ASME Y14.8-2022

[Revision of ASME Y14.8-2009 (R2014)]

Castings, Forgings, and Molded Parts

Engineering Product Definition and Related Documentation Practices

AN AMERICAN NATIONAL STANDARD

The American Society of Mechanical Engineers

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The American Society of Mechanical Engineers

Two Park Avenue • New York, NY • 10016 USA

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

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FOREWORD

This is a revision of ASME Y14.8-2009 (R2014), Castings, Forgings, and Molded Parts. Based on guidance from the ASME Y14 Committee, the material formerly in Section 1 has been reorganized into Sections 1 through 3, and the subsequent Sections have been renumbered. The scope of the Standard has expanded to include full feature concepts as a result of draft. Changes to both the text and figures have been made to better illustrate drafting practices pertaining to drawings of cast, forged, and molded parts.

Figures for plus draft, minus draft, and draft included have been improved. A new symbol for full feature has been created, and figures have been added to show application of the full feature symbol. Customized datum references are shown to demonstrate control of specific degrees of freedom due to process variations, such as mismatch and die closure. The effect of applying profile of a surface with datum references to surfaces containing datum targets is continued for the increased use of form tolerancing in model-based computer-aided design (CAD) systems. Text and figures have been revised to reflect these changes.

The successful revision of this Standard is attributed to the commitment of the committee members and the support of their sponsoring companies. Their time commitment and their contributed expertise are gratefully acknowledged. Don E. Day, former chair of the ASME Y14.8 Subcommittee, is acknowledged for his tireless leadership, commitment, and knowledge, which have made this revision possible.

This Standard was approved by the American National Standards Institute as an American National Standard on March 17, 2022.

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Engineering Product Definition and Related Documentation Practices

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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 proposing revisions or a case and attending Committee meetings. Correspondence should be addressed to:

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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.

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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 Y14 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 Y14 Standards Committee. Future Committee meeting dates and locations can be found on the Committee Page at http://go.asme.org/Y14committee.

Section 1
Scope

This Standard covers definitions of terms and features unique to casting, forging, and molded-part technologies with recommendations for their uniform specification on engineering drawings and related documents. Castings, forgings, and molded parts are delineated as "part" or "parts" throughout the Standard.

1.1 GENERAL

Unless otherwise specified, any reference to features, parts, or processes shall be interpreted as applying to castings, forgings, and molded parts. Sections 2 through 5 establish related references, definitions, drawing presentation methods, and drafted feature considerations. Datum referencing is presented in Section 6, and Section 7 provides drawing notes and drawing items. Additional information unique to castings, forgings, and moldings is located in Nonmandatory Appendices A through C. The information in Nonmandatory Appendix D is provided to assist in the interpretation of existing drawings on which practices in previous editions of ASME Y14.8 may appear.

1.2 DIMENSIONING AND TOLERANCING

The methods of dimensioning and tolerancing shall be in accordance with ASME Y14.5 and this Standard.

1.3 ASME Y14 SERIES CONVENTIONS

The conventions in paras. 1.3.1 through 1.3.12 are used in this and other ASME Y14 standards.

1.3.1 Mandatory, Recommended, Guidance, and Optional Words

(a) The word "shall" establishes a 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.

1.3.2 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 other ASME Y14 standards in the text without a date following the standard designator indicates the edition of the standard identified in the References section (Section 2) shall be used to meet the requirement.

(b) Reference to other 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.

1.3.3 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 invoked for guidance only. For example, "For gaging principles, see ASME Y14.43" is only for guidance and no portion of the standard is invoked.

1.3.4 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/.

1.3.5 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.

1.3.6 Notes

Notes depicted in this Standard in ALL UPPERCASE letters are intended to reflect actual product 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.

1.3.7 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 is used thereafter throughout the text.

1.3.8 Units

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

1.3.9 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 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. Some figures are illustrations of models in a three-dimensional (3D) environment. The absence of dimensioning and tolerancing annotations in a view may indicate that the product definition is defined in three dimensions. Dimensions that locate or orient and are not shown are considered basic and shall be queried to determine the intended requirement. 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. Multiview drawings contained within figures are third-angle projection.

1.3.10 Precedence of Standards

The following are ASME Y14 standards that are basic engineering drawing standards:

ASME Y14.1, 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.

1.3.11 Use of an ASME Y14 Case

Where product definition and engineering documentation are based on an ASME Y14 Case, this fact shall be noted on the documentation or in a referenced document.

1.3.12 Product Definition Without Reference to a Standard

When a product definition is created without a reference document (company, regional, national, or international) or contractually imposed documents, the drawing shall be interpreted in accordance with ASME PDS-1.1–2013.

Section 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 PDS-1.1–2013, Dimensioning, Tolerancing, Surface Texture, and Metrology Standards — Rules for Drawings With Incomplete Reference to Applicable Drawing Standard
ASME Y14.1-2020, Drawing Sheet Size and Format
ASME Y14.2-2014 (R2020), Line Conventions and Lettering
ASME Y14.3-2012 (R2018), Orthographic and Pictorial Views
ASME Y14.5-2009, Dimensioning and Tolerancing
ASME Y14.24-2020, Types and Applications of Engineering Drawings
ASME Y14.34-2013 (R2018), Associated Lists
ASME Y14.35-2014 (R2019), Revision of Engineering Drawings and Associated Documents
ASME Y14.38-2019, Abbreviations and Acronyms for Use on Drawings and Related Documents
ASME Y14.41-2019, Digital Product Definition Data Practices
ASME Y14.43-2011 (R2020), Dimensioning and Tolerancing Principles for Gages and Fixtures
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)

IEEE/ASTM SI 10, Standard for Use of the International System of Units (SI): The Modern Metric System
Publisher: Institute of Electrical and Electronics Engineers, Inc. (IEEE), 445 Hoes Lane, Piscataway, NJ 08854 (www.ieee.org)

Section 3
Definitions

3.1 DEFINITIONS

The following terms are defined as their use applies in this Standard. Nonmandatory Appendix A defines other commonly used terms for castings, forgings, and molded parts.

3.1.1 Casting

casting:

(a) a part obtained by solidification of material in a die or mold.

(b) a process by which liquid material is introduced into a mold, is allowed to solidify inside the mold, and is subsequently removed, resulting in a part.

3.1.2 Corner Radius

corner radius: the convex radius on the surface of a part connecting two or more surfaces. Also called *edge radius*. See Figure 3-1.

3.1.3 Die

die: any of various tools or devices for imparting a desired shape, form, or finish to a material or for impressing an object or material.

3.1.4 Die Closure

die closure: allowable part variation caused by inconsistent mating of opposing segments of a mold or die. See Figure 5-15, illustrations (a), (b), and (c).

3.1.5 Draft

draft: the angle given to a feature on a part so that it can be withdrawn from the mold or die. See Figure 5-14, illustration (a).

3.1.5.1 Drafted Fillet and Corner Radii

drafted fillet and corner radii: filleted and cornered surfaces normal to the forging plane or parallel to the die removal direction where draft is required.

NOTES:

(1) Drafted fillet surfaces occur when the fillet connects two or more adjacent draft surfaces with the same draft angle. These drafted fillets and corners create a conical surface. See Figure 3-2, illustration (a).

(2) Fillets and corners that connect two or more adjacent drafted surfaces with different draft angles do not create a conical fillet surface; they create a canted cylindrical surface. See Figure 3-2, illustration (b).

3.1.6 Drawing

drawing: an engineering document or data set that discloses, directly or by reference, by means of graphic or textual or graphic and textual presentations, the physical or functional requirements of an item (ASME Y14.100).

3.1.7 Fillet Radius

fillet radius: the concave radius on the surface of a part connecting two or more surfaces. See Figure 3-1.

NOTE: Fillet radii are intended to minimize stress concentrations, aid in proper fill, and minimize defects.

3.1.8 Flash

flash: excess material that results from leakage between mating surfaces of a mold or die. See Figure 5-11, illustrations (a) and (b).

3.1.9 Flash Extension

flash extension: allowable flash remnant. See Figure 5-11, illustrations (a) and (b).

3.1.10 Forging

forging:

(a) a part created by plastically deforming metal.

(b) the process of creating a part by plastically deforming metal (normally preheated) with impact or pressure into a specific shape.

3.1.11 Forging Plane

forging plane: a plane perpendicular to the opening and closing direction of the die. See Figure 5-2, illustrations (a) and (b).

3.1.12 Full Feature Axis

full feature axis: similar to feature axis as defined in ASME Y14.5-2009, except that the unrelated actual mating envelope is a conical drafted feature, with its included angle being the sum of draft angles.

3.1.13 Full Feature Center Plane

full feature center plane: similar to feature center plane as defined in ASME Y14.5-2009, except that the unrelated actual mating envelope is a wedge of a drafted feature, with its included angle being the sum of draft angles.

3.1.14 Gate

gate: a channel in a mold through which material flows into the mold cavity.

3.1.15 Grain Direction

grain direction: the predominant orientation of the fibrous crystalline structural units of wrought materials. See Figure 5-23.

3.1.16 Grain Flow

grain flow: the directional elongation in the grain structure of the material, and its nonhomogenous constituents, resulting from the forging process. Also called *flow lines*.

NOTE: Grain flow follows the direction of working during forging and is usually revealed by polishing and etching sections of the forging.

3.1.17 Match Draft

match draft: draft allowance permitted on matching surfaces at parting lines when the normal draft allowance would result in an offset of the surfaces at the parting line. See Figure 5-10.

3.1.18 Mismatch

mismatch: the offset of features on a part caused by misalignment of opposing segments of a mold or die. See Figure 5-16, illustrations (a) and (b).

3.1.19 Mold

mold:

(a) a form made of sand, metal, or other material.

(b) the process of pouring or injecting material into a form to produce a part.

3.1.20 Mold Line

mold line: a line generated by the intersection of projected surfaces. See Figure 5-14, illustration (a).

3.1.21 Mold Line Center Line

mold line center line: the line centered between a pair of mold lines, where the mold lines lie in a common plane. See Figure 5-14, illustration (d).

3.1.22 Mold Line Center Point

mold line center point: the point of the center of a circular mold line. See Figures 5-12 and 5-13.

3.1.23 Parting Line

parting line:

(a) the location on the part where mold or die segments separate.

(b) a line on the drawing representing the mating surfaces of the die or mold segments.

See Figure 5-14, illustration (a); Figure 5-15, illustrations (a) through (c); Figures 5-17 and 5-18; and Nonmandatory Appendix C.

3.1.24 Parting Plane

parting plane: a plane perpendicular to the opening and closing directions of the die segments. See also *forging plane*.

3.1.25 Parting Surface

parting surface: the mating surfaces of die segments.

3.1.26 Pattern

pattern: a form made of wood, metal, or other material around which sand or other suitable material is placed to make a mold.

3.1.27 Riser

riser: a reservoir built into a mold to prevent shrinkage cavities in the part.

3.1.28 Riser Stub

riser stub: the allowable remaining riser.

3.1.29 Round

round: see *corner radius*.

3.1.30 Scale

scale: an encrustation formed on the surface of a hot worked metal (forging) as a result of oxidation.

3.1.31 Sink

sink: a shallow depression in the surface of a cast or molded part due to internal shrinkage.

3.1.32 Vent Marks

vent marks: small protrusions on the surface of a part caused by material entering the vents (air escape passages) in the mold or die.

3.1.33 Wall

wall: a solid feature at any physical orientation comprised of opposing surfaces having a nominally uniform thickness. See Figure 5-24.

3.1.34 Wall Thickness

wall thickness: the actual local size between all sets of opposing points on the surfaces of a wall. See Figure 5-24.

NOTE: Perfect form at maximum wall thickness tolerance is not a requirement [i.e., Rule #1 (see ASME Y14.5, para. 5.8.1) does not apply to "wall thickness" tolerance].

Figure 3-1
Fillet and Corner Radius

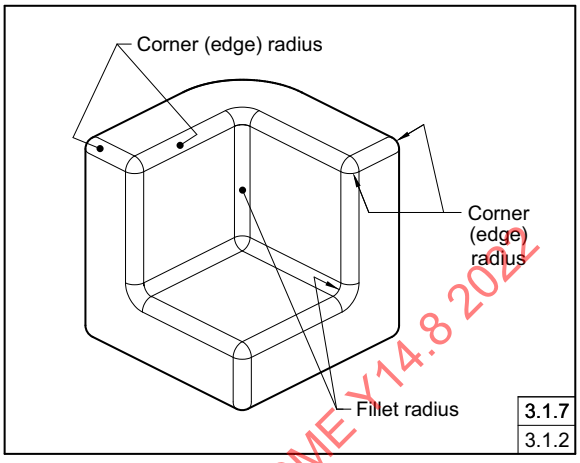
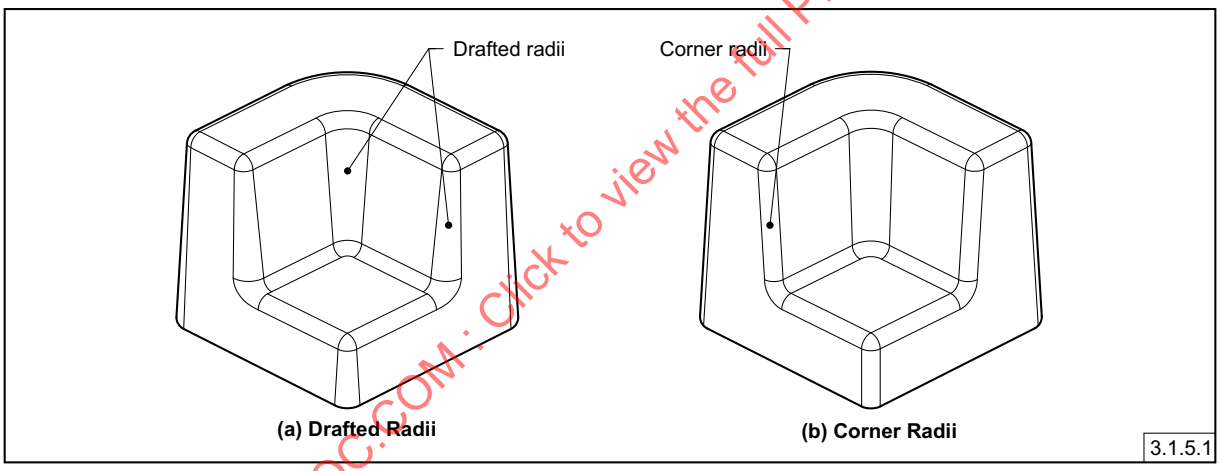


Figure 3-2
Fillet and Corner Radii



Section 4
Drawing Presentation Methods

4.1 GENERAL

This Section establishes methods of preparing drawings for castings, forgings, and molded parts.

See ASME Y14.24 for complete descriptions and suggested use of types and applications of engineering drawings.

4.2 SEPARATE REQUIREMENTS DRAWING METHOD

This method shows requirements for castings, forgings, and molded parts and for postprocessing (finishing, machining, etc.) on separate drawings in the same document or in separate documents. See Figure 4-1.

4.3 COMBINED REQUIREMENTS DRAWING METHOD

This method shows the part requirements and the end-item requirements in superimposed views. Phantom lines may be used to show the casting, forging, or molded-part outline. See Figure 4-2.

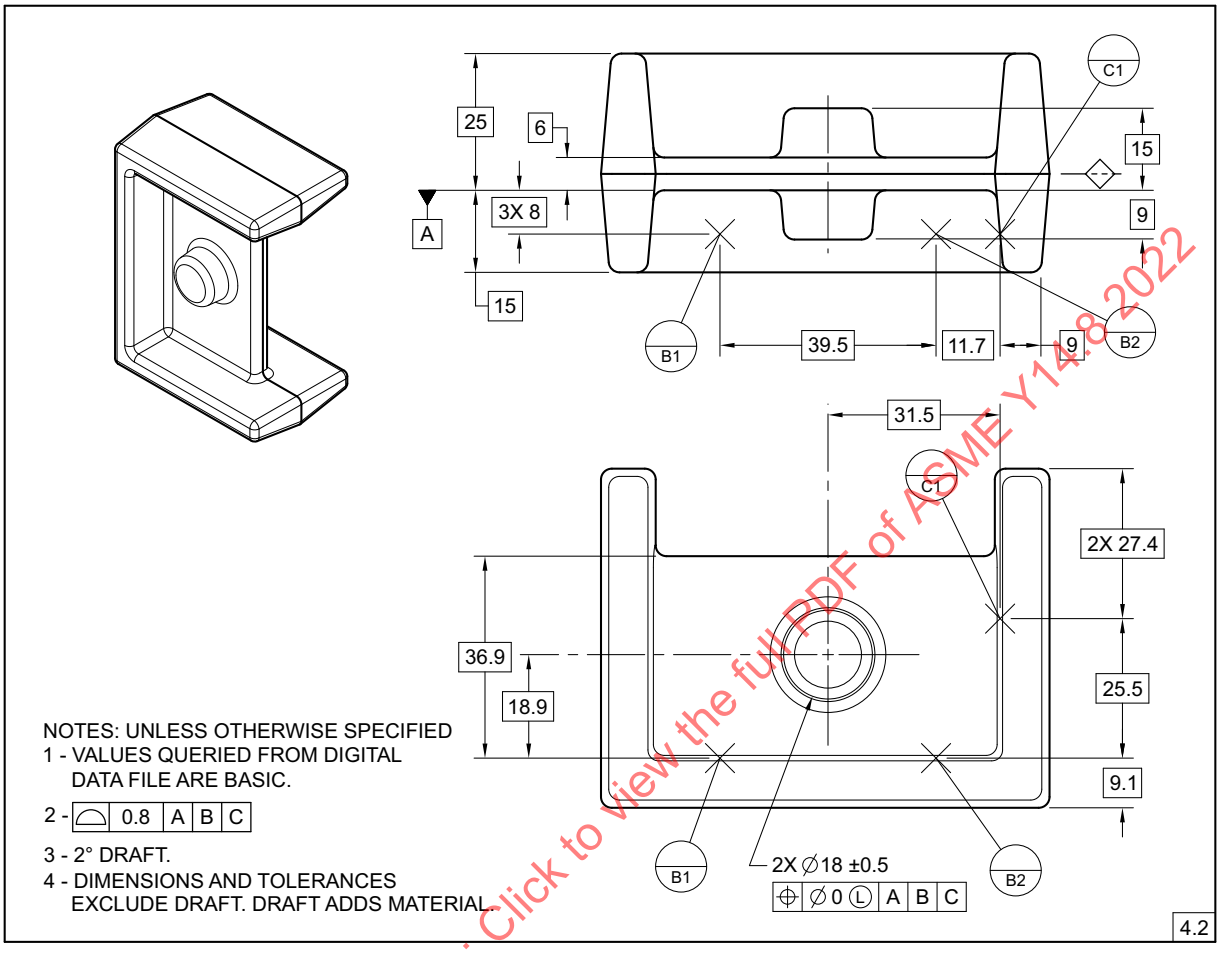
Casting, forging, and molded-part notes shall be separated from end-item notes.

CAUTION: The decision to combine drawings should be made cautiously. Potential disadvantages resulting from combining drawings include

***(a)* increased complexity of the drawing, which may diminish clarity and usefulness**

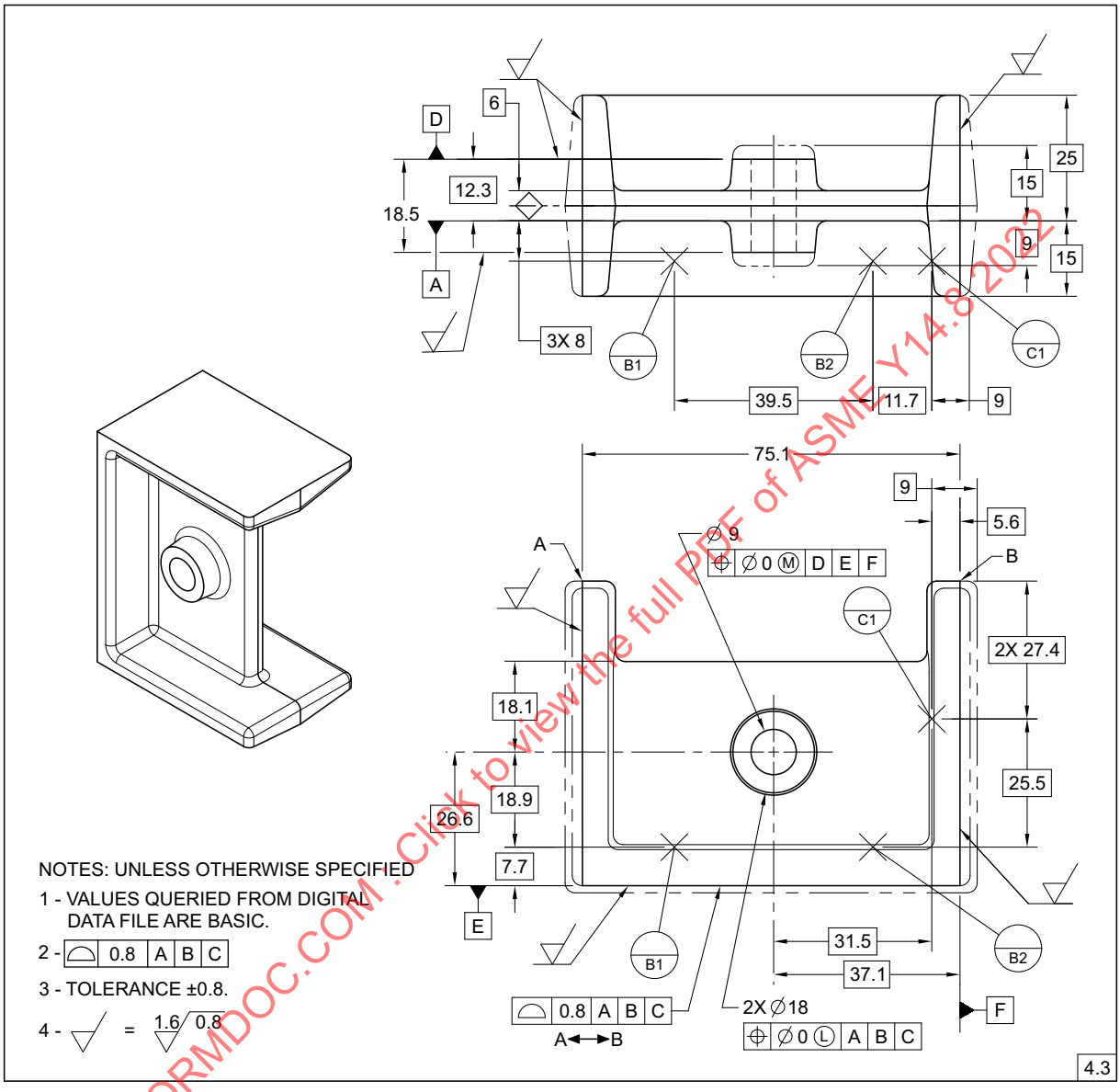
***(b)* frequent change activity to the drawing, which may increase the need to update associated records, material control data, manufacturing planning, etc.**

Figure 4-1
Separate Drawing Method — Casting, Forging, and Molding Requirements



GENERAL NOTE: This is the recommended practice.

Figure 4-2
Combined Drawing Method



GENERAL NOTE: This is not the recommended practice. See Caution in para. 4.3.

Section 5
Drawing Practices

5.1 GENERAL

This Section establishes items unique to castings, forgings, and molded parts not defined by other standards that should be defined on the drawing.

5.2 AS-CAST, AS-FORGED, AND AS-MOLDED SURFACES

Drawings shall specify whether machining as-cast, as-forged, and as-molded surfaces is permitted or prohibited other than for removing gates, risers, flash, etc. Where machining is permitted, the surface texture shall be specified. Gates, riser stubs, flash, etc. may exceed tolerance boundaries unless otherwise specified. Where a surface may retain gates, riser stubs, flash, etc. beyond the tolerance boundary, a specified limit beyond the tolerance boundary shall be defined.

5.3 CORNER AND FILLET RADII

Corner and fillet radii values and tolerances shall be specified on the drawing.

5.4 DRAFT

Draft shall be defined and should be shown for clarity; see Figure 5-1. Forged-part draft angles are related to the forging plane; see Figure 5-2, illustrations (a) and (b). Cast- and molded-part draft angles are related to mold parting action. Draft may exceed the perfect form boundary at maximum material condition (MMC) unless otherwise specified. Draft angles specified are per side (not included angle) unless otherwise specified. Draft angles may be specified with a stated value or a maximum value or defined in a 3D model. Drafted features may be controlled using profile of a surface. See Figures 5-1 and 5-3.

5.4.1 Methods for Specifying Dimensions Affected by Draft

Provision for draft may be addressed either directly on the field of the drawing or by a general note, or both on the field of the drawing and in a note. The note should state "DRAFT ADDS MATERIAL" to specify that material is added relative to the dimensions; see Figure 5-4. Exceptions to this are

(a) application of the local note "DRAFT REDUCES MATERIAL"; see Figure 5-5.

(b) application of the "DFT INCL" (draft included) symbol, which indicates that any draft shall be contained within the stated tolerance; see Figure 5-6.

(c) controlling drafted features by applying toleranced dimensions at specific locations; see Figure 5-7.

(d) application of the "+DFT" (plus draft) symbol, which indicates that the dimension may increase due to draft in addition to the increase allowed by any applicable tolerance applied to the considered feature. The value of the draft angle may be included before DFT. See Figure 5-8.

(e) application of the "−DFT" (minus draft) symbol, which indicates that the dimension may decrease due to draft in addition to the decrease allowed by any applicable tolerance applied to the considered feature. The value of the draft angle may be included before DFT. See Figure 5-9.

(f) a size dimension with a geometric form or orientation tolerance, such as cylindricity or perpendicularity, which indicates that the feature has zero draft.

CAUTION: The symbols for "+DFT" and "−DFT" and the notes "DRAFT ADDS MATERIAL" and "DRAFT REDUCES MATERIAL" should not be used for critical features due to the limitations of measurements being only near, rather than at, a given end of the feature.
For fully defined features, a geometric tolerance such as a profile of a surface tolerance should be used.

5.4.2 Match Draft

Match draft shall be specified where applicable. See Figure 5-10.

5.5 FLASH EXTENSION

The limits of permissible flash extension shall be specified on the drawing. See Figure 5-11, illustrations (a) and (b).

5.6 EJECTOR PIN DEPRESSIONS OR PROTRUSIONS

The limits of permissible ejector pin marks (e.g., height or depth) shall be specified on the drawing.

5.7 MOLD LINE DIMENSIONS AND TOLERANCES

Dimensions and tolerances (size, position, etc.) shall be applied at the mold line(s) at one end of features unless otherwise specified. See Figures 5-12 and 5-13, and Figure 5-14, illustration (a).

5.7.1 Circular Mold Line

The following text is a description of design intent, where a position tolerance is associated with a drafted feature dimensioned to mold lines, on drawings of parts prepared in accordance with this Standard.

Basic dimensions (shown or not shown) in the figures of this Section locate and orient the tolerance zone relative to the specified datum reference frame. Because of the draft, the feature is dimensioned to the mold line (the theoretical line generated at the intersection of the extended draft and the dimensioned end of the feature). The feature will be a circular mold line. The mold line center point must fall within the tolerance zone defined in the feature control frame associated with the size dimension. The tolerance zone is a circle. See Figures 5-12 and 5-13.

5.7.2 Parallel Line Elements at a Mold Line

The following text is a description of design intent, where a position tolerance is associated with a drafted feature dimensioned to mold lines, on drawings of parts prepared in accordance with this Standard.

Basic dimensions (shown or not shown) in the figures of this Section locate and orient the tolerance zone relative to the specified datum reference frame. Because of the draft, the feature is dimensioned to the mold lines (the theoretical lines generated at the intersection of the extended draft and the dimensioned end of the feature). The feature comprises the two parallel mold lines. The mold line center line must fall within the tolerance zone defined in the feature control frame associated with the size dimension. The tolerance zone is two parallel lines. See Figure 5-14, illustration (d).

5.8 FULL FEATURE MODIFIER

The note "FULL FEATURE" or a full feature symbol is used as a modifier applied to a position or orientation tolerance (perpendicularity, parallelism, or angularity). A full feature modifier may be applied to a feature control frame that is placed below a size tolerance that applies to one mold line of a drafted feature. When a full feature modifier is applied, the associated tolerance controls the feature's full feature axis or full feature center plane, rather than the default control, which applies to the mold line center point or mold line center line at one end of the drafted feature. The tolerance zone for the position or orientation tolerance is bounded by the mold line at each end of the feature. The full feature modifier shall only be applied at regardless of feature size. See Figure 5-14, illustrations (b) through (e).

5.9 DIE CLOSURE

Die closure tolerance is applied to the appropriate dimensions as shown in Figure 5-15, illustrations (a), (b), and (c). Unless otherwise specified, die closure tolerances shall be included in the dimensional limits.

5.10 SURFACE IRREGULARITIES CAUSED BY MANUFACTURING PROCESSES

The limits of surface irregularities created by processing (such as ejector pin marks, flash extension, and gate marks) may exceed tolerance boundaries unless otherwise specified. If limited, these limits shall be identified on the drawing or in a related document; see Figure 5-11, illustrations (a) and (b) and notes in Section 7 for examples. Features created for tooling considerations, such as ejector pin pads or bosses, shall be specified.

5.11 ORIENTATION OF FORGING PLANE

The orientation of the forging plane shall be indicated by defining a basic angular relationship between the forging plane and an appropriate datum reference frame. See Figure 5-2, illustrations (a) and (b).

5.12 MARKING

The part drawing shall define applicable identification information. See paras. 7.2(k) and 7.4(e).

5.13 MISMATCH

Mismatch tolerance, where applied, shall be specified as a maximum value. Mismatch may cause features to exceed specified tolerance limits unless otherwise specified. See Figure 5-16, illustrations (a) and (b).

5.14 PARTING LINES

Parting lines shall be depicted on drawings as a phantom line extending beyond the part in applicable views, with the parting line symbol added. See Figure 5-2, illustrations (a) and (b), and Figures 5-17 and 5-18. Parting line symbols shall be shown on drawings of closed die forged parts or when the all around this side of parting line symbol (see para. 5.14.1) or the all over this side of parting line symbol (see para. 5.14.2) is used. The parting line symbol may be required for other applications, such as when trimming specifications are indicated.

5.14.1 All Around This Side of Parting Line

To apply a requirement to all features all around one side of a parting line, the symbol for all around this side of parting line shall be indicated on the leader line; see

Figure 5-19. The symbol is shown in the view or section showing the desired basic profile. The all around this side of parting line symbol shall be applied to orthographic views. See Figure 5-20.

5.14.2 All Over This Side of Parting Line

To apply a requirement to all features all over one side of a parting line, the symbol for all over this side of parting line shall be indicated on the leader line; see Figure 5-21. An example of the application of the symbol is given in Figure 5-22.

5.15 SHARP CORNERS

Drawings shall specify the requirements of corners and fillets that are shown sharp. See paras. 7.3(m), 7.3(n), 7.3(y), and 7.3(z).

5.16 GRAIN DIRECTION

Where a grain direction requirement is specified on the drawing, it shall be shown in the appropriate view. See Figure 5-23.

5.17 PRODUCT DEFINITION DATA REQUIREMENTS

This subsection establishes the minimum requirements for product definition data and items unique to castings, forgings, and molded parts not defined by other standards. See ASME Y14.41.

5.17.1 3D Model

Product definition data that defines the entire casting, forging, or molded part directly or by reference is required. The product definition data shall include the location of the parting line(s), all draft, all radii, and finish machine stock where needed.

5.17.2 Controlling Document

If product definition data and a drawing graphic sheet are both supplied, the controlling document shall be clearly identified before the start of the tool build.

5.17.3 Concurrent Engineering

It is recommended for the product design team for the casting, forging, or molding to consult with their supplier prior to finalizing the product definition data. During this review, the supplier can assist in locating the desired parting line and make design recommendations for the best casting, forging, and molding practices.

5.18 WALL THICKNESS AS A REFINEMENT OF PROFILE OF A SURFACE

Where it is necessary to maintain a uniform wall thickness throughout a region of a part or an entire part, a local or general note indicating the wall thickness shall be included. The wall thickness is a refinement of the control provided by the profile of a surface tolerance.

In Figure 5-24, the basic thickness of the part walls from the product definition data is 4 mm. The profile of a surface tolerance applied to the basic digital data allows the wall thickness to range from 3.2 mm to 4.8 mm with respect to the datum reference frame. The addition of the wall thickness general note of 4±0.2 limits the range of the wall thickness from 3.8 mm to 4.2 mm. Since the wall thickness tolerance is an actual local size specification, it does not relate to the datum reference frame and is free to float but is bounded by the profile of a surface tolerance of 0.8 mm.

Figure 5-1
Draft Angle

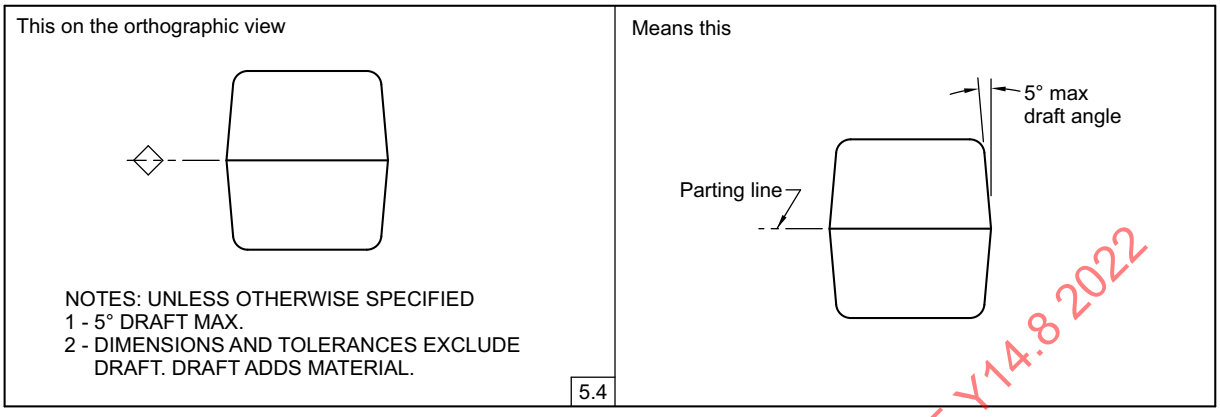


Figure 5-2
Datum Plane and Forging Plane

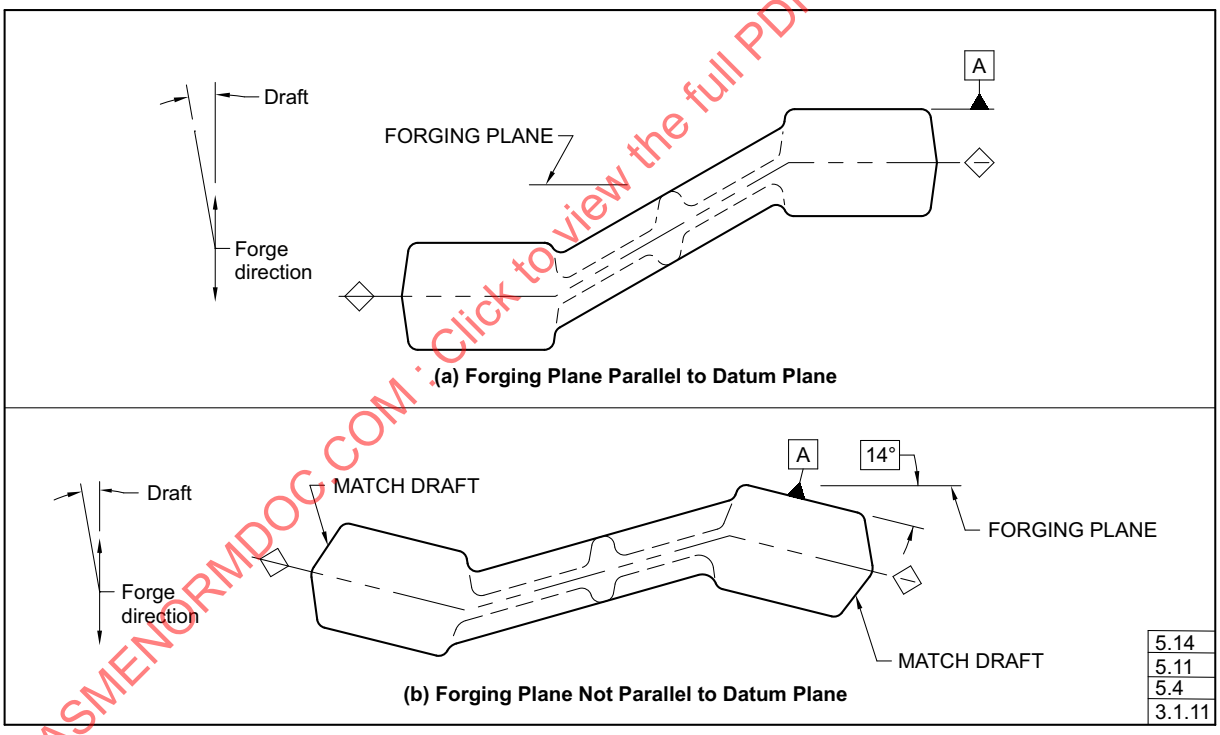


Figure 5-3
Drafted Surfaces Controlled With Profile of a Surface

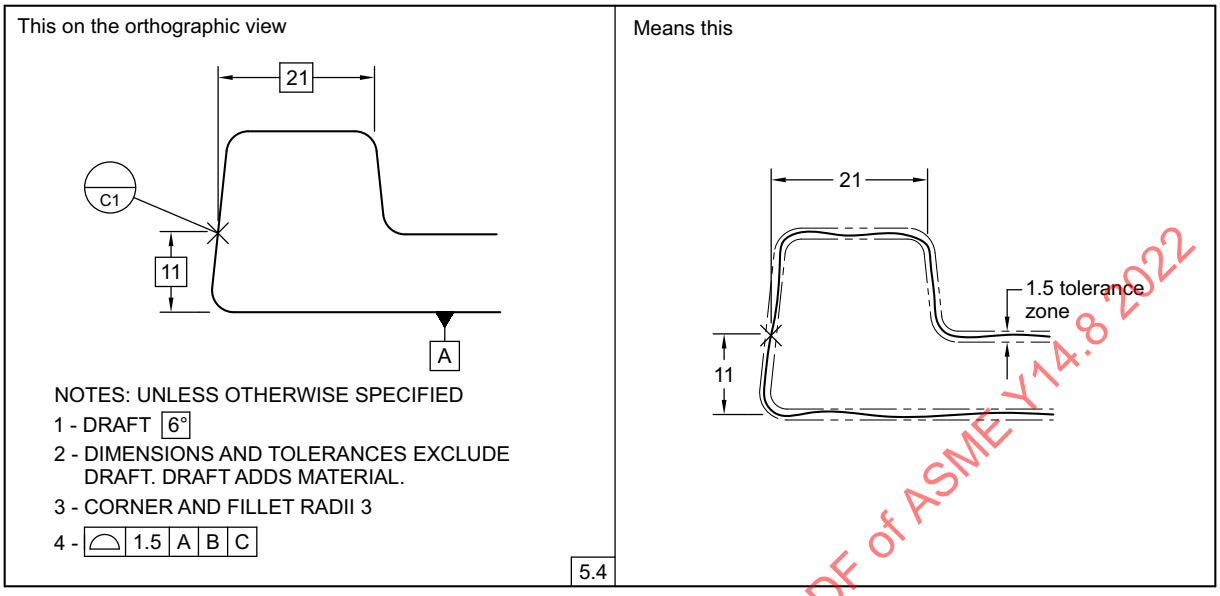


Figure 5-4
Draft Adds Material

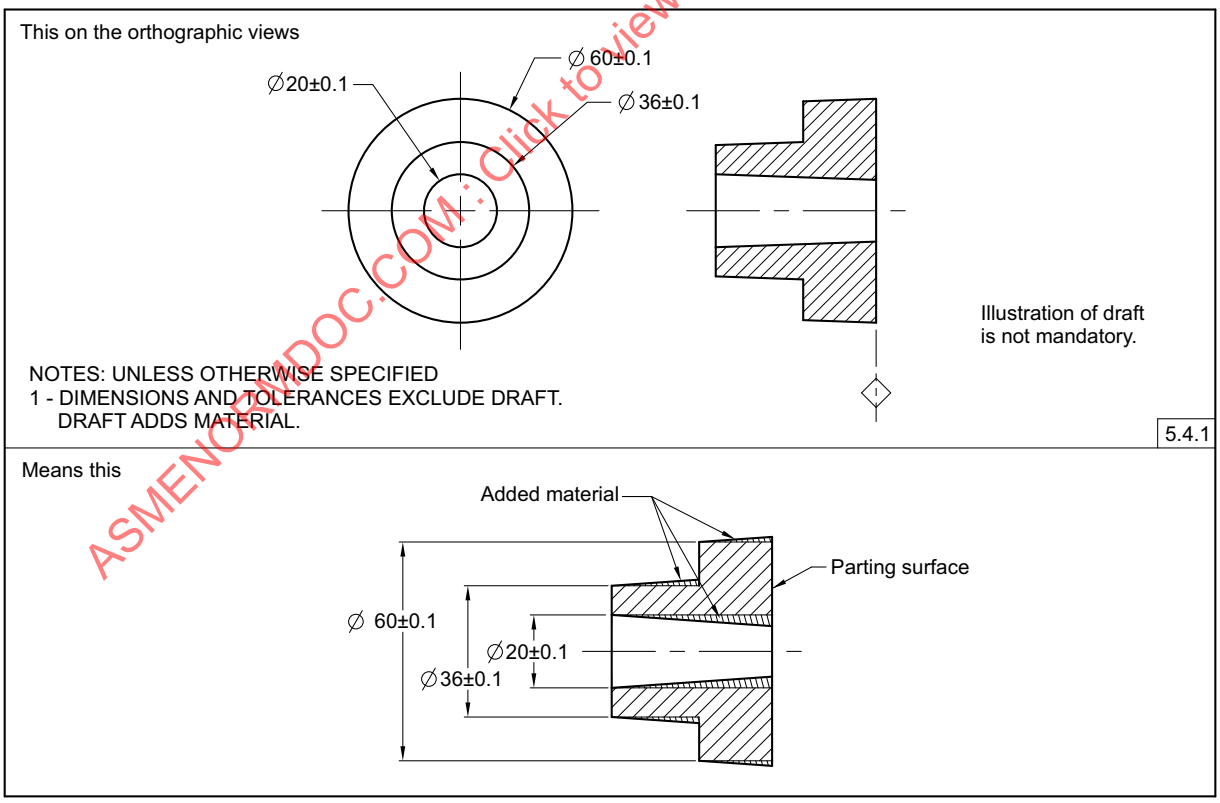


Figure 5-5
Draft Reduces Material

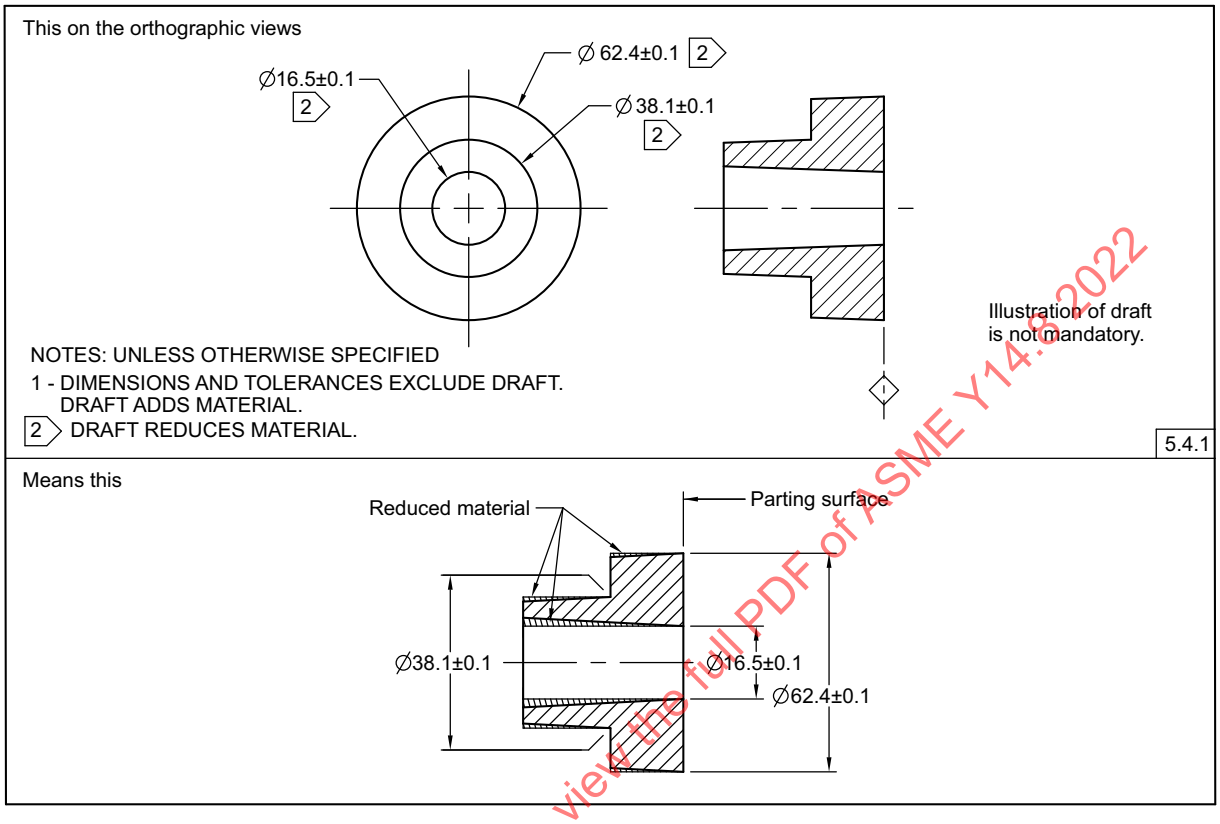


Figure 5-6
"DFT INCL" Example

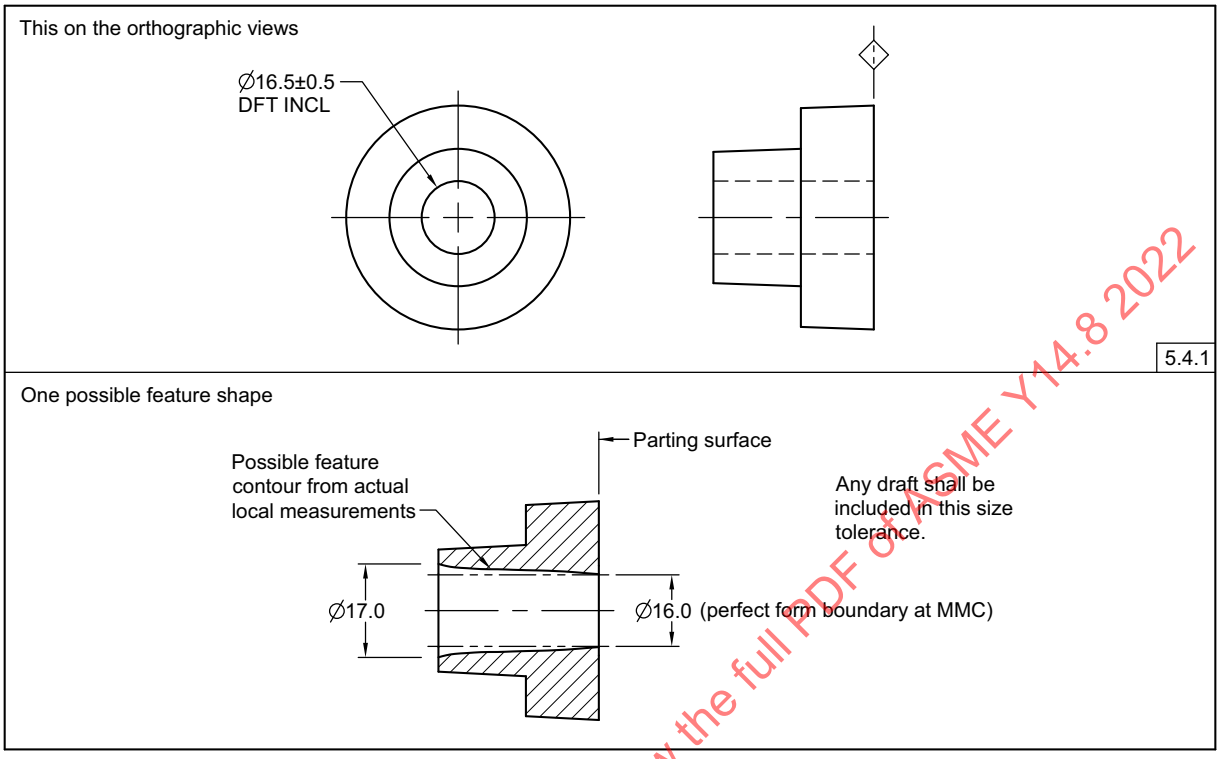


Figure 5-7
Size at Specific Locations for a Drafted Feature

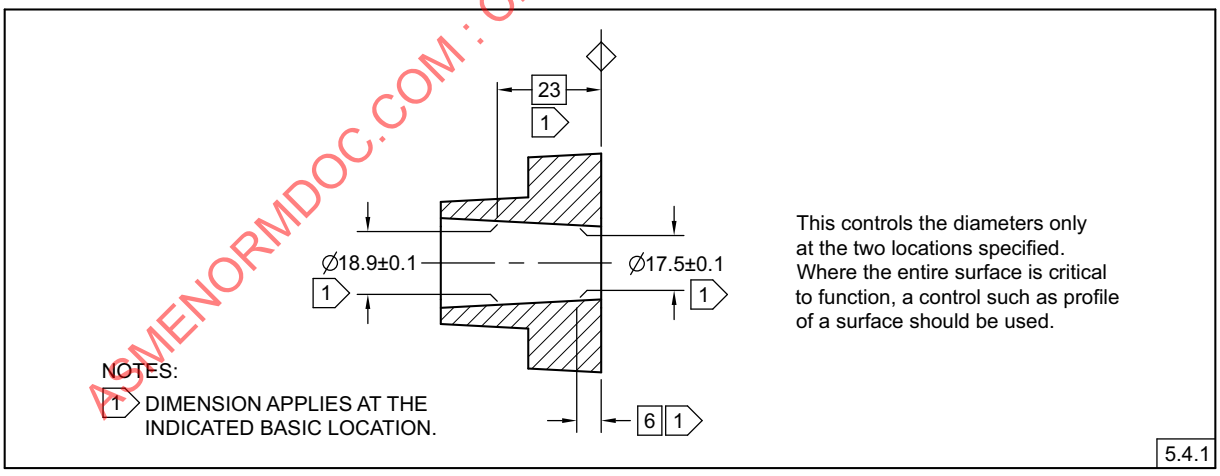


Figure 5-8
"+DFT" Example

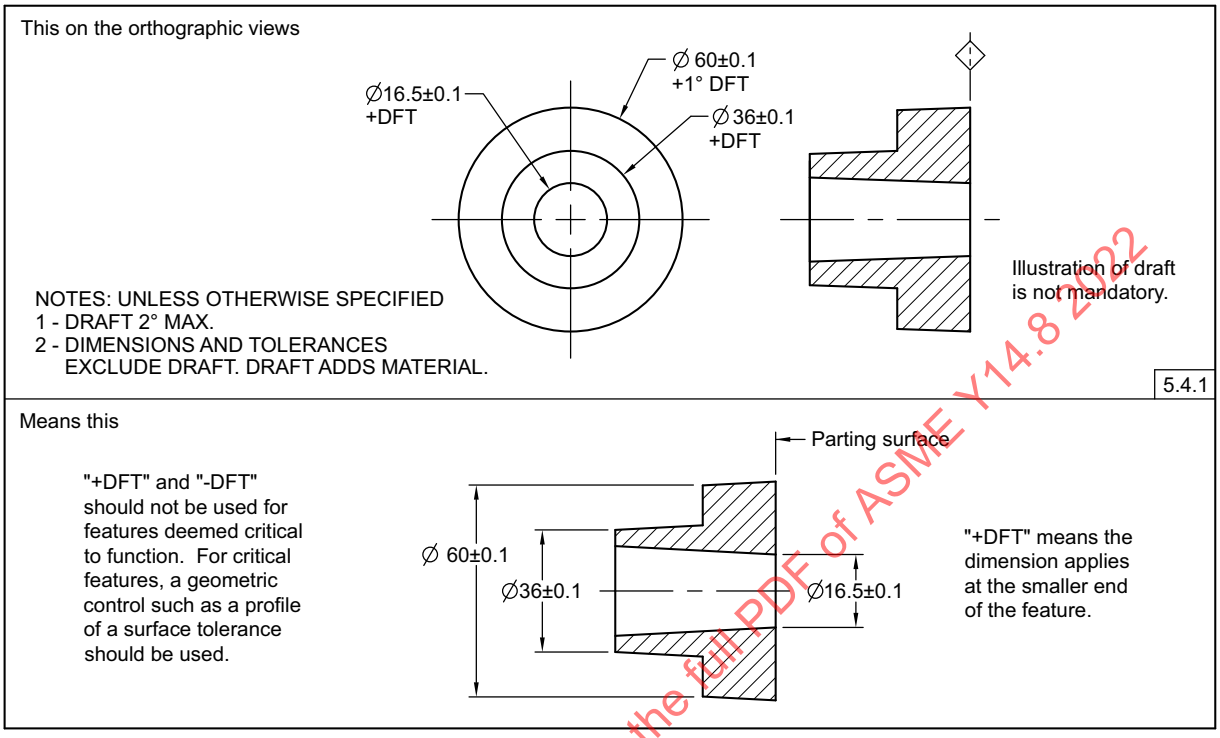


Figure 5-9
"–DFT" Example

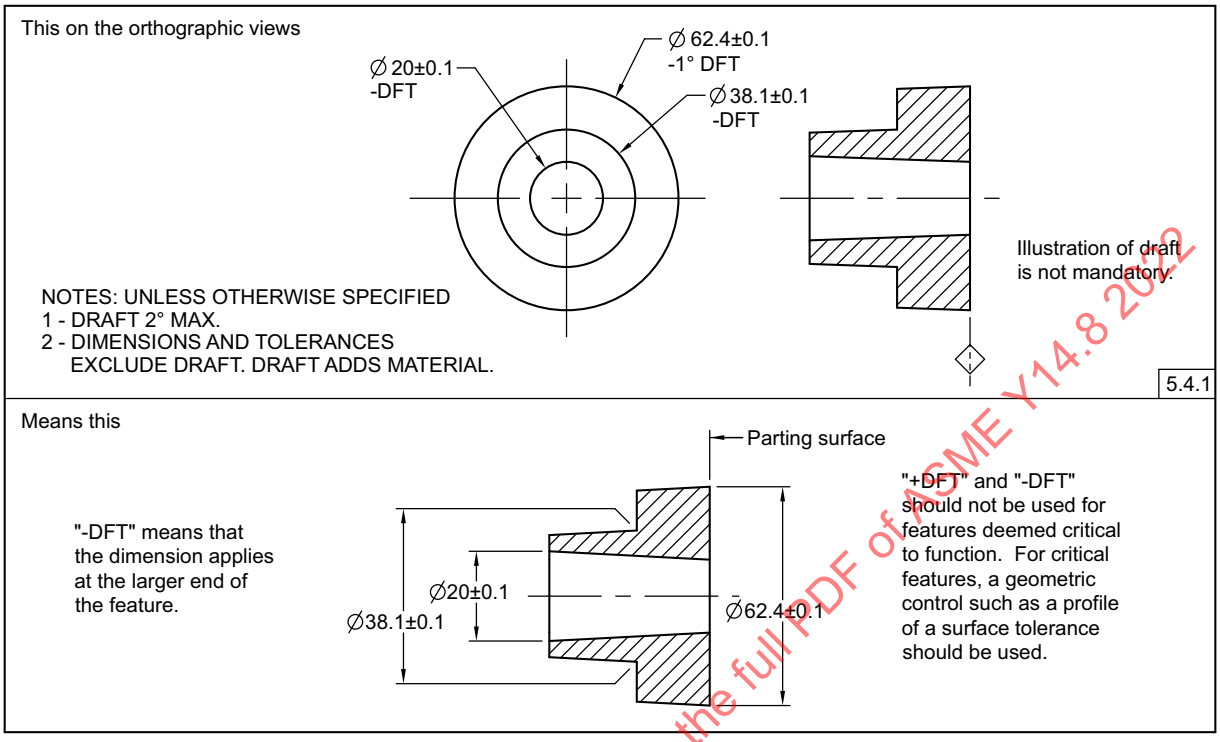


Figure 5-10
Match Draft

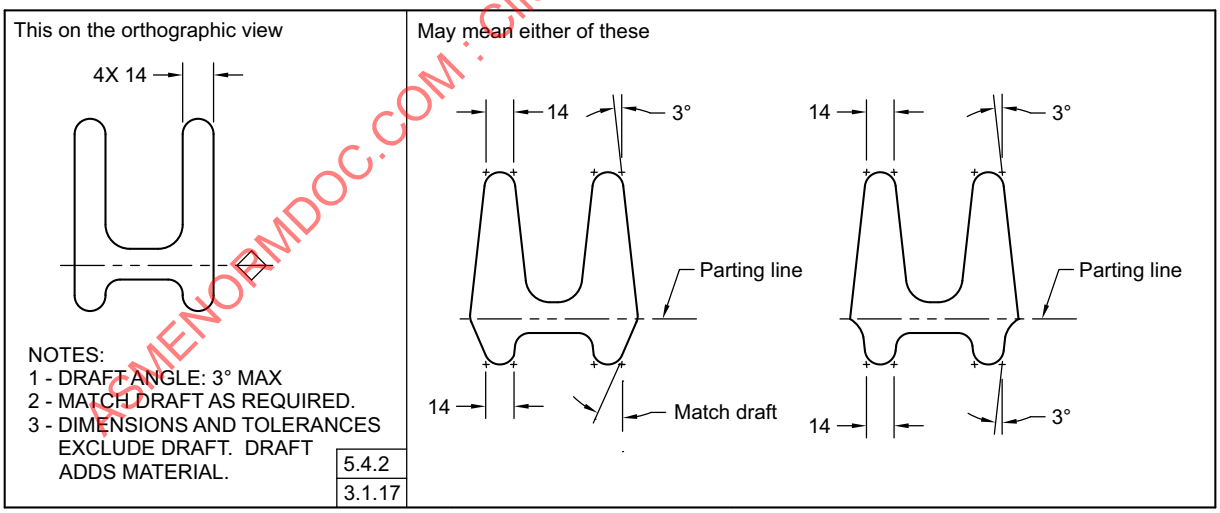
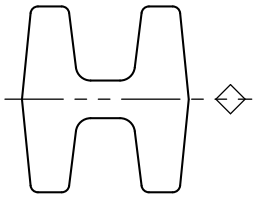


Figure 5-11
Flash Extension

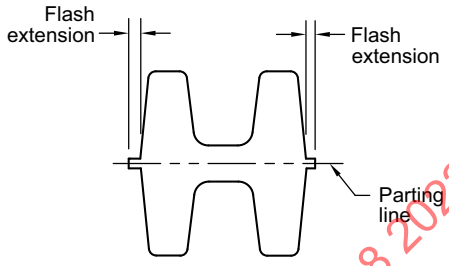
This on the orthographic view



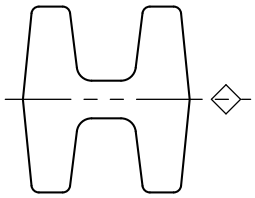
NOTES:
1 - FLASH EXTENSION: 2 MAX

(a) Flash Extension

Means this



This on the orthographic view



NOTES:
1 - FLASH EXTENSION: 2 MAX
2 - MISMATCH: 3 MAX

(b) Flash Extension and Mismatch

5.10
5.5
3.1.9
3.1.8

Means this

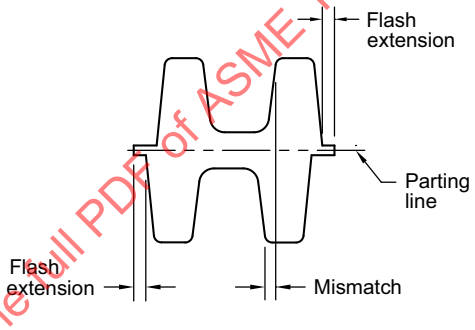
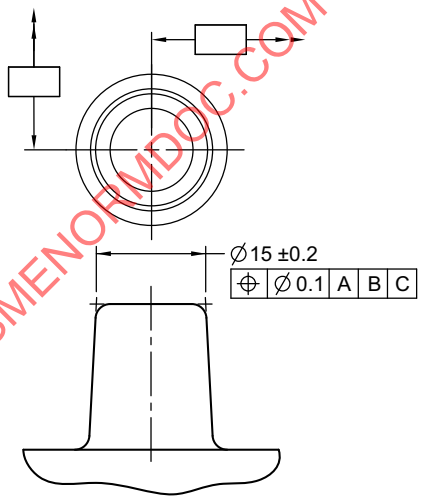


Figure 5-12
Boss Circular Mold Line and Mold Line Center Point

This on the orthographic views



NOTES: UNLESS OTHERWISE SPECIFIED
1 - DRAFT ANGLE 3° MAX.
2 - DIMENSIONS AND TOLERANCES EXCLUDE DRAFT. DRAFT ADDS MATERIAL.

5.7.1
5.7
3.1.22

Means this

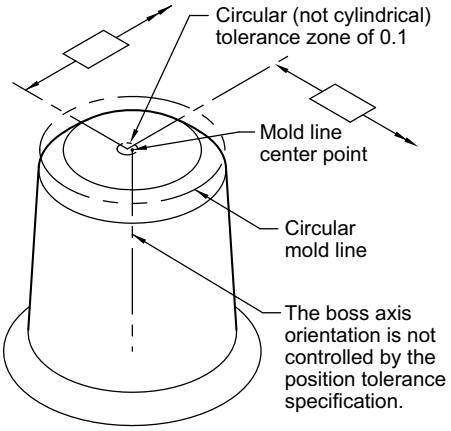
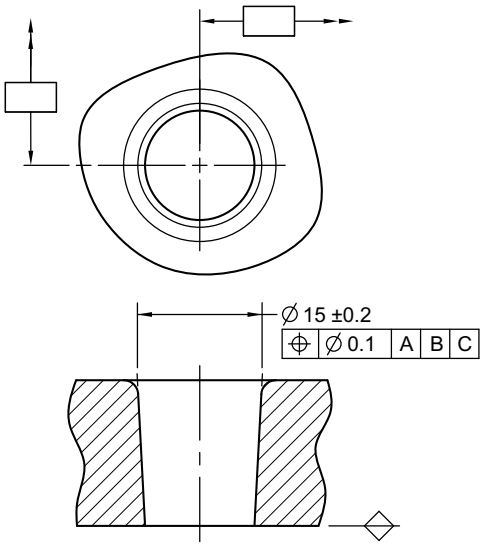


Figure 5-13
Hole Circular Mold Line and Mold Line Center Point

This on the orthographic views



NOTES: UNLESS OTHERWISE SPECIFIED
1 - DRAFT ANGLE 3° MAX.
2 - DIMENSIONS AND TOLERANCES EXCLUDE DRAFT. DRAFT ADDS MATERIAL.

5.7.1
5.7
3.1.22

Means this

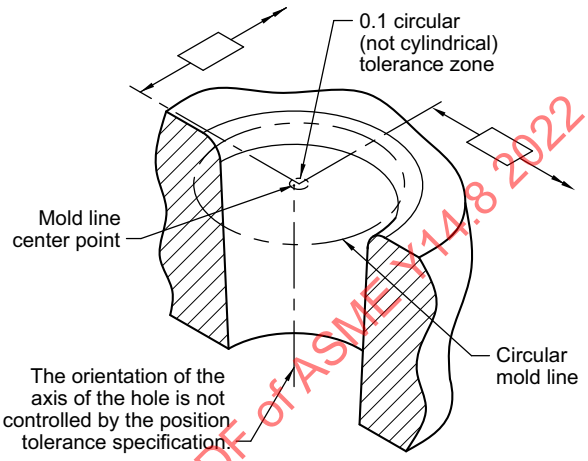
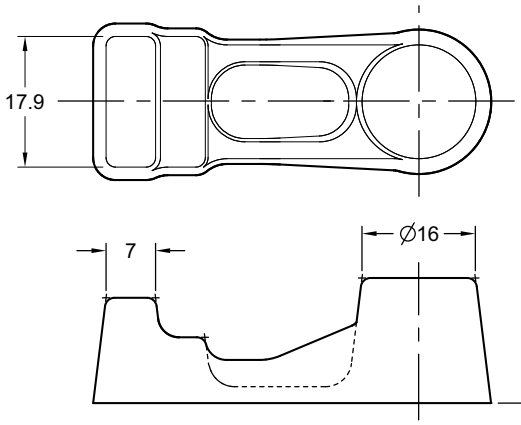


Figure 5-14
Use of Full Feature Modifier

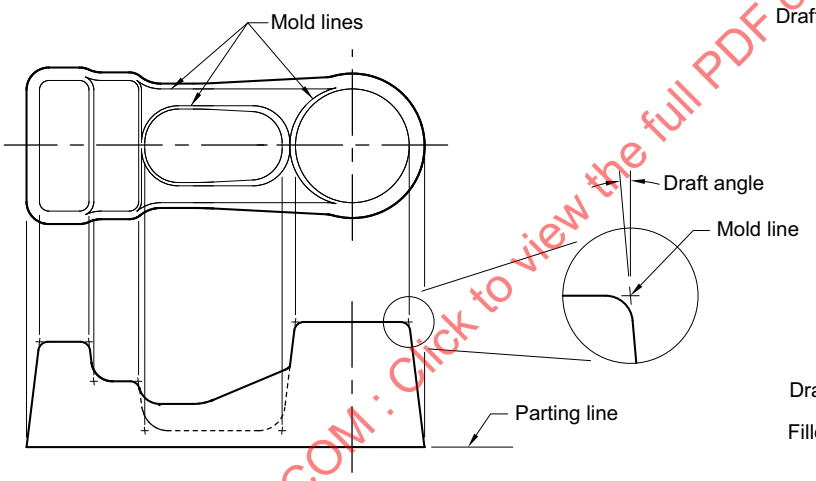
This on the orthographic views

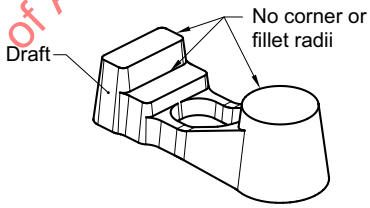


(a) Draft Construction and Mold Lines

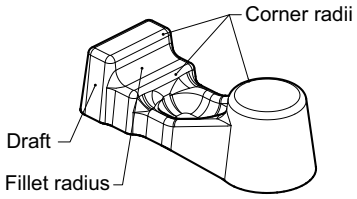
5.7
3.1.23
3.1.20
3.1.5

Means this





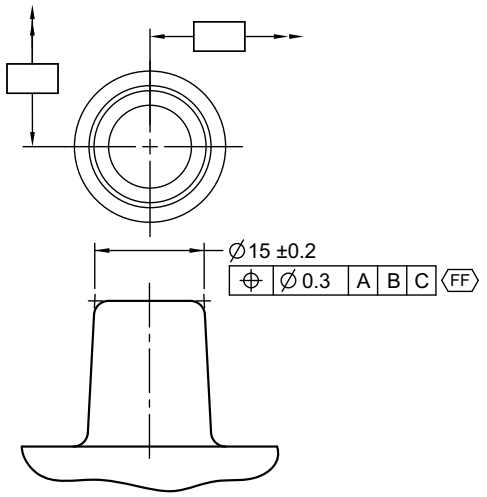
Solid model without corner and fillet radii to show mold lines



Completed solid model with corner and fillet radii

Figure 5-14
Use of Full Feature Modifier (Cont'd)

This on the orthographic views

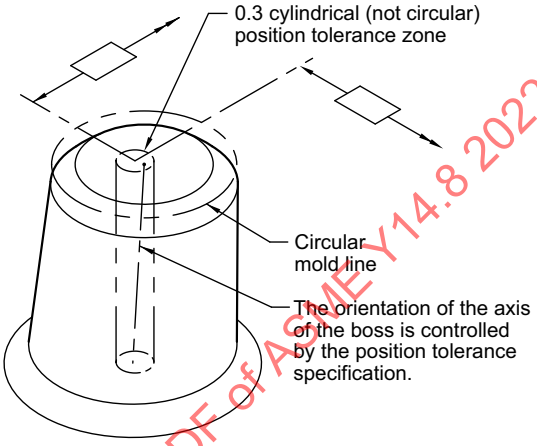


NOTES: UNLESS OTHERWISE SPECIFIED
1 - DRAFT 3° MAX.
2 - DIMENSIONS AND TOLERANCES EXCLUDE DRAFT. DRAFT ADDS MATERIAL.

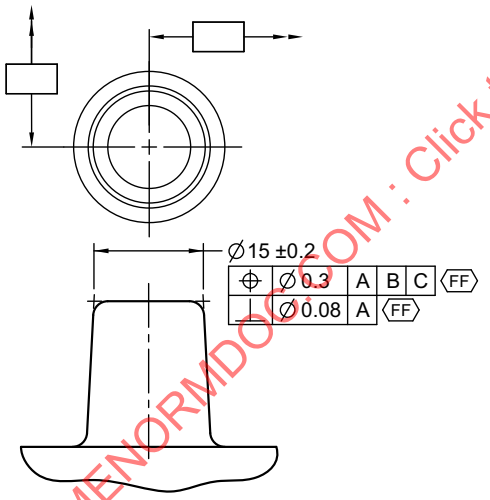
(b) Boss Full Feature Modifier for Position

5.8

Means this



This on the orthographic views



NOTES: UNLESS OTHERWISE SPECIFIED
1 - DRAFT 3° MAX.
2 - DIMENSIONS AND TOLERANCES EXCLUDE DRAFT. DRAFT ADDS MATERIAL.

(c) Boss Full Feature Modifier for Orientation

5.8

Means this

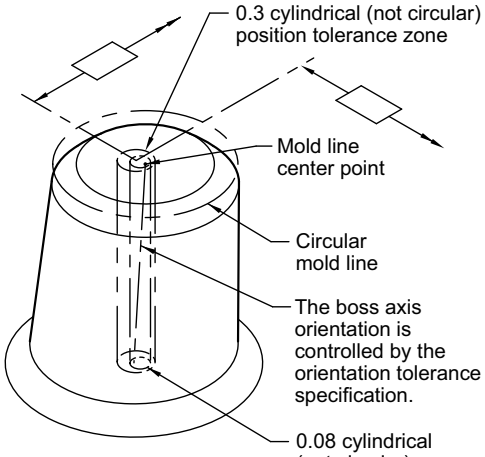
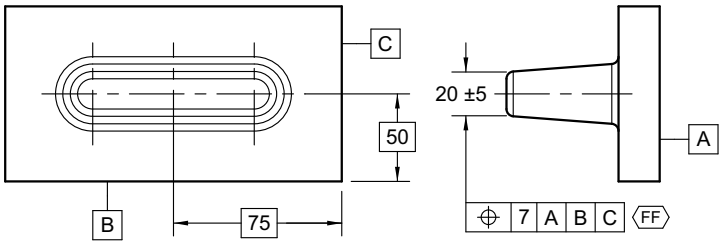


Figure 5-14
Use of Full Feature Modifier (Cont'd)

This on the orthographic views

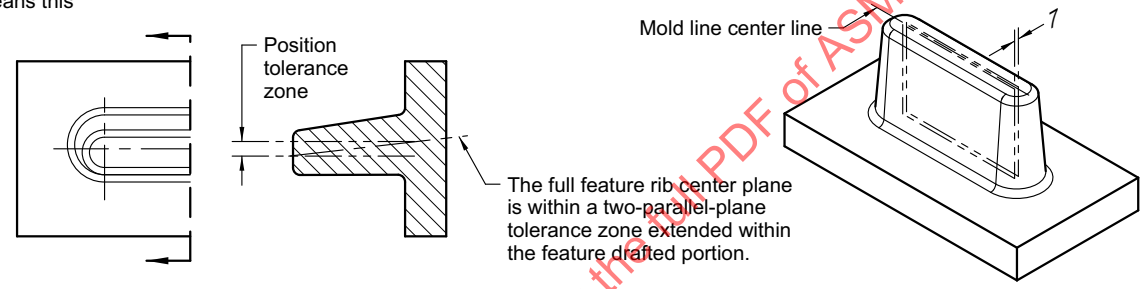


NOTES:
1 - DIMENSIONS AND TOLERANCES EXCLUDE DRAFT.
2 - DRAFT ADDS MATERIAL.

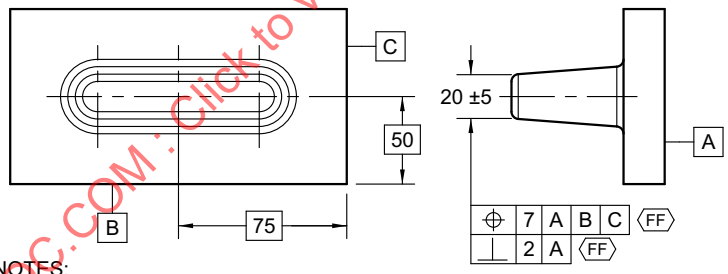
(d) Rib Full Feature Modifier for Position

5:8
5.7.2
3.1.21

Means this



This on the orthographic views



NOTES:
1 - DIMENSIONS AND TOLERANCES EXCLUDE DRAFT.
2 - DRAFT ADDS MATERIAL.

(e) Rib Full Feature Modifier for Orientation

5.8

Means this

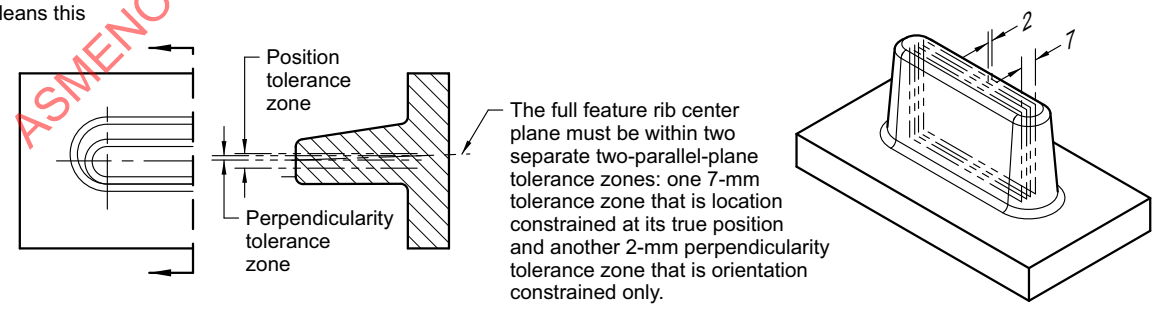
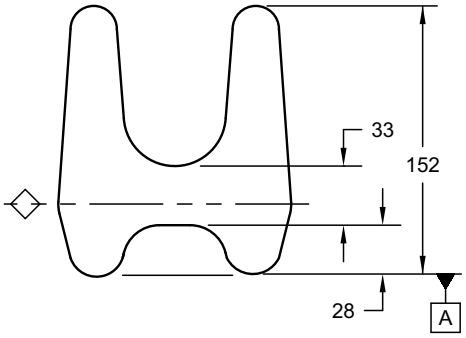


Figure 5-15
Die Closure

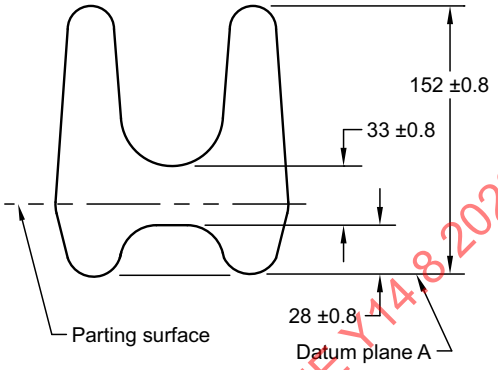
This on the orthographic view



NOTES: UNLESS OTHERWISE SPECIFIED
1 - TOLERANCE: ±0.8

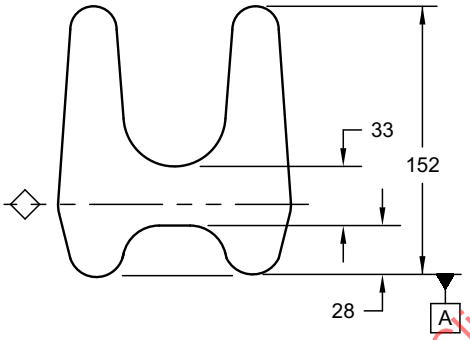
(a) Die Closure Not Specified and Parting Line Indicated

Means this



Die closure allowance is included in dimensions shown.

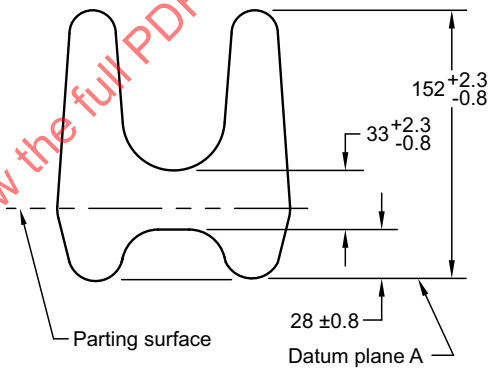
This on the orthographic view



NOTES: UNLESS OTHERWISE SPECIFIED
1 - TOLERANCE: ±0.8
2 - DIE CLOSURE TOLERANCE: +2.3

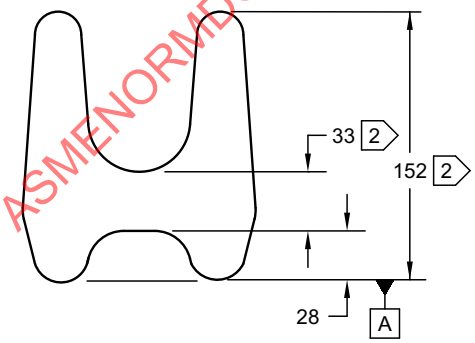
(b) Die Closure Specified and Parting Line Indicated

Means this



Die closure tolerance applies to
dimensions that cross the parting line.

This on the orthographic view

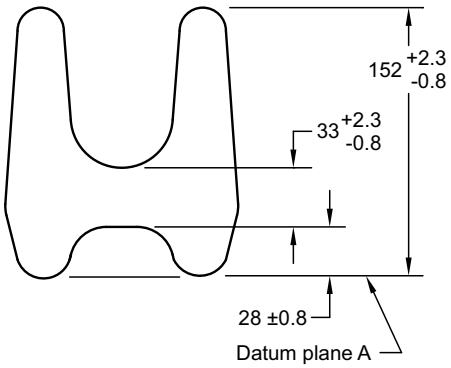


NOTES: UNLESS OTHERWISE SPECIFIED
1 - TOLERANCE: ±0.8
2 DIE CLOSURE TOLERANCE: +2.3

(c) Die Closure Specified and Parting Line Not Indicated

5.9
3.1.23
3.1.4

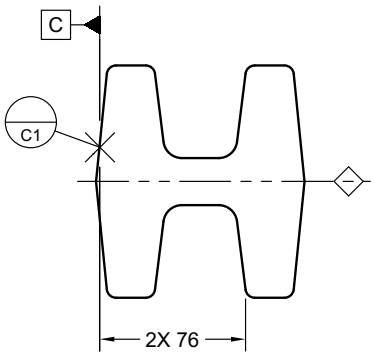
Means this



Die closure tolerance applies
to flagged dimensions.

Figure 5-16
Mismatch

This on the orthographic view

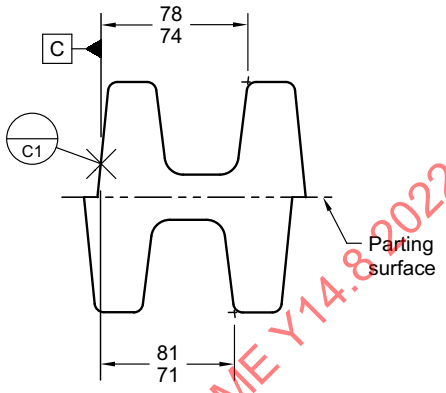


NOTES: UNLESS OTHERWISE SPECIFIED
1 - TOLERANCE: ±2
2 - MISMATCH: 3 MAX

(a) Mismatch Tolerance

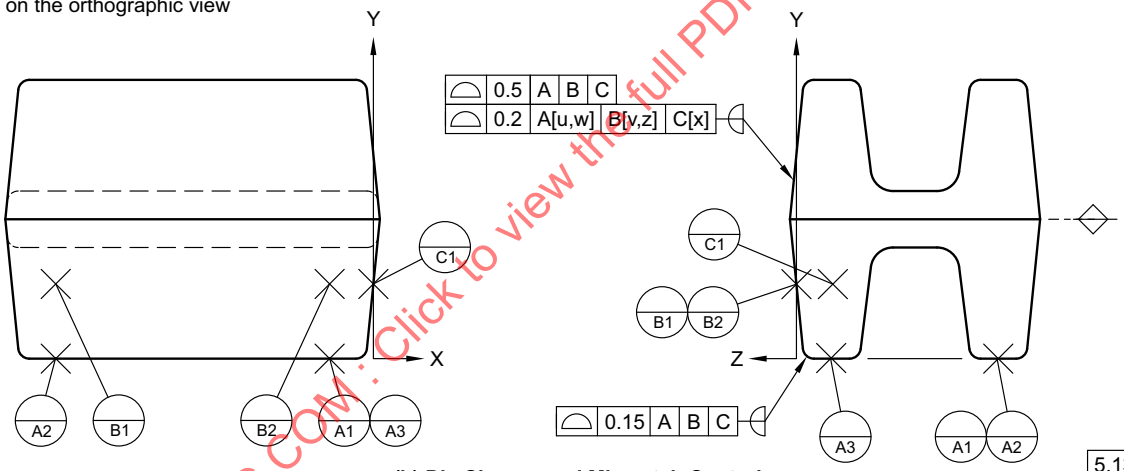
5.13
3.1.18

Means this



Mismatch is additive to dimensional limits.

This on the orthographic view



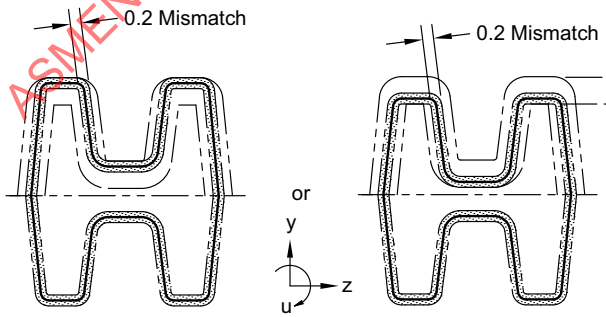
(b) Die Closure and Mismatch Control

5.13
3.1.18

Means this

0.5	A	B	C	
0.2	A[u,w]	B[v,z]	C[x]	

← Die closure
← Size, form, orientation, and location in the x and z directions mismatch.



		Degrees of Freedom Constrained					
		X	Y	Z	u	v	w
Datum Features	A		○		△		△
	B			△		△	
	C	△					

△ = Constraint for customized datum reference frame

○ = Added constraint for noncustomized datum reference frame

Figure 5-17
Parting Line Locations

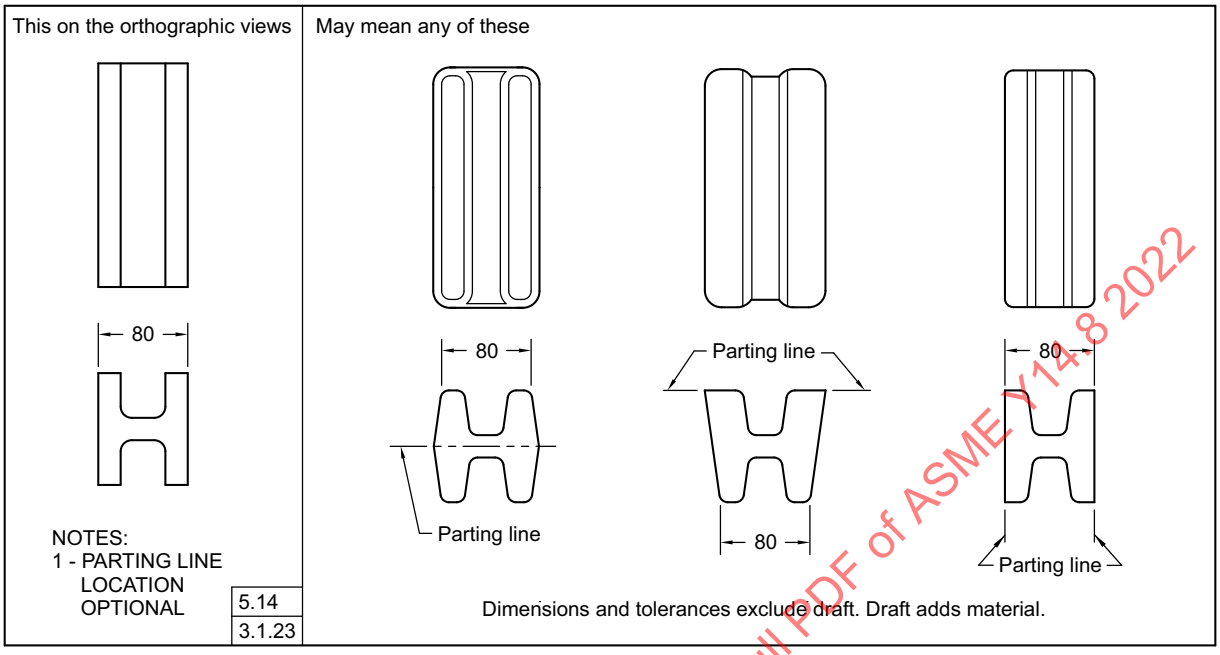


Figure 5-18
Parting Line Symbol Application

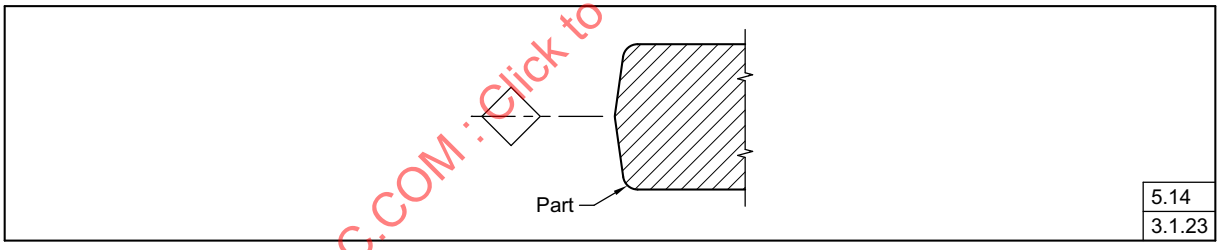


Figure 5-19
All Around This Side of Parting Line Symbol Application

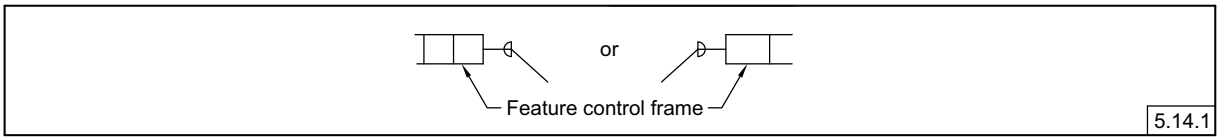


Figure 5-20
All Around This Side of Parting Line

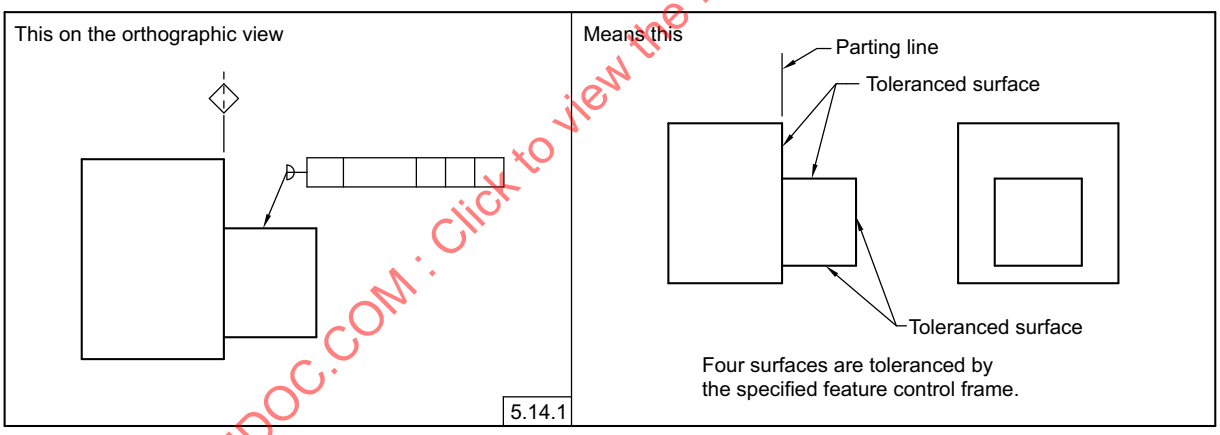


Figure 5-21
All Over This Side of Parting Line Symbol Application

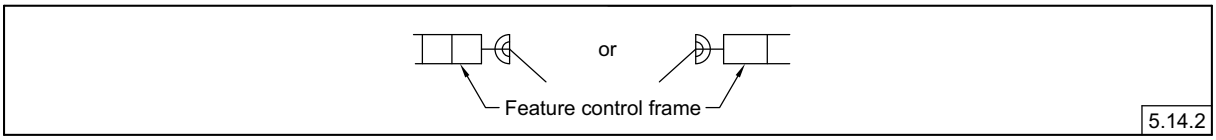


Figure 5-22
All Over This Side of Parting Line

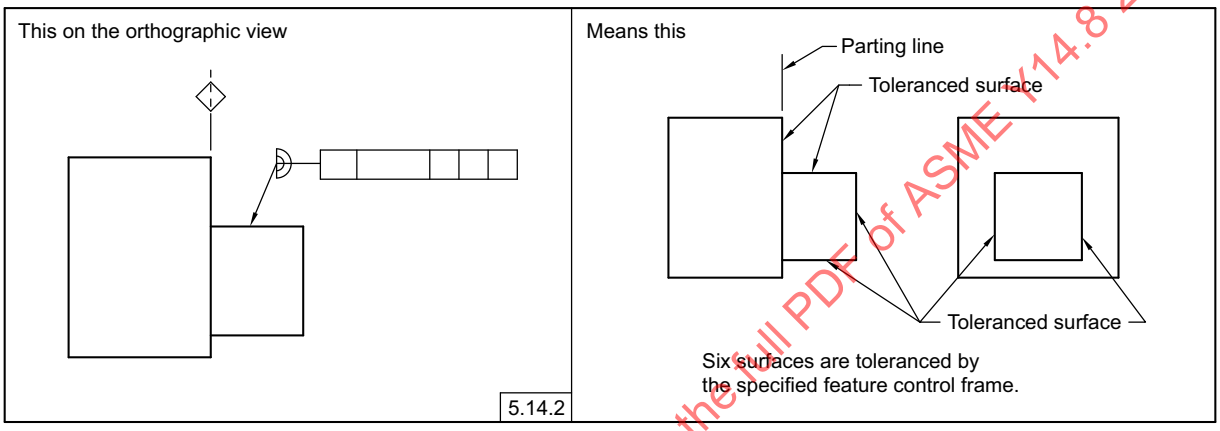


Figure 5-23
Grain Direction Specified

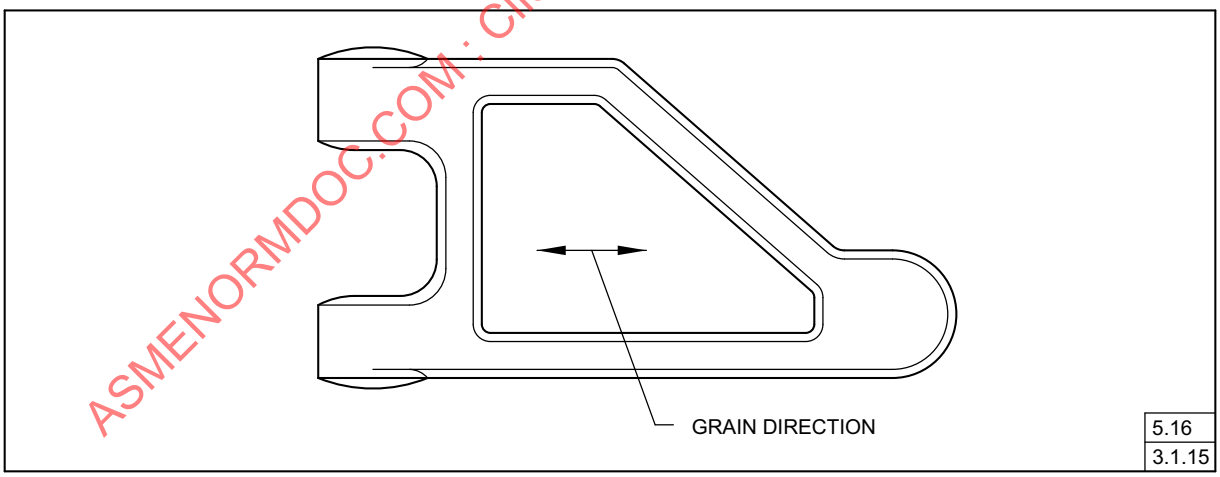
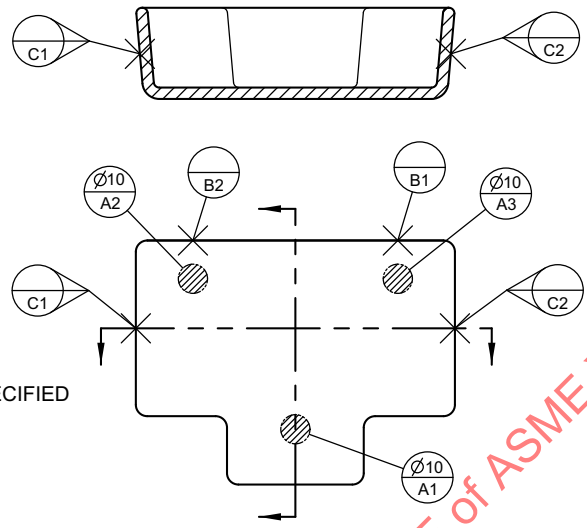


Figure 5-24
Profile Refined by Wall Thickness

This on the orthographic views



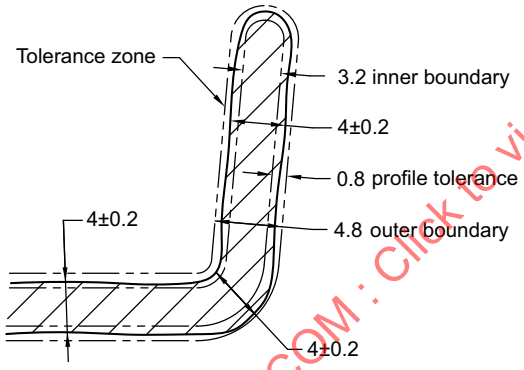
NOTES: UNLESS OTHERWISE SPECIFIED
1 - VALUES QUERIED FROM THE DIGITAL DATA FILE ARE BASIC.

⌓	0.8	A	B	C

2 - WALL THICKNESS: 4±0.2

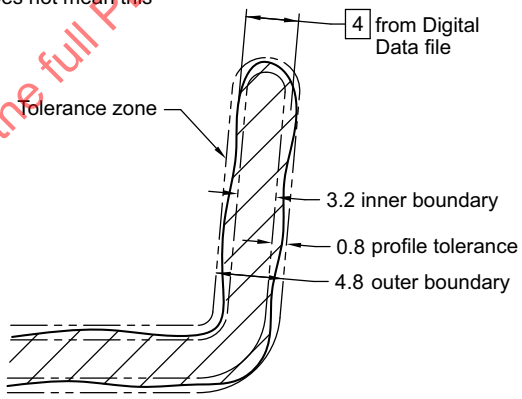
5.18
3.1.34
3.1.33

Means this



The wall thickness specification means that the surfaces of the part may float within the profile of a surface tolerance but the wall thickness dimension may not measure less than 3.8 or greater than 4.2.

Does not mean this



Without the wall thickness dimension, the thickness could vary by the total profile of a surface tolerance. In this case, it could vary from 3.2 to 4.8.

Wall thickness is not a feature of size; it is a local size between any two opposing points.

Section 6
Datum Referencing

6.1 GENERAL

This Section establishes the principles of datum referencing for cast, forged, and molded parts. It contains the criteria for selecting and designating features to establish the datum reference frame and relate it to the finished part. See Figure 6-1.

6.2 DATUM TARGETS

Because of inherent irregularities, the entire surface of some features cannot be effectively used to establish a datum. Such surfaces are common on castings, forgings, and molded parts. Datum targets and datum features may be combined to establish a datum reference frame.

6.2.1 Datum Target Location

Datum targets should be located as follows:

(a) on features produced by one segment of a die or pattern, except in the case of equalizing datum targets; see Figures 6-2 through 6-4

(b) on features opposite machining cuts that establish a subsequent machining datum reference frame; see Figure 6-5

(c) on features not subject to processing variables, such as parting lines and flash extensions

(d) on features not subsequently altered or removed

(e) with optimum spacing considering function and producibility; see Figure 6-4

(f) on noncoplanar features where the area or location requires one or more datum targets offset from the datum plane; see Figure 6-6

6.2.2 Effect of Draft and Parting Lines

The relationship of features of a part to datums established by targets can be affected by draft and parting lines. See Figure 6-7.

6.2.3 Placement of Local Dimensions

Dimensions not intended to be from the datum reference frame (local dimensions) shall be clearly indicated to apply to the mold lines. See Figure 6-8.

6.3 MACHINED DATUM FEATURES

Machined datum features should be controlled relative to a datum reference frame established from cast, forged, or molded datum features. The cast, forged, or molded surfaces selected as datum features should be surfaces that will exist after machining, maintaining dimensional traceability back to cast, forged, or molded datums.

6.3.1 Machined Datum Reference Frame

Machined datum features should be established relative to a cast, forged, or molded-part datum reference frame. The relationship between datum features of multiple datum reference frames shall be specified. See Figure 6-9.

6.3.2 Machined Datum Reference Frame Established With Optimization of Material

Where the complexity of design or variations in tooling capabilities preclude the assignment of datum targets to as-cast, as-forged, or as-molded parts, machined datum targets may be created by optimizing the material distribution in a simulated datum reference frame. Where only machined datum features are shown on the drawing, the part mass shall be optimized to establish the location of these machined datum features. See Figure 6-10.

NOTE: Use of the optimized material distribution method of creating datum targets will require close cooperation between all elements of the concurrent engineering team.

6.4 EQUALIZING DATUMS

Where it is desirable to center a casting, forging, or molded part, the application of equalizing datums should be considered; see Figure 6-4. The associated datum plane or axis may be identified by a note. Figure 6-11 illustrates the application of an equalizing datum established by two fixed datum targets, B1 and B2, and two movable target simulators, C1 and C2. Figure 6-12 illustrates the establishment of two equalizing datums resulting from datum B targets being movable. Figure 6-13 illustrates the establishment of two equalizing datums resulting from six fixed datum targets.

6.4.1 Movable Target Simulators

The readability of a drawing may be improved by indicating movable datum target simulators (per ASME Y14.5). Figures 6-3, 6-11, and 6-12 illustrate applications of the movable datum target symbol. Where movable target simulators are used, regardless of material boundary (RMB) applies. Where the direction of the simulator motion is not indicated, a direction normal to the undrafted true surface is used.

6.5 DATUM TARGETS AND PROFILE TOLERANCING

Where a profile tolerance has been applied to a surface containing datum targets, the true profile of the surface and the extremities of the surface that contact the datum target simulators are coincident. Unless otherwise specified, the profile tolerance zone is equally disposed about the true profile. See Figure 6-14.

The underlying concepts are as follows:

(a) The profile tolerance zone is located and oriented to the datum reference frame.

(b) The datum reference frame is located and oriented by the datum target simulators.

(c) When the part is mated with the datum target simulators, the remaining surface of the considered feature is then related to the tolerance zone.

Figure 6-14, illustration (a) shows the workpiece with a midpoint surface element (neither a high point nor a low point) coincident with the datum target simulators. All surface elements shown are contained within the profile tolerance zone.

Figure 6-14, illustration (b) shows the same workpiece with a high point contacting the datum target simulators if the drawing specifies them in a different location. Several portions of the surface elements are now outside of the profile tolerance zone.

In Figure 6-14, illustrations (a) and (b), the true profile of the tolerance zone is aligned to the datum target simulators. The tolerance zone is then related to the feature surface by the contact of the workpiece surface with the datum target simulators. The feature's form error does not exceed 0.8; however, the actual contact of the datum target simulator and the workpiece surface sets the location of the true profile relative to the workpiece. The use of targets to establish a datum reference frame and the variation from workpiece to workpiece may result in having some portions of the actual surface extend beyond the tolerance zone. When a balanced surface relative to the targets is desired, this method should be considered.

Allowable variation of the actual workpiece surface coincident with the target area is one-half the specified profile tolerance for an equal bilateral distribution, or the amount of the specified profile tolerance in the direction that removes material specified in an unequal bilateral control. Target lines and datum target points act in a similar fashion, except that the variation restriction applies to areas with the control being along the line or point of the simulator. The remaining surface variation outside the target area or line shall be contained within the total profile tolerance zone as established from the true profile related to the datum target location.

Profile of a surface without a datum reference frame specified could be applied to datum feature A and would control the surface variation independent of, and prior to, the establishment of any datum. This may be applied in cases when a forged or cast part is being evaluated to determine whether sufficient material is available to produce a machined part. An "ALL OVER" profile of a surface control without datum features referenced could also be applied, when the tolerance zone for the entire surface is intended to be best-fit to the workpiece, rather than fixed relative to a datum reference frame.

Figure 6-1
Datum Targets Establishing a Datum Reference Frame

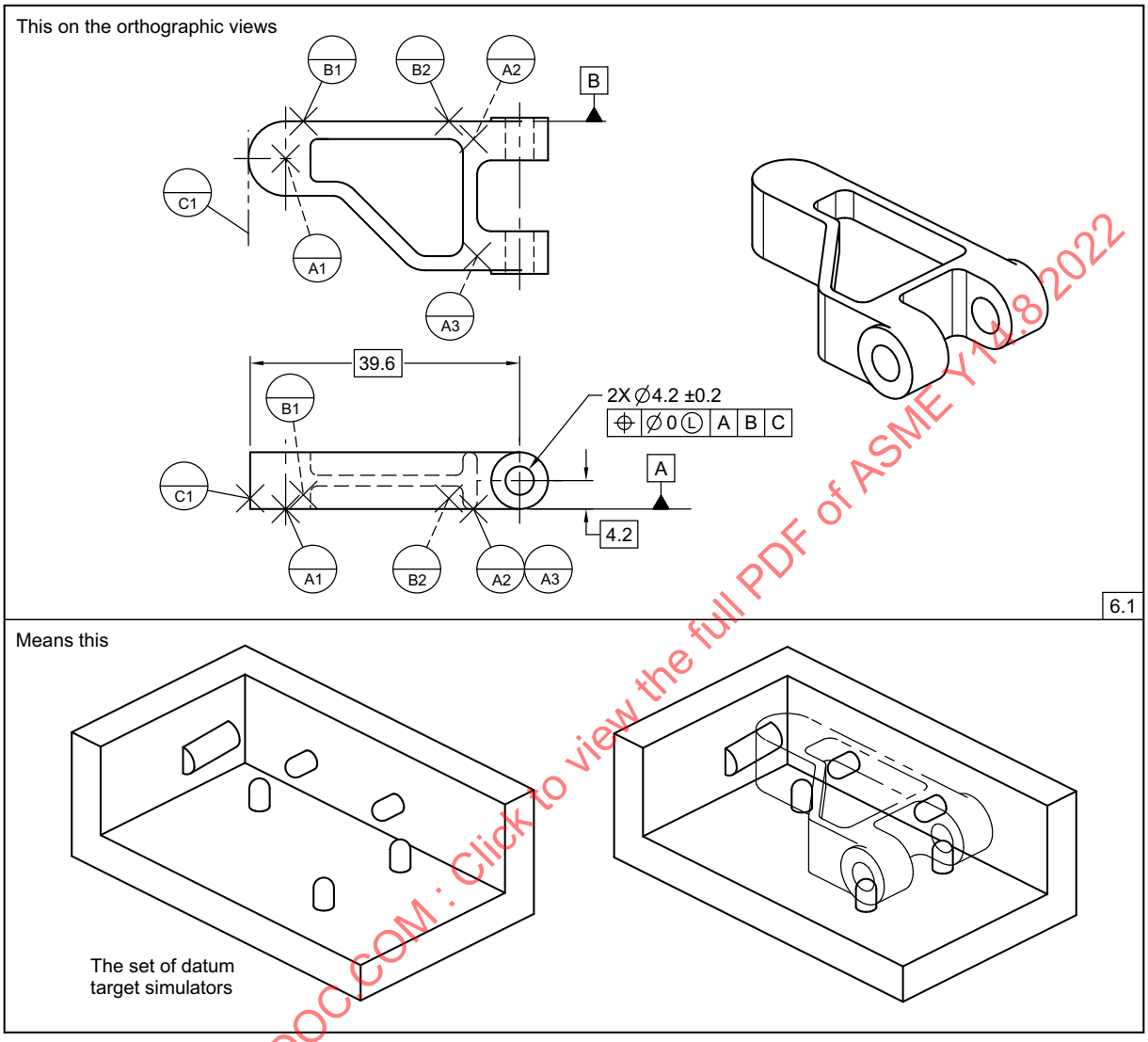


Figure 6-2
Datum Targets Within the Same Die Segment

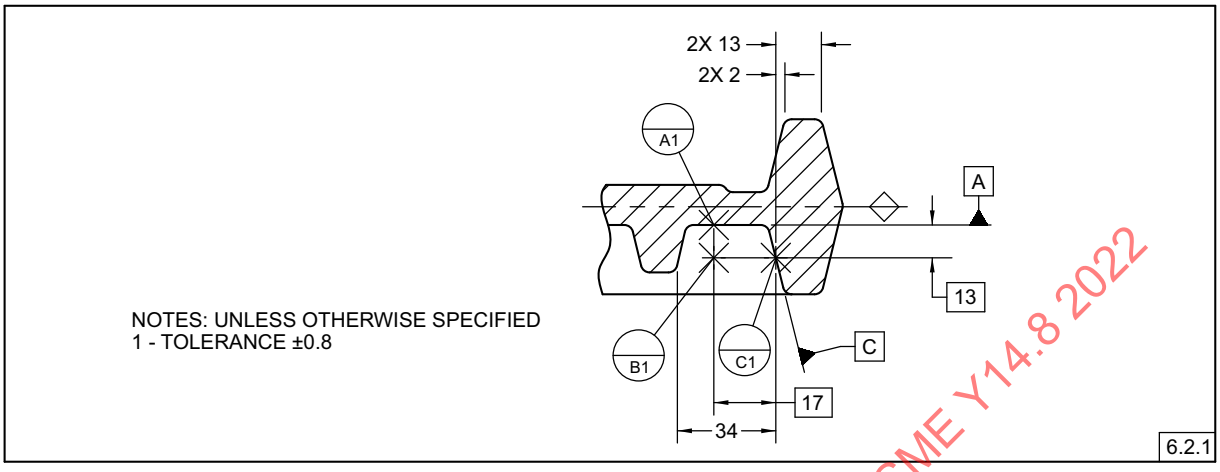


Figure 6-3
Equalizing Datums

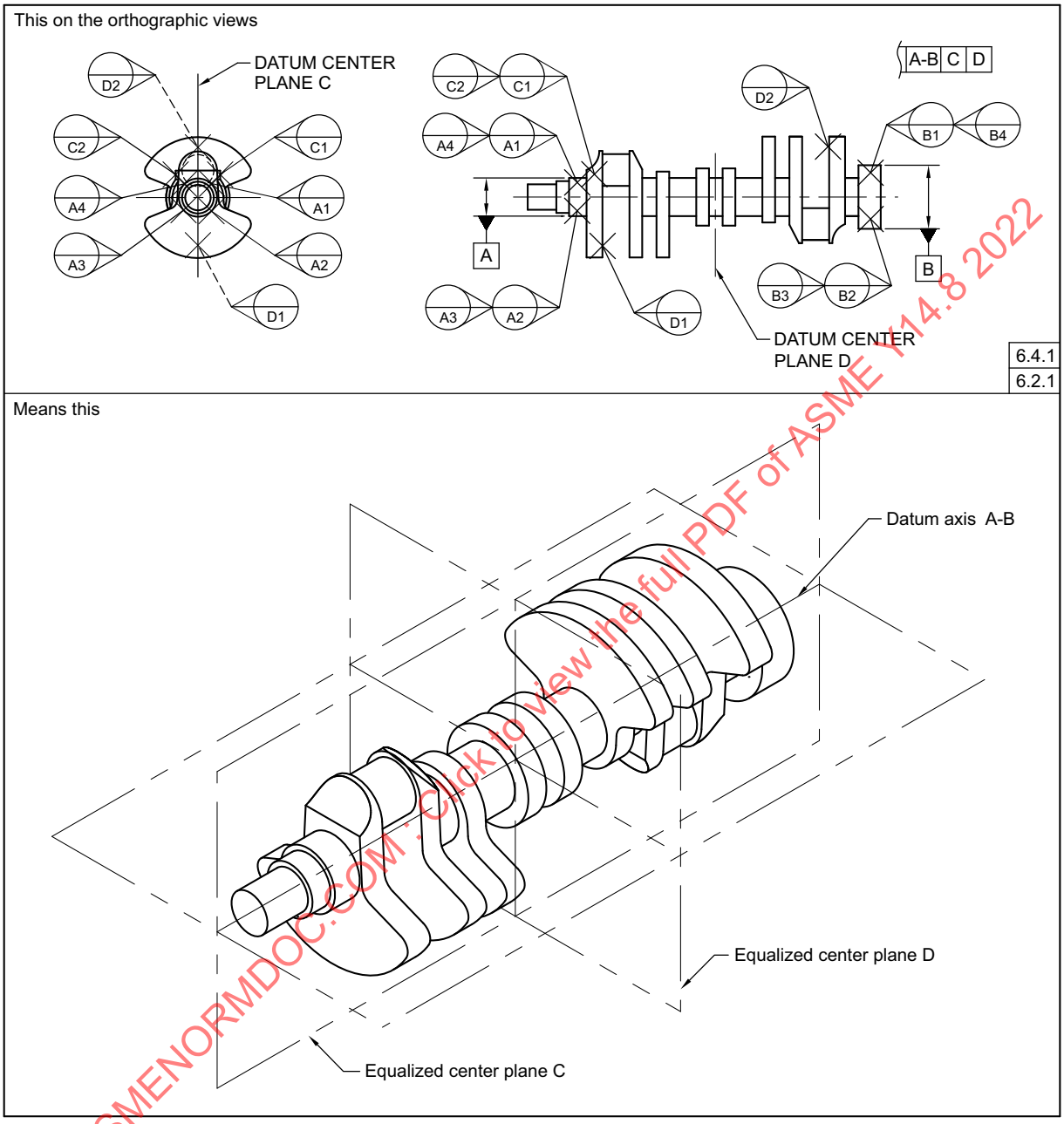


Figure 6-4
Optimum Location of Equalizing Datum Targets

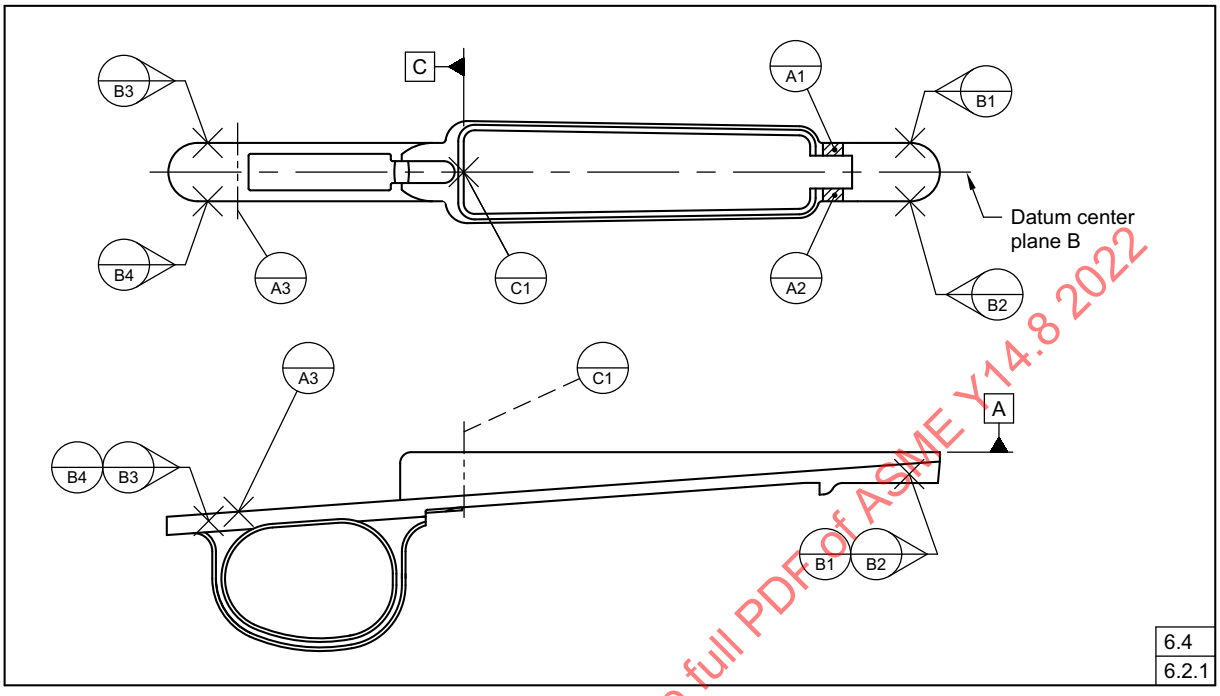


Figure 6-5
Datum Targets Located Opposite Machined Surfaces

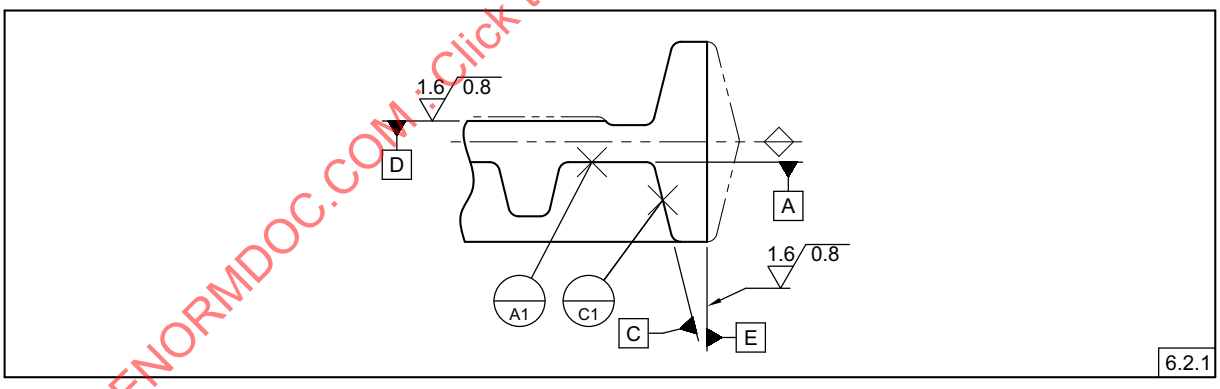


Figure 6-6
Datum Targets Offset From the Datum Plane

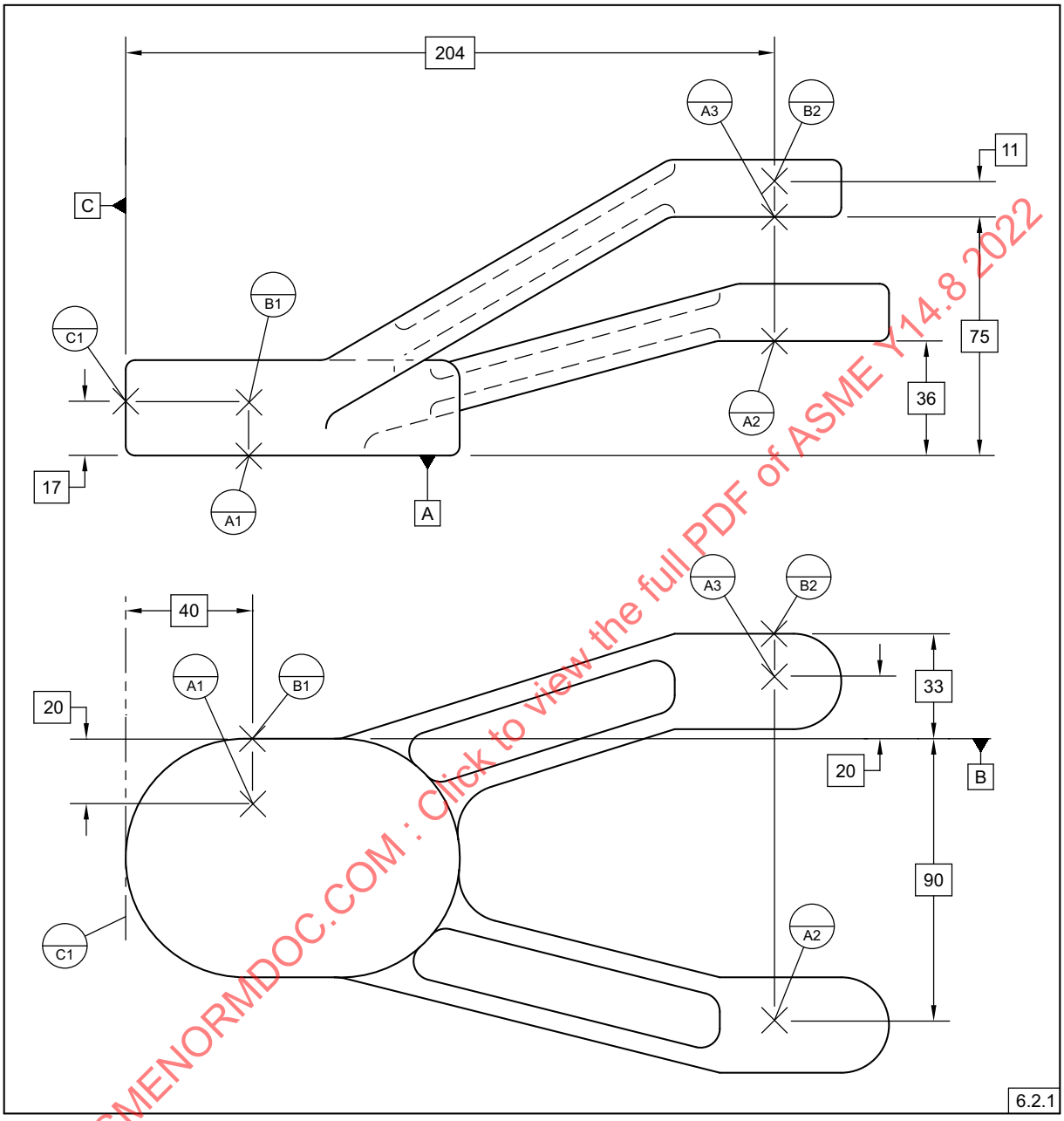


Figure 6-7
Effect of Draft and Parting Line on Datums

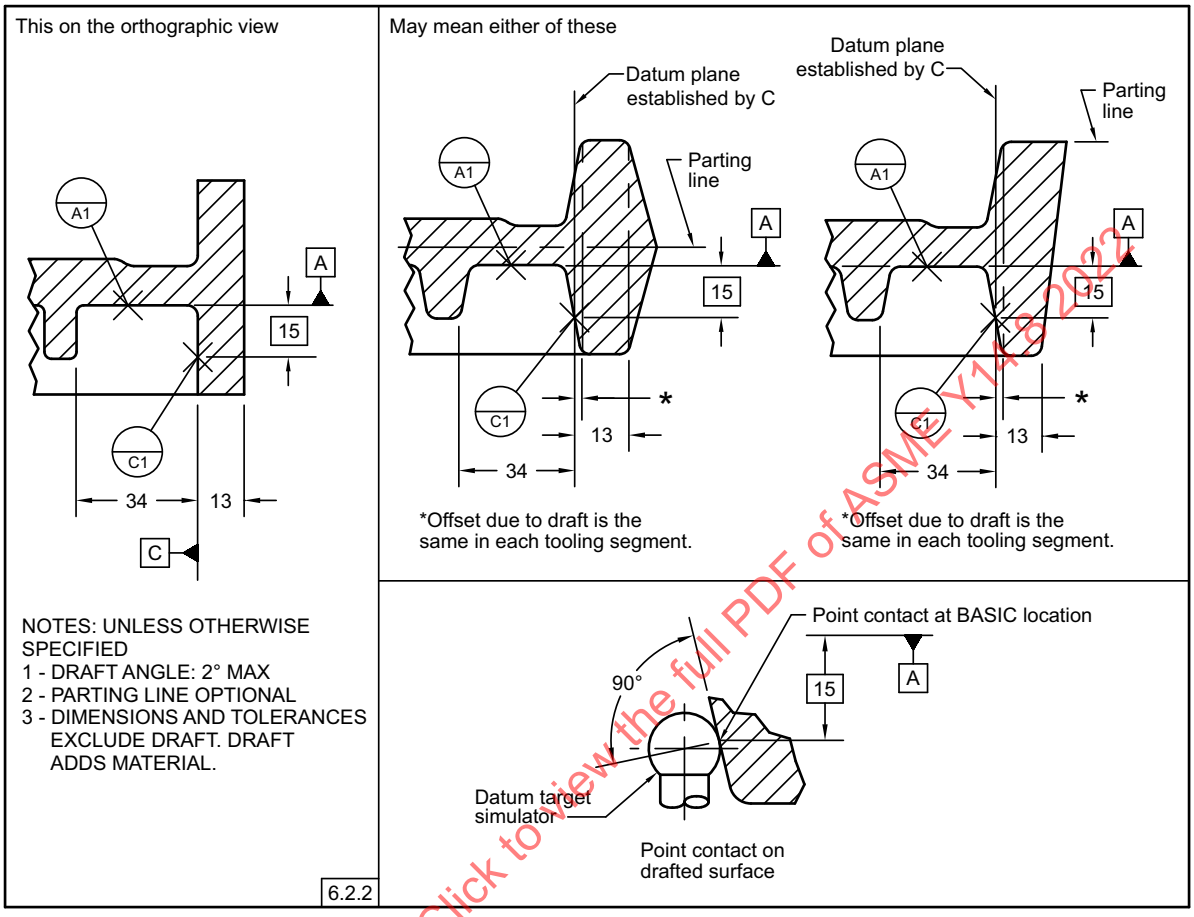


Figure 6-8
Placement of Local Dimensions Affected by Draft

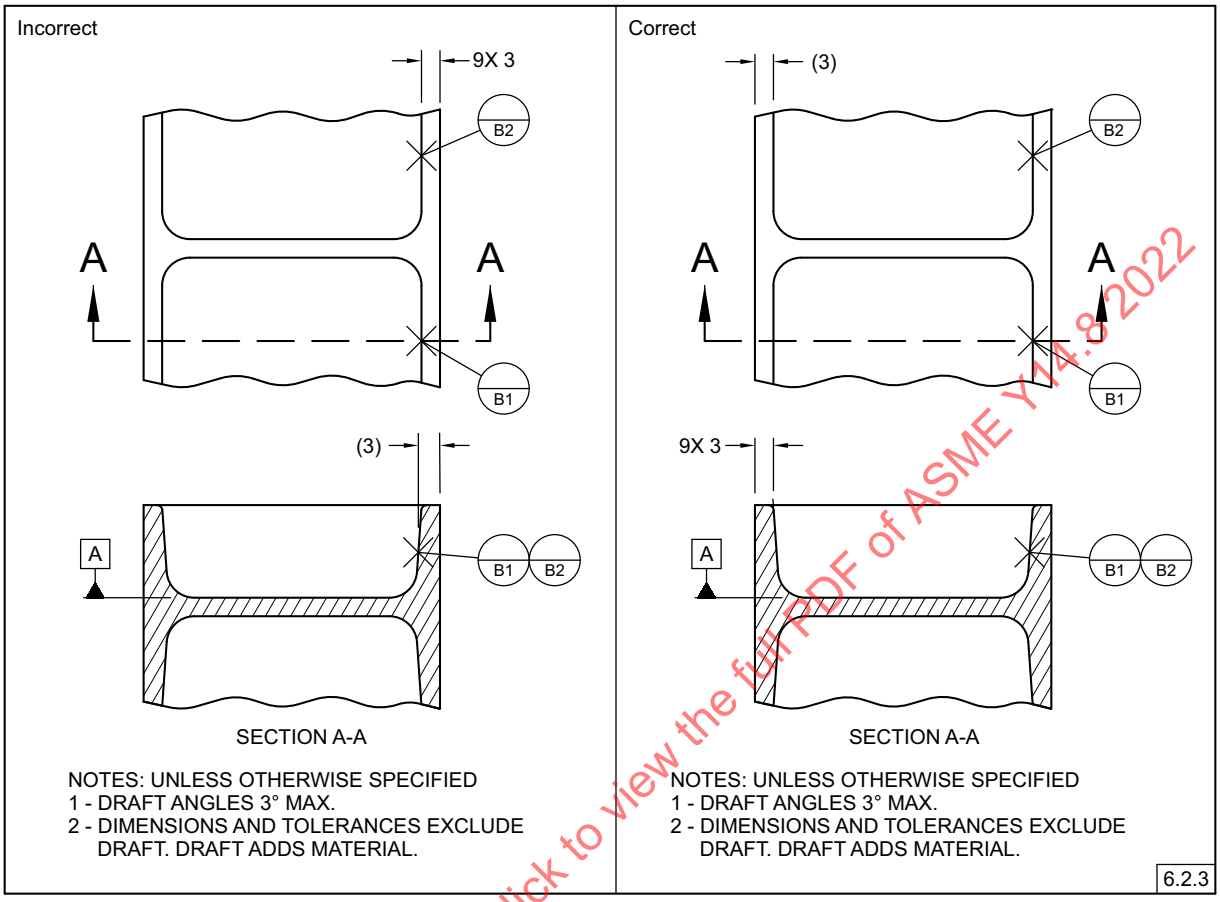


Figure 6-9
Machined Datum Features Located From Cast, Forged, or Molded Datum Features

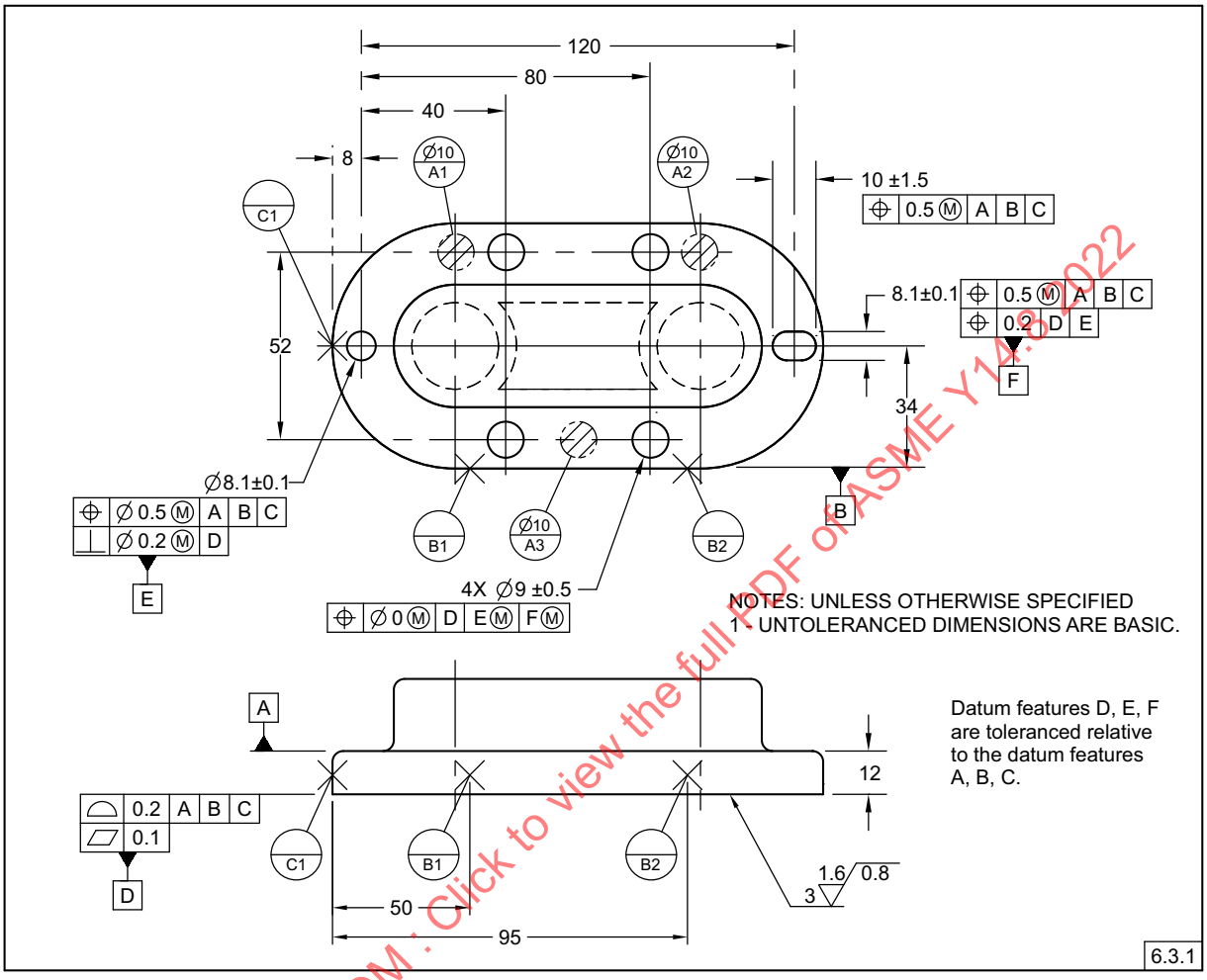


Figure 6-10
Machine Centers Establishing a Datum Axis

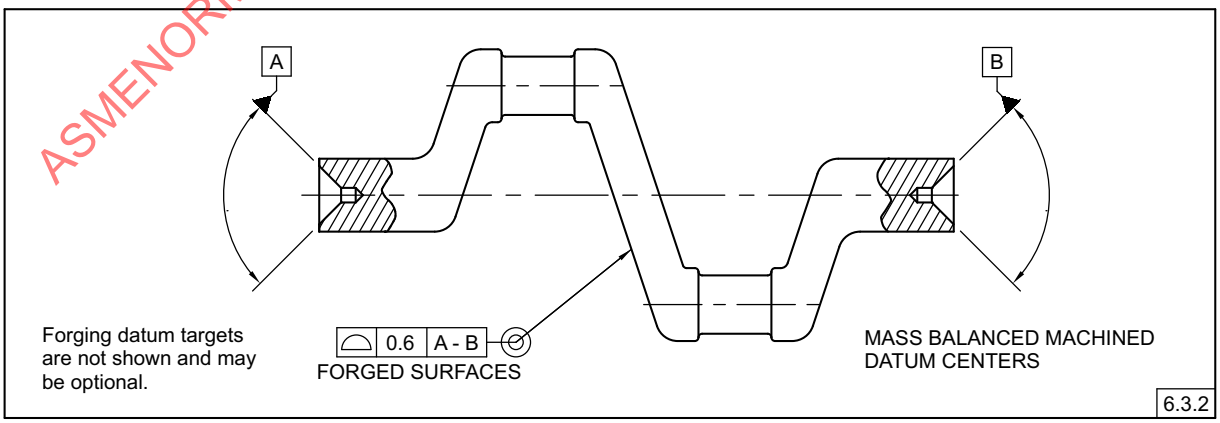
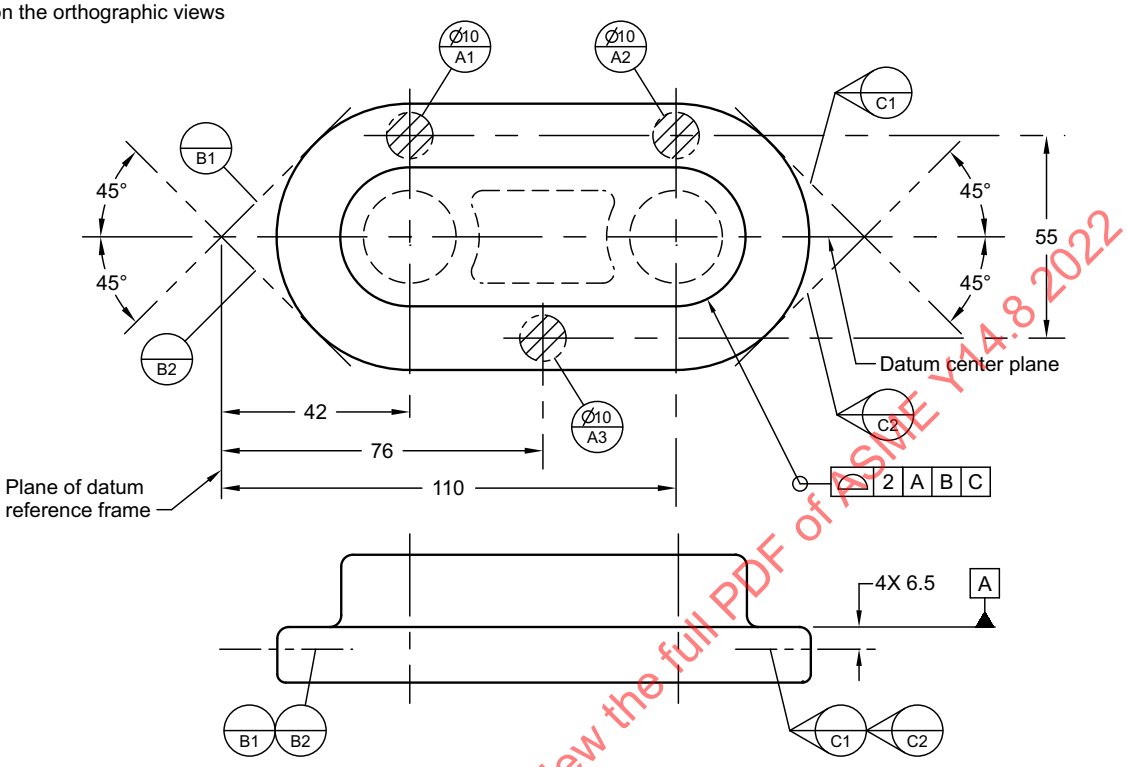


Figure 6-11
Movable Datum Target Simulators Establishing a Datum Center Plane

This on the orthographic views



NOTES: UNLESS OTHERWISE SPECIFIED
1 - UNTOLERANCED DIMENSIONS ARE BASIC.

6.4.1
6.4

Means this

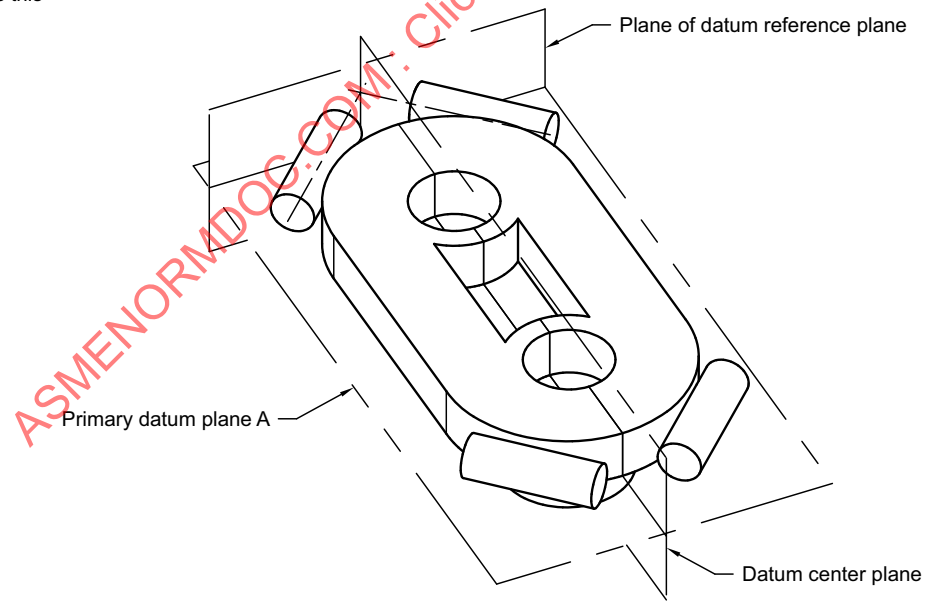


Figure 6-12
Movable Datum Target Simulators Establishing Two Datum Center Planes

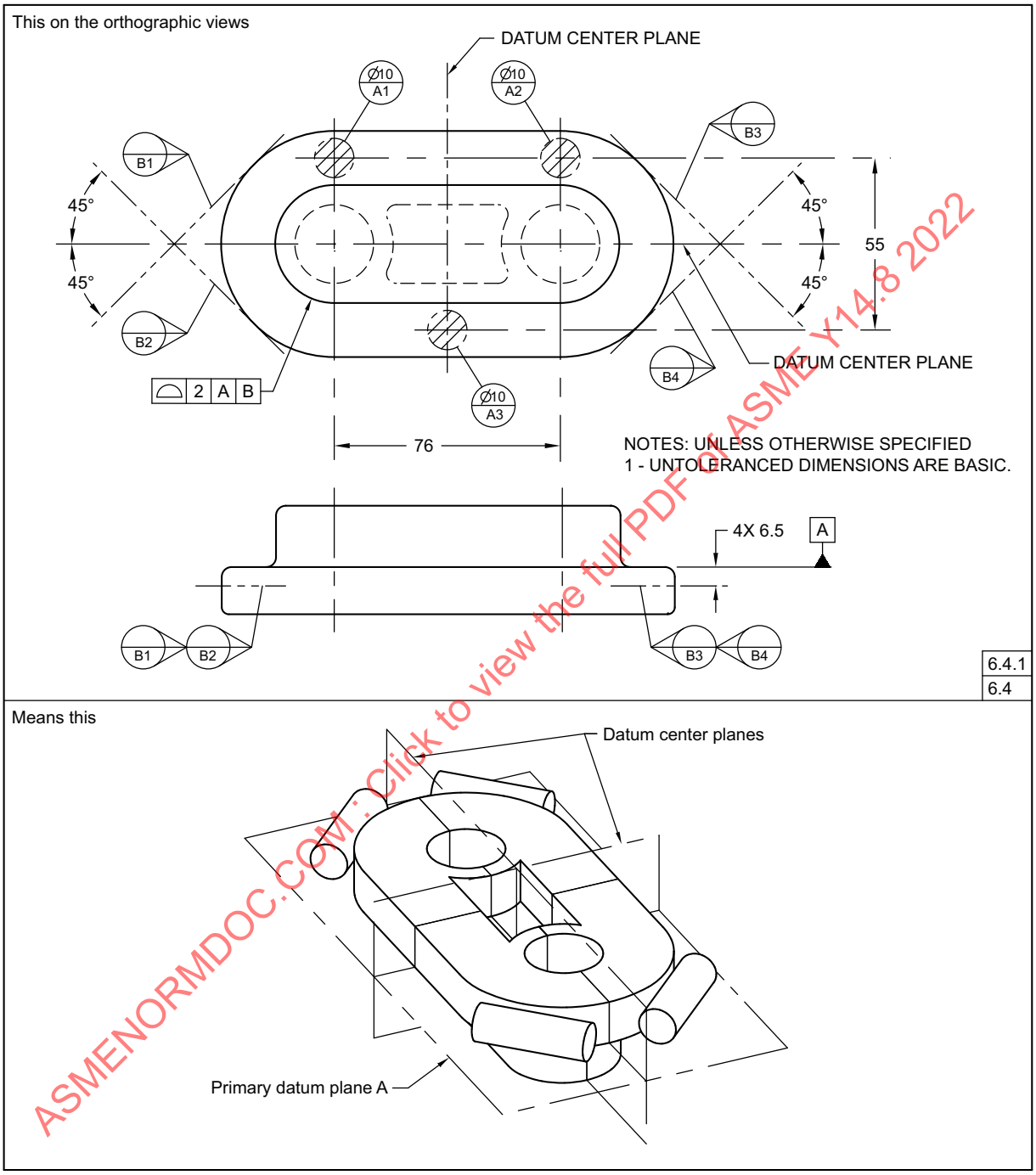


Figure 6-13
Equalized Datums Established by Fixed Datum Target Simulators

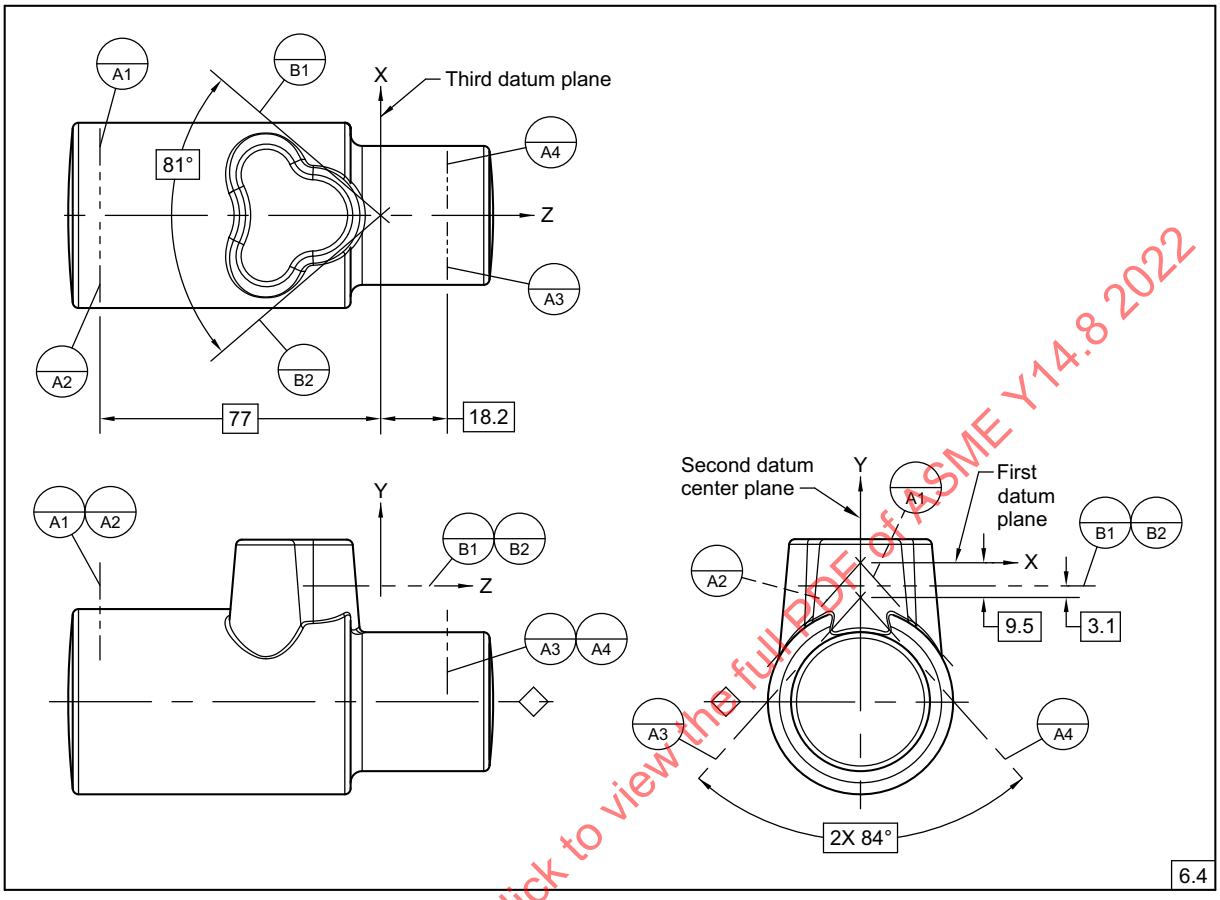
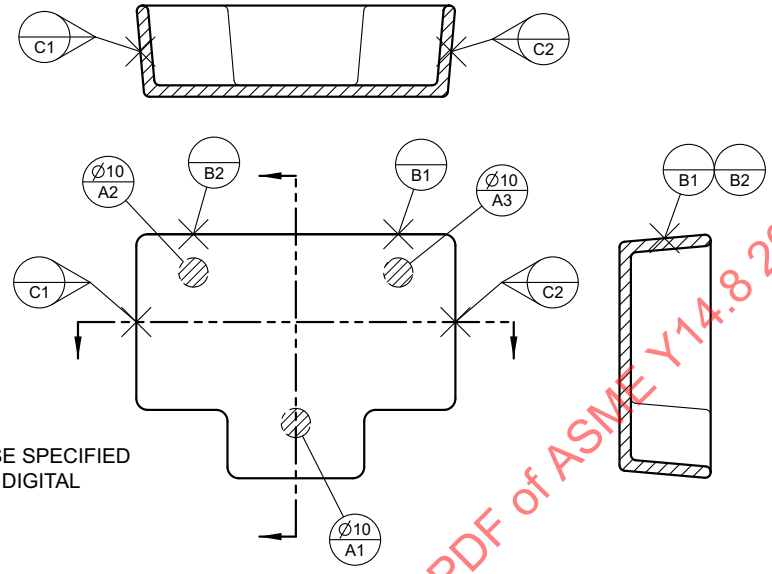


Figure 6-14
Datum Targets and Profile Tolerancing

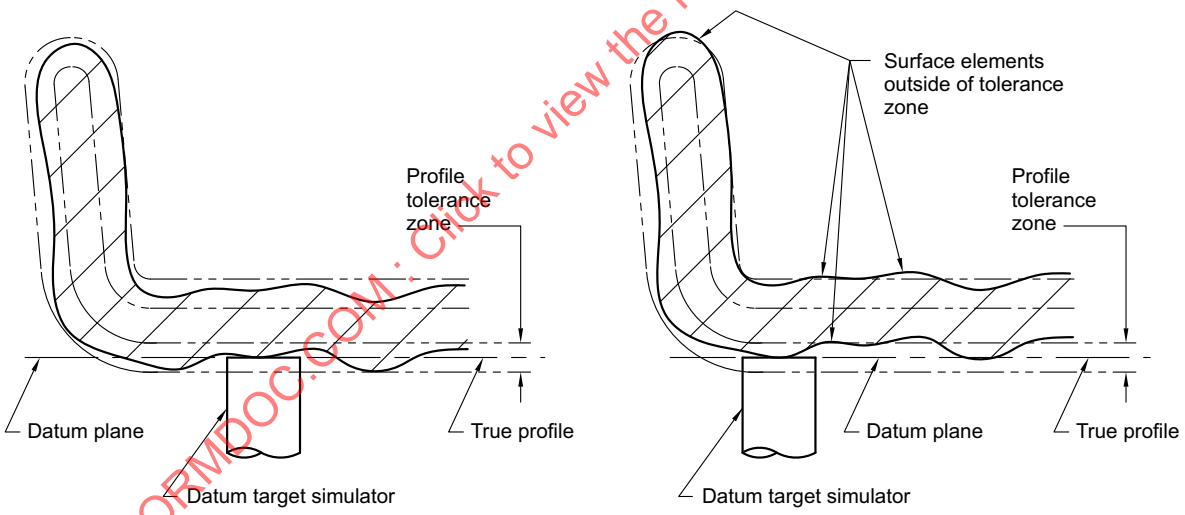
This on the orthographic views



NOTES: UNLESS OTHERWISE SPECIFIED
1 - VALUES QUERIED FROM DIGITAL DATA FILE ARE BASIC.
2 - ⌓ | 0.8 | A | B | C

6.5

Two possibilities depending on where the datum target simulator contacts the part



(a) Target Residing on a Midpoint Surface Element

(b) Target Residing on a High Point Surface Element