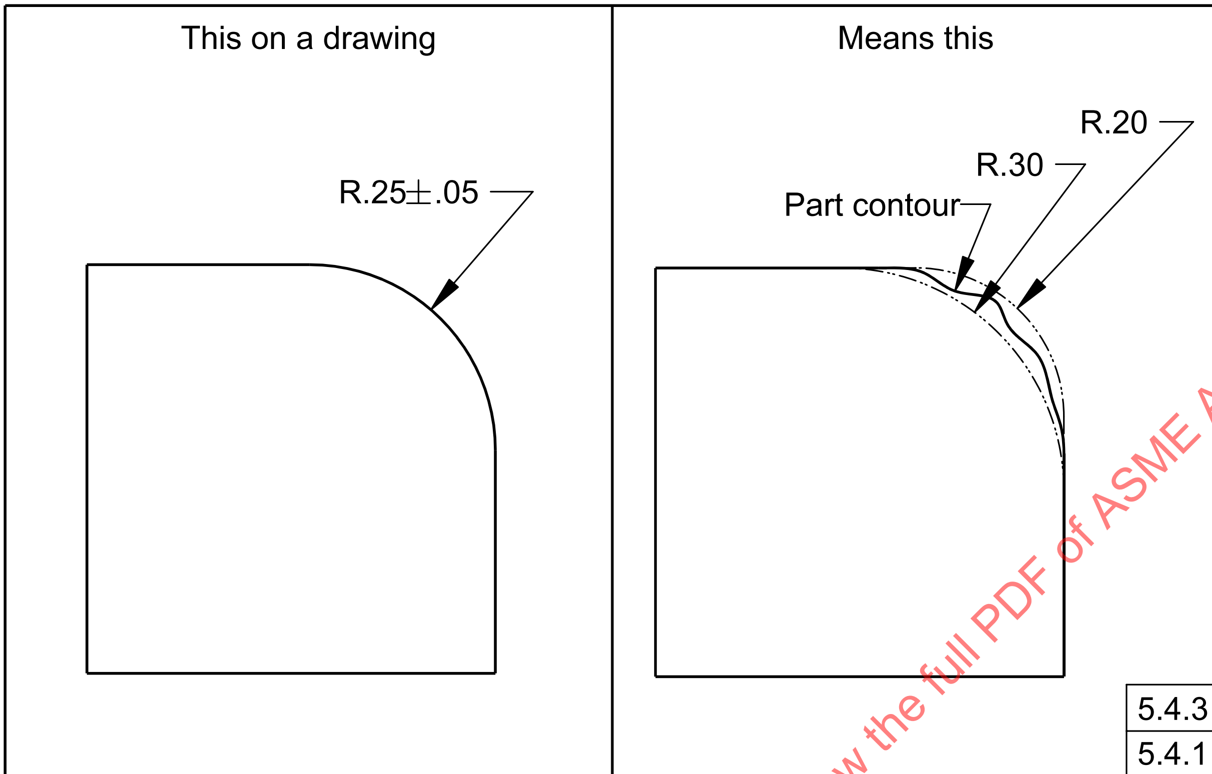


Figure 5-7
Controlled Radius

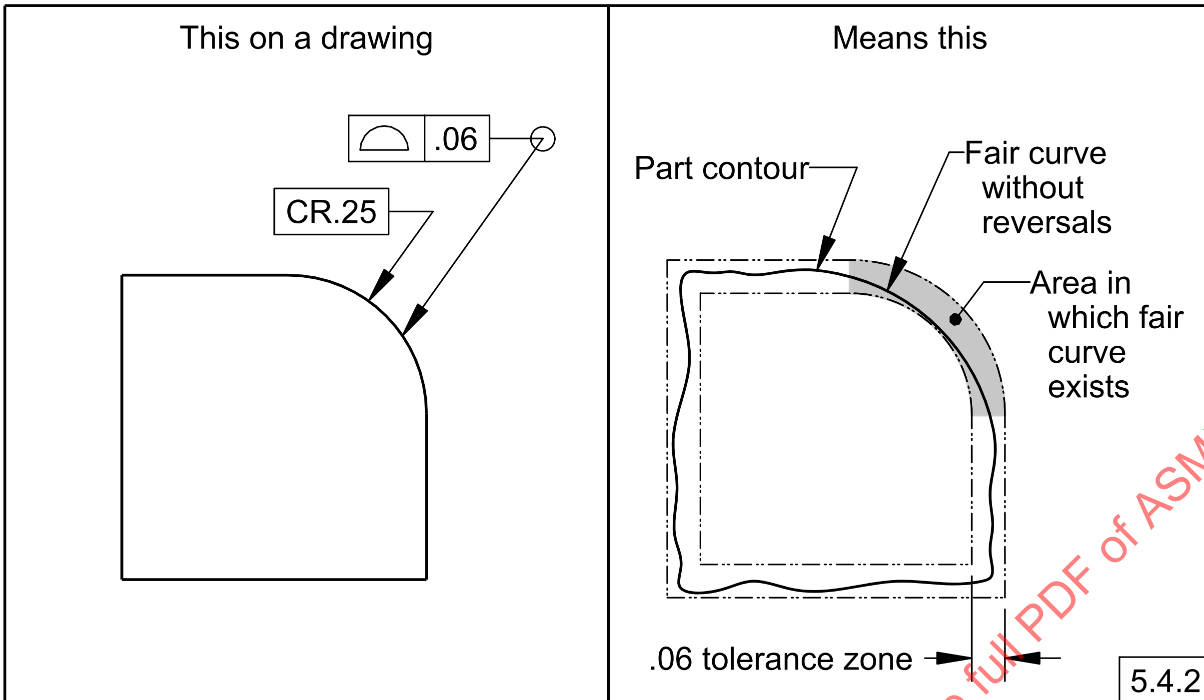


Figure 5-8
External Radius — Permissible Termination

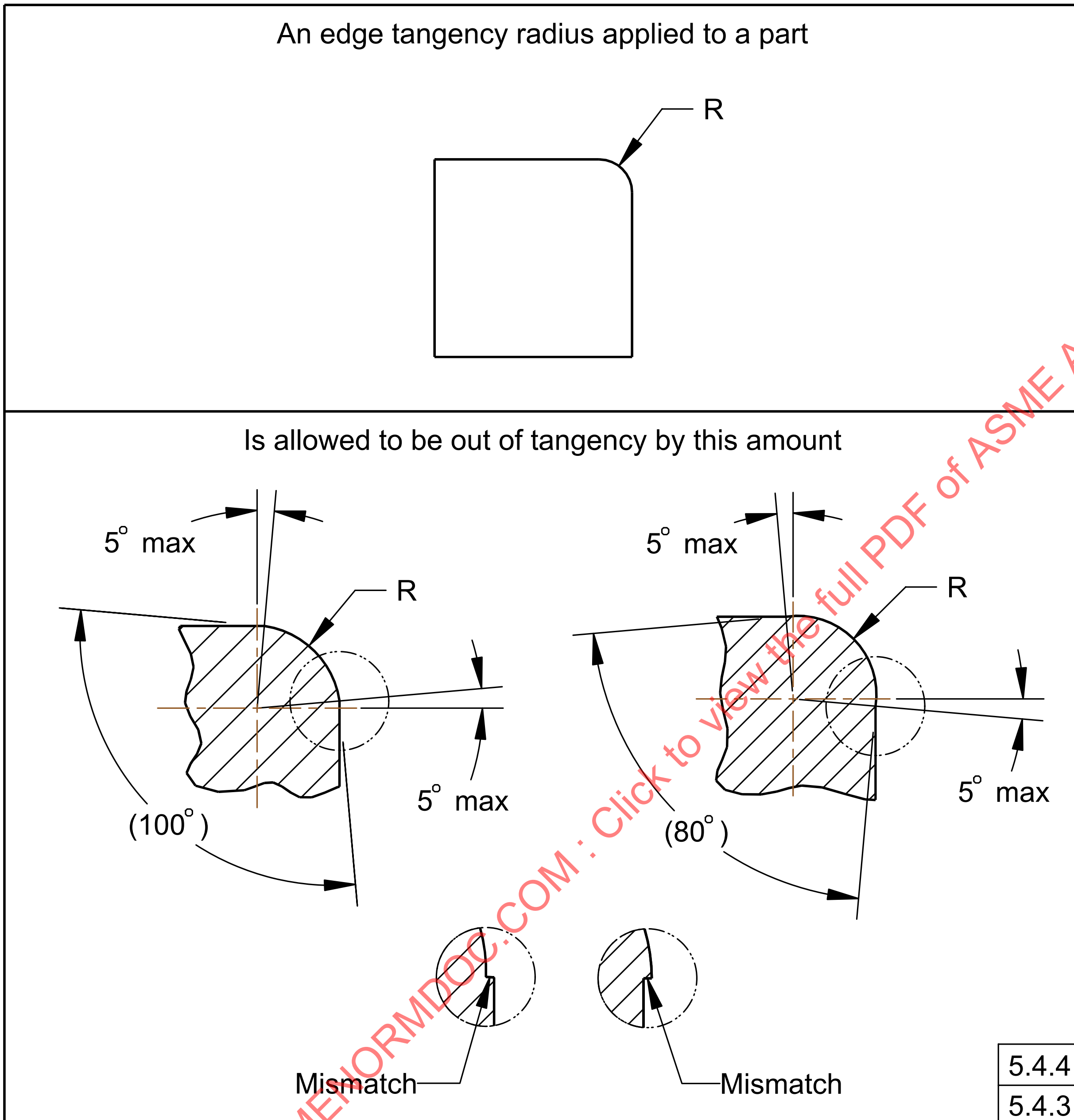


Figure 5-9
Internal Radius — Permissible Termination

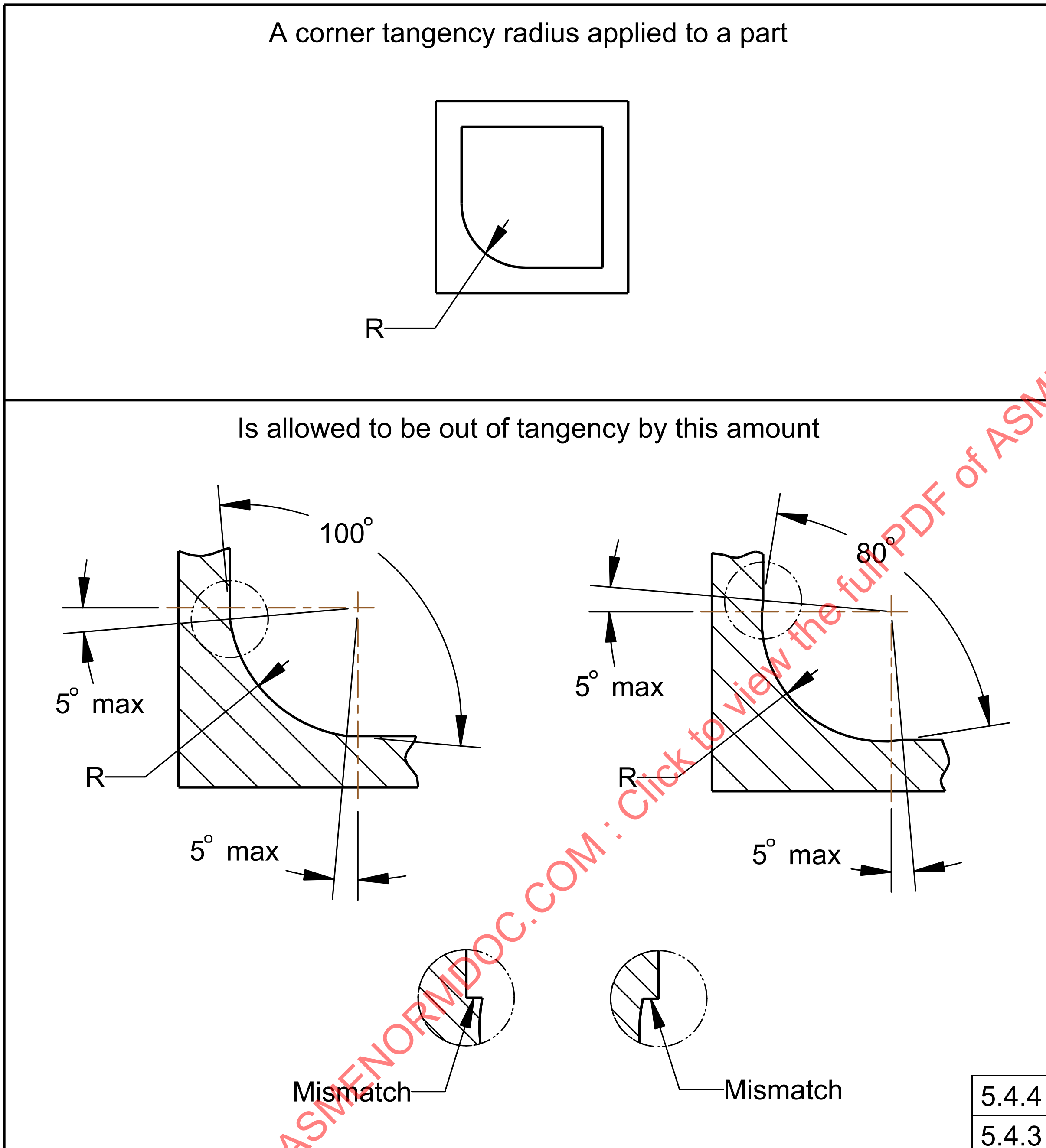


Figure 5-10
Parabola Tangency

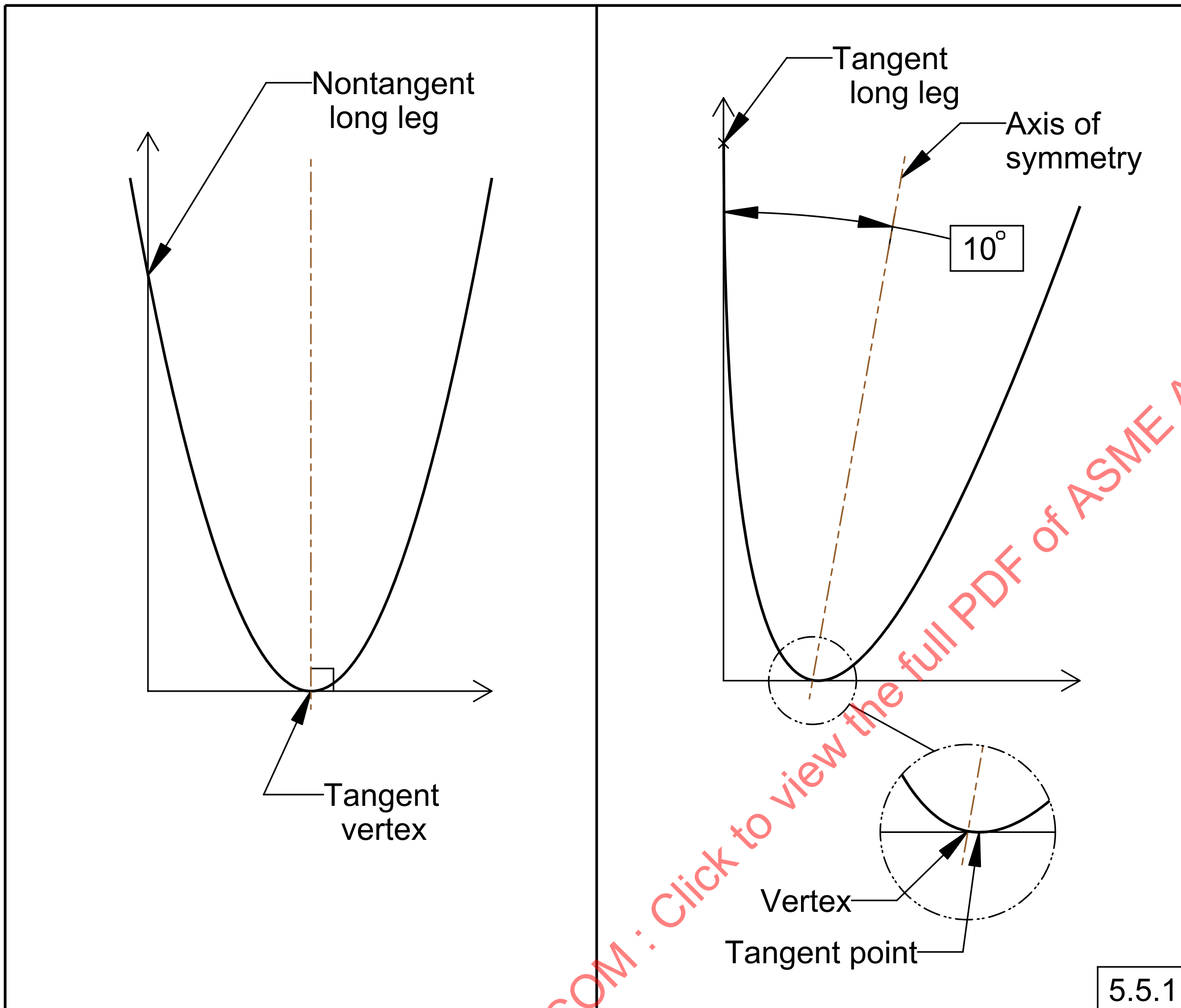


Figure 5-11
Parabolic Fillet — Example Specification

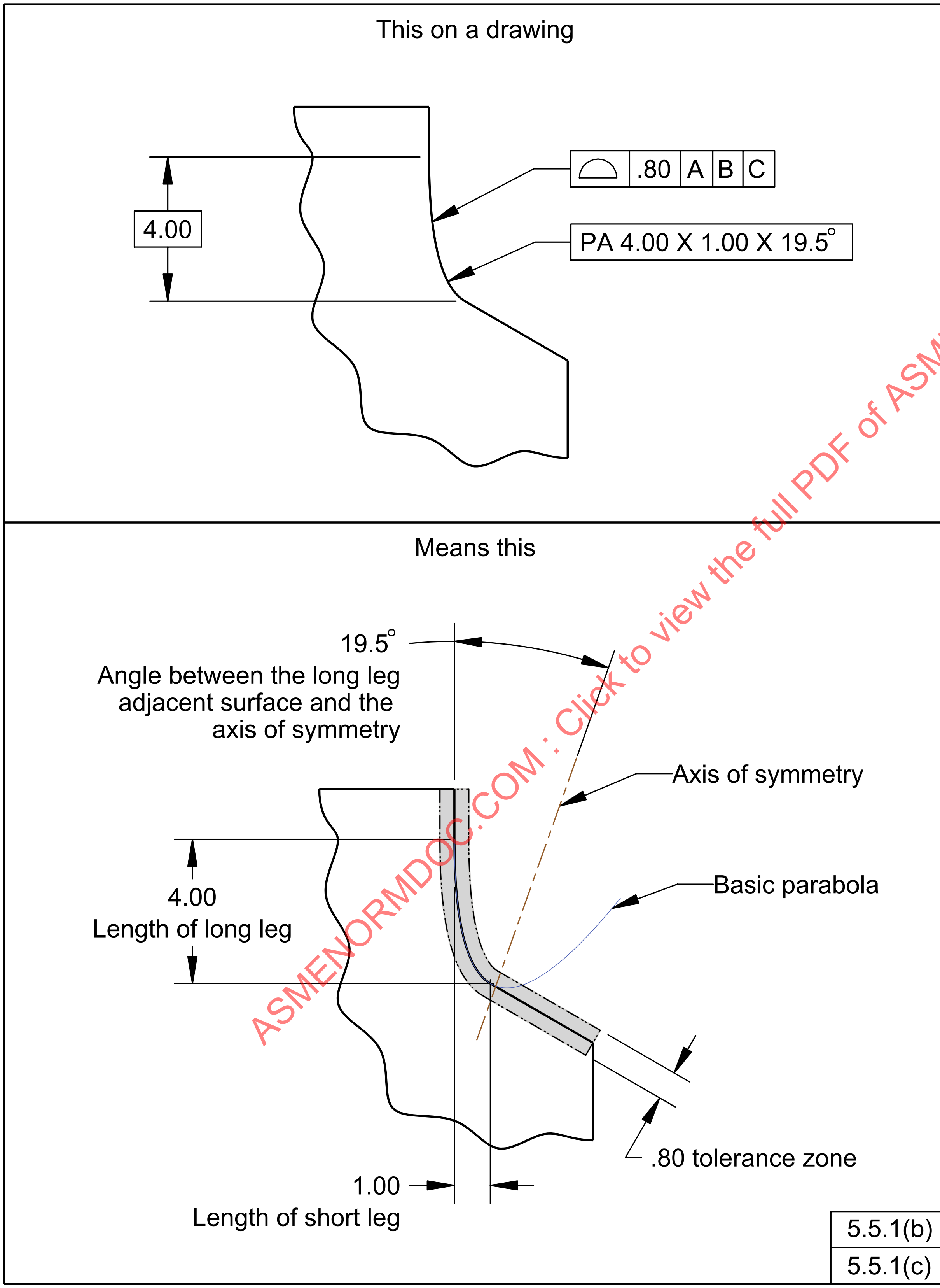


Figure 5-12
Elliptical Fillet — Example Specification

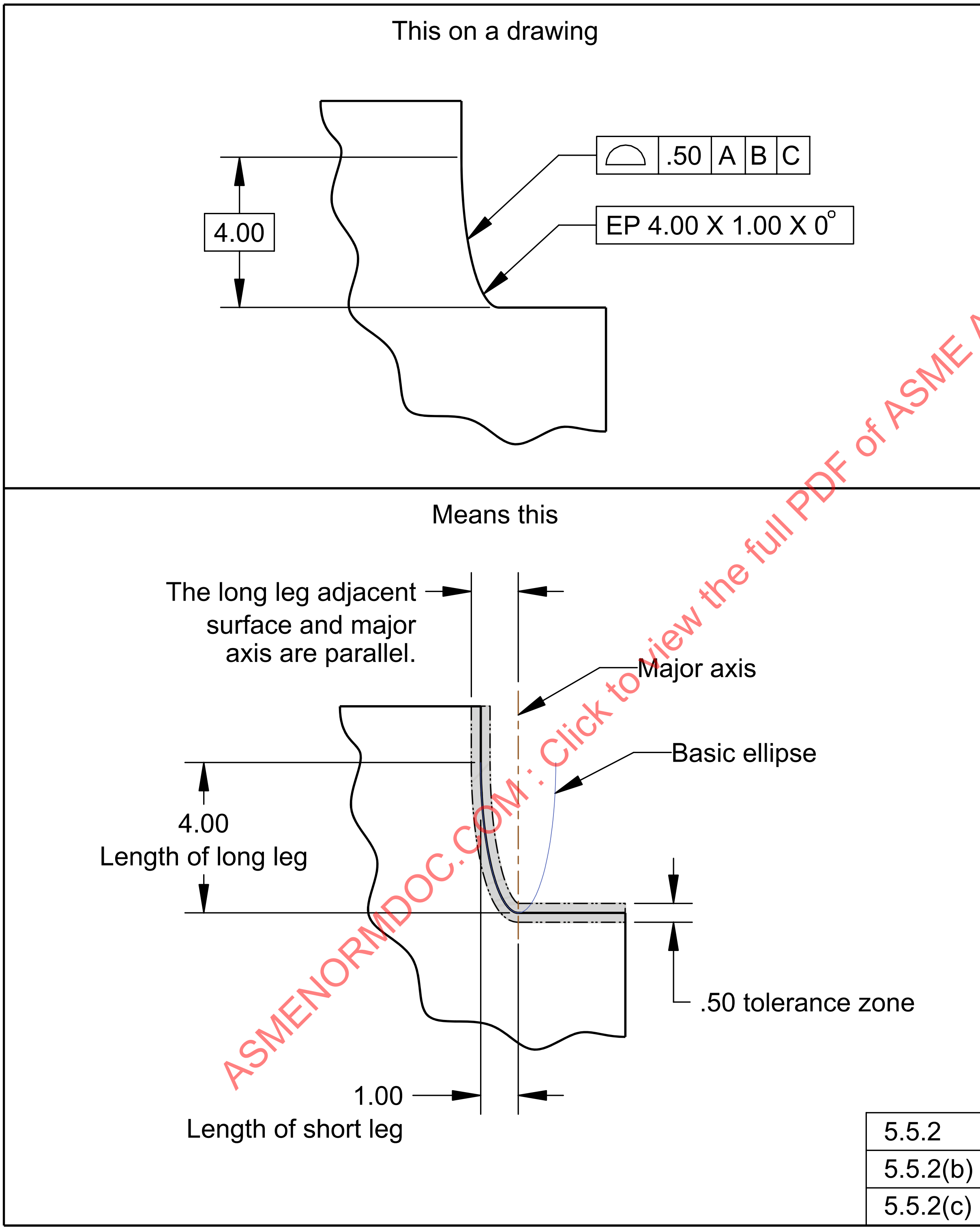


Figure 5-13
Blind Hole

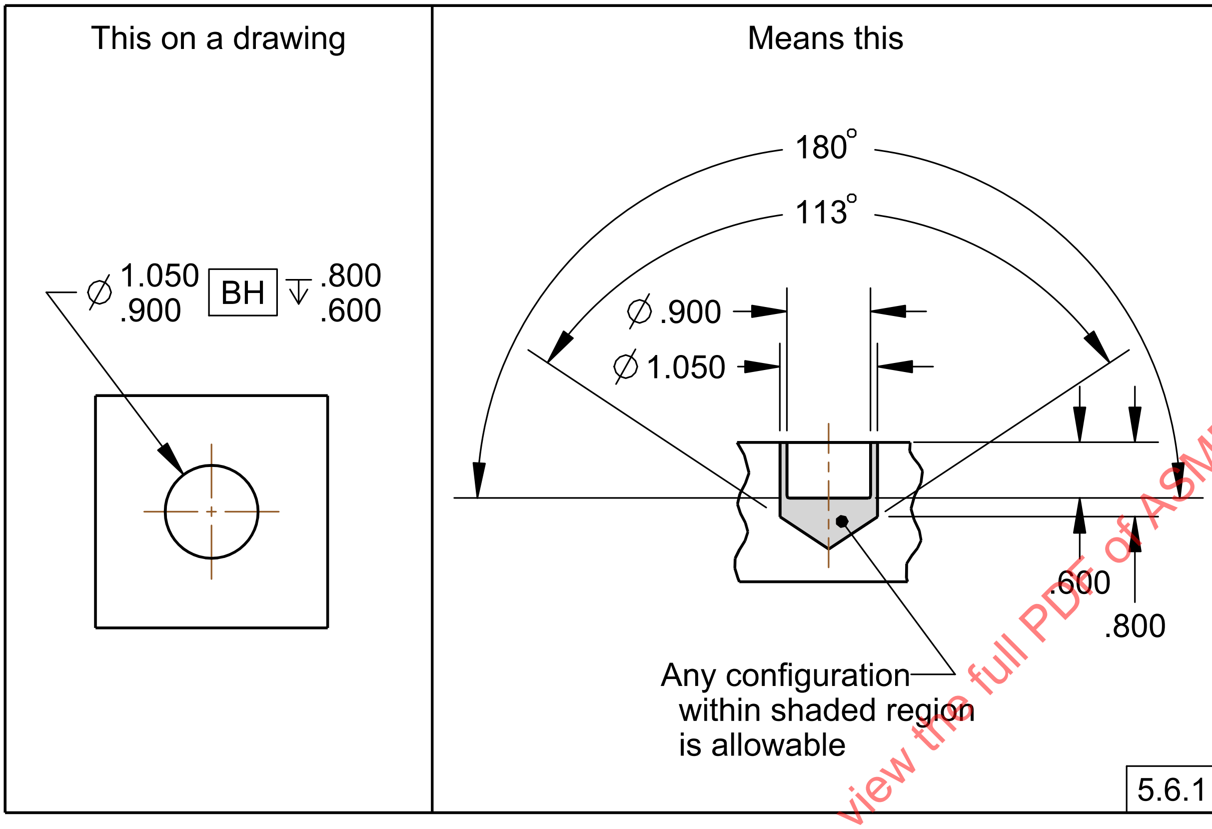


Figure 5-14
Blind Hole With a Through Hole

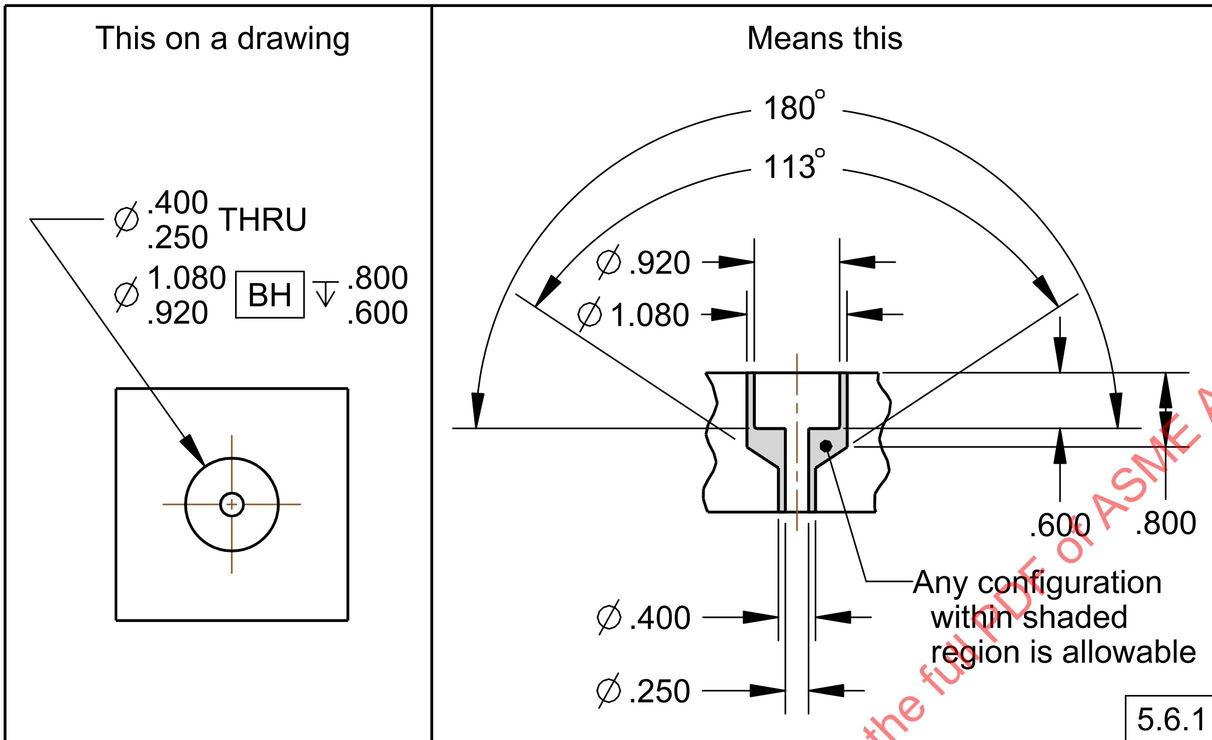


Figure 5-15
113-deg Drill Point Callout Example

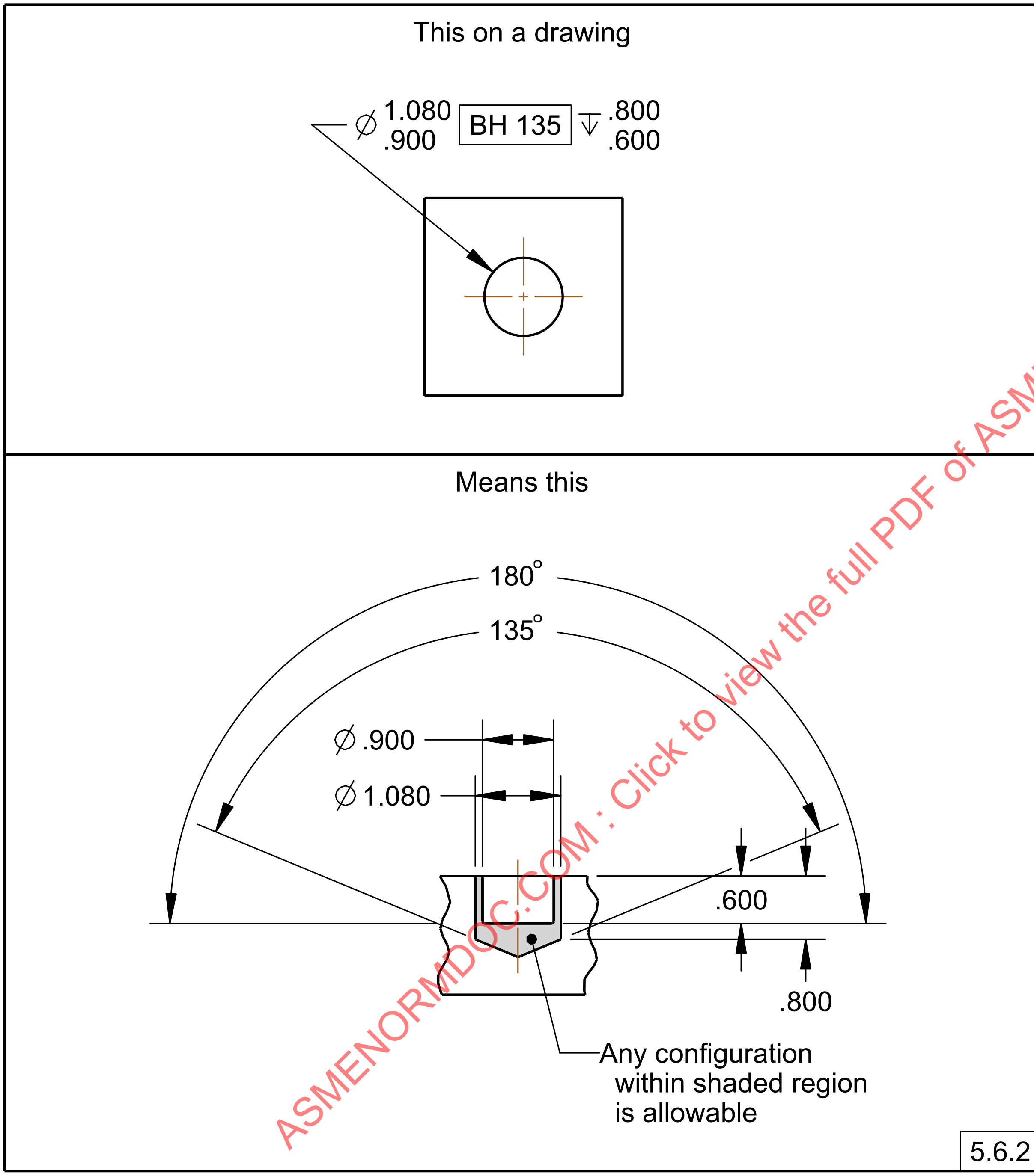


Figure 5-16
Edge Break Examples

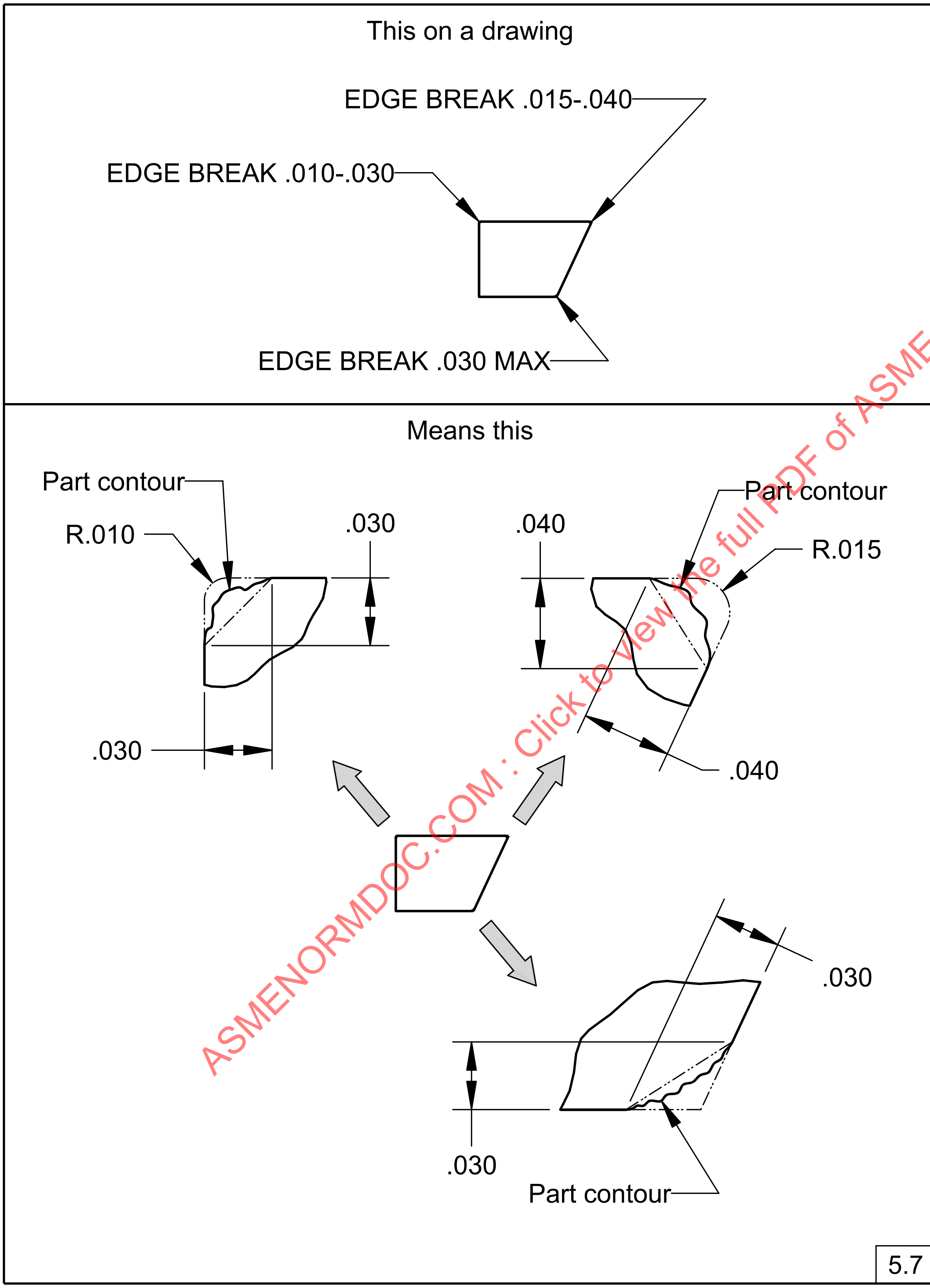


Figure 5-17
Near-Side Projected Tolerance Zone

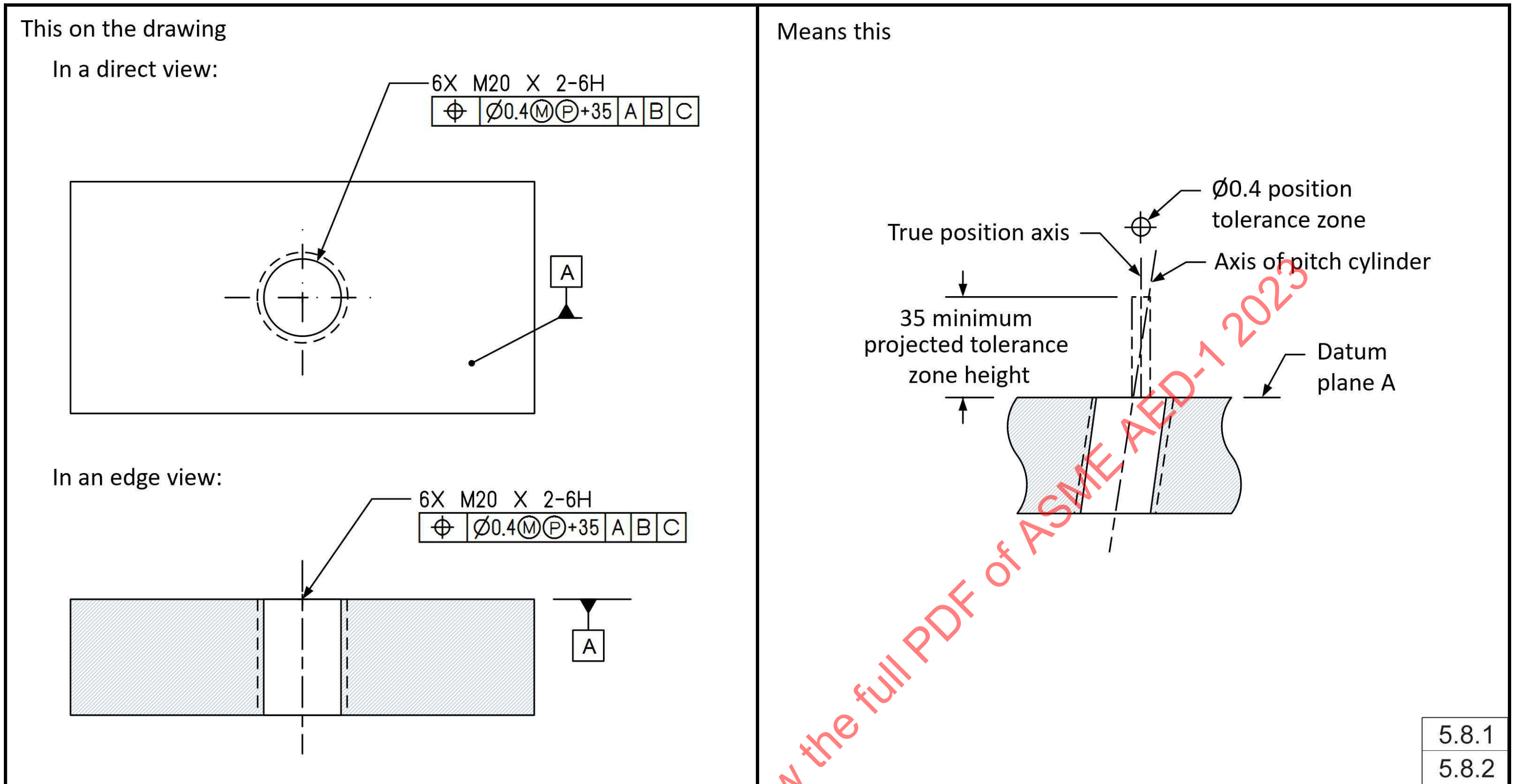


Figure 5-18
Far-Side Projected Tolerance Zone

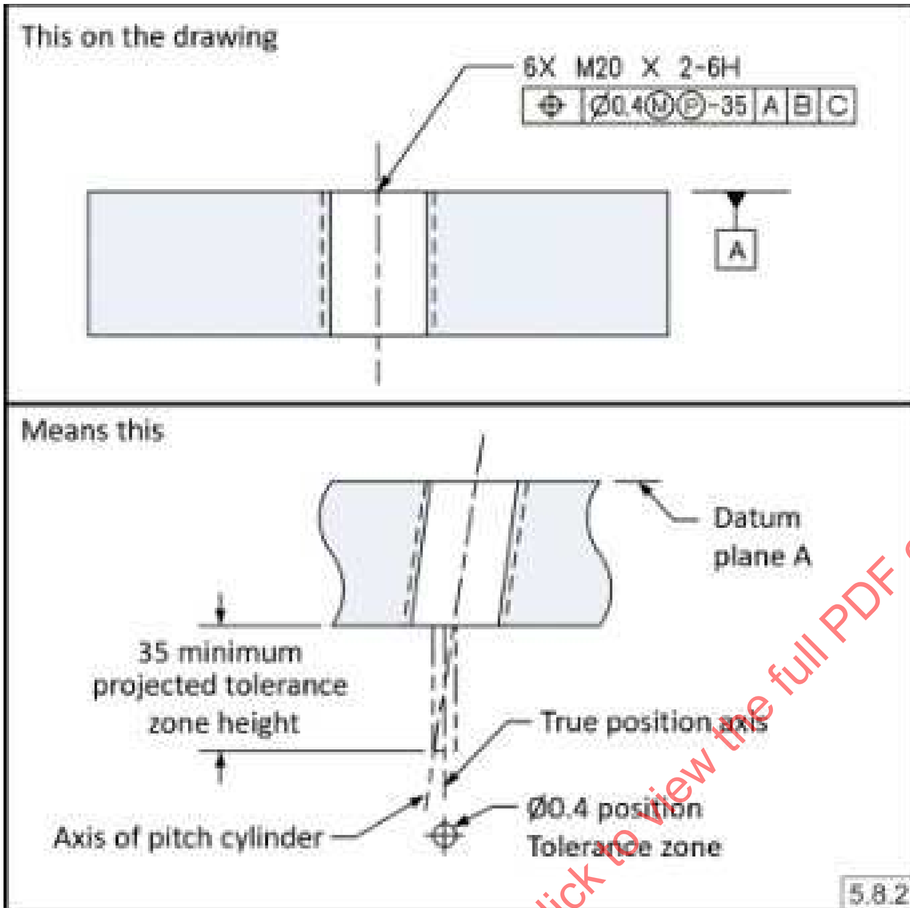


Figure 5-19
Both-Side Projected Tolerance Zone

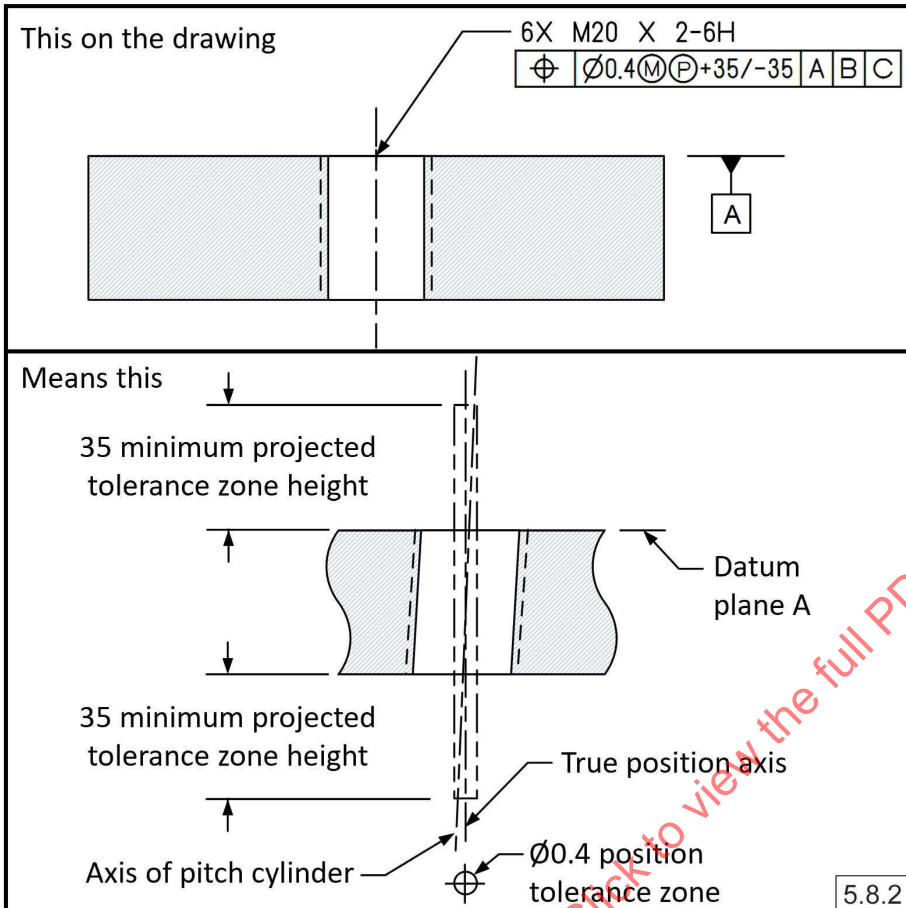


Figure 5-20
Both-Side Projected Tolerance Zone With Different Projected Lengths