PROGRAM

- Welcoming remarks – 8:30 A.M.
    - Archie R. Anderson, Seminar Leader
      Immediate Past Chair, Y14/SC 5
      Owner, Dimensional Dynamics, LLC
    - Frank Bakos, Chairman, Y14 Committee
      Consultant

- Introduction to Y14 Standardization – Archie R. Anderson
    - Companion Standards Y14.5.1 and Y14.5.2
    - Associated Standards Y14.43 and Y14.41

- Development process and overview of the changes to Y14.5
  Presenter: Archie R. Anderson

- Changes to Section 1 [Scope, Definitions, and General Dimensioning]
  Presenter: Martin P. Wright, Design Engineer Senior Staff, Lockheed Martin Aeronautics Company
    - Q & A

- Changes to Section 2 [General Tolerancing and Related Principles]
  Presenter: Norman W. Cutler, President, Dimensional Management Inc.
    - Q & A

- Section 4 [Datum Reference Frames]
  Presenters:
  Alvin G. Neumann, President/Director, Tech Consultants Inc.
  Donald E. Day, President, Tec-Ease, Inc.
    - Q & A

- Section 5 [Tolerances of Form]
  Presenter: Don W. Shepherd, PhD, Owner, Shepherd Industries
    - Q & A

- Section 6 [Tolerances of Orientation]
  Presenter: Don W. Shepherd, PhD
    - Q & A

- Section 7 [Tolerances of Location]
  Presenter: Bruce A. Wilson, Technical Fellow, The Boeing Company
    - Q & A

- Section 8 [Tolerances of Profile]
  Presenter: Archie R. Anderson
    - Q & A

- Section 9 [Tolerances of Runout]
  Presenter: Don W. Shepherd, PhD
    - Q & A

- Questions and Answers
  Moderator: Archie R. Anderson

- Closing Remarks – Archie R. Anderson

- Expected Closing Time: 5 P.M.

Development Process and Overview of Y14.5-2009 Changes

ASME

SETTING THE STANDARD

Archie R. Anderson
Chairman Y14.5 - 2009

Development Process

- ASME Y14.5 Dimensioning and Tolerancing is a subcommittee of Y14 Engineering Drawing Practices under the auspices of the American Society of Mechanical Engineers (ASME).

ASME
SETTING THE STANDARD

Development Process

- Y14 has 25 active subcommittees covering all aspects of Engineering Drawing Practices i.e. Drawing Sheet Size and Format, Pictorial Drawings, Dimensioning and Tolerancing, Certification, Digital Product Definition Data Practices, and many more.

Development Process

- All standardization is voluntary and the subcommittee consists of individuals from industry, academia, National Laboratories, self-employed individuals and the Department of Defense.

Development Process

- Industries represented
  - Aerospace
  - Automotive (active and retired)
  - Earth Moving Machinery Manufactures
  - Office Furniture (retired)
  - Photographer / Office Reproduction (retired)
  - Dimensional Analysis Software Development

Development Process

- National Laboratories
  - Los Alamos National Laboratory

- Universities
  - Ohio University, Athens, Ohio
  - Purdue University

ASME
SETTING THE STANDARD

Development Process

- Development of Y14.5-2009 started in January 1995 in Sarasota, Florida.
- The Y14.5 subcommittee meets in May and October each year at different cities around the country.

Development Process

- The process of the revising Y14.5 began with resolving deferred comments from the 1994 Public Review and proposals from the membership of Y14.5.
- The subcommittee was organized in Working Groups with a leader appointed for each WG.

Development Process

- Working Group Leaders
  - Section 1 Marty Wright – Lockheed Martin Aeronautics Co.
  - Section 2 Norm Cutler – Dimensional Management, Inc. (Retired Poloraid)
  - Section 3 Archie Anderson – Dimensional Control Systems, Inc. / Dimensional Dynamics, LLC (Retired GM)

ASME
SETTING THE STANDARD

Development Process

- Working Group Leaders (Cont.)
  - Section 4 Al Neumann – Technical Consultants, Inc.
  - Section 4 – Assisted by Don Day-Tec-Ease, Inc (Formerly Monroe Community College)
  - Sections 5,6,9 Dr. Don Shepherd – Shepherd Industries (Formerly Northern Illinois University)

ASME
SETTING THE STANDARD

Development Process

- Working Group Leaders (Cont.)
  - Section 7 Bruce Wilson – The Boeing Company
  - Section 8 Dennis Karl – Karl Engineering (Retired Ford)

Development Process

- Working Group Leaders (Cont.)
  - Appendix A, C, D Archie Anderson
  - Appendix B Paul Drake Jr. – MechSigma Consulting, Inc. (Formerly Texas Instrument)
  - Appendix E Frank Bakos – Frank Bakos Associates (Formerly Kodak)

Overview of Revisions

- One of the major revisions is the reformatting of the Standard, the order of the sections and material within the sections without reversing any of the 1994 principles.

  - Section 1 Scope, Definitions, and General Dimensioning

  - Section 2 General Tolerancing and Related Principles

  - Section 3 Symbology

  - Section 4 Datum Reference Frames

Overview of Revisions

- Section 5 Tolerances of Form
- Section 6 Tolerances of Orientation
- Section 7 Tolerances of Location
- Section 8 – Tolerances of Profile
- Section 9 – Tolerances of Runout

Overview of Revisions

- Appendix A – Principle Changes and Improvements
- Appendix B – Formula for Positional Tolerancing
- Appendix C – Form, Proportion, and Comparison of Symbols
- Appendix D – Former Practices
- Appendix E – Decision Diagram for Geometric Controls

Overview of Revisions

- The sections were rearranged to make the flow of the material flow from the basic concepts to the more complex.
- Principle Revisions and Improvements
  - Revised definitions
  - New terms
  - Fundamental rules added

Overview of Revisions

Principle Revisions and Improvements (Cont)

- New symbology
- Degrees of Freedom emphasized for datums
- Datum selection example on a small assembly

Overview of Revisions

Principle Revisions and Improvements (Cont)

- Ability to add the value of a datum simulator in the feature control frame
- Datum shift added
- Added restrained parts
- Simplified Orientation explanation

Overview of Revisions

Principle Revisions and Improvements (Cont)

- Major reformatting of section on location tolerances
- Expanded coverage of composite positional tolerancing
- Expanded coverage of profile tolerancing

ASME
SETTING THE STANDARD

Overview of Revisions

- There will be an opportunity for questions following each presentation. The answers and comments are the opinion of the presenters and do not constitute an official ASME interpretation of the standard.

SETTING THE STANDARD

2nd DRAFT

Section 1
Scope, Definitions and General Dimensioning

Martin P. Wright

Section 1 Work Group Leader

Definition Consolidation

The following are terms that have been traditionally defined within the body of the standard, but now appear in paragraph 1.3 with a cross-reference to the paragraph where the definition is found.

- **Angularity**
- **Circularity (Roundness)**
- **Coaxiality**
- **Concentricity**
- **Coplanarity**
- **Cylindricity**
- **Datum Reference Frame**
- **Diameter, Average**
- **Feature Control Frame**
- **Feature-Relating Tolerance Zone Framework (FRTZF)**
- **Free-State Variation**
- **Flatness**
- **Parallelism**
- **Pattern-Locating Tolerance Zone Framework (PLTZF)**
- **Perpendicularity**
- **Position**
- **Profile**
- **Runout**
- **Simultaneous Requirement**
- **Straightness**
- **Statistical Tolerancing**
- **Symmetry**
- **True Profile**

ASME SETTING THE STANDARD

Revised Definitions

- The following terms have been revised for editorial clarification and/or to accommodate new practices added under this revision of the standard.

  - **Datum**
  - **Datum Feature**
  - **Datum Feature Simulator**
  - **Datum Target**
  - **Dimension**
  - **Dimension, Basic**
  - **Dimension, Reference**
  - **Envelope, Actual Mating**
  - **Feature**
  - **Feature Axis**
  - **Feature, Center Plane of**
  - **Feature of Size**
  - **Plane, Tangent**
  - **Resultant Condition**
  - **Size, Actual Local**

Newly Added Definitions

- The following new terms have been added to support new practices added under this revision of the standard.

- **Boundary, Least Material (LMB)**
- **Boundary, Maximum Material (MMB)**
- **Complex Feature**
- **Envelope, Actual Minimum Material**
- **Free State**
- **Non-Uniform Tolerance Zone**
- **Pattern**
- **Regardless of Material Boundary (RMB)**
- **Uniform Tolerance Zone**

- New and revised terms will be illustrated under the sections where the terms are applied.

ASME SETTING THE STANDARD

Fundamental Rules - Revised

- Updated para 1.4(b) to include "Values may be expressed in an engineering drawing or in a CAD product definition set (see ASME Y14.41)".

- For para. 1.4(f), the structure of the rule was changed so that the rule is stated first and then examples are given.

Fundamental Rules - New

- Added para 1.4(k) “A zero basic dimension applies where axes, centerplanes or surfaces that are shown congruent on a drawing and geometric controls establish the relationship between the features.”

- Added para 1.4(p) “Where a coordinate system is shown on the drawing, it shall be right-handed unless otherwise specified. Each axis shall be labeled and the positive direction shown.”

Dimensioning - Dimensions Not to Scale

- Para. 1.7.9 Dimensions Not to Scale. Subparagraph (a) Added a requirement for indicating that a basic dimension is not shown to scale:

"Where a basic dimension symbol is used, the line is placed beneath the symbol."

Dimensioning - Rounded Ends and Slotted Holes

- Methods for dimensioning Rounded Ends and Slotted Holes have been consolidated under paragraph 1.8.4 and Figures 1-29 & 1-30.

Fig. 1-29 Slotted Holes

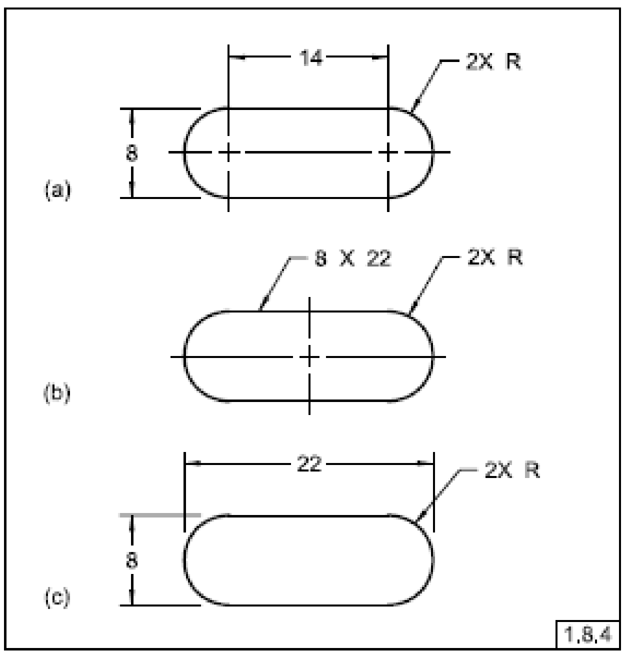
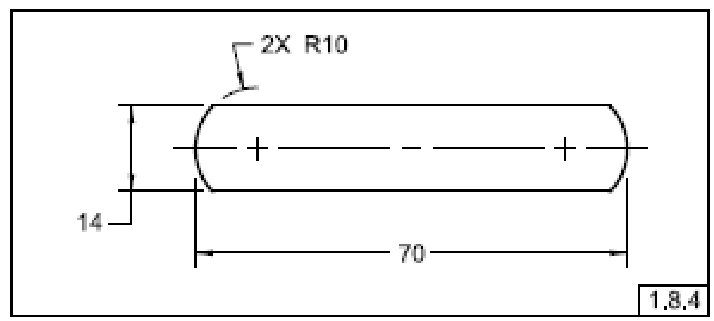


Fig. 1-30 Partially Rounded Ends



Dimensioning - Round Holes

- Para 1.8.10 Round Holes. Added a statement that additional clarification may be required when indicating THRU holes for multiple features.

"**Where it is not clear that a hole goes through, the notation THRU follows a dimension. <u>Where multiple features are involved, additional clarification may be required</u>.**"

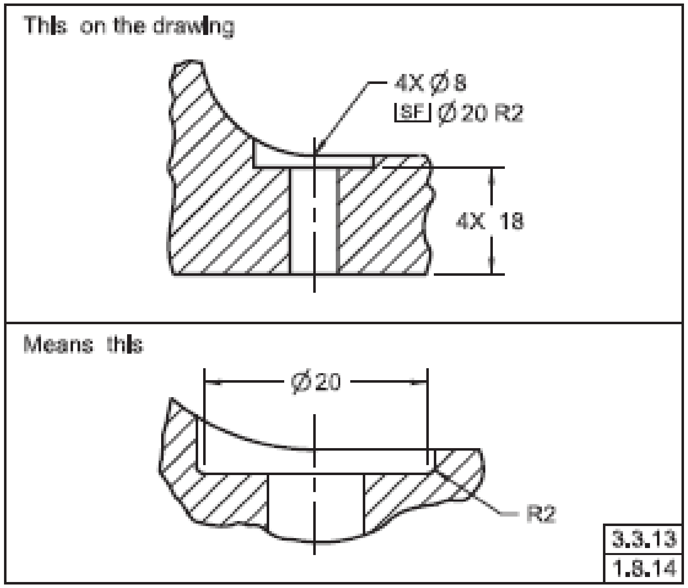
ASME
SETTING THE STANDARD

Dimensioning - Spotfaces

- Para 1.8.14 Spotfaces was updated to add the following:
  - **Where applicable, a fillet radius may be indicated for the spotface.**
  - **In some cases, such as with a through hole, a notation may be necessary to indicate the surface to be spotfaced.**

- Figure 1-41 has been updated to illustrate the new symbol for indicating a spotface.

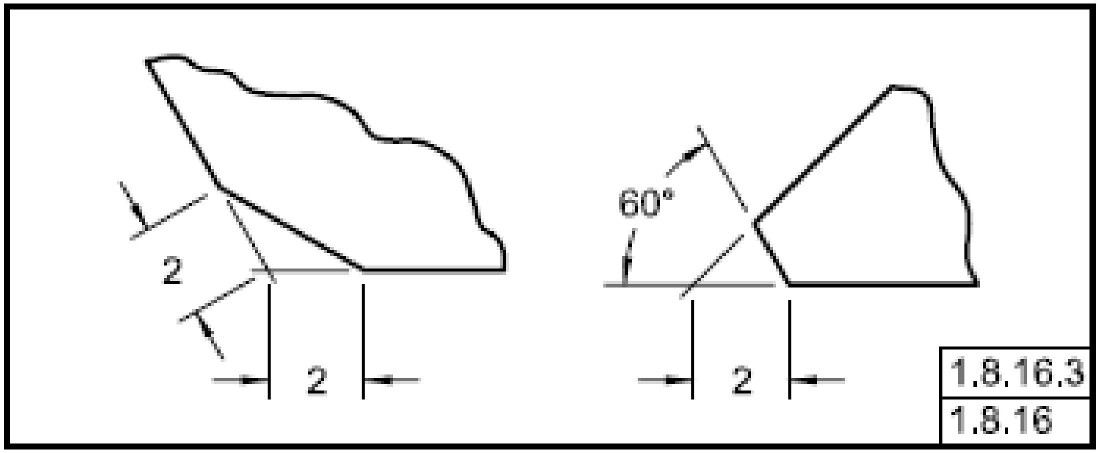
Fig. 1-41 Spotfaced Holes



Dimensioning - Chamfers

- Para 1.8.16.3 This new paragraph introduces two methods of dimensioning chamfers for surfaces intersecting at angles other than right angles.

Fig. 1-45 Chamfers Between Surfaces at Other Than 90°



Questions??

Section 2
General Tolerancing and Related Principles

Norman W. Cutler
President

Dimensional Management Inc.

10/2009

Paragraph 2.1.1 has been revised to emphasize /encourage the use of basic dimensions and geo- metric tolerancing as the preferred method of controlling the form, orientation, and location of features.

The use of direct tolerancing methods, limit dimensioning and plus minus tolerancing should be used to control the size of features only.

2.1.1 Application

Tolerances may be expressed as follows:

(a) as direct limits or as tolerance values applied directly to a dimension. See para. 2.2

(b) as a geometric tolerance as described in Sections 5 through 9.

(c) in a note or table referring to specific dimensions.

(d) as specified in other documents referenced on the drawing for specific features or processes.

(e) in a general tolerance block referring to all dimensions on a drawing for which tolerances are not otherwise specified.

Paragraph 2.1.1.2   Basic Dimensions paragraph has been added for more clarification on basic dimensions.

2. 1.1.1 Basic dimensions,   Basic dimensions may be indicated on the drawing in the following ways:

(a) applying the basic dimensions symbol to each of the basic dimensions . See fig. 7-1, illustrations (a) and (b).

(b) specifying on the drawing (or in a document reference on the drawing) a general note such as: untoleranced dimension are basic. See fig. 7 illustration (c)

Note: Where using this method a plus/ minus general tolerance is not allowed.

(c) For specifying and querying basic dimensions on models or digital drawings with models, see ASME Y. 14.41

Paragraph 2.1.1.4 on implied 90 degree angles has been expanded to include basically dimensioned angles controlled by geometric tolerancing

2.1.1.4 Implied 90° or 0° Basic Angle. Where center lines and surfaces are depicted on 2-D orthographic engineering drawings intersecting at right angles or parallel to each other and basic dimensions or geometric tolerances have been specified, implied 90° or 0° basic angles are understood to apply. The tolerance on the feature associated with these implied 90° or 0° basic angles is provided by feature control frames that govern the location, orientation, profile, or runout of features. See para. 1.4(J) and(K).

Zero's for infinities

Paragraph 2.4 on interpretation of limits has been revised to eliminate the discussion on deviation outside the limiting value of a dimension.

2.4 INTERPRETATION OF LIMITS

All limits are absolute. Dimensional limits regardless of the number of decimal places, are used as if they were continued with zeros.

Example

12.2 means 12.20.... 0

12.0 means 12.00....0

12.01 means 12.010....0

The following paragraph has been a eliminated.

To determine conformance within limits, the measured value is compared directly with the specified value and any deviation outside the specified limiting value signifies nonconformance with the limits.

Zero tolerance accumulation with basic dim.

In para.2.6, a note has been added on the effect of tolerance accumulation when using basic dimensioning.

NOTE: When basic dimensions are used. There is no accumulation of tolerances. A geometric tolerance is required to create the tolerance zone. In this case, the style of dimensioning (chain, baseline, direct) is up to the discretion of the user. Locating features using directly tolerance dimensions is not recommended.

In Para. 2.7.1, Rule#1 has been expanded for clarification.

2.7.1 variations of form(Rule #1: Envelope Principle)

The form of an individual regular feature of size is controlled by its limits of size to the extent prescribed in the following paragraphs and illustrated in Fig.2-6.

(a)The surface our surfaces of a regular feature of size shall not extend beyond a boundary (envelope) of perfect form at MMC . This boundary is the true geometric form represented by the drawing. No variation in form is permitted if the regular feature of size is produced at its MMC limit of size unless a straightness or flatness tolerance is associated with the size dimension or the Independency symbol is applied per para. 2.7.3. See Fig. 2-7.

(b) Where the actual local size of a regular feature of size as the product from MMC towards LMC, a local variation in form is allowed equal to the amount of such departure.

(c) There is no default requirements for a boundary of perfect form at LMC. Thus a regular feature of size produced at its LMC limit on size is permitted to very from true form to the maximum variation allowed by the boundary of perfect form at MMC.

(d) In cases where a geometric tolerances specified to apply at LMC, perfect form at LMC is required. See Para. 7.3.5

In Para. 2.8.4, Perfect form at LMC when a tolerance applies at LMC is clarified.

2.8.4 Effect of LMC

Where a geometric tolerance is applied on an LMC basis, perfect form at LMC is required. Perfect form at MMC is not required. This is the reciprocal of the MMC concept. See figure 2 - 11. Where a geometric tolerance is applied on LMC basis, the allowed tolerance is dependant on the unrelated actual minimum material envelope on the considered feature. The tolerance is limited to the specified value if the feature is produced at its LMC limit the size. Where the unrelated actual minimum material envelope of the future has departed from LMC, an increase in the tolerance equal to the amount of such departure is allowed. The total permissible variation in position is maximum when the feature is at MMC unless a maximum is specified. See Figs. 7-14 and 7-15.

A new definition added

Limits of size of a continuous feature are defined in Para. 2.7.5

2 .7 .5 Limits of Size and Continuous Features of Size
The note “CONTINUOUS FEATURE” or continuous future symbol is used to identify a group of two or more features of size where there is a requirement that they be treated geometrically as a single feature of size. When using the continuous feature symbol, extension lines between the features may be shown or omitted; however, extension lines by themselves do not indicate a continuous future. See Figs. 2 -8 through 2-10.

New definition added

The placement of the tangent plane symbol within the feature control frame is specified in Para. 2.16

2.16 TANGENT PLANE

Where it is desired to control a tangent plane established by the contacting points of a surface, the tangent plane symbols shall be added in the feature control frame after the stated tolerance. See Fig. 6 - 18. If the tangent plane is unstable it may be optimized. See Para. 4.11.2 and ASME Y. 14.5.1 M.

Removed Para. 2.11.1

Paragraph 2.11.1 in ASME Y14.5M- 1994 (Virtual Condition) has been removed.

This concept is covered in Para. 2.11, Boundary Conditions

Removed Para. 2.11.2

Paragraph 2.11.2 in ASME Y14.5M-1994 (Resultant Condition) has been removed. This concept is covered in Para. 2.11, Boundary Conditions

Removed Par. 2.11.3

Paragraph 2.11.3 in ASME Y14.5M-1994 (Datum Features at Virtual Condition) has been removed. This Concept is covered in Para. 4.11.9 (Datum Feature Shift / Displacement) and Para. 7.3.6 (Datum Feature Modifiers In Positional Tolerancing).

ASME SETTING THE STANDARD

New Figures

Figure 2-7 Independency and Flatness Application

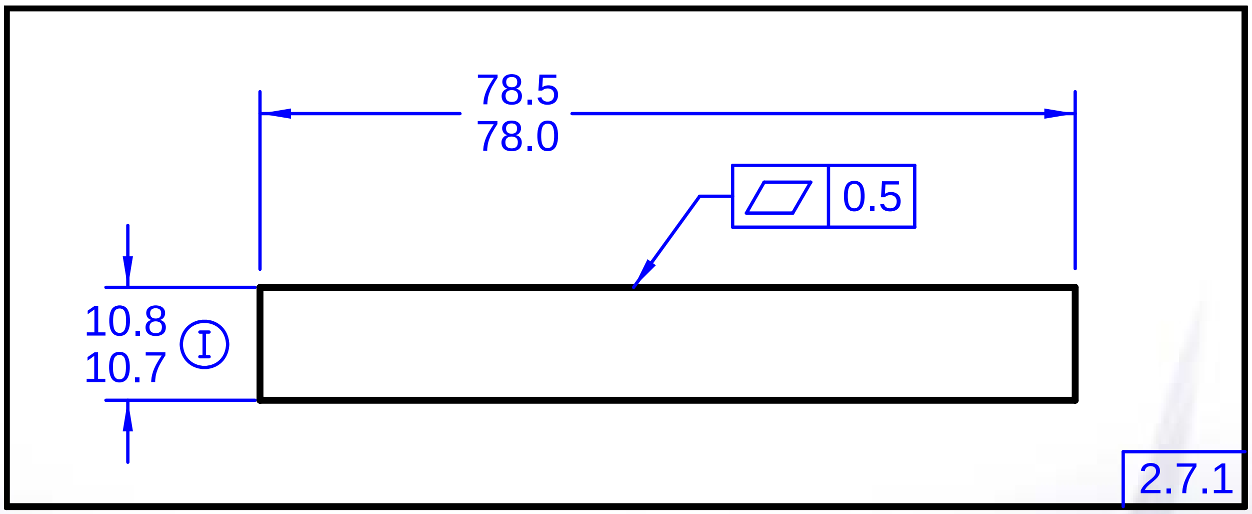


FIG. 2-7 INDEPENDENCY AND FLATNESS APPLICATION

Figure 2-8 Continuous Feature, External Cylindrical

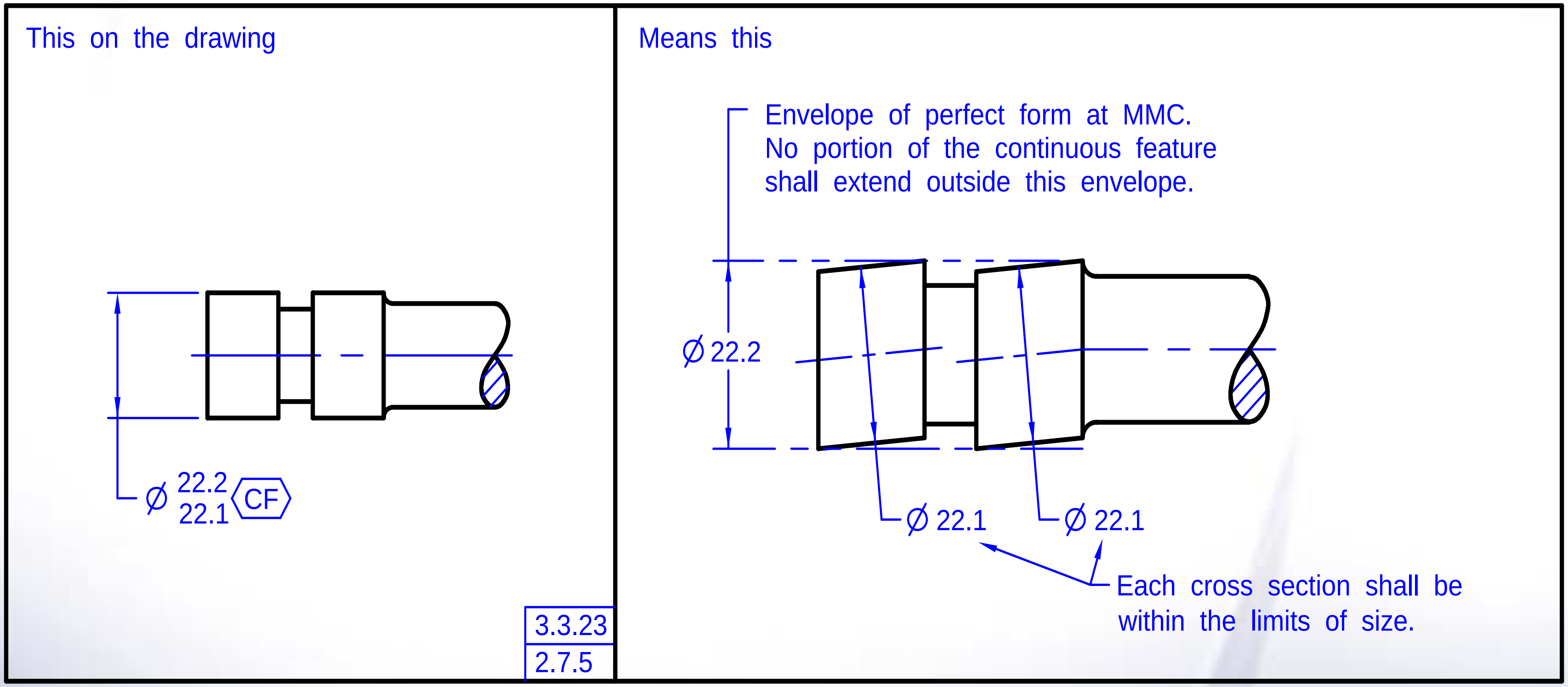


FIG. 2-8 CONTINUOUS FEATURE, EXTERNAL CYLINDRICAL

Figure 2-9 Continuous Feature, Internal Cylindrical

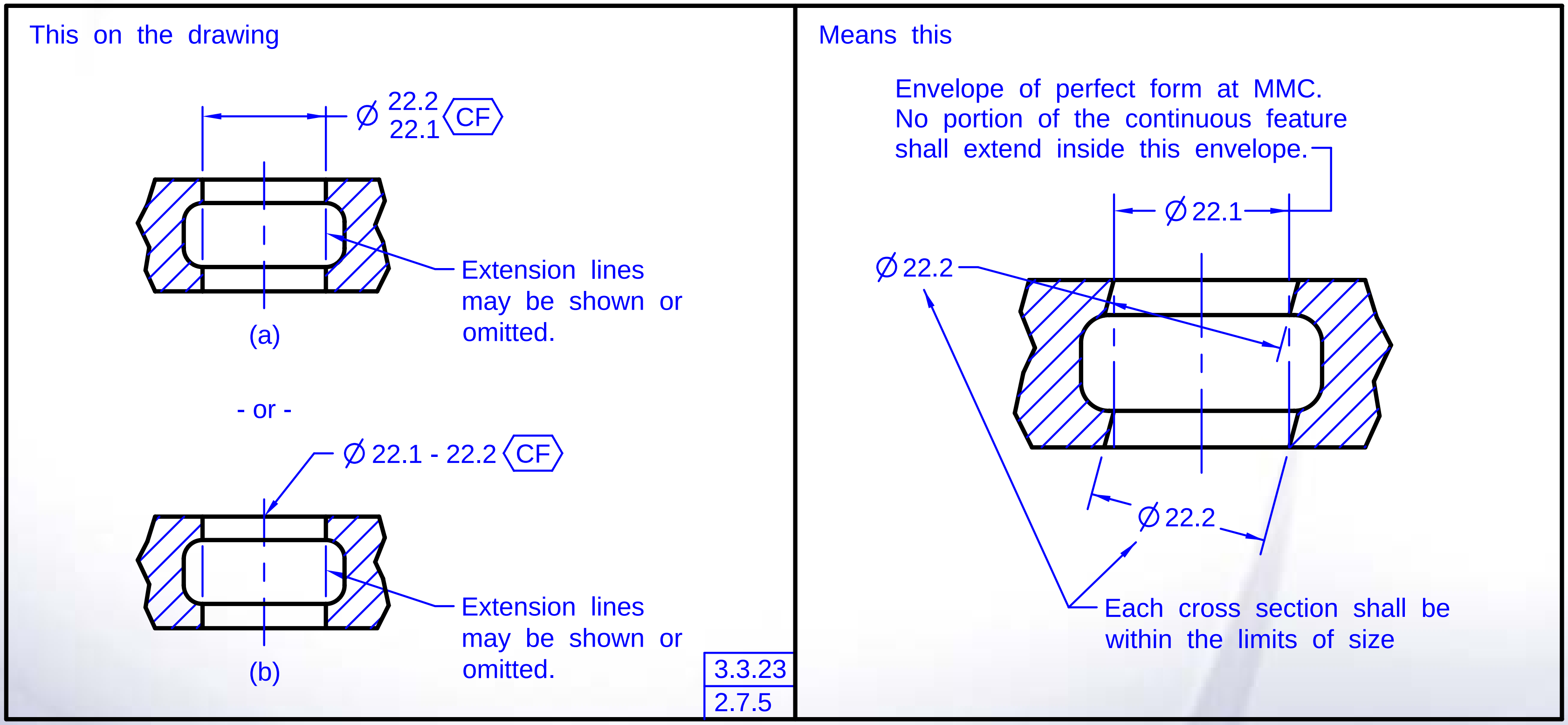


FIG. 2-9 CONTINUOUS FEATURE, INTERNAL CYLINDRICAL

Figure 2-10 Continuous Feature, External Width

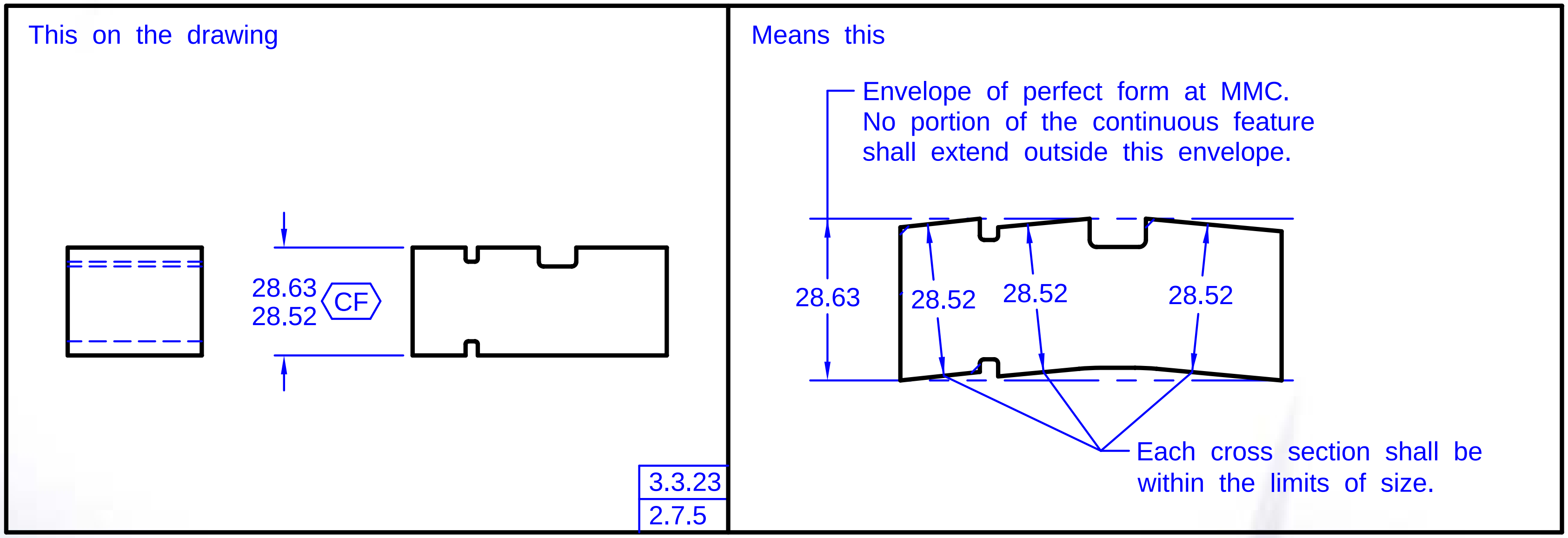


FIG 2-10 CONTINUOUS FEATURE, EXTERNAL WIDTH

Figure 2-11 Extreme variation of Form Allowed by a geometric tolerance – Perfect Form at LMC

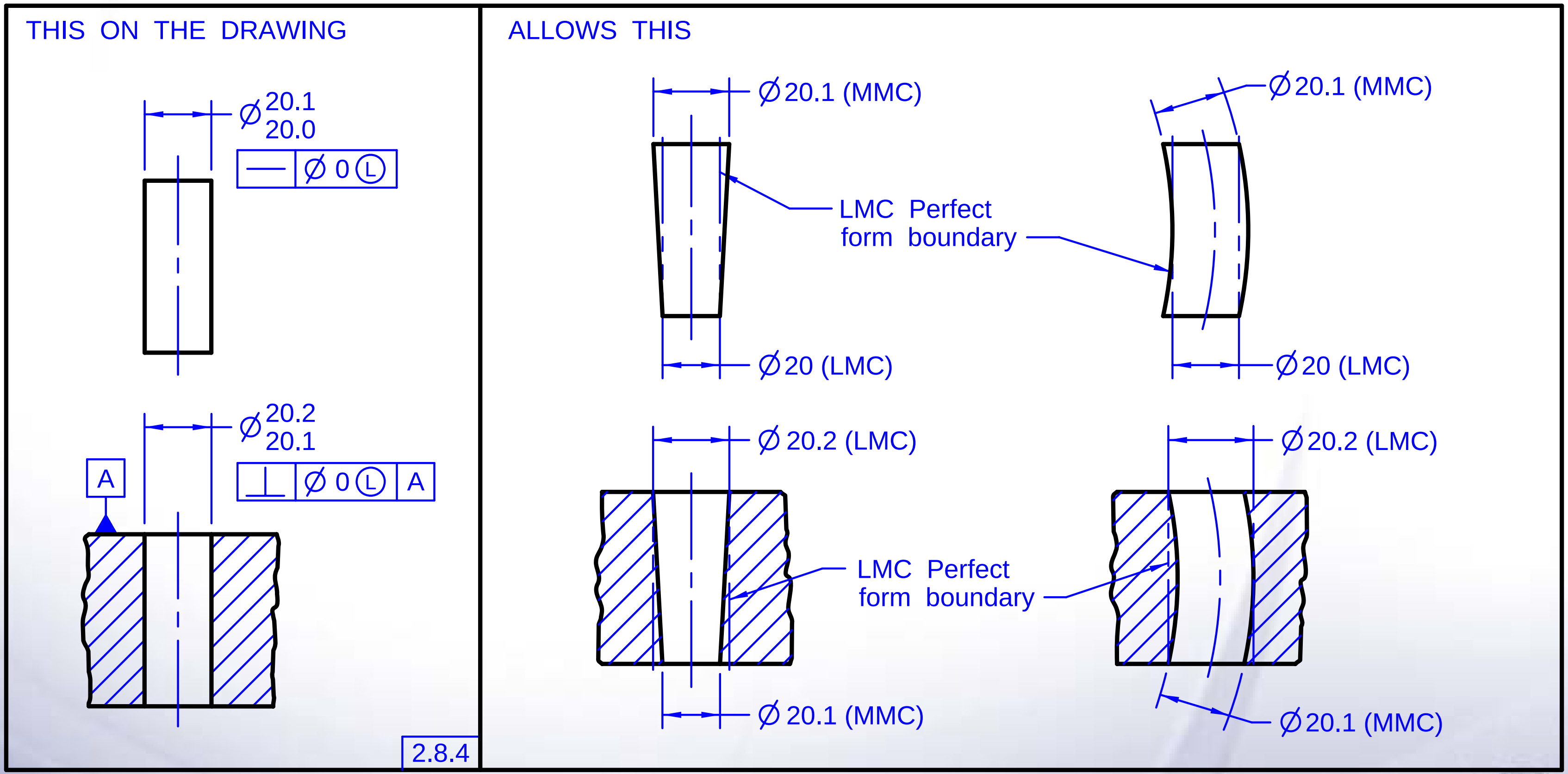


FIG. 2-11 EXTREME VARIATIONS OF FORM ALLOWED BY A GEOMETRIC TOLERANCE - PERFECT FORM AT LMC

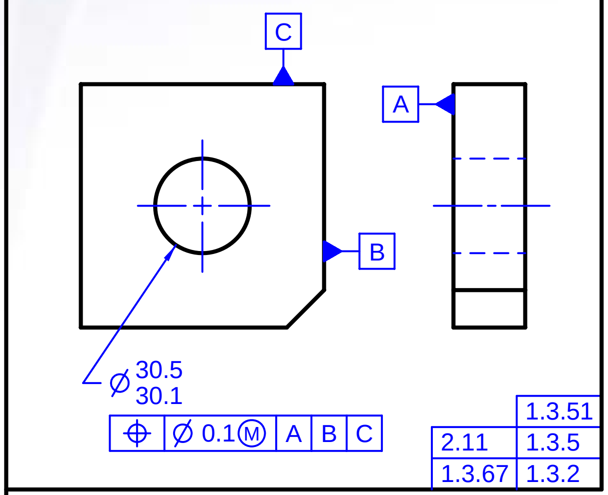
Consolidated Figures

Consolidated Old Figures 2.7 ,2.8, 2.9, 2.10, 2.11, 2.12

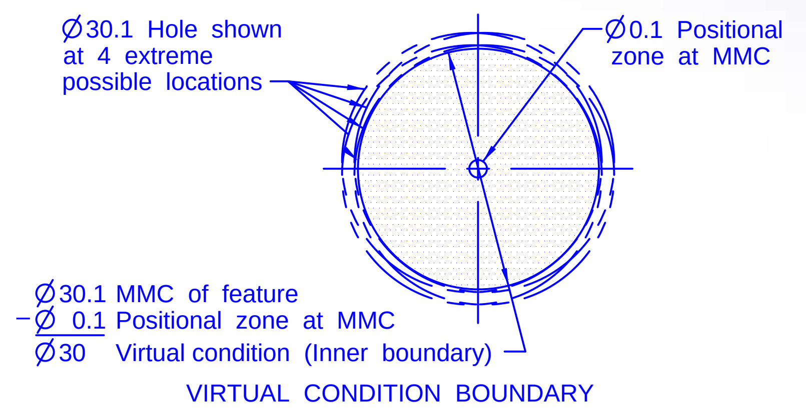
into

New figures 2.12, 2.13, 2.14, 2.15, 2.16, 2.16, 2.17

This on the drawing



Means this



	Ø HOLE	Ø TOL	VC	RC
LMC	30.5	0.5	30	31
	30.4	0.4		
	30.3	0.3		
	30.2	0.2		
MMC	30.1	0.1		

The virtual condition (VC) of the internal feature is a constant value equal to its maximum material condition MINUS its applicable geometric tolerance.

The resultant condition (RC) of the internal feature is a single value equal to its least material condition PLUS its applicable geometric tolerance.

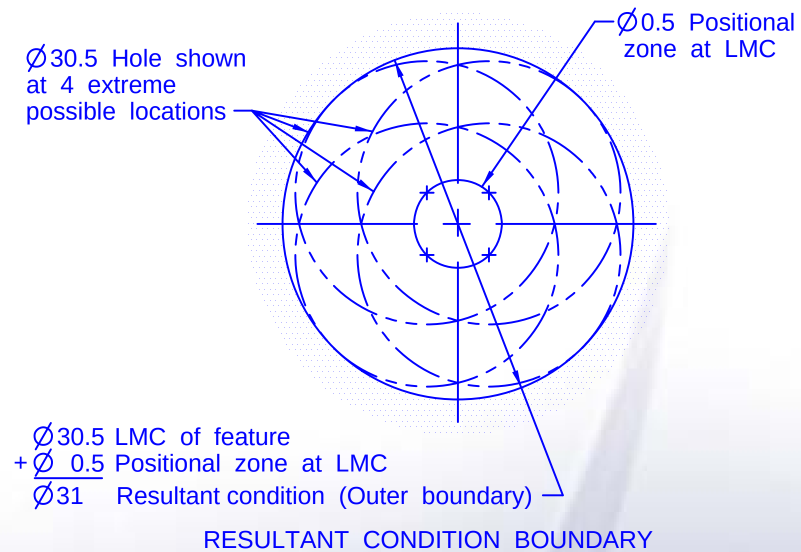
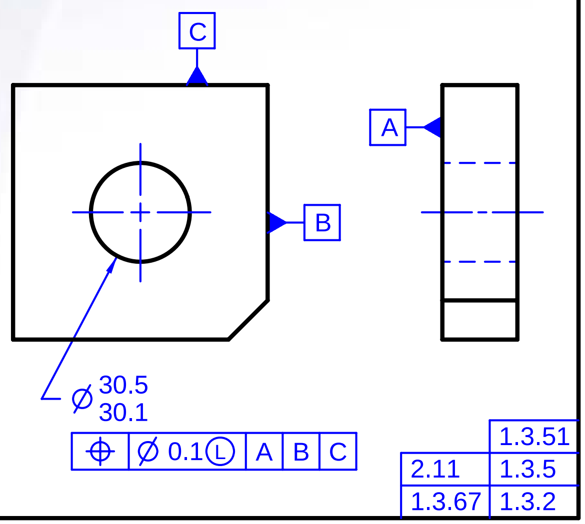
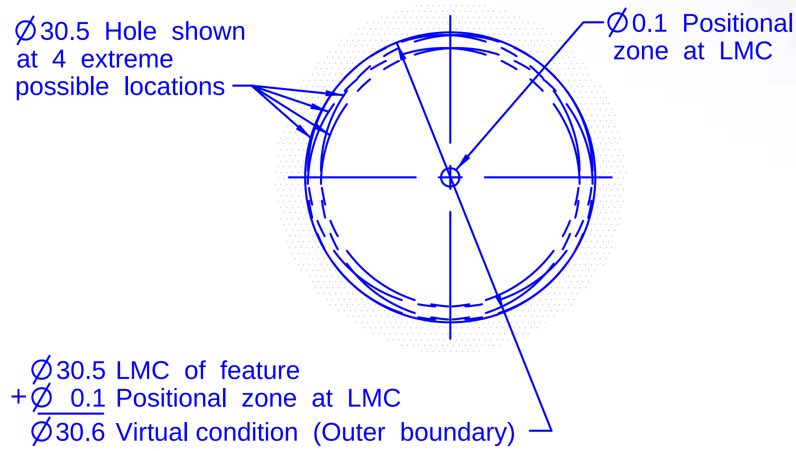


FIG. 2-12 VIRTUAL AND RESULTANT CONDITION BOUNDARIES USING MMC CONCEPT-INTERNAL FEATURE

This on the drawing



Means this

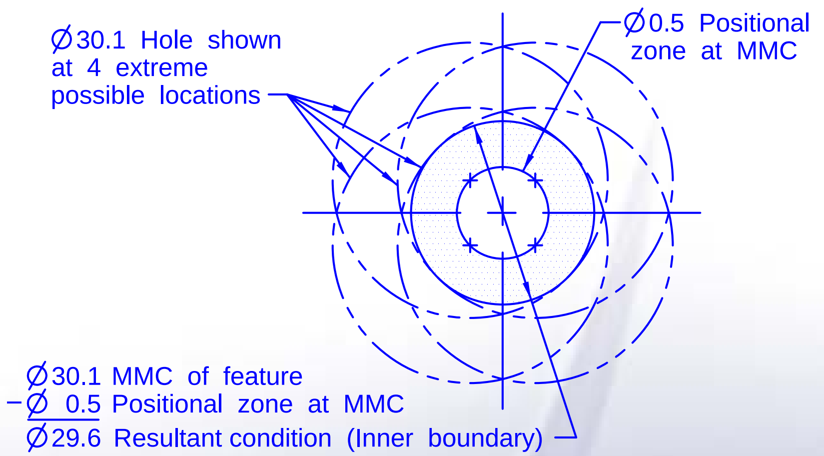


VIRTUAL CONDITION BOUNDARY

	Ø HOLE	Ø TOL	VC	RC
LMC	30.5	0.1		
	30.4	0.2		
	30.3	0.3	30.6	
	30.2	0.4		
MMC	30.1	0.5		29.6

The virtual condition (VC) of the internal feature is a constant value equal to its least material condition PLUS its applicable geometric tolerance.

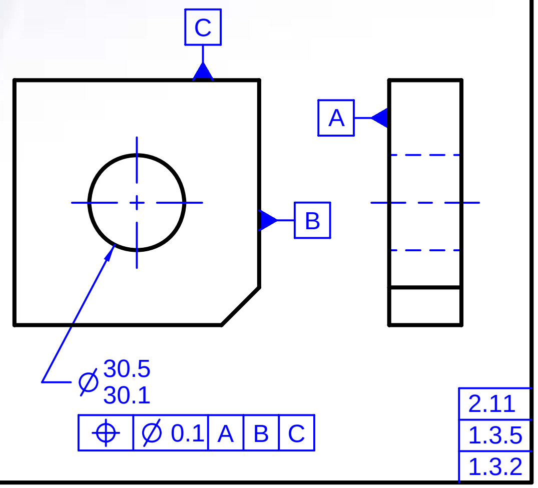
The resultant condition (RC) of the internal feature is a single value equal to its maximum material condition MINUS its applicable geometric tolerance.



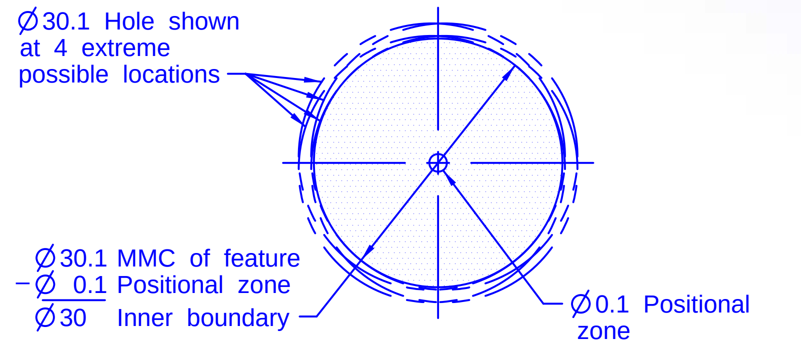
RESULTANT CONDITION BOUNDARY

FIG. 2-13 VIRTUAL AND RESULTANT CONDITION BOUNDARIES USING LMC CONCEPT-INTERNAL FEATURE

This on the drawing



Means this



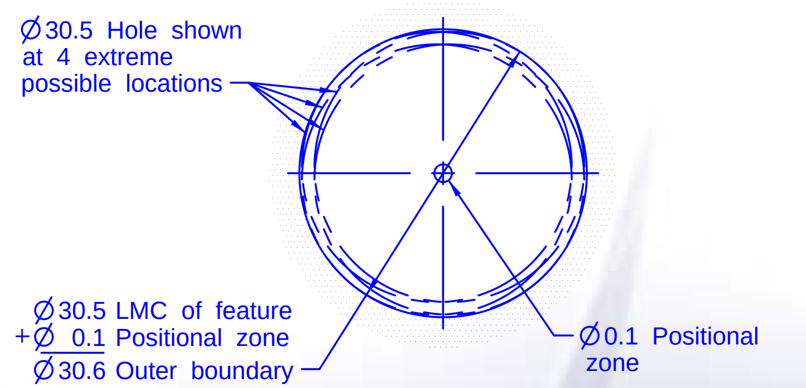
MINIMUM INNER BOUNDARY

	ØHOLE	ØTOL	IB	OB*
LMC	30.5			30.6
	30.4			
	30.3	0.1		
	30.2			
MMC	30.1		30	

The inner boundary (IB) of the internal feature is a single value equal to its maximum material condition MINUS its stated geometric tolerance.

The outer boundary (OB) of the internal feature is a single value equal to its least material condition PLUS its stated geometric tolerance.

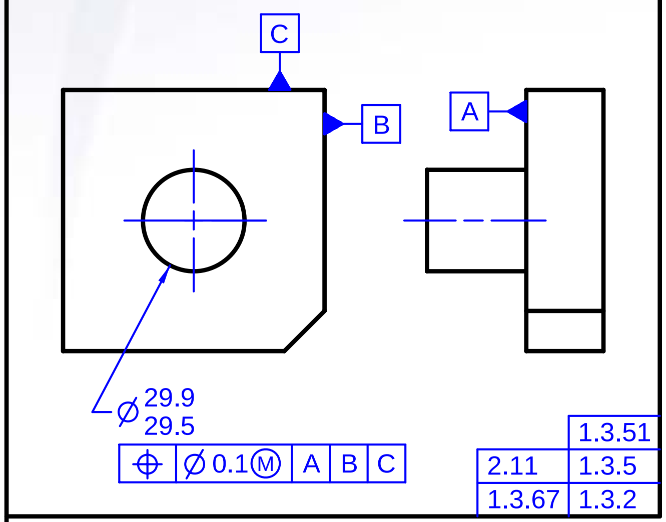
* Calculaions do not include form variations of the feature.



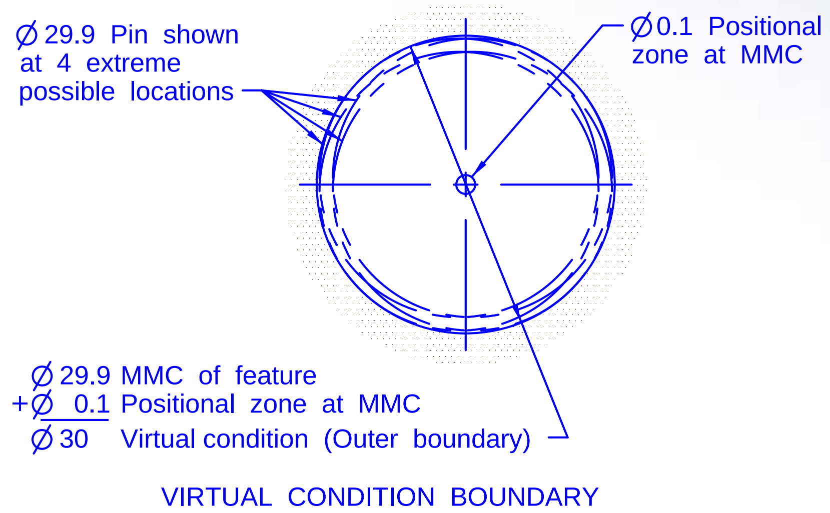
MAXIMUM OUTER BOUNDARY

FIG. 2-14 INNER AND OUTER BOUNDARIES USING RFS CONCEPT - INTERNAL FEATURE

This on the drawing



Means this



	⌀ PIN	⌀ TOL	VC	RC
MMC	29.9	0.1		
	29.8	0.2		
	29.7	0.3	30	
	29.6	0.4		
LMC	29.5	0.5		29

The virtual condition (VC) of the external feature is a constant value equal to its maximum material condition PLUS its applicable geometric tolerance.

The resultant condition (RC) of an external feature is a single value equal to its least material condition MINUS its applicable geometric tolerance.

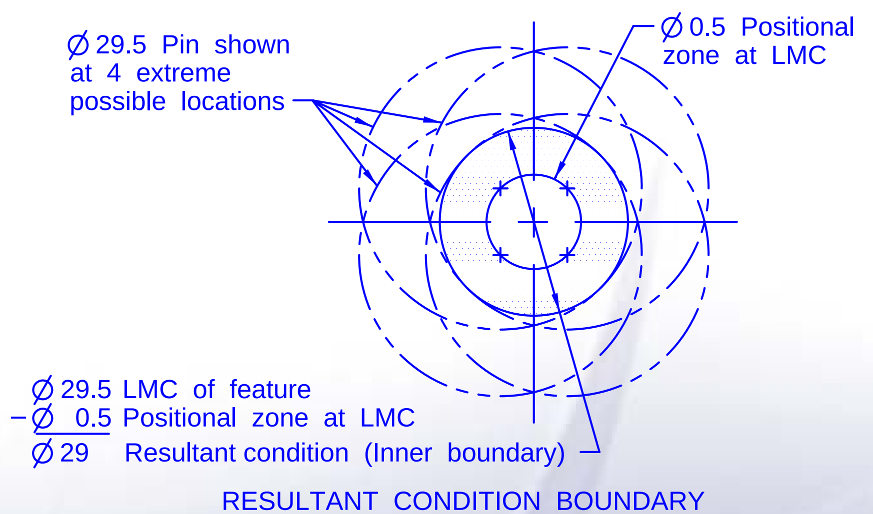
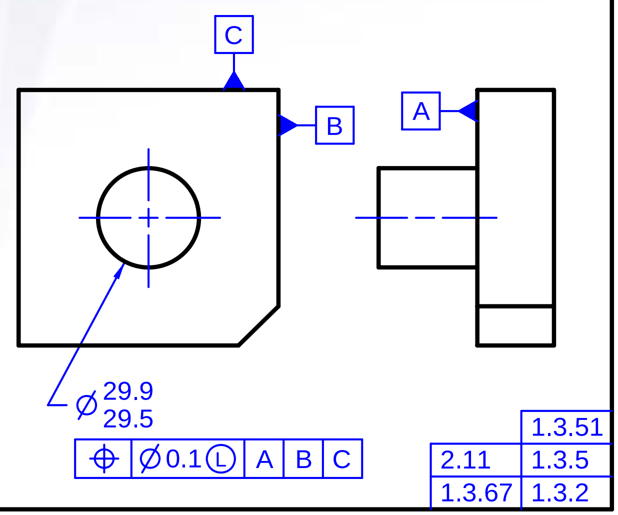
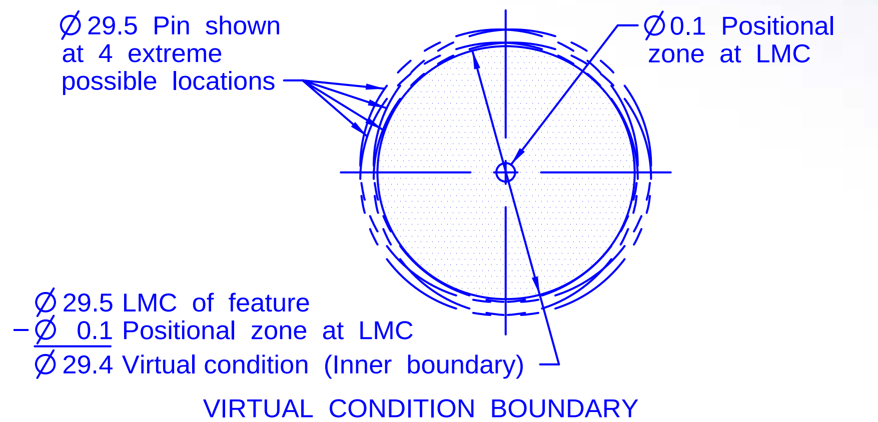


FIG. 2-15 VIRTUAL AND RESULTANT CONDITION BOUNDARIES USING MMC CONCEPT-EXTERNAL FEATURE

This on the drawing



Means this



	Ø PIN	Ø TOL	VC	RC
MMC	29.9	0.5		30.4
	29.8	0.4		
	29.7	0.3	29.4	
	29.6	0.2		
LMC	29.5	0.1		

The virtual condition (VC) of the external feature is a constant value equal to its least material condition MINUS its applicable geometric tolerance.

The resultant condition (RC) of the external feature is a single value equal to its maximum material condition PLUS its applicable geometric tolerance.

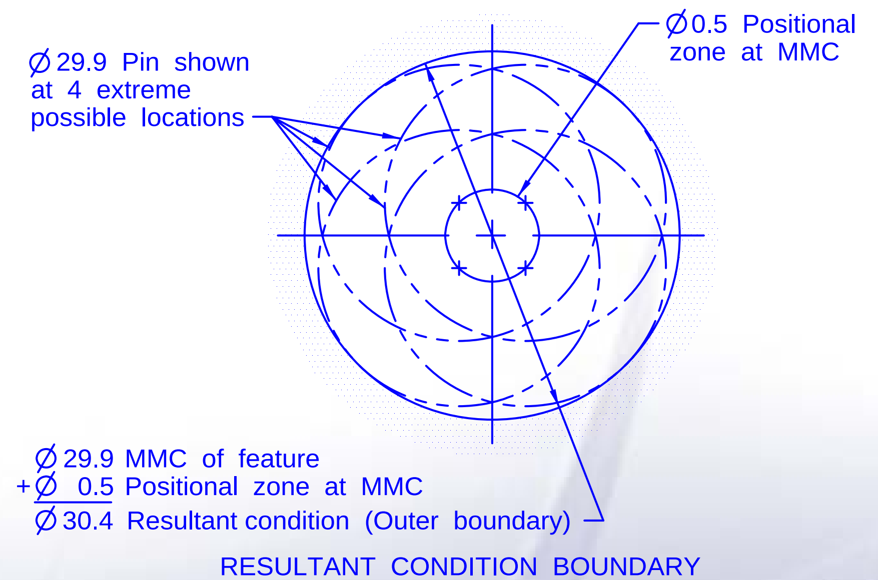
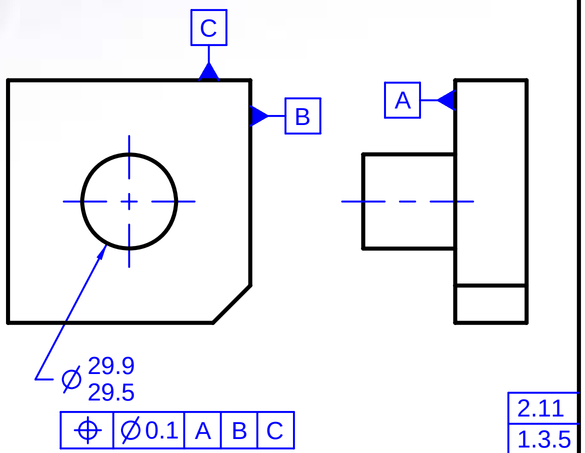


FIG. 2-16 VIRTUAL AND RESULTANT CONDITION BOUNDARIES USING LMC CONCEPT-EXTERNAL FEATURE

This on the drawing



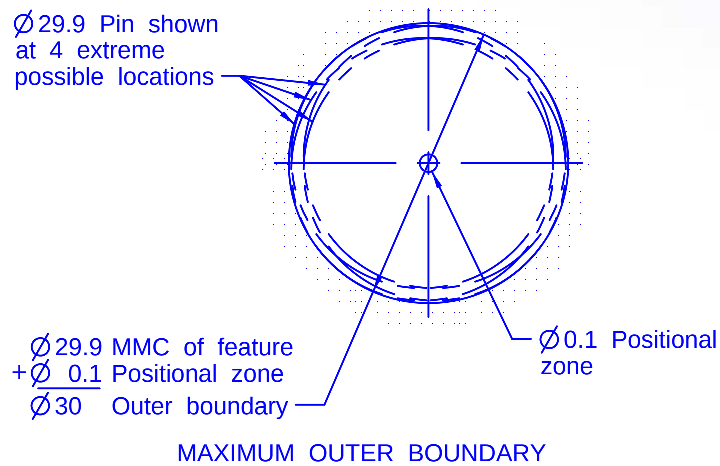
Means this

	ØPIN	ØTOL	OB	IB*
MMC	29.9	0.1	30	
	29.8			
	29.7			
	29.6			
LMC	29.5			29.4

The outer boundary (OB) of the external feature is a single value equal to its least material condition PLUS its stated tolerance of location.

The inner boundary (IB) of the external feature is a single value equal to its least material condition MINUS its stated tolerance of location.

* Calculaions do not include form variations of the feature.



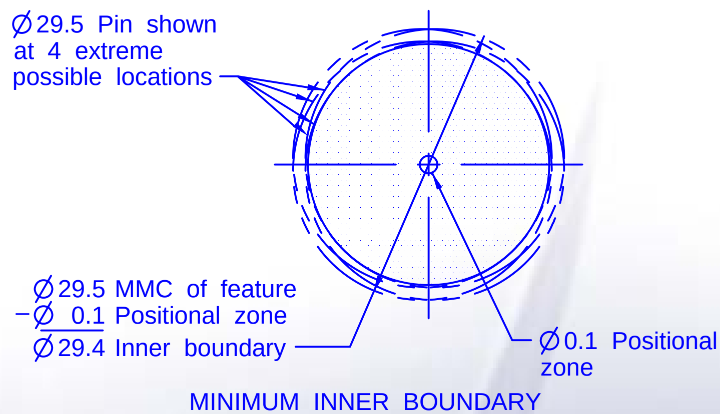


FIG. 2-17 INNER AND OUTER CONDITION BOUNDARIES USING RFS CONCEPT - EXTERNAL FEATURE

Deleted figures

Figure 2.14 Tolerancing an Angular Surface with a Basic Angle.

Figure 2.15 Specifying a Basic Diameter

Questions:

Answers represent the opinion of the respondent and do not represent ASME.

ASME

SETTING THE STANDARD

Section 4
Datum Reference Frames

Al Neumann

Section 4 - Work Group Leader

ASME

SETTING THE STANDARD

4.2 DEGREES OF FREEDOM

All parts have six degrees of freedom, three translational and three rotational, which may be constrained by datum feature references in a feature control frame. The three translational degrees of freedom are termed X, Y, and Z. The three rotational degrees of freedom are termed u, v, and w. See Figs. 4-1, 4-2, illustration (c); 4-2, illustration (d); and 4-2, illustration (e).

Fig. 4-1 Datum Reference Frame

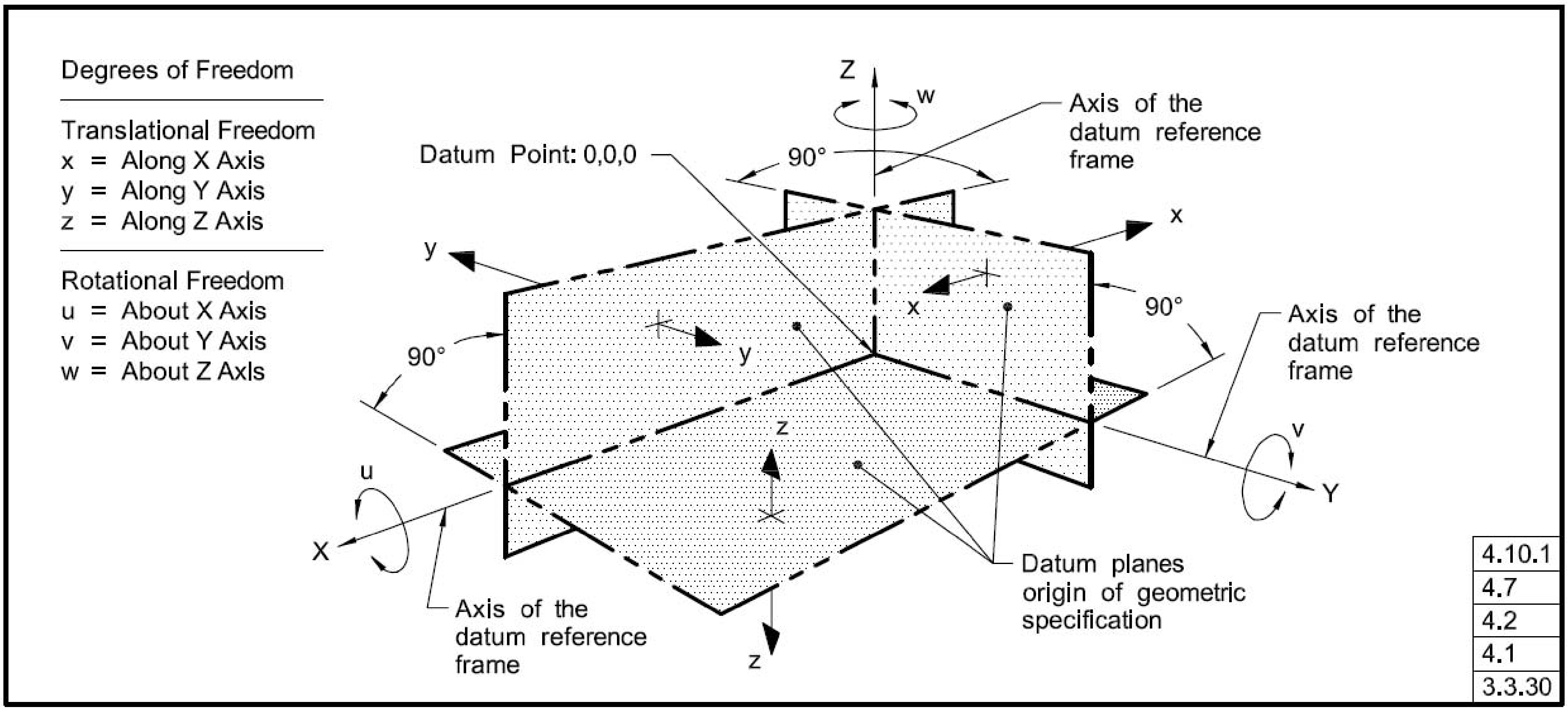


Fig. 4-2 Sequence of Datum Features Relates Part to Datum Reference Frame

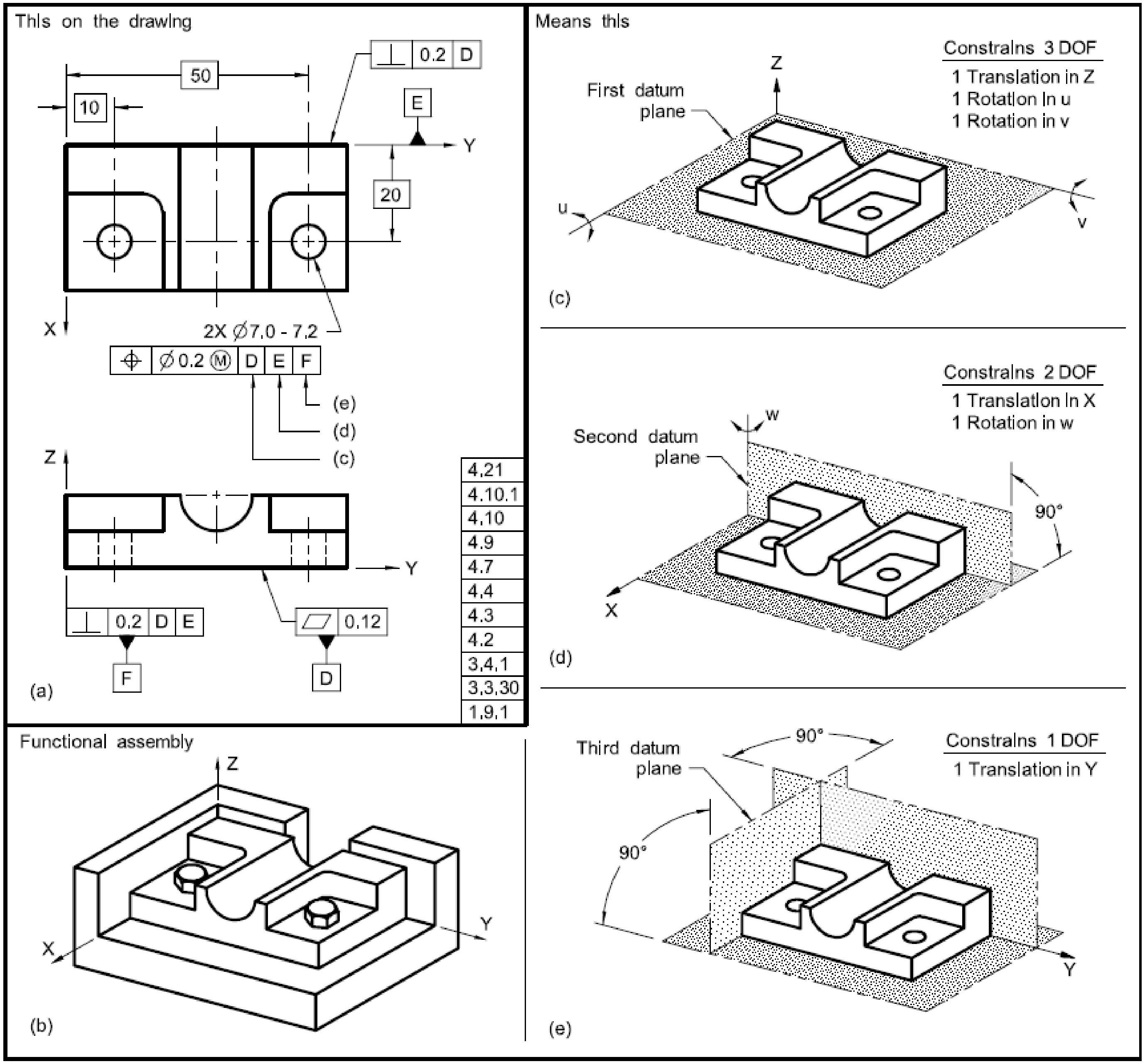


Fig. 4-7 Inclined Datum Features

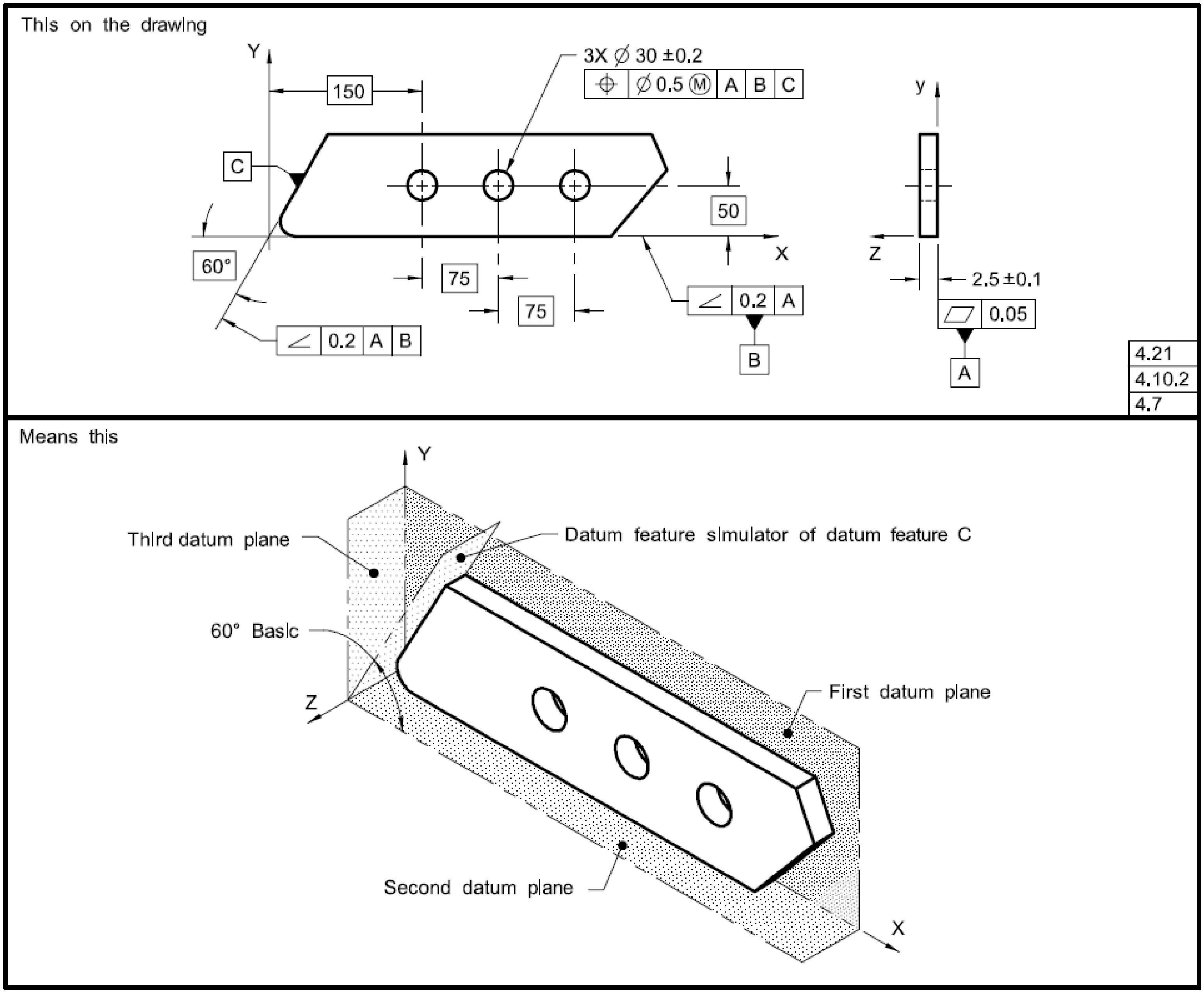
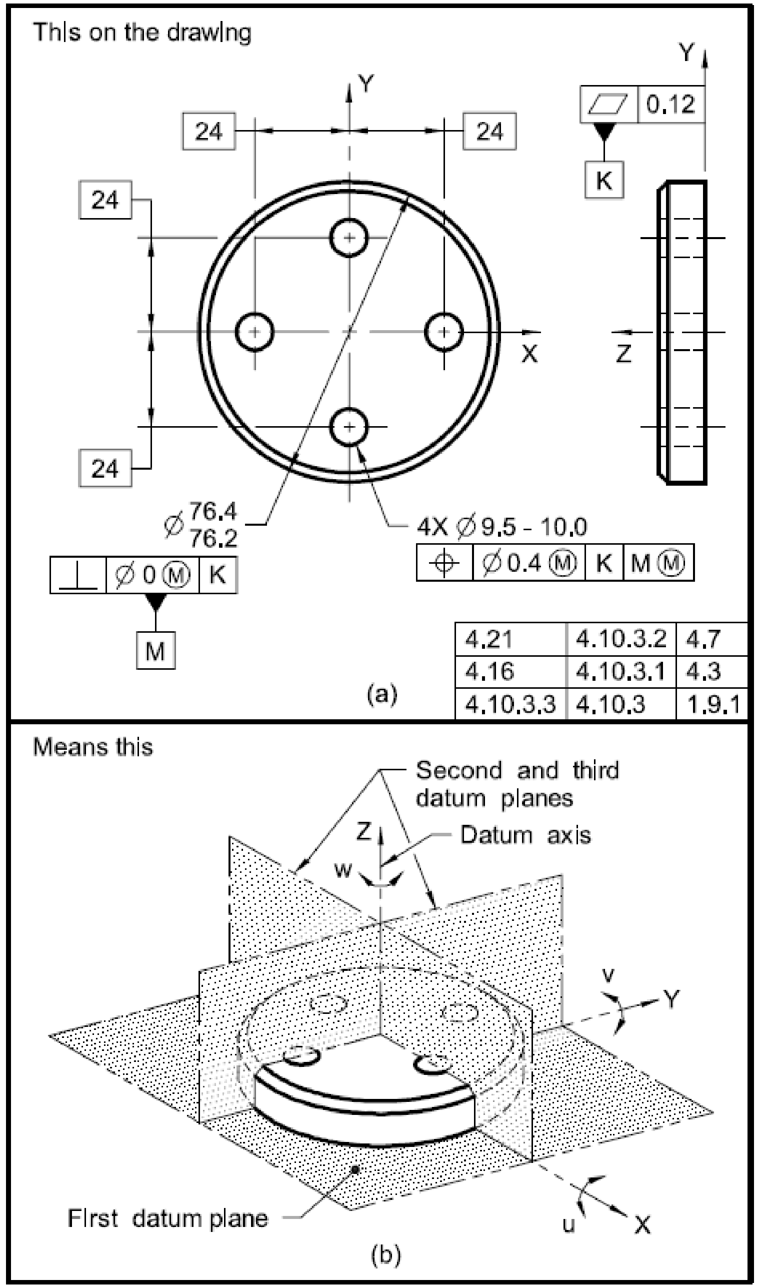


Fig. 4-8 Part With Cylindrical Datum Feature



The following primary datums are derived from the associated datum feature simulator:

a) a planar datum feature (nominally flat) establishes a datum feature simulator that creates a datum plane and constrains three degrees of freedom (one translation and two rotations). See Fig. 4-3, illustration (a).

(b) a width as a datum feature (two opposed parallel surfaces) establishes a datum feature simulator thatcreates a datum center plane and constrains three degrees of freedom (one translation and two rotations). See Fig. 4-3, illustration (b).

(c) a spherical datum feature establishes a datum feature simulator that creates a datum center point andconstrains three translational degrees of freedom. See Fig. 4-3, illustration (c).

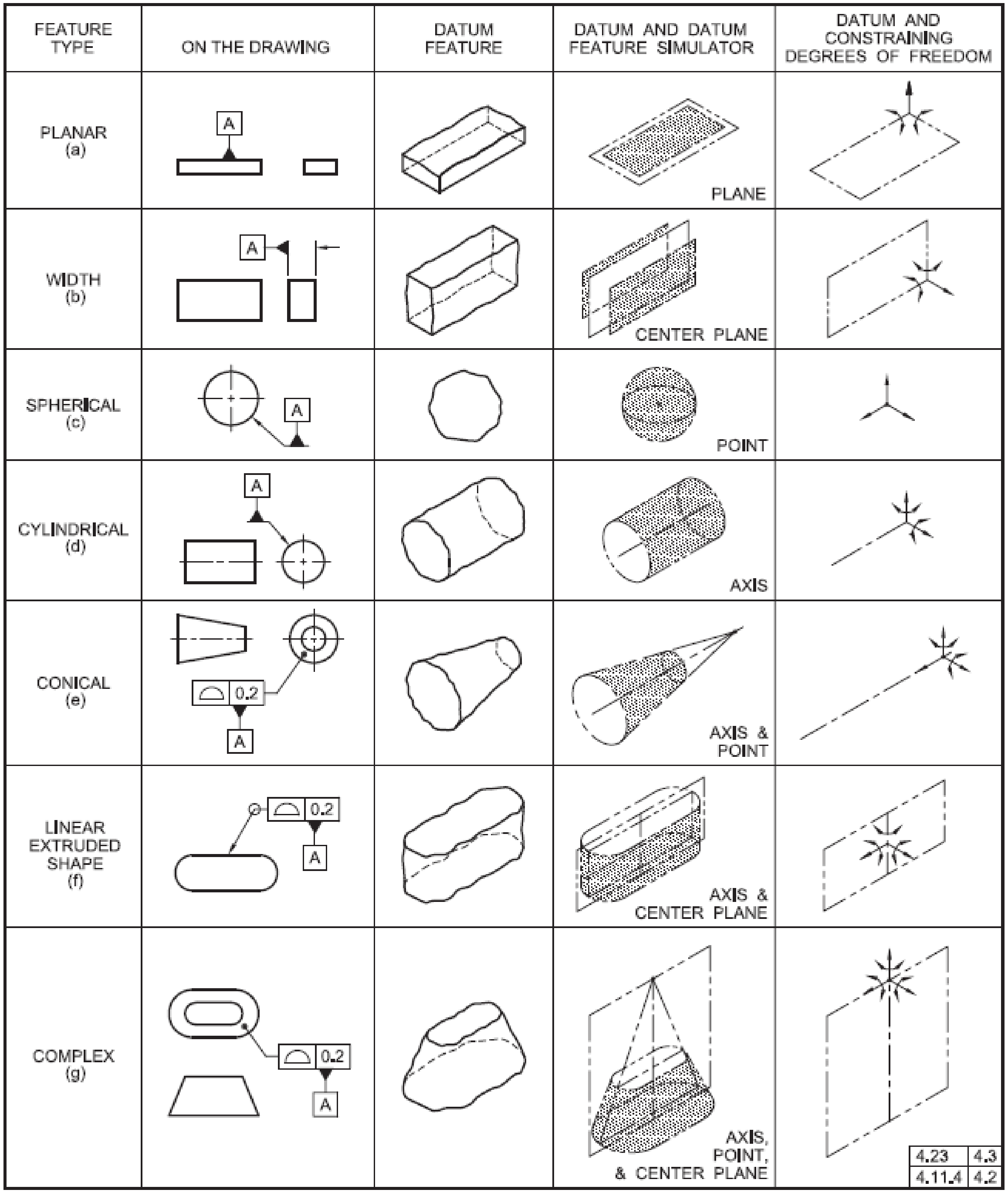
(d) a cylindrical datum feature establishes a datum feature simulator that creates a datum axis (line) and constrains four degrees of freedom (two translations and two rotations). See Fig. 4-3, illustration (d).

(e) a conical shaped datum feature establishes a datum feature simulator that creates a datum axis anda datum point and constrains five degrees of freedom (three translations and two rotations). See Fig. 4-3, illustration (e).

(f) a datum feature of linear extruded shape establishes a datum feature simulator that creates a datum plane and a datum axis and constrains five degrees of freedom (two translations and three rotations). See Fig. 4-3, illustration (f).

(g) a complex datum feature establishes a datum feature simulator that creates a datum plane, datum point, and a datum axis and constrains six degrees of freedom (three translations and three rotations). See Fig. 4-3, illustration (g).

Fig. 4-3 Constrained Degrees of Freedom for Primary Datum Features



4.5 DATUM FEATURE SIMULATOR

A datum feature simulator, as defined in para. 1.3.17, shall be the inverse shape of the datum feature, unless otherwise specified. See Figs. 4-10, 4-11, 4-12, 4-13, and 4-14.

1.3.17 Datum Feature Simulator

datum feature simulator: encompasses two types: theoretical and physical. See paras. 1.3.17.1 and 1.3.17.2.

1.3.17.1 Datum Feature Simulator (Theoretical). ***datum*** *feature simulator (theoretical): the theoretically perfect* boundary used to establish a datum from a specified datum feature.

1.3.17.2 Datum Feature Simulator (Physical). ***datum*** *feature simulator (physical): the physical boundary used* to establish a simulated datum from a specified datum feature.

NOTE: For example, a gage, fixture element, or digital data (such as machine tables, surface plates, a mandrel, or mathematical simulation) —although not true planes — are of sufficient quality that the planes derived from them are used to establish simulated datums. Physical datum feature simulators are used as the physical embodiment of the theoretical datum feature simulators during manufacturing and inspection. See ASME Y14.43.

4.5.1 Examples

A datum feature simulator may be one of the following:

(a) a maximum material boundary (MMB)
(b) a least material boundary (LMB)
(c) an actual mating envelope
(d) a minimum material envelope
(e) a tangent plane
(f) a datum target(s)
(g) a mathematically defined contour

NOTE: Whenever the term “datum feature simulator” is used in this Standard, it refers to the theoretical, unless specifically otherwise indicated.

Fig. 4-11 Establishment of Datums — For External Cylindrical Feature — RMB

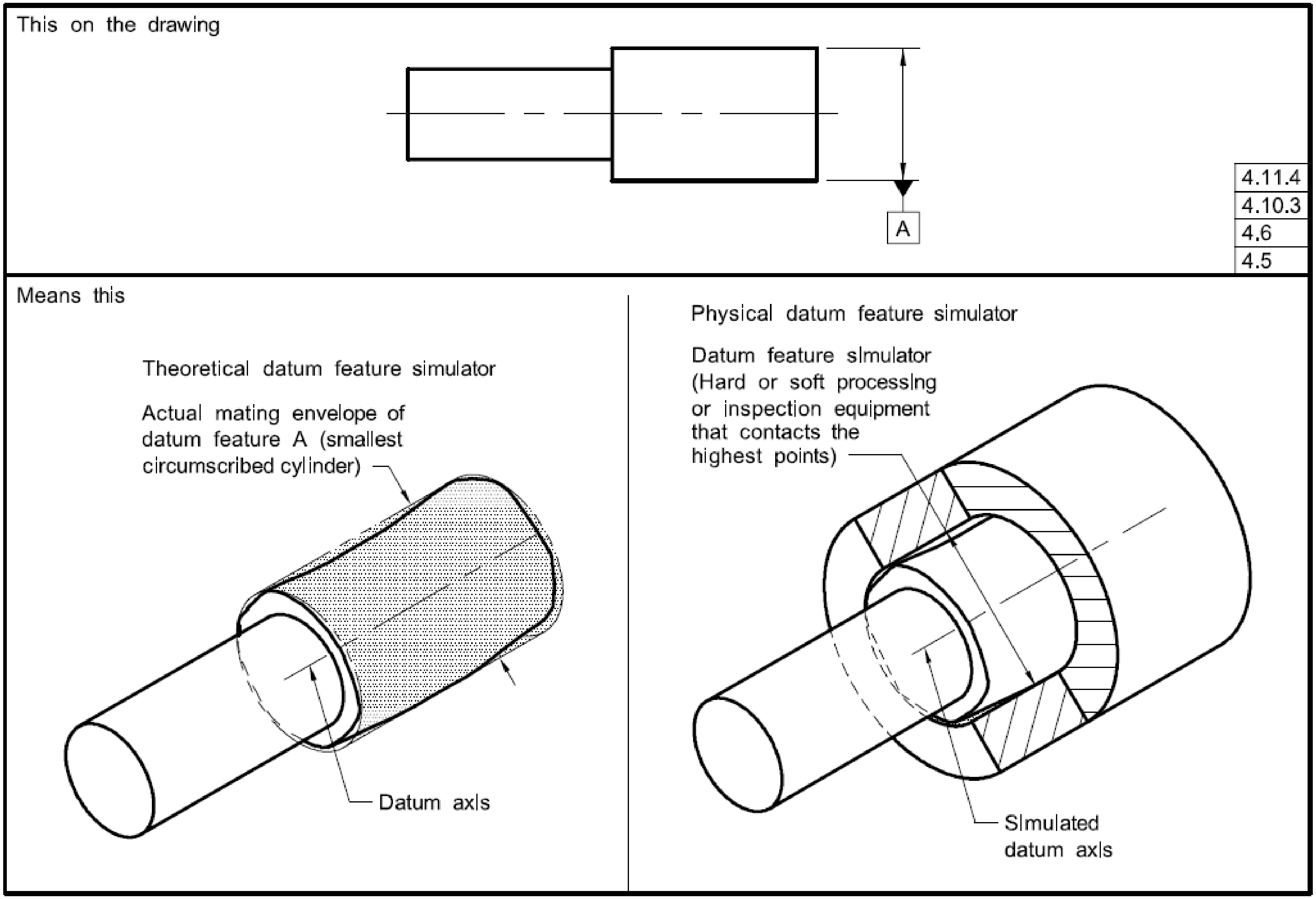


Fig. 4-12 Establishment of Datums — For Internal Cylindrical Feature — RMB

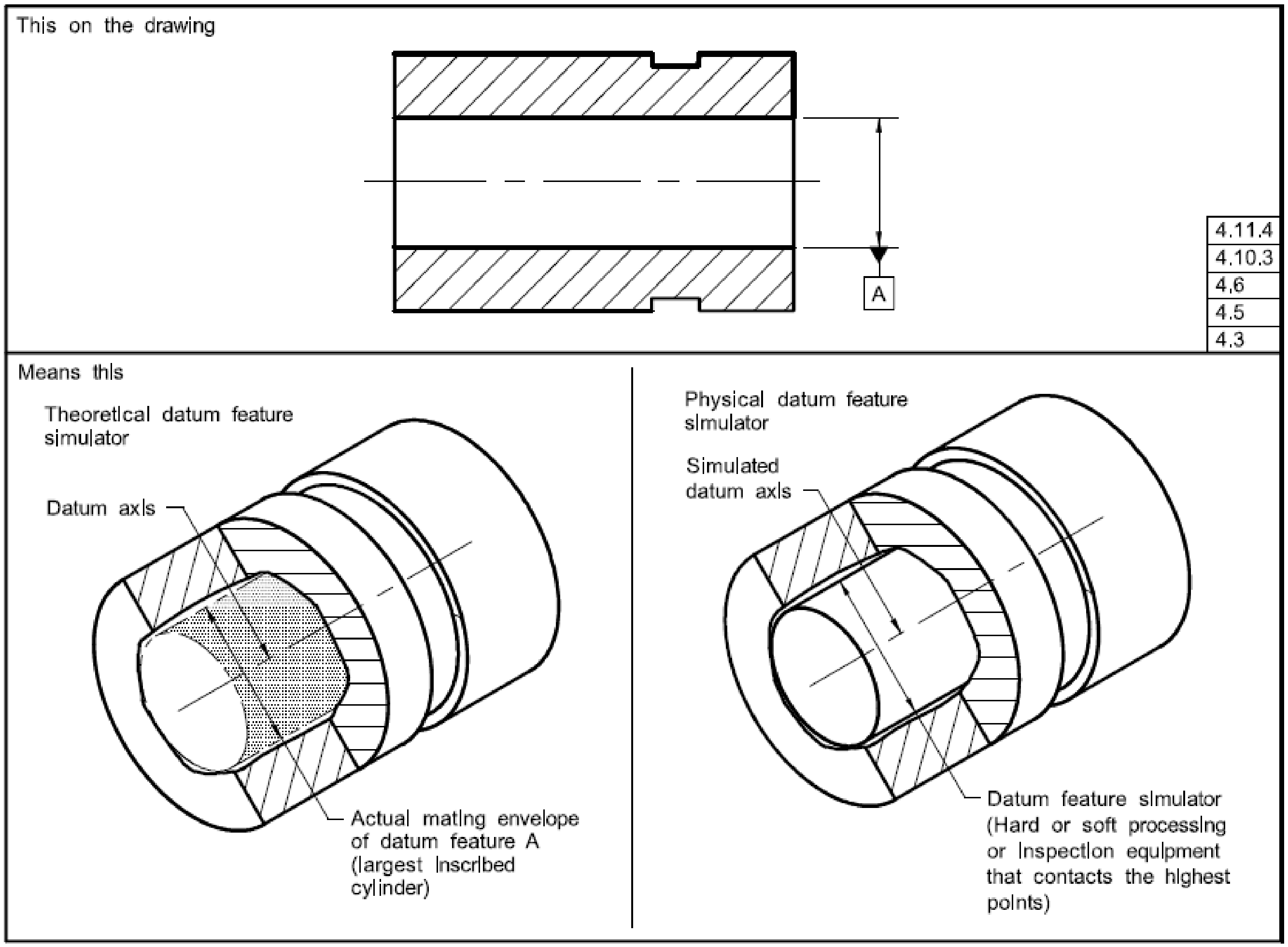


Fig. 4-13 Establishment of Datums — For External Datum Width — RMB

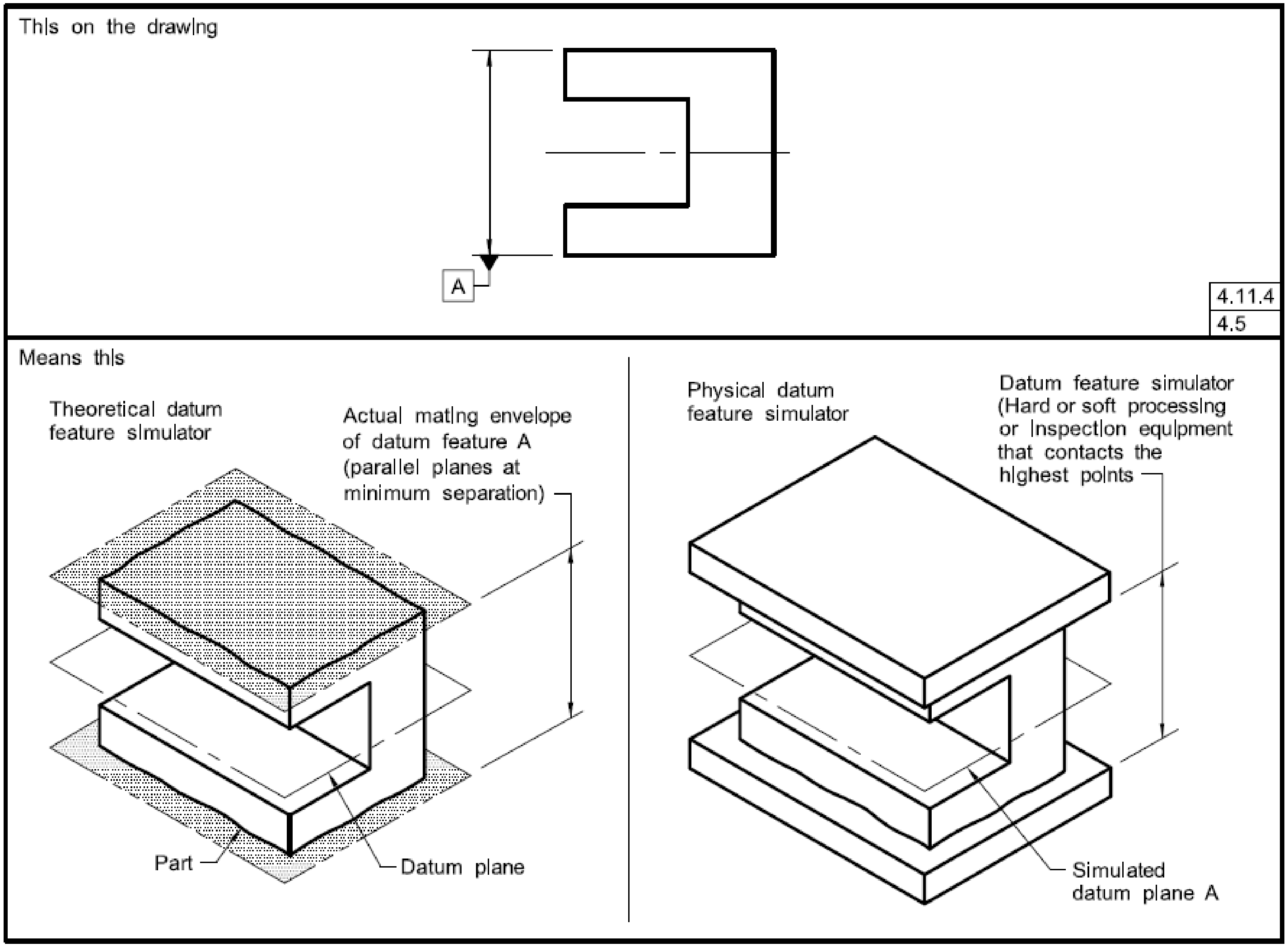
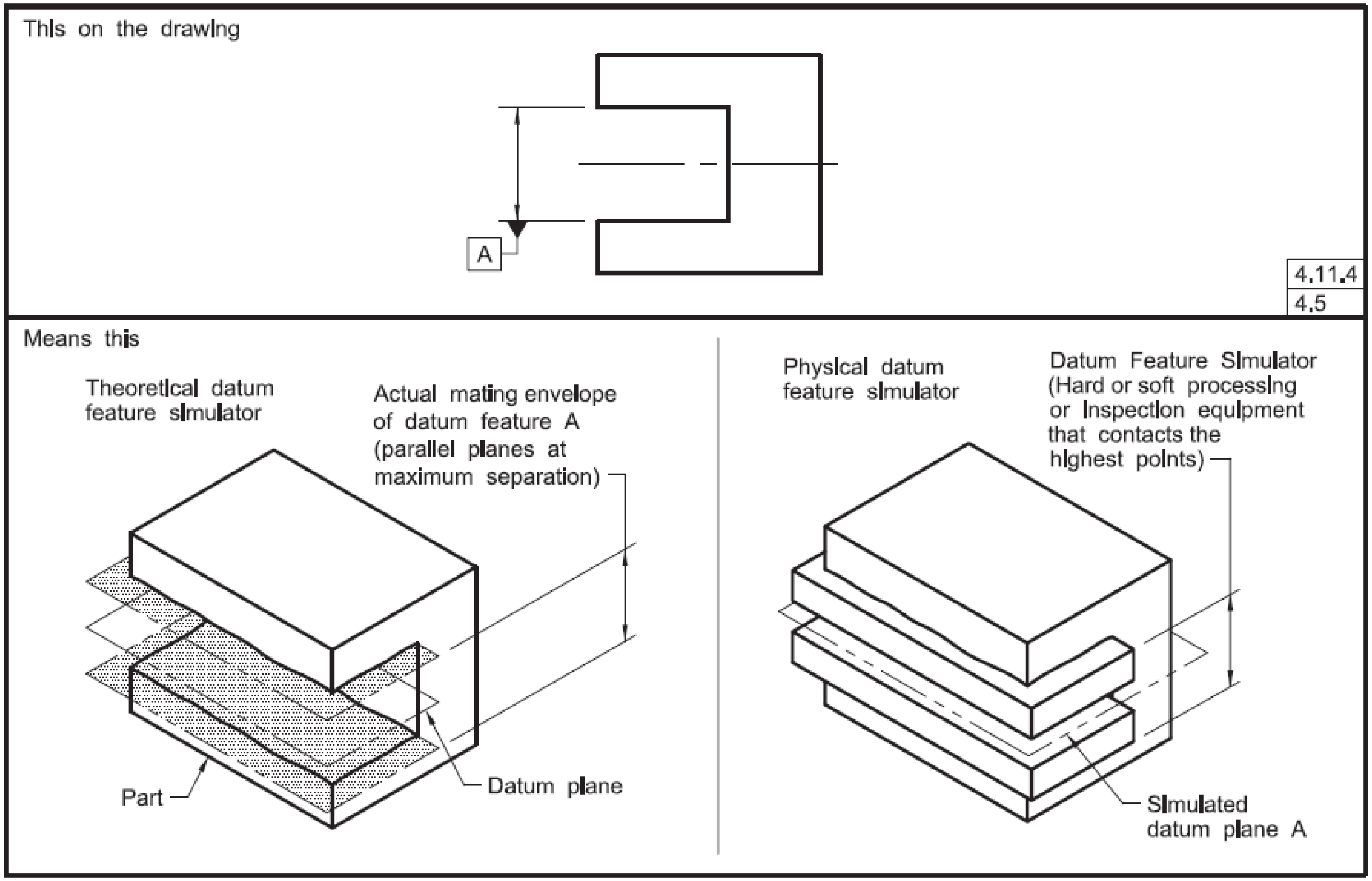


Fig. 4-14 Establishment of Datums — For Internal Datum Width — RMB



4.5.2 Requirements

Datum feature simulators shall have the following requirements:

(a) perfect form.

(b) basic orientation relative to one another for all the datum references in a feature control frame.

(c) basic location relative to other datum feature simulators for all the datum references in a feature control frame, unless a translation modifier or movable datum target symbol is specified. See Figs. 4-9, 4-19, and 4-32, illustration (a).

(d) movable location when the translation modifier or the movable datum target symbol is specified. See Figs. 4-19, 4-32, illustration (b), and 4-49.

(e) fixed at the designated size, when MMB or LMB is specified.

(f) adjustable in size, when the datum feature applies at RMB.

4.6 THEORETICAL AND PHYSICAL APPLICATION OF DATUM FEATURE SIMULATORS

Theoretical datum feature simulators formerly called True Geometric Counterpart (TGC)

4.9 DATUM FEATURE CONTROLS

Geometric tolerances related to a datum reference frame do not take into account any variations in form, orientation, or location of the datum features. Datum features shall be controlled directly by applying appropriate geometric tolerances or indirectly by dimensions such as the size of a primary datum feature of size. This in turn makes it possible to calculate the datum feature simulator boundaries of each datum feature in a datum reference frame. The relationships between datum features to be considered are the

(a) form of the primary datum feature(s) (see Figs. 4-2 and 4-5) and/or the location between features in a pattern used to establish the primary datum. See Figs. 4-24 and 4-25.
(b) secondary datum features' orientation and/or location as applicable, to higher precedence datums. See Figs. 4-2, 4-5, 4-26, and 4-30.
(c) tertiary datum features' orientation and/or location to higher precedence datums as applicable. See Figs. 4-2 and 4-5.

Fig. 4-2 Sequence of Datum Features Relates Part to Datum Reference Frame

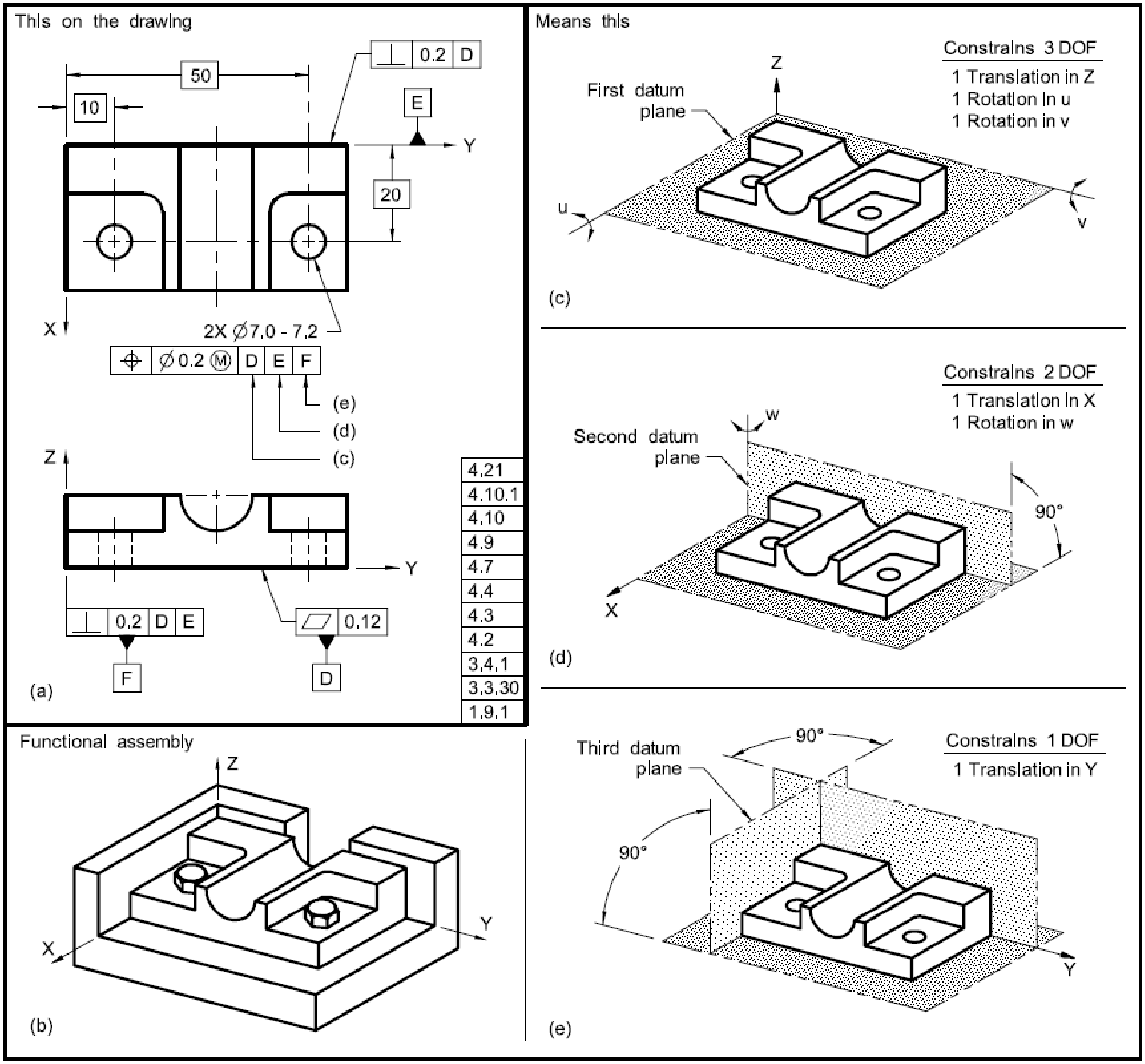


Fig. 4-5 Part Where Rotational Constraint Is Important

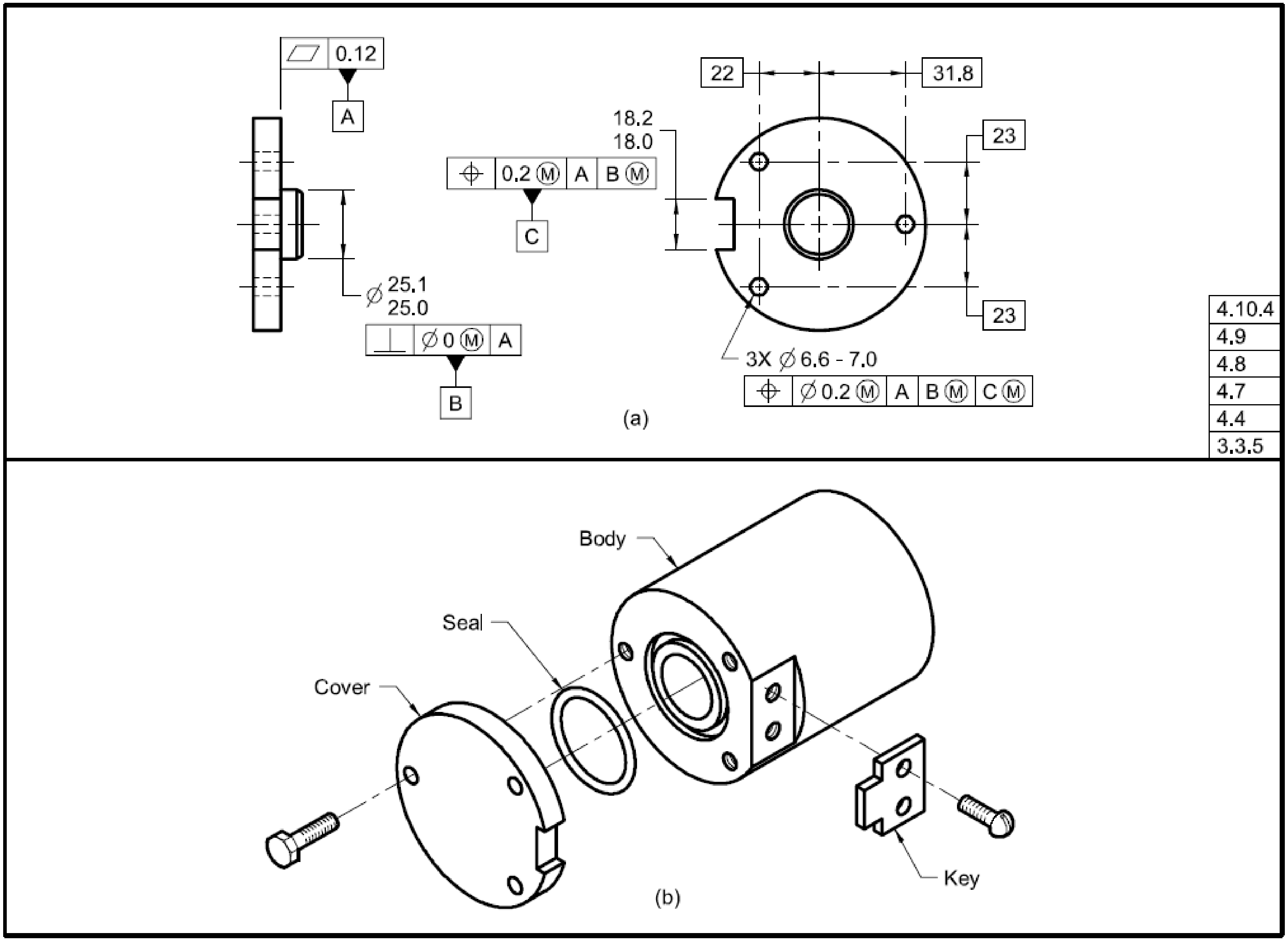
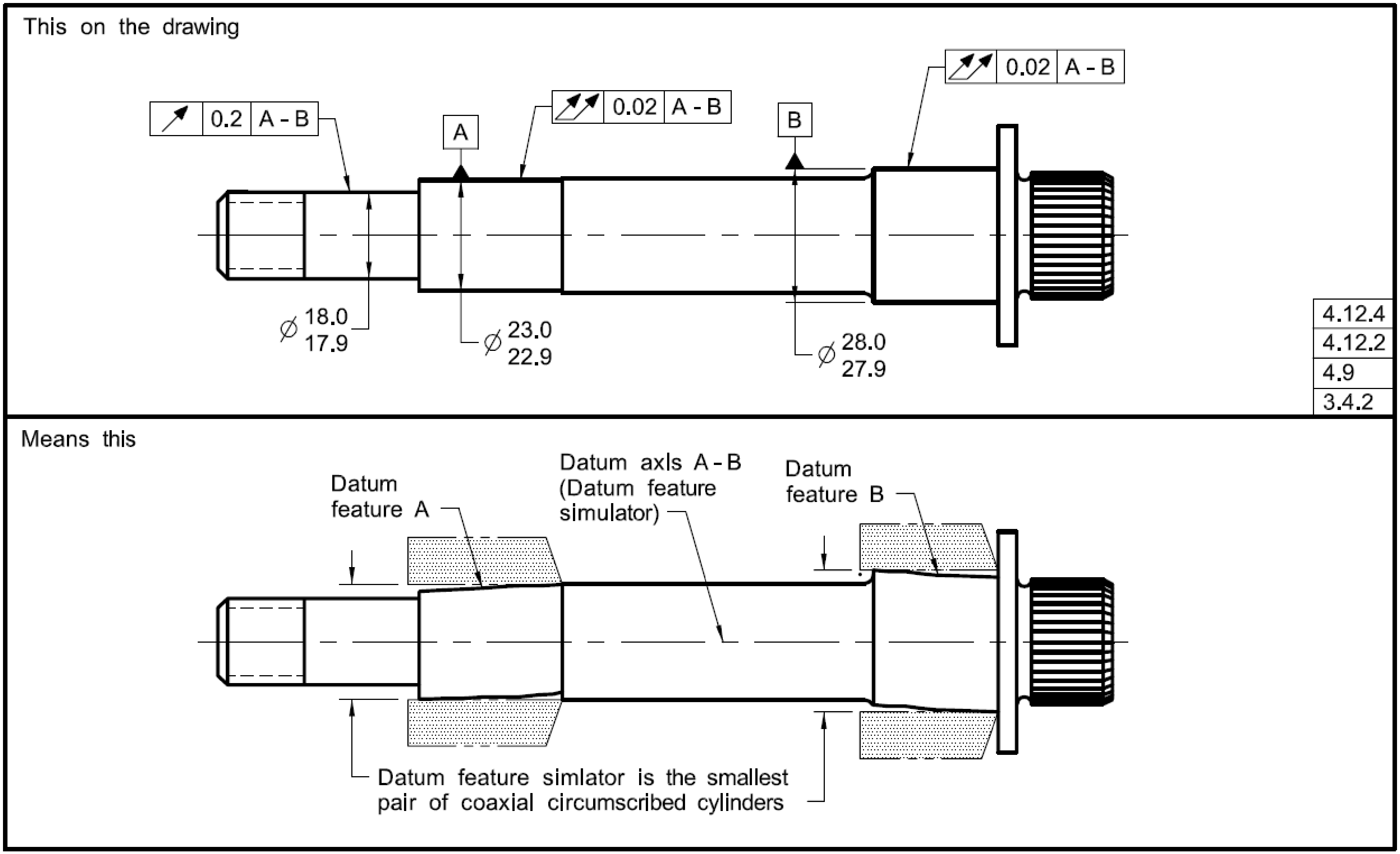


Fig. 4-25 Two Datum Features at RMB, Single Datum Axis



4.11.2 Irregularities on Datum Features

If irregularities on a datum feature are such that the part is unstable (that is, it rocks) when brought into contact with the corresponding datum feature simulator, the default stabilization procedure is per the candidate datum set as outlined in ASME Y14.5.1M. If a different procedure is desired (Chebychev, least squares,

4.11.3 Effect of Material Boundary Modifiers Applied to Datum Feature References

MMB, LMB, and RMB conditions may be applied/implied to any datum feature reference in a feature control frame. Modifiers applicable to datum features referenced in a feature control frame will affect the relationship of the part to the datum reference frame. See Figs. 4-20 and 4-21.

4.11.4 Specifying Datum Features RMB

Where a datum feature is referenced at RMB in a feature control frame, the datum feature simulator geometry originates at the MMB and progresses proportionally through the tolerance zone to make maximum possible contact with the extremities of the datum feature or collection of features. If another fitting routine is required, it shall be stated on the drawing.

(a) Primary Datum Feature: Diameter RMB. The datum is the axis of the datum feature simulator of the datum feature. The datum feature simulator (or unrelated actual mating envelope) is the smallest circumscribed (for an external feature) or largest inscribed (for an internal feature) perfect cylinder that makes maximum possible contact with the datum feature surface. See Figs. 4-3, illustration (d); 4-11; and 4-12.

(b) Primary Datum Feature: Width RMB. The datum is the center plane of the datum feature simulator of the datum feature. The datum feature simulator (or unrelated actual mating envelope) is two parallel planes at minimum separation (for an external feature) or maximum separation (for an internal feature) that makes maximum possible contact with the corresponding surfaces of the datum feature. See Figs. 4-3, illustration (b); 4-13; and 4-14.

Fig. 4-11 Establishment of Datums — For External Cylindrical Feature — RMB

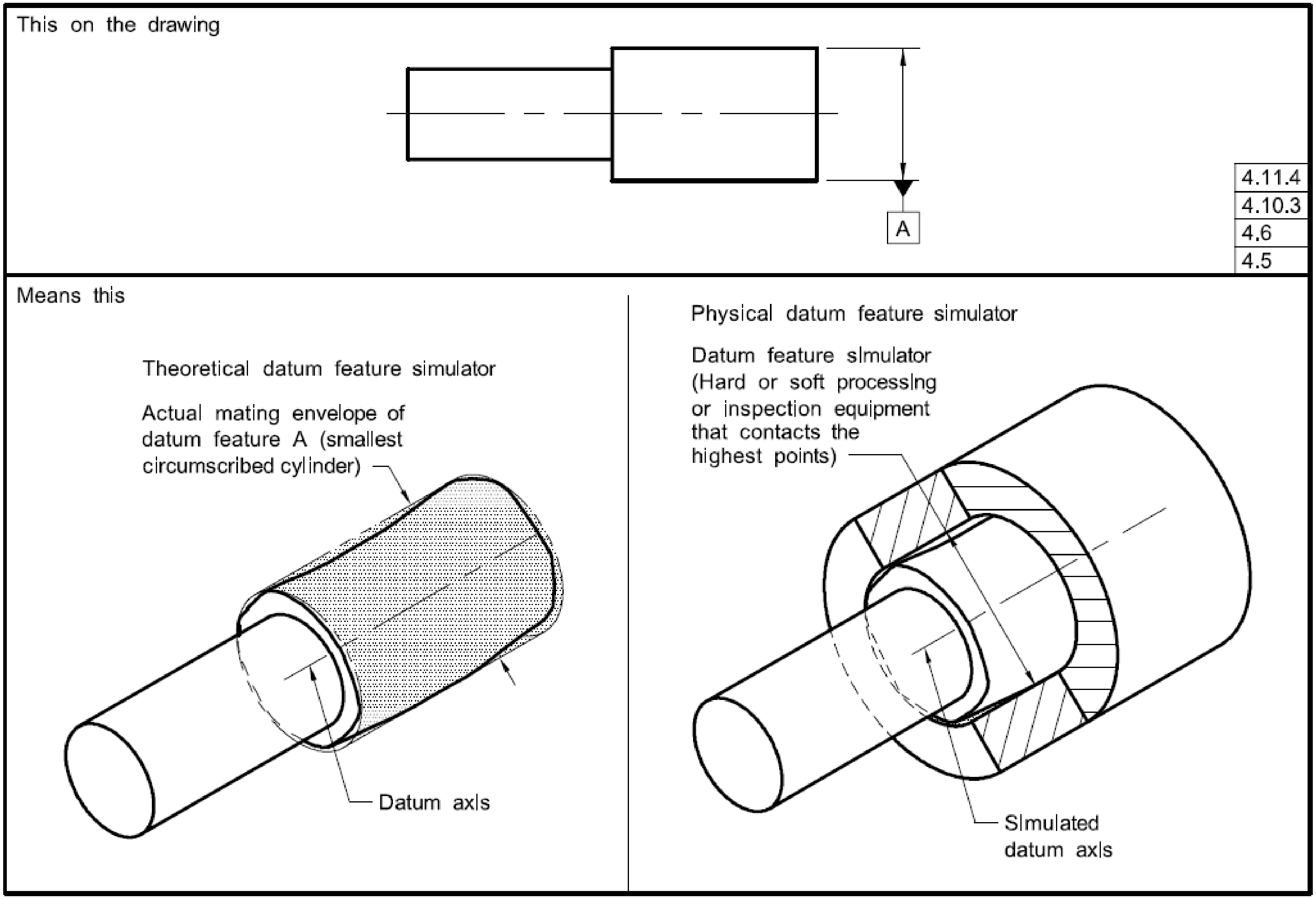


Fig. 4-12 Establishment of Datums — For Internal Cylindrical Feature — RMB

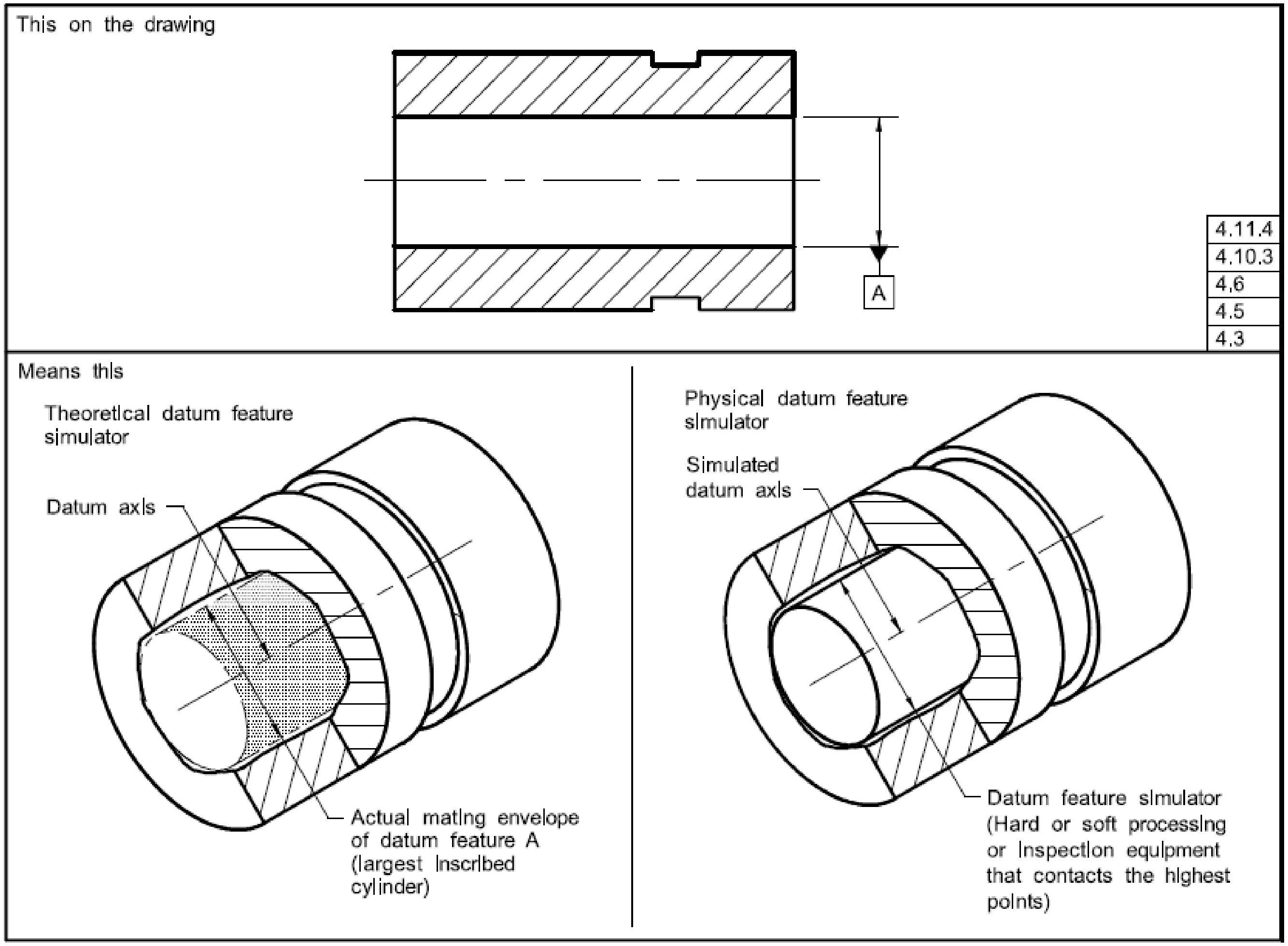


Fig. 4-13 Establishment of Datums — For External Datum Width — RMB

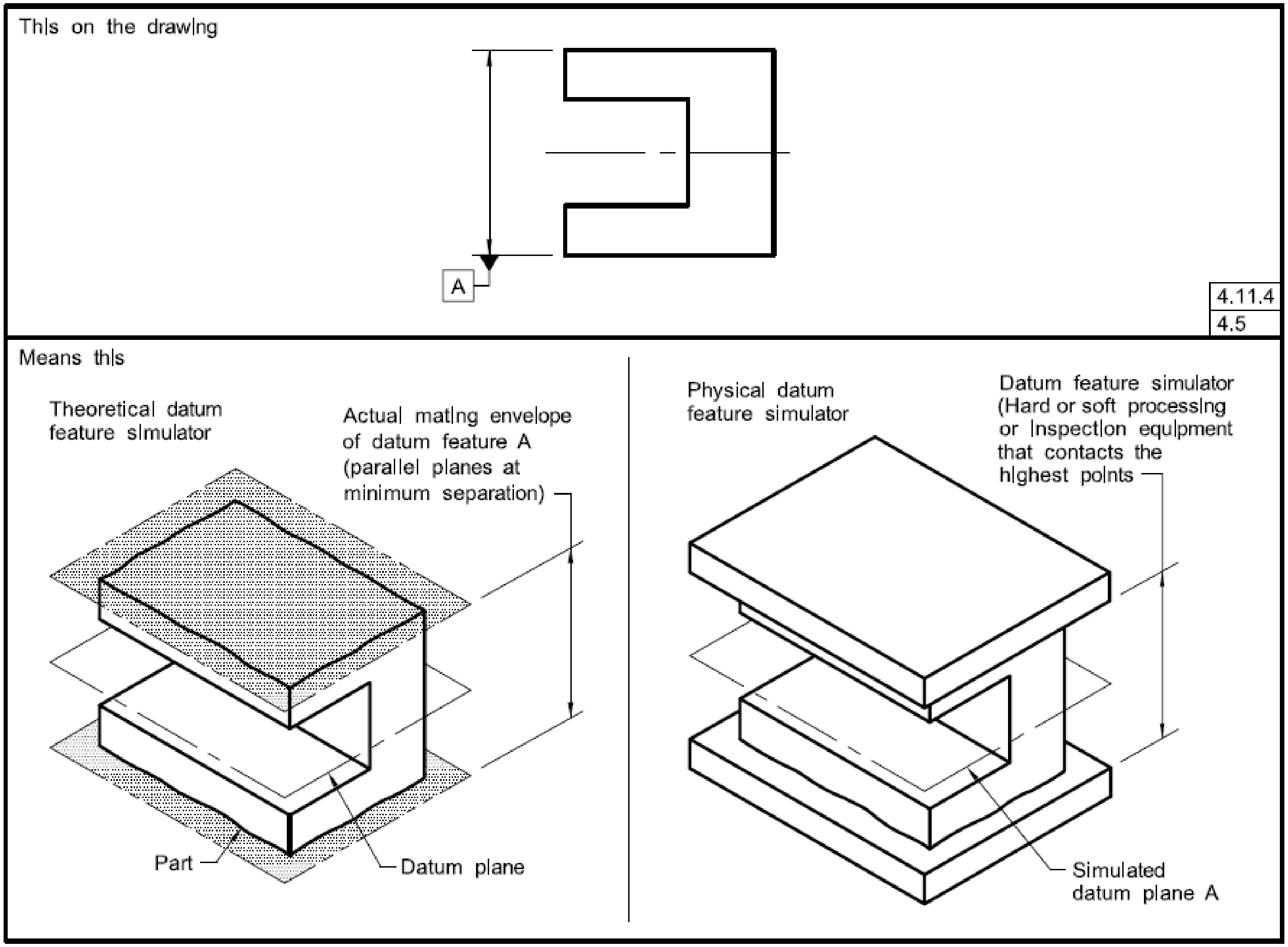
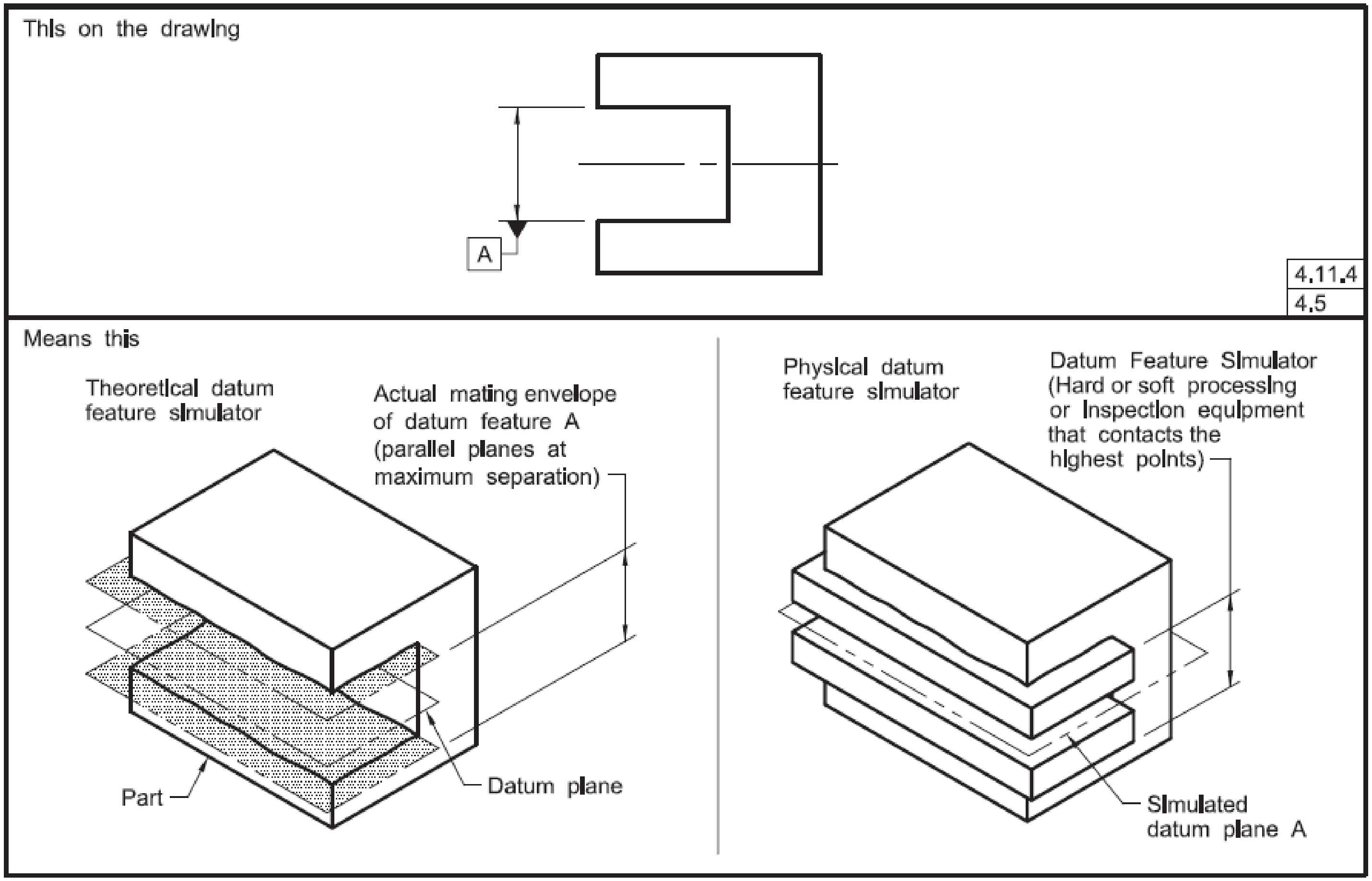


Fig. 4-14 Establishment of Datums — For Internal Datum Width — RMB



4.11.5 Specifying Datum Features at MMB

Where MMB is applied to a datum feature referenced in a feature control frame it establishes the datum feature simulator of the appropriate boundary. The appropriate boundary is determined by its collective effects of size, and any applicable geometric tolerances relative to any higher precedence datums. As a practical example, where a datum feature is applied on an MMB basis, machine and gaging elements in the processing equipment that remain constant may be used to simulate a datum feature simulator of the feature and to establish the simulated datum. To determine the applicable boundary, see para 4.11.6

4.11.6 Determining Size of Datum Feature Simulators at MMB

An analysis of geometric tolerances applied to a datum feature is necessary in determining the size of its datum feature simulator. A feature of size or pattern of features of size serving as a datum feature may have several MMB. These include the MMC of a datum feature of size or the collective effects of MMC and geometric tolerances. Datum feature precedence shall be respected, except in the case of a customized datum reference frame. See para. 4.22. Therefore, the appropriate MMB for determining the size of the datum feature simulator for an

(a) internal datum feature of size is the largest MMB that the datum feature(s) of size will contain while respecting the datum feature precedence.

(b) external feature of size is the smallest MMB that will contain the datum feature(s) of size while respecting the datum feature precedence. See Fig. 4-16 for examples of calculating the size of MMB.

4.11.6.1 Determining the Correct Maximum Material Boundary (MMB).
Datum feature D in Fig. 4-16 has three MMB. For an external feature of size the appropriate MMB is the smallest value that will contain the datum feature of size while respecting datum feature precedence.

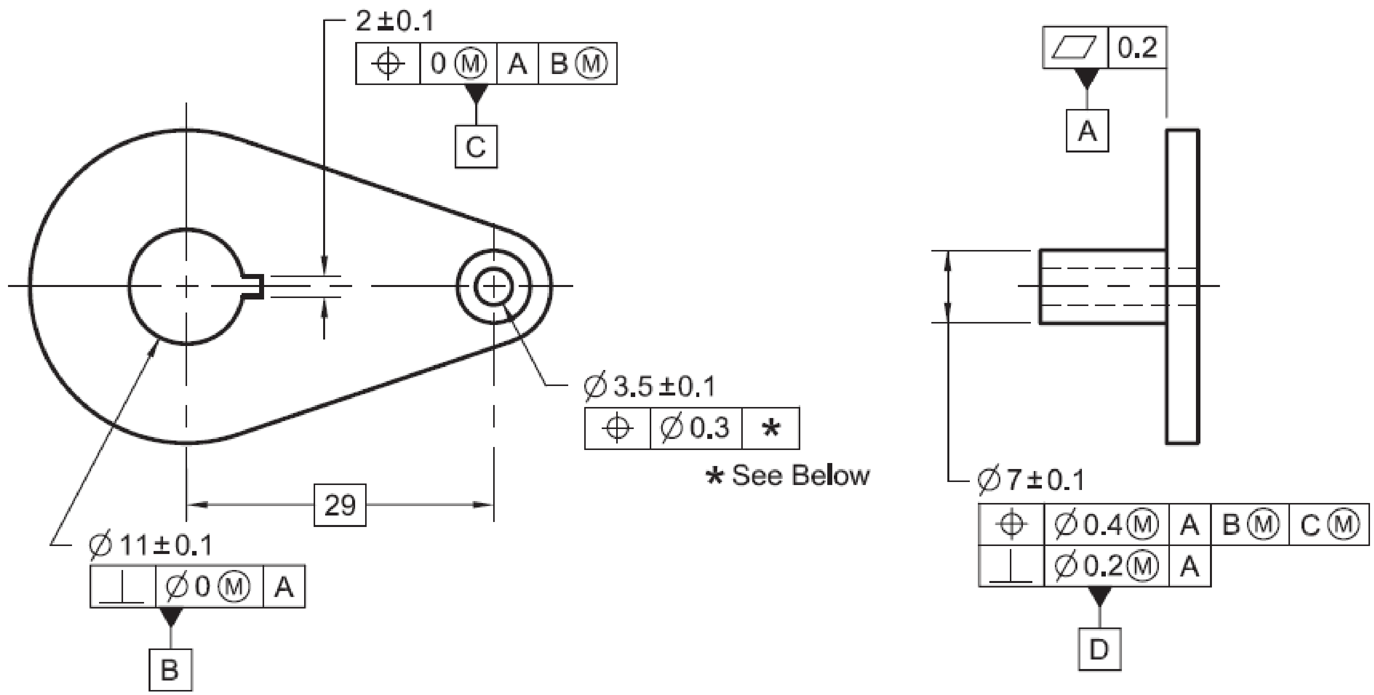
(a) In option (a) where datum feature D is referenced as primary, the appropriate MMB is the MMC of the feature or 7.1 mm (Rule #1).

(b) In option (b), where datum feature D is referenced as secondary to ensure that datum precedence is not violated, the collective effects of the MMC (7.1 mm diameter) and the perpendicularity tolerance (0.2 mm diameter) establishes an MMB of 7.3 mm diameter.

(c) In option (c), where datum feature D is referenced as tertiary to ensure that datum precedence is not violated, the collective effects of the MMC (7.1 mm diameter) and the position tolerance (0.4 mm diameter) establishes an MMB of 7.5 mm diameter. Since the perpendicularity tolerance is a refinement of the position tolerance, it is not additive.

4.11.6.2 Calculations for the MMB.
For the position tolerance applied to datum feature D, the appropriate MMB for datum features B and C are 10.9 mm diameter (10.9 minus 0 perpendicularity tolerance) and 1.9 mm (1.9 MMC minus 0 position tolerance), respectively.

Fig. 4-16 Example Calculations of Maximum Material Boundary



Option	Geometric Control	Maximum Material Boundary of datum feature D
(a)	⌖ ⌀0.3 D Ⓜ	7.1
(b)	⌖ ⌀0.3 A D Ⓜ	7.1 + 0.2 = 7.3
(c)	⌖ ⌀0.3 A B Ⓜ D Ⓜ	7.1 + 0.4 = 7.5

4.11.6.3
4.11.6.1
4.11.6

4.11.6.3 Clarifying Applicable MMB.

In cases where the boundary is not clear, or another boundary is desired, the value of the boundary shall be stated, enclosed in brackets, following the applicable datum feature reference and any modifier in the feature control frame. The term “BSC” or “BASIC” may be used to indicate that the datum feature simulator is located at the basic location of the datum feature. See Fig. 4-31, illustration (b).

EXAMPLE:

or

Where an MMB equal to MMC is the design requirement for a given datum feature, a zero geometric tolerance at MMC is specified to the datum feature as shown on datum features B and C in Fig. 4-16. See para. 7.3.4 and Fig. 6-14.

4.11.7 Specifying Datum Features at LMB

Where LMB is applied to a datum feature referenced in a feature control frame it establishes the datum feature simulator at the appropriate boundary. The appropriate boundary is determined by its collective effects of size, and any applicable geometric tolerances relative to any higher precedence datums. See para. 2.11 and Fig. 4-17. This example illustrates both secondary and tertiary datum features specified at LMB and simulated at LMB.

4.11.8 Multiple LMBs

A feature or pattern of features serving as a datum feature may have several LMB. These include the LMC of a feature or the collective effects of LMC and geometric tolerances. Datum precedence may not be violated, except in the case of a customized datum reference frame. In cases where the boundary is not clear, or another boundary is desired, the value of the boundary shall be stated following the applicable datum feature reference any modifier in the feature control frame.

EXAMPLE:

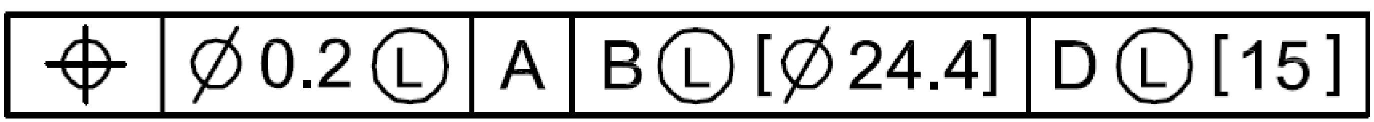
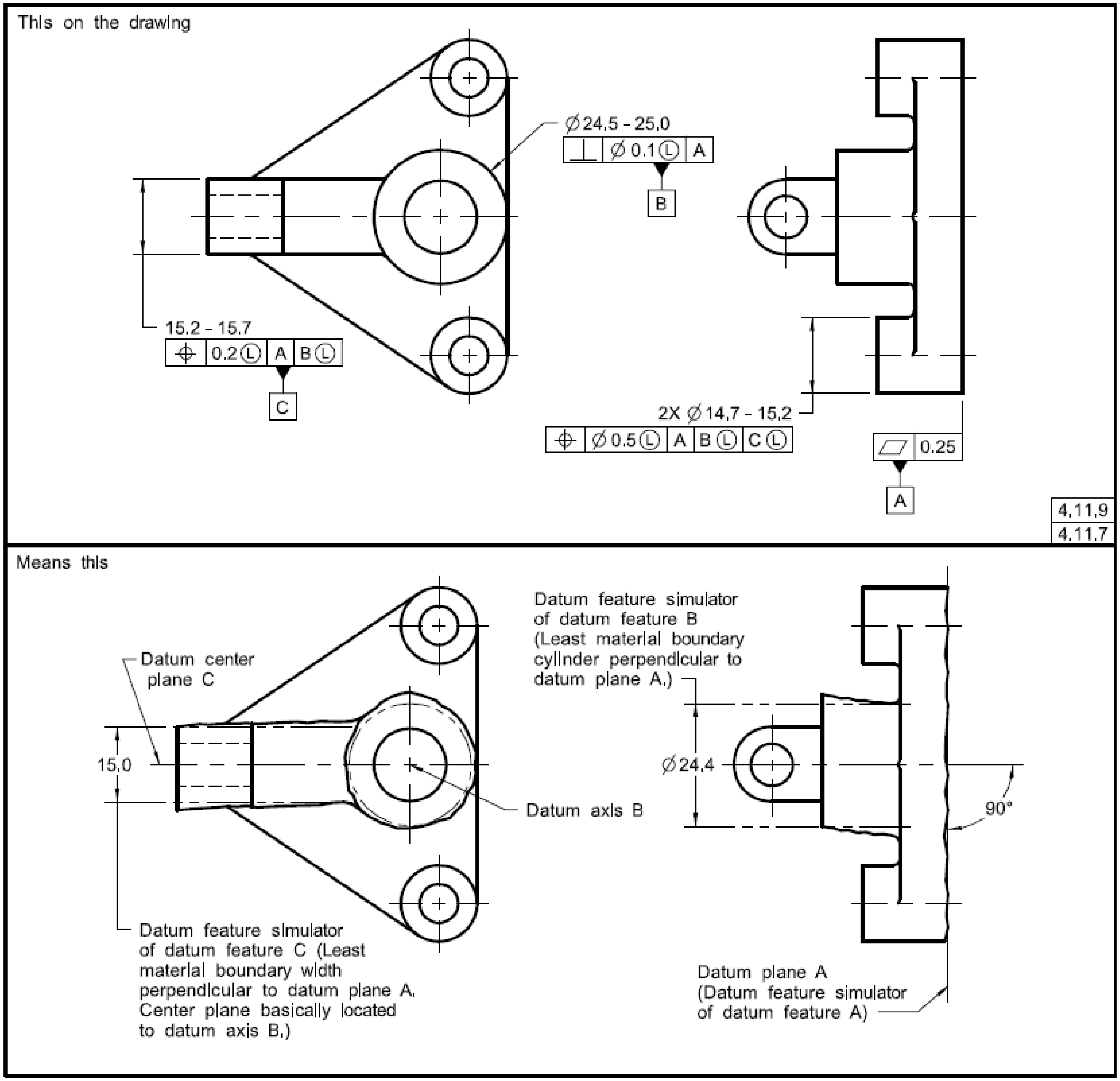


Fig. 4-17 Secondary and Tertiary Datum Features at LMB



4.11.9 Datum Feature Shift/Displacement

MMB or LMB modifiers applied to the datum feature reference will allow the datum feature to shift/displace from the boundary established by the datum feature simulator in an amount that is equal to the difference between the applicable (unrelated or related) actual mating envelope for MMB, actual minimum material envelope for LMB, or surface of the feature and the datum feature simulator. The datum reference frame is established from the datum feature simulator and not the datum features. See Fig. 4-17 for LMB, Figs. 4-18 and 4-24, datum feature B in Fig. 4-26 for MMB, and Fig. 4-30, illustration (b) for the surface. The datum feature shift/displacement shall always be limited or constrained by the datum feature simulator. If the datum feature simulator geometry is such that it does not fully limit or constrain the feature such as rotating away from the datum feature simulator beyond the established boundary limits, as shown in Fig. 4-31, illustration (c), then the feature must remain in contact with the datum feature simulator, and datum shift or displacement is not allowed. See para. 4.16.7 and datum feature A in Fig. 4-28.

Fig. 4-18 Secondary and Tertiary Datum Features at MMB

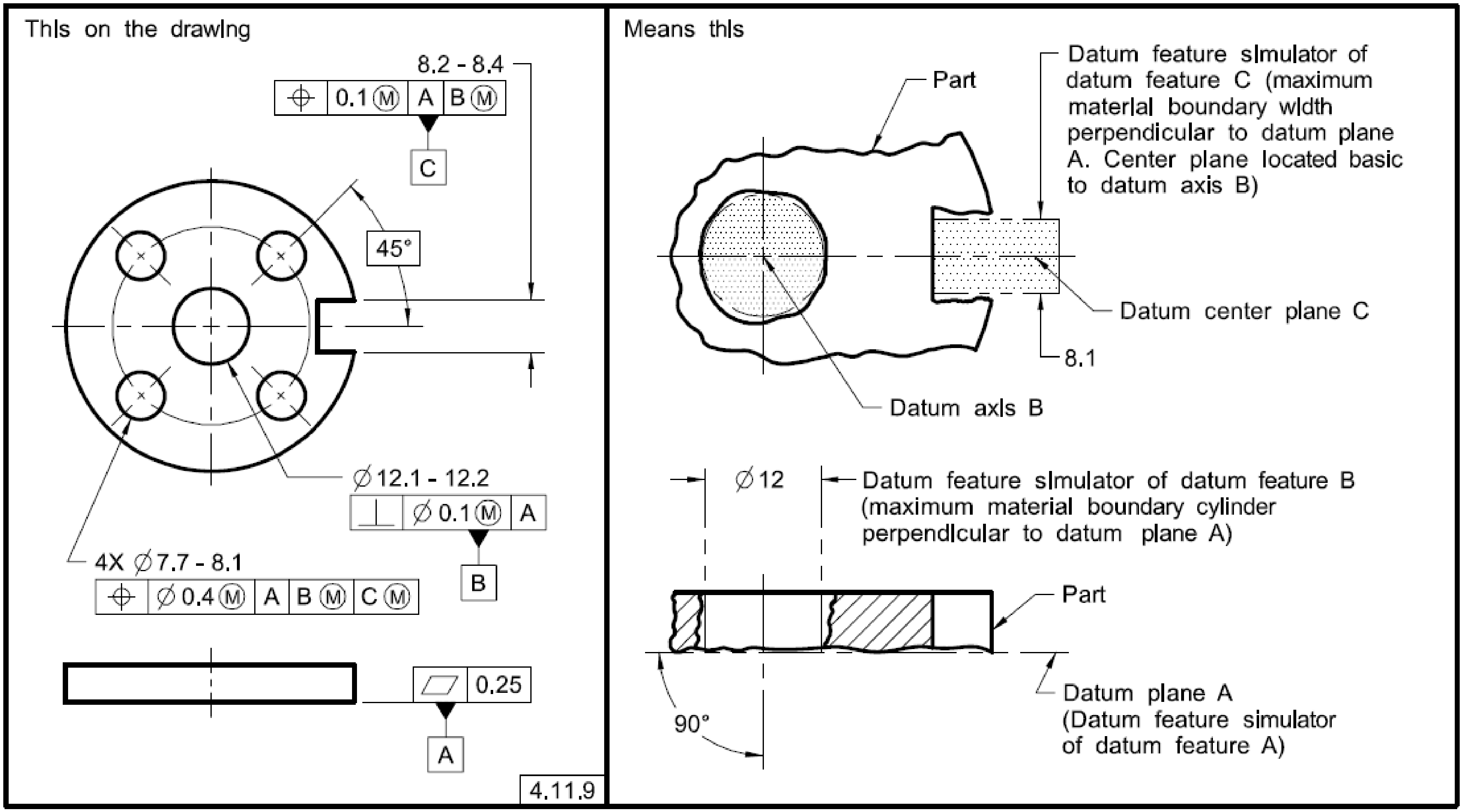


Fig. 4-26 Hole Pattern Identified as Datum

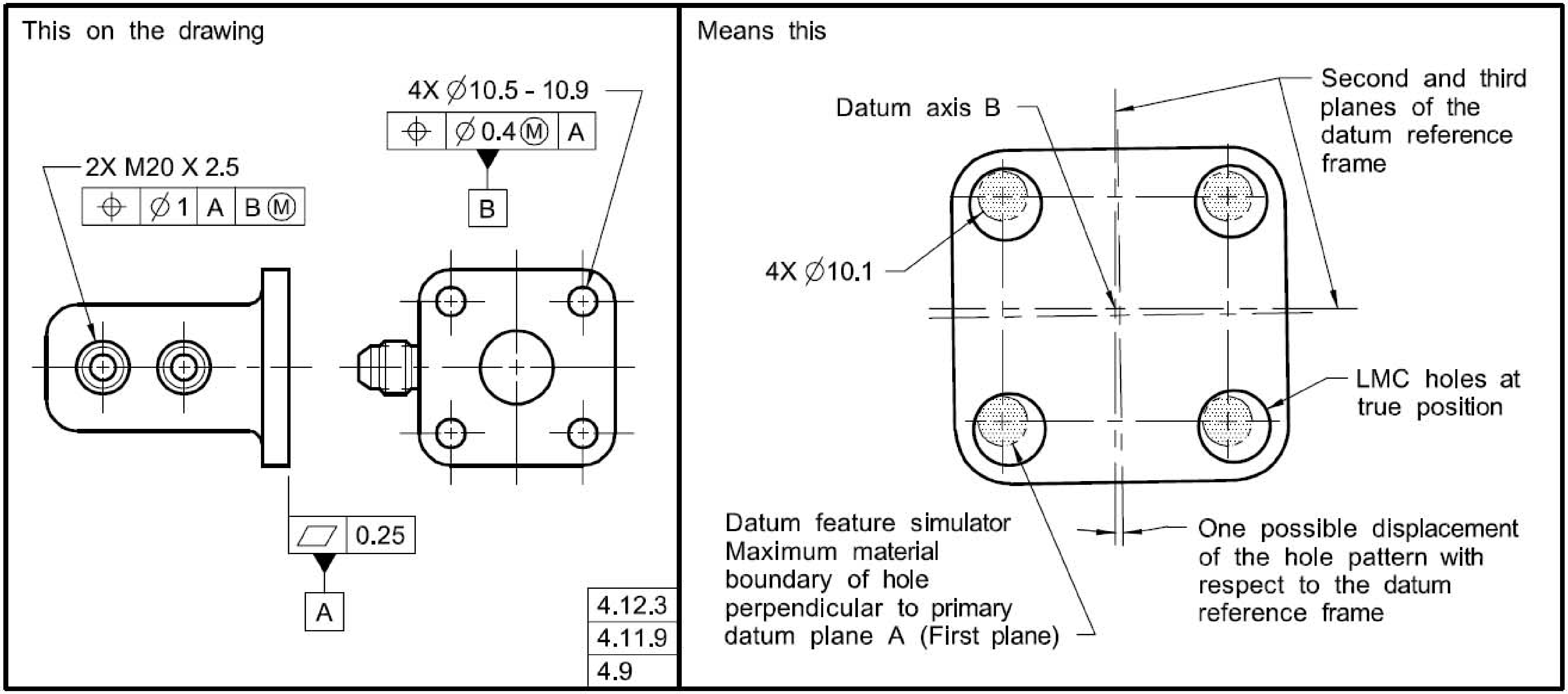


Fig. 4-30 Planar Datum Feature Constraining a Rotational Degree of Freedom

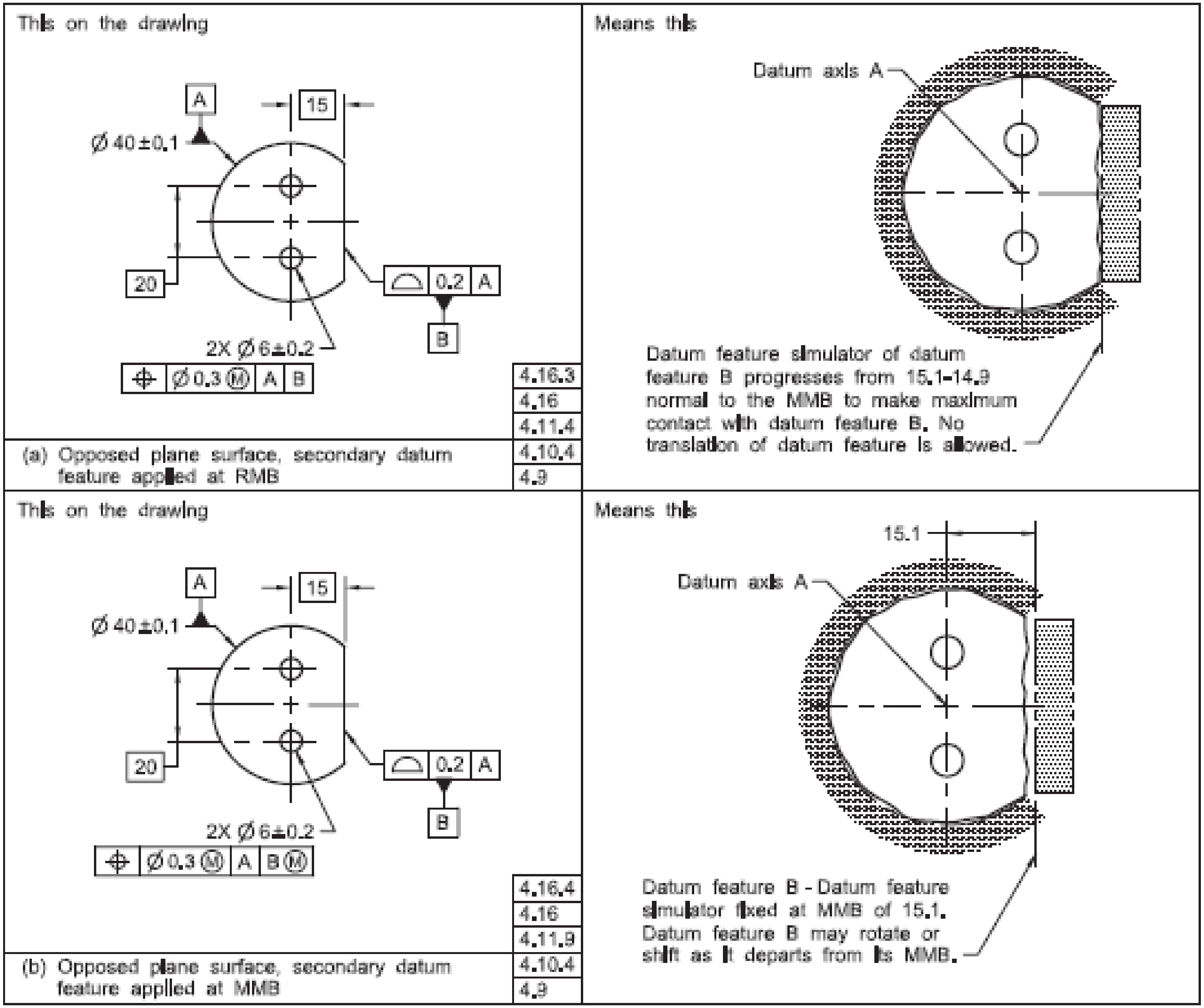
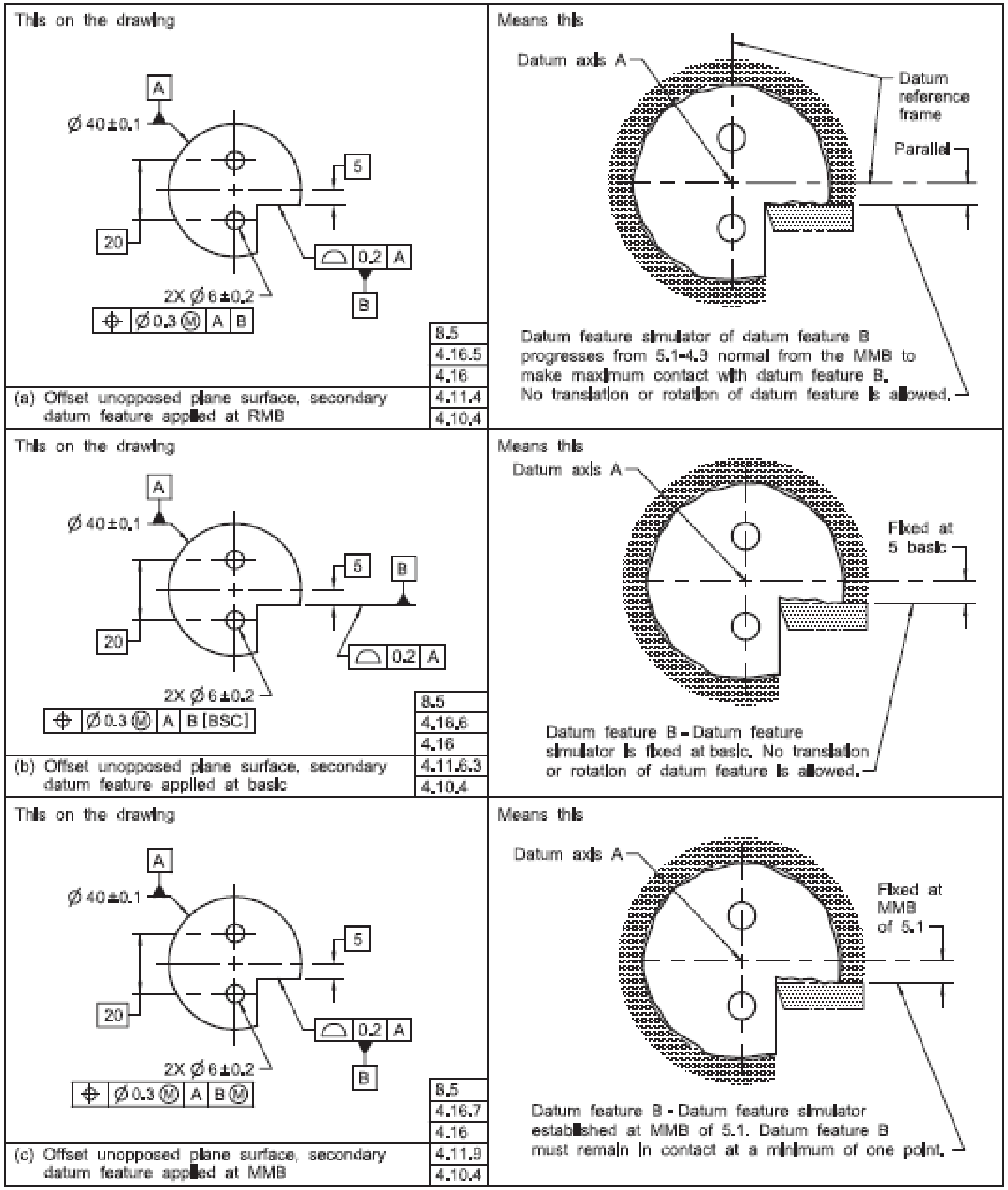


Fig. 4-31 Datum Modifier Effects — Plane Surface



4.11.10 Translation Modifier

Where it is necessary to indicate that the basic location of the datum feature simulator is unlocked and the datum feature simulator is able to translate within the specified geometric tolerance to fully engage the feature, the translation modifier is added to the feature control frame following the datum feature reference and any other applicable modifiers. See Figs. 4-19 and 4-32, illustration (b), and para. 3.3.26. When the translation modifier is applicable and the direction of movement is not clear, movement requirements shall be specified.

Fig. 4-9 Development of a Datum Reference Frame

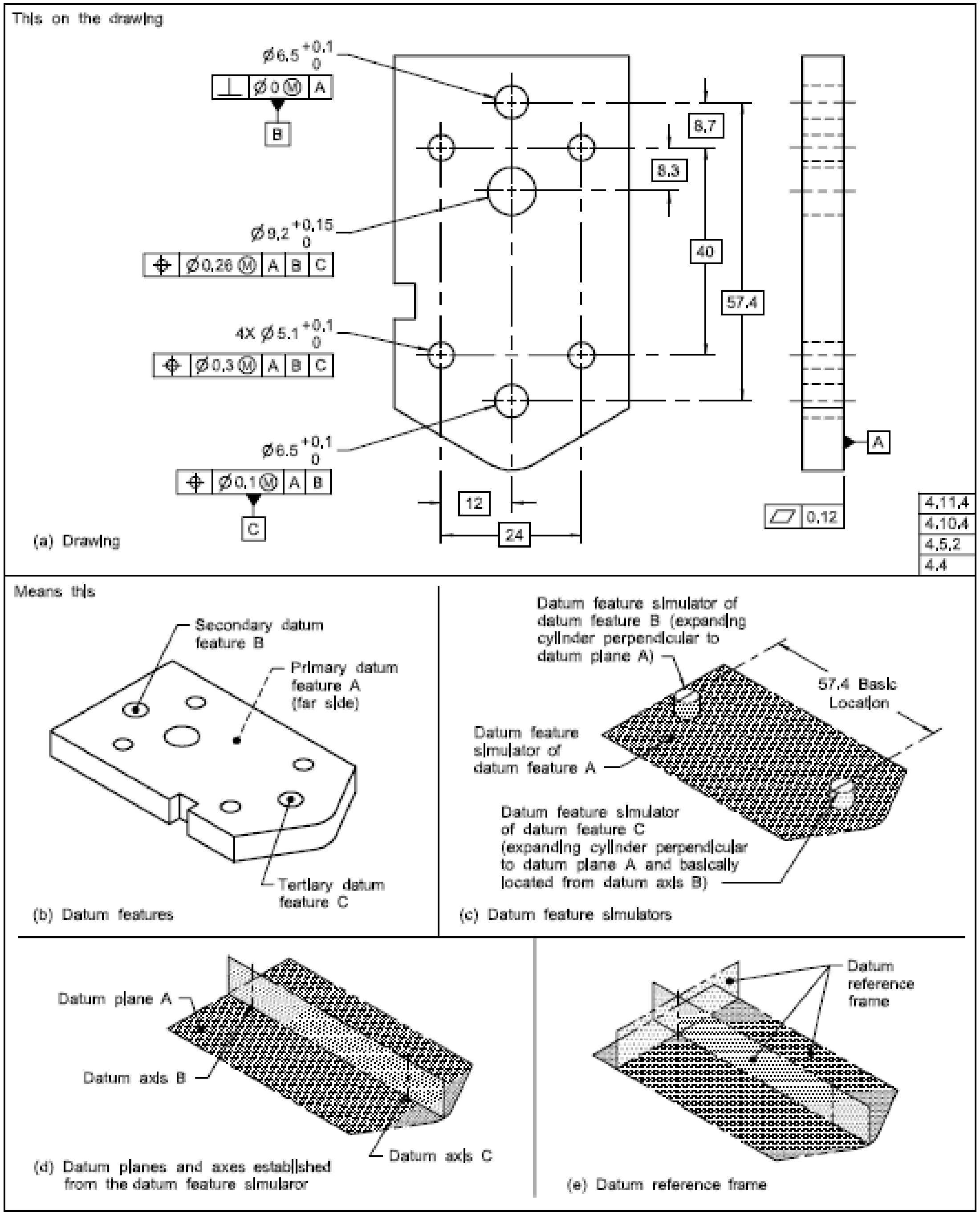


Fig. 4-19 Development of a Datum Reference Frame With Translation Modifier

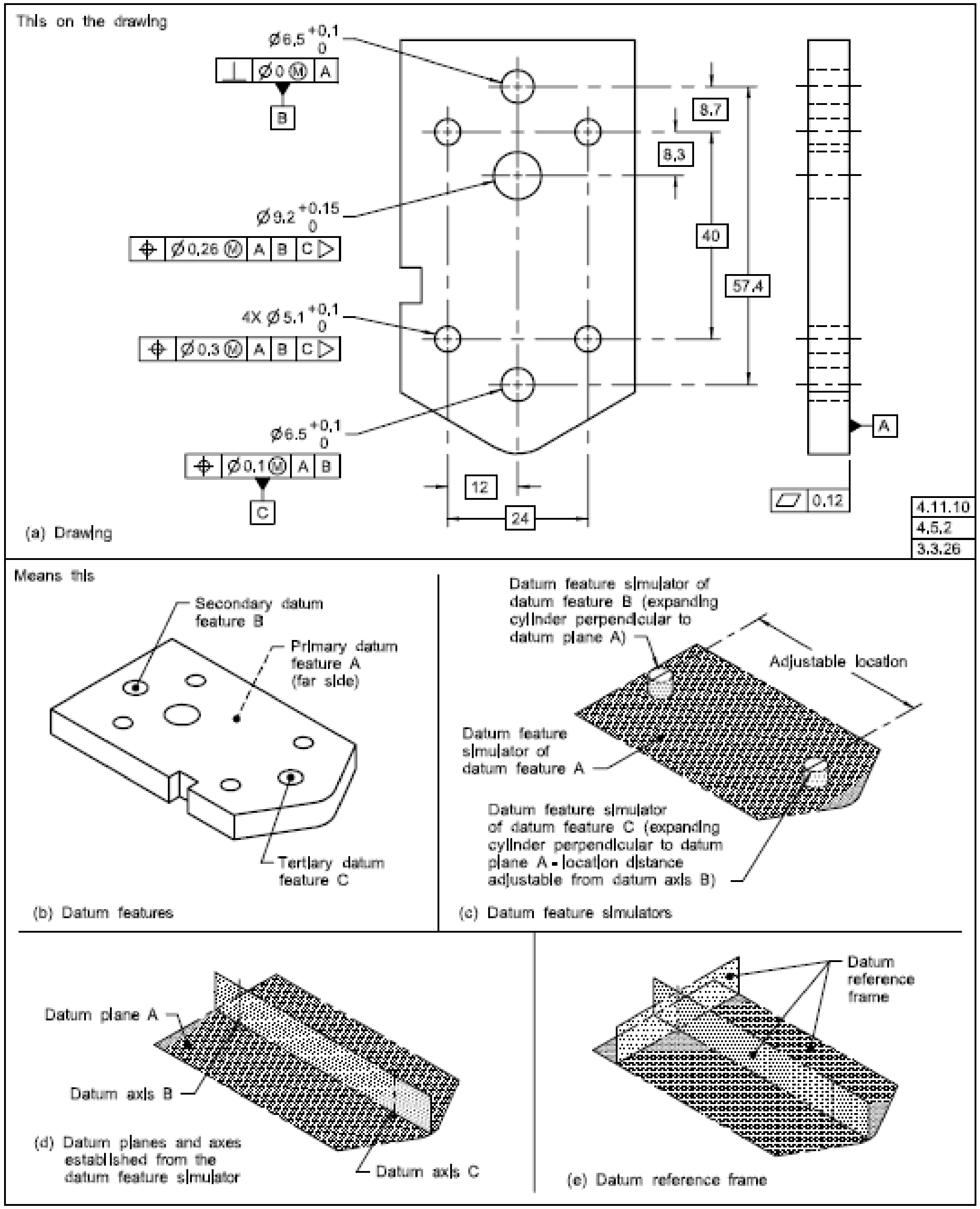
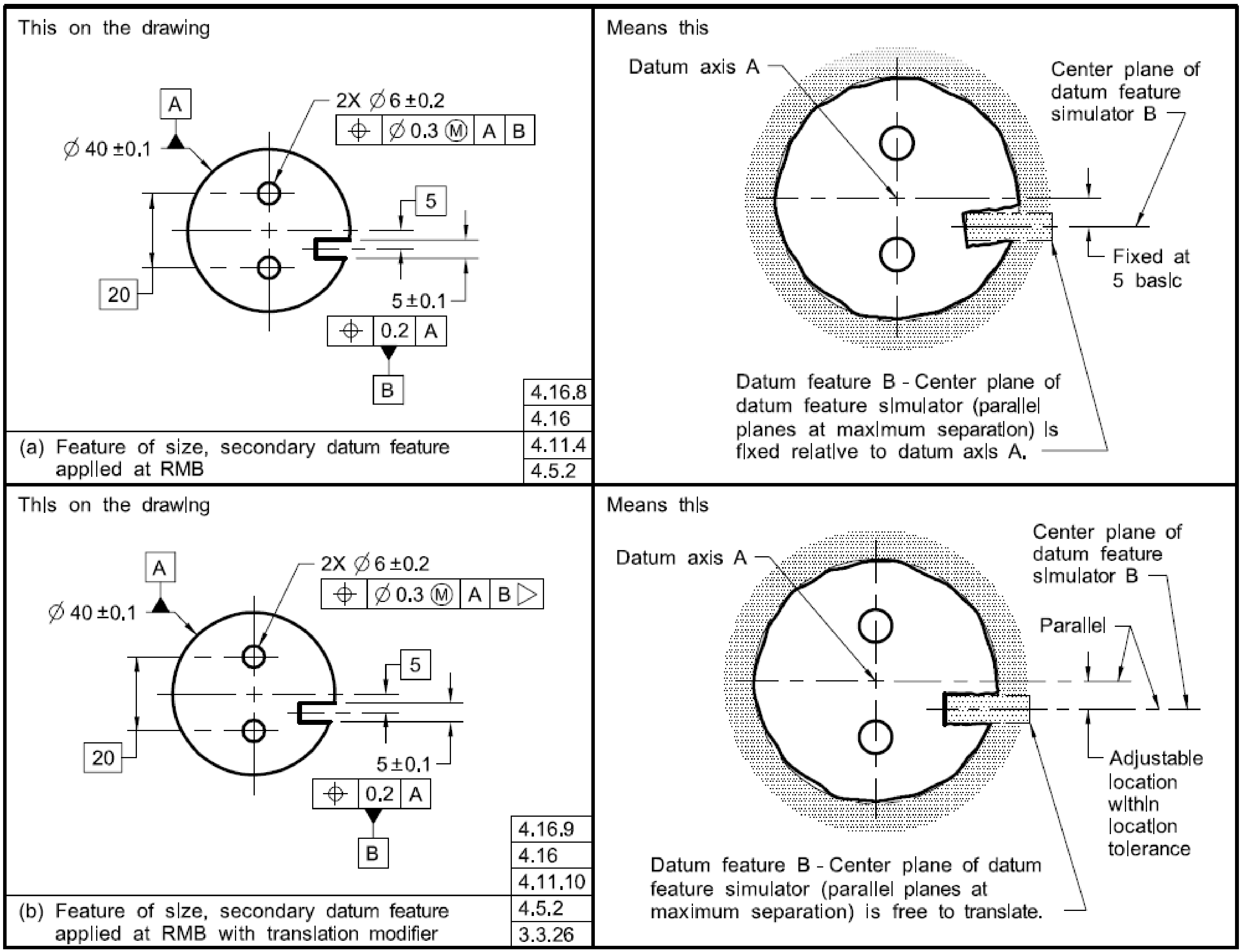


Fig. 4-32 Datum Modifier Effects — Size Feature



4.11.11 Effects of Datum Precedence and Datum Feature Material Boundary Conditions

Where datums are specified in an order of precedence, the material boundary condition at which the datum feature applies must be determined. The effect of its material boundary condition and order of precedence should be considered relative to fit and function of the part. Figures 4-20 and 4-21 illustrate a part with a pattern of holes located in relation to diameter A and surface B. As indicated by asterisks, datum

Fig. 4-20 Effect of Datum Modifier

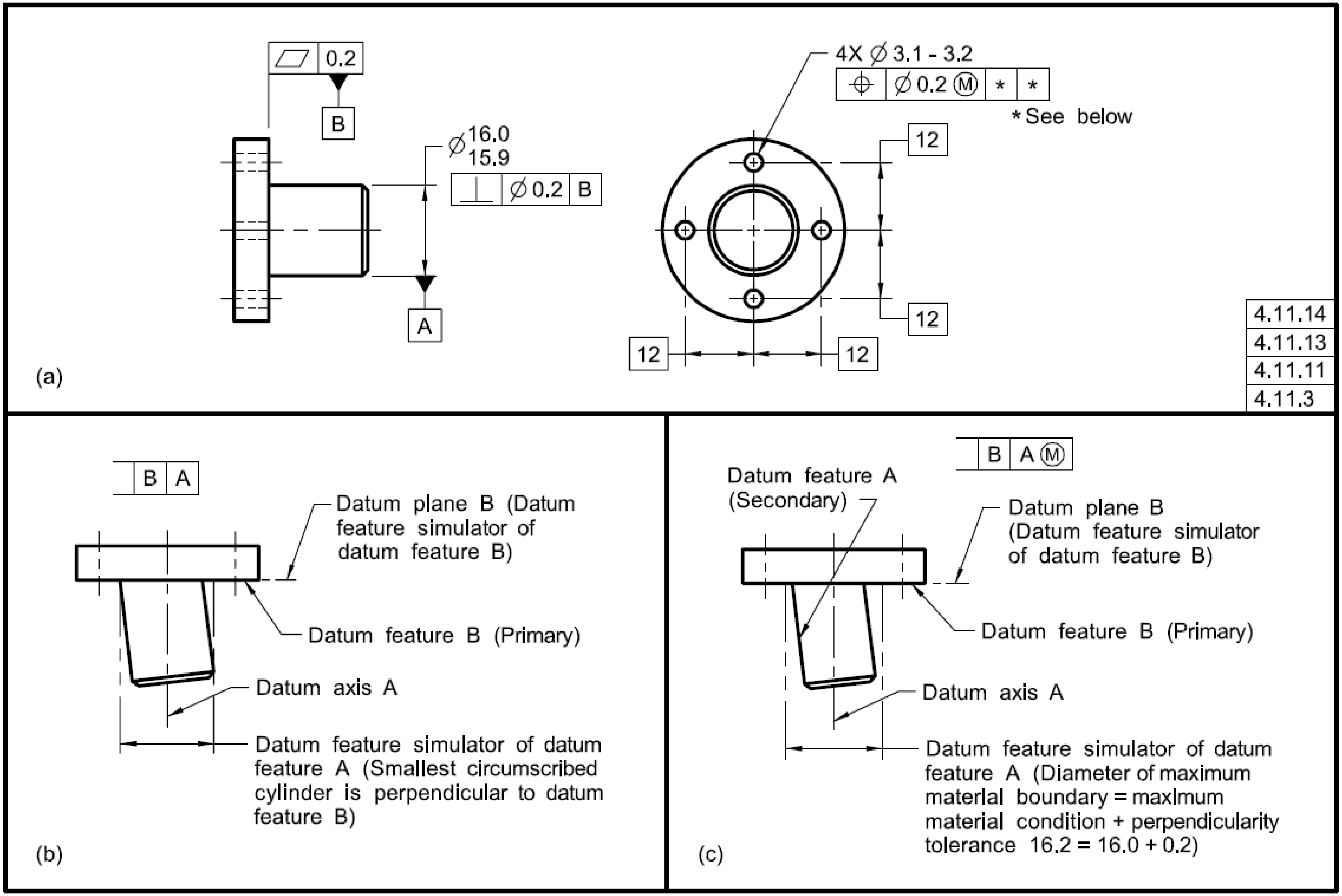
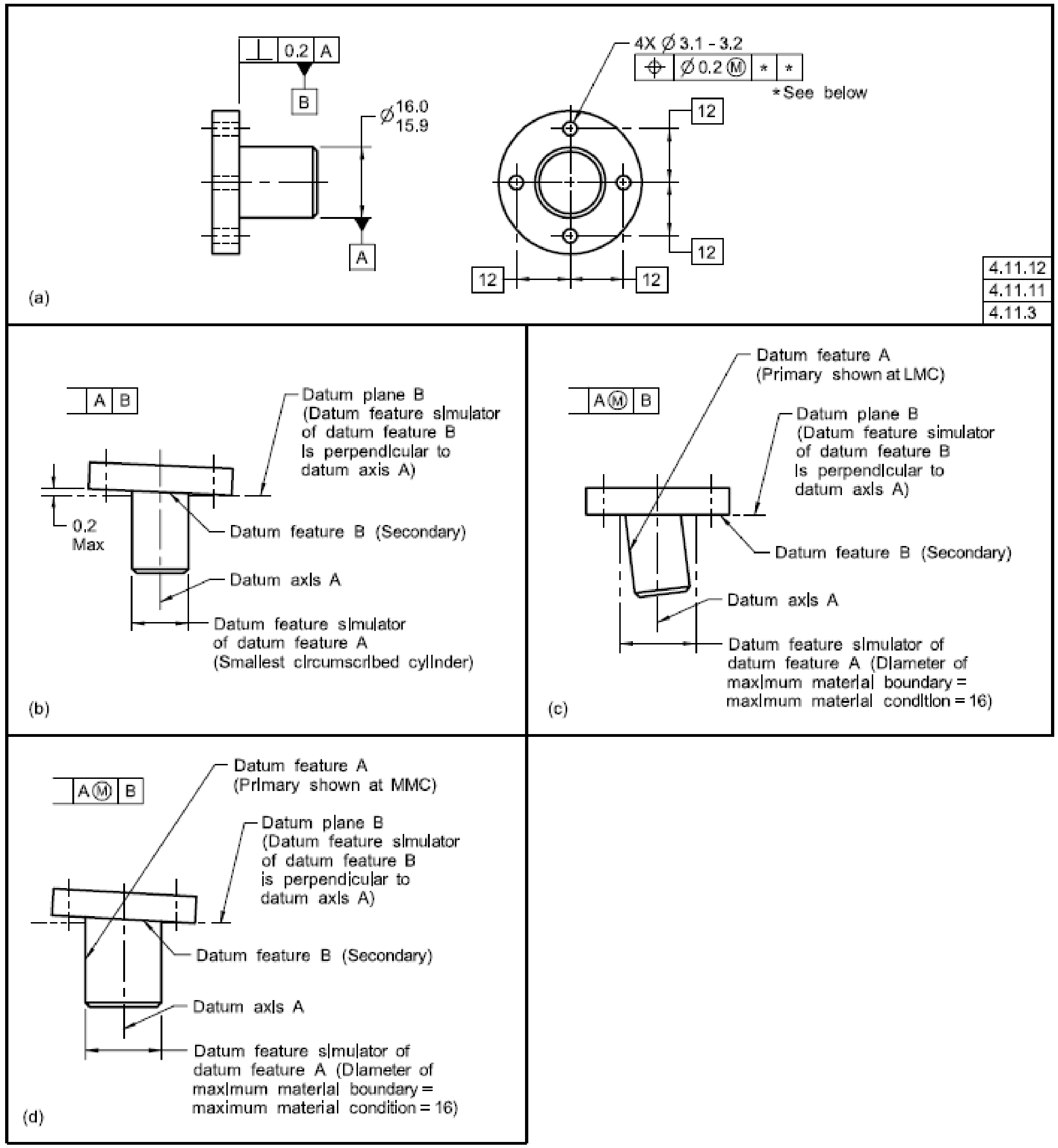


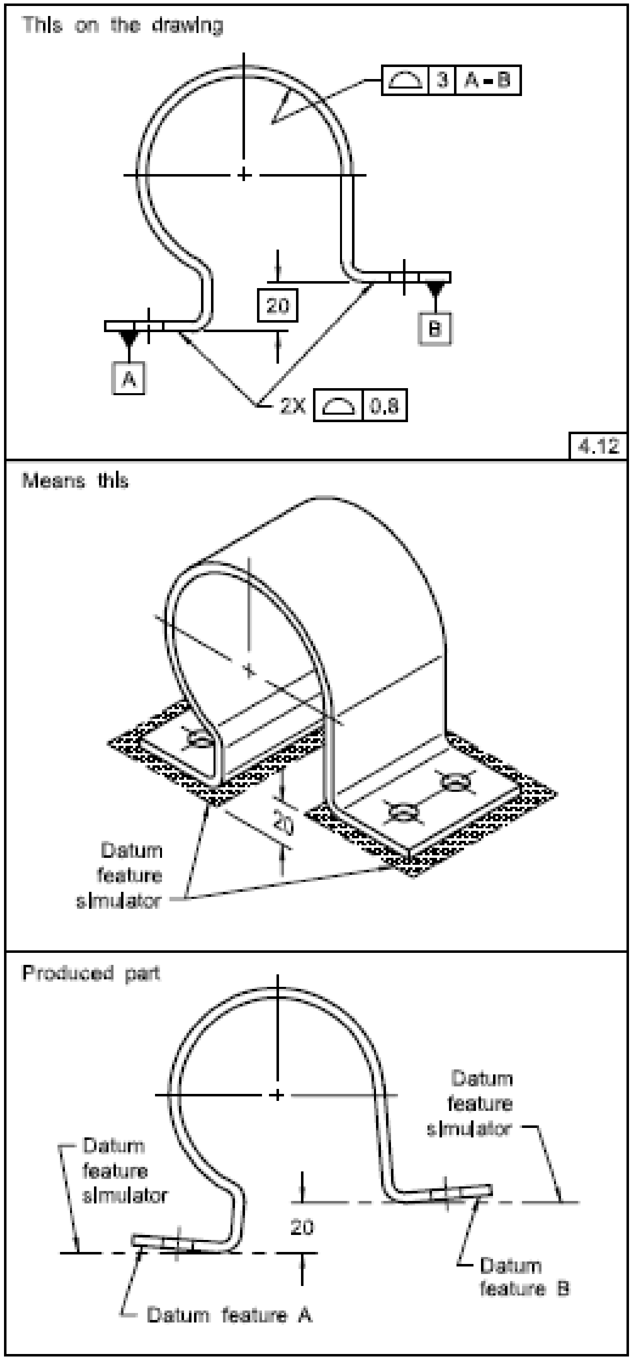
Fig. 4-21 Effect of Material Condition



4.12 MULTIPLE DATUM FEATURES

Where more than one datum feature is used to establisha datum feature simulator for a single datum, the appropriate datum feature reference letters and associated modifiers, separated by a dash, are entered in one compartment of the feature control frame. See para. 3.4.2 and Fig. 4-22. Since the datum features have equal importance, datum feature reference letters may be entered in any order within this compartment. Where the intent is clear, a datum feature reference letter may be used to define the multiple surfaces as a single datum feature.

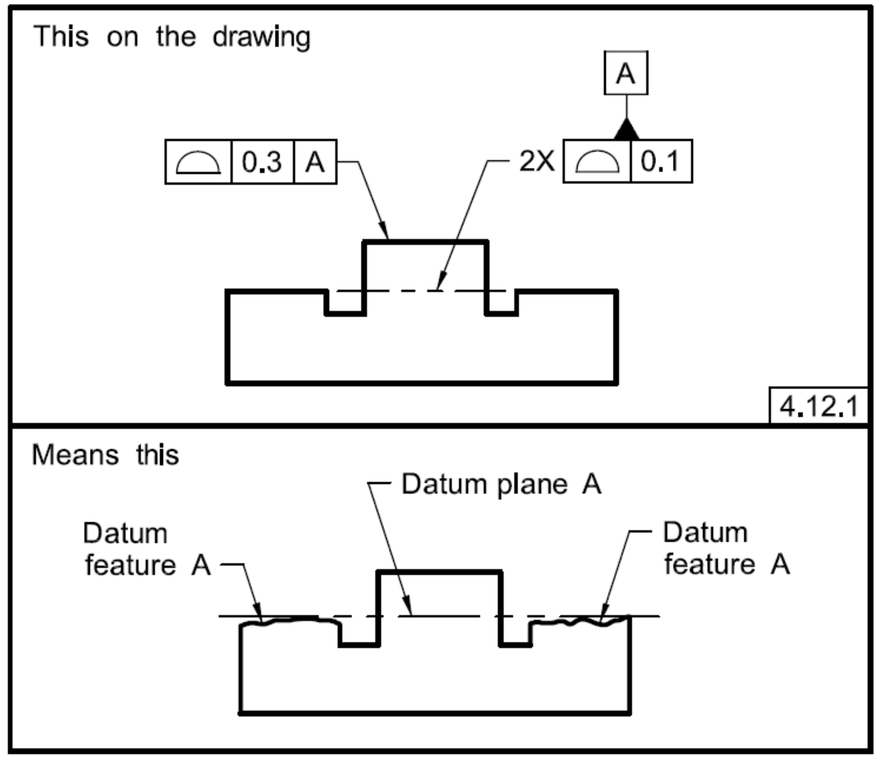
Fig. 4-22 Planar Multiple Datum



4.12.1 Simulation of a Single Datum Plane

Figure 4-23 is an example of a single datum plane simulated, as explained in para. 4.11.1, by coinciding with the datum feature simulator that simultaneously contacts the high points of two surfaces. Identification of two features to establish a single datum plane may be required where separation of the features is caused by an obstruction, such as in Fig. 4-23, or by a comparable opening (e.g., a slot). For controlling coplanarity of these surfaces, see Fig. 4-23 and para. 8.4.1.1. A single datum feature symbol may also be used to indicate that offset surfaces establish a single datum.

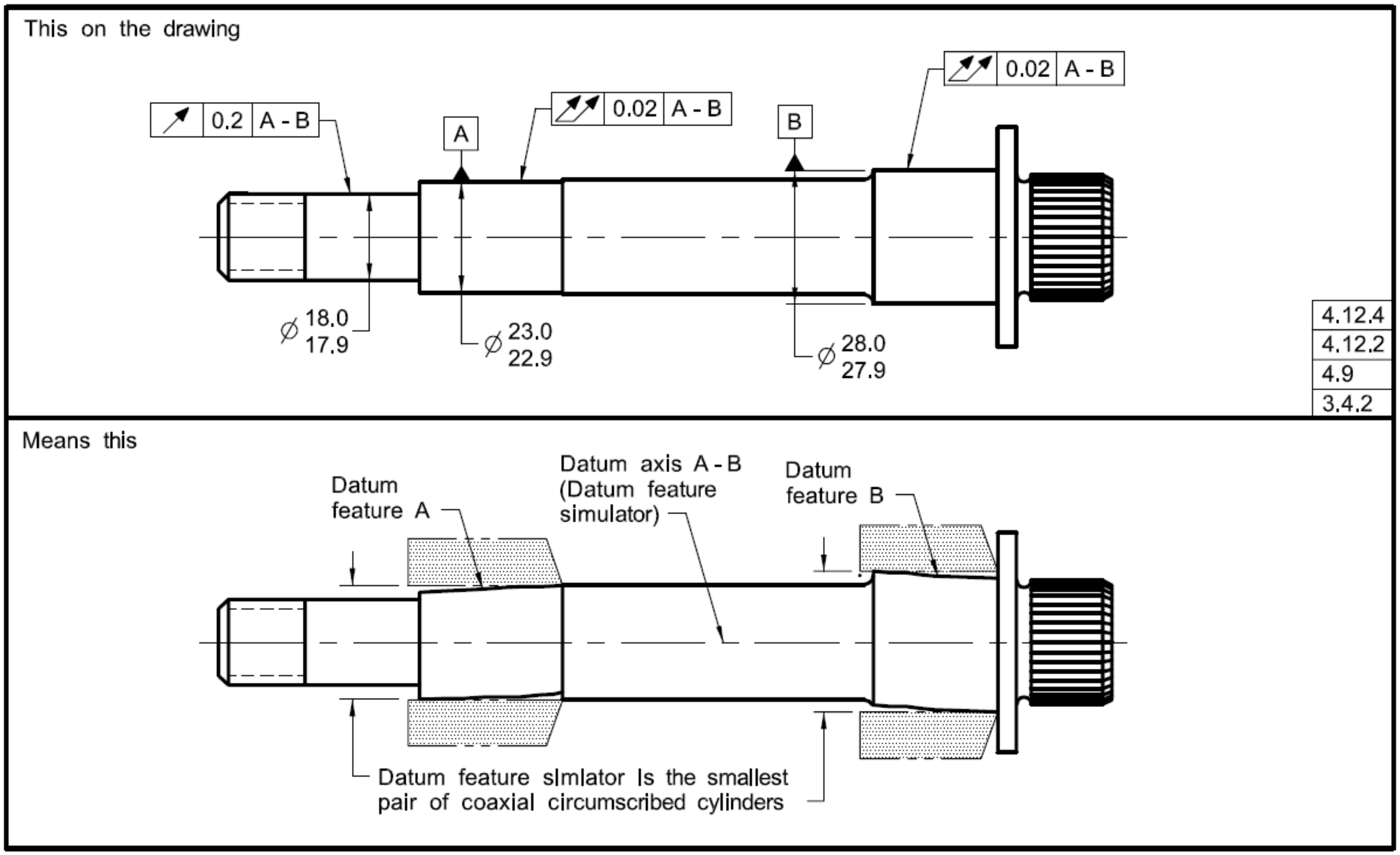
Fig. 4-23 Two Datum Features Establishing a Single Datum Plane



4.12.4 Pattern of Features of Size at RMB

Where RMB is applied in a feature control frame to multiple datum features of size used to establish a single datum, the datum feature simulator of each feature shall be fixed in a location relative to one another. The datum feature simulators shall expand or contract simultaneously from their MMB to their LMB until the datum feature simulators make maximum possible contact with the extremities of the datum feature(s). See Fig. 4-25.

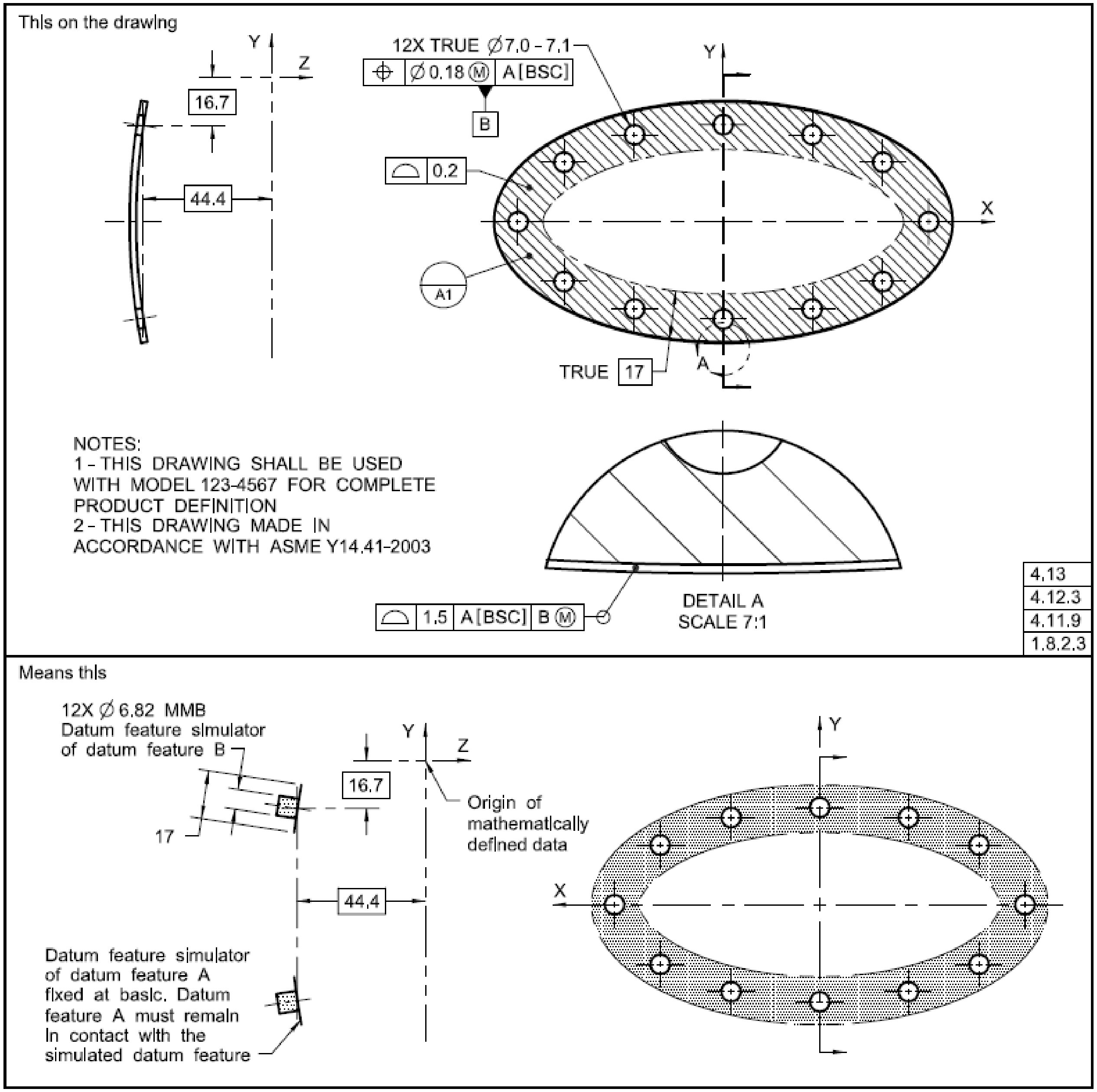
Fig. 4-25 Two Datum Features at RMB, Single Datum Axis



4.13 MATHEMATICALLY DEFINED SURFACE

It is sometimes necessary to identify a compound curve or a contoured surface as a datum feature. A mathematically defined feature shall be defined within a three-dimensional coordinate system. Where such a feature is specified as a datum feature, its datum feature simulator (derived from the math data) is used in establishing the datum reference frame. Aligning the high points of the datum feature with its datum feature simulator restricts movement of the part to the datum reference frame. Where the datum feature alone will not adequately restrict the required degrees of freedom of the part, additional datum features will be required. See Fig. 4-28.

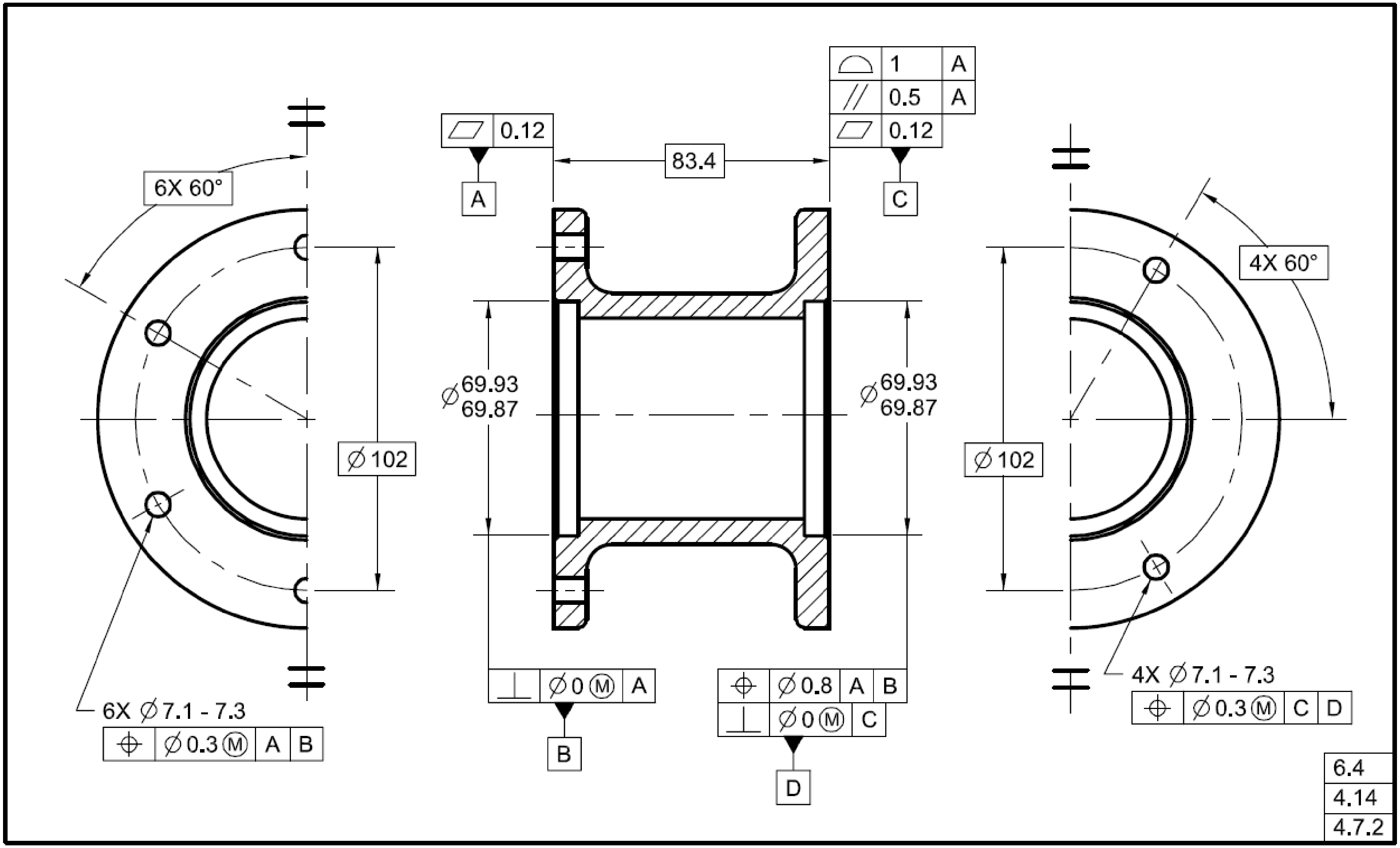
Fig. 4-28 Contoured Surface as a Datum Feature



4.14 MULTIPLE DATUM REFERENCE FRAMES

More than one datum reference frame may be necessary for certain parts, depending upon functional requirements. Where more than one datum reference frame is used and it is necessary to determine the relationships and calculate boundaries between the reference frames, the relationship between the datum reference frames shall be specified. In Fig. 4-4, datum features A and B establish one datum reference frame, while datum features C and D establish a different datum reference frame.

Fig. 4-4 Multiple Datum Reference Frames and Their Interrelationships



4.15 FUNCTIONAL DATUM FEATURES

Only the required datum features should be referenced in feature control frames when specifying geometric tolerances. An understanding of the geometric control provided by these tolerances (as explained in Sections 5 through 9) is necessary to determine effectively the number of datum feature references required for a given application. The functional requirements of the design should be the basis for selecting the related datum features to be referenced in the feature control frame. Figures 4-36 through 4-38 illustrate parts in an assembly where geometric tolerances are specified, each having the required number of datum feature references.

Fig. 4-36 Mating Parts for Functional Datum Selection

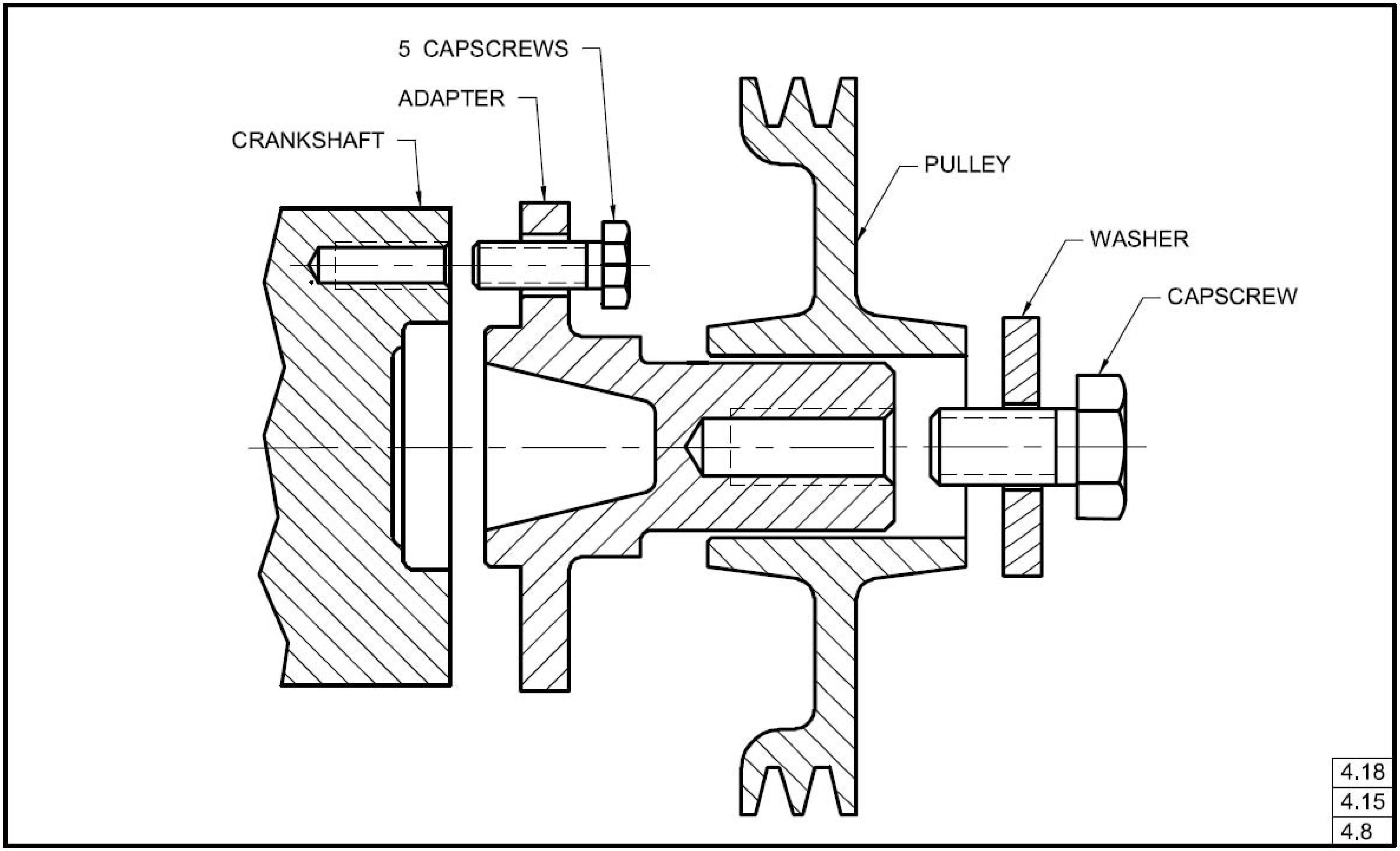


Fig. 4-37 Functional Datum Application — Pulley

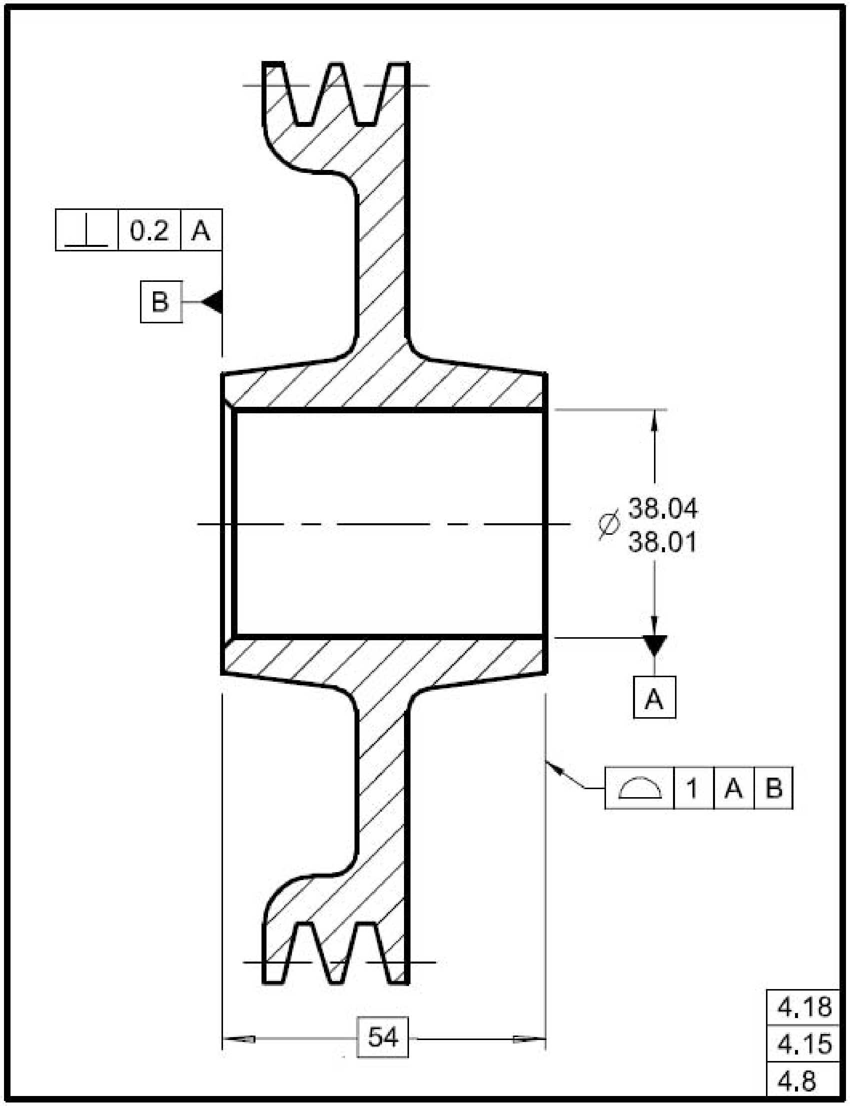
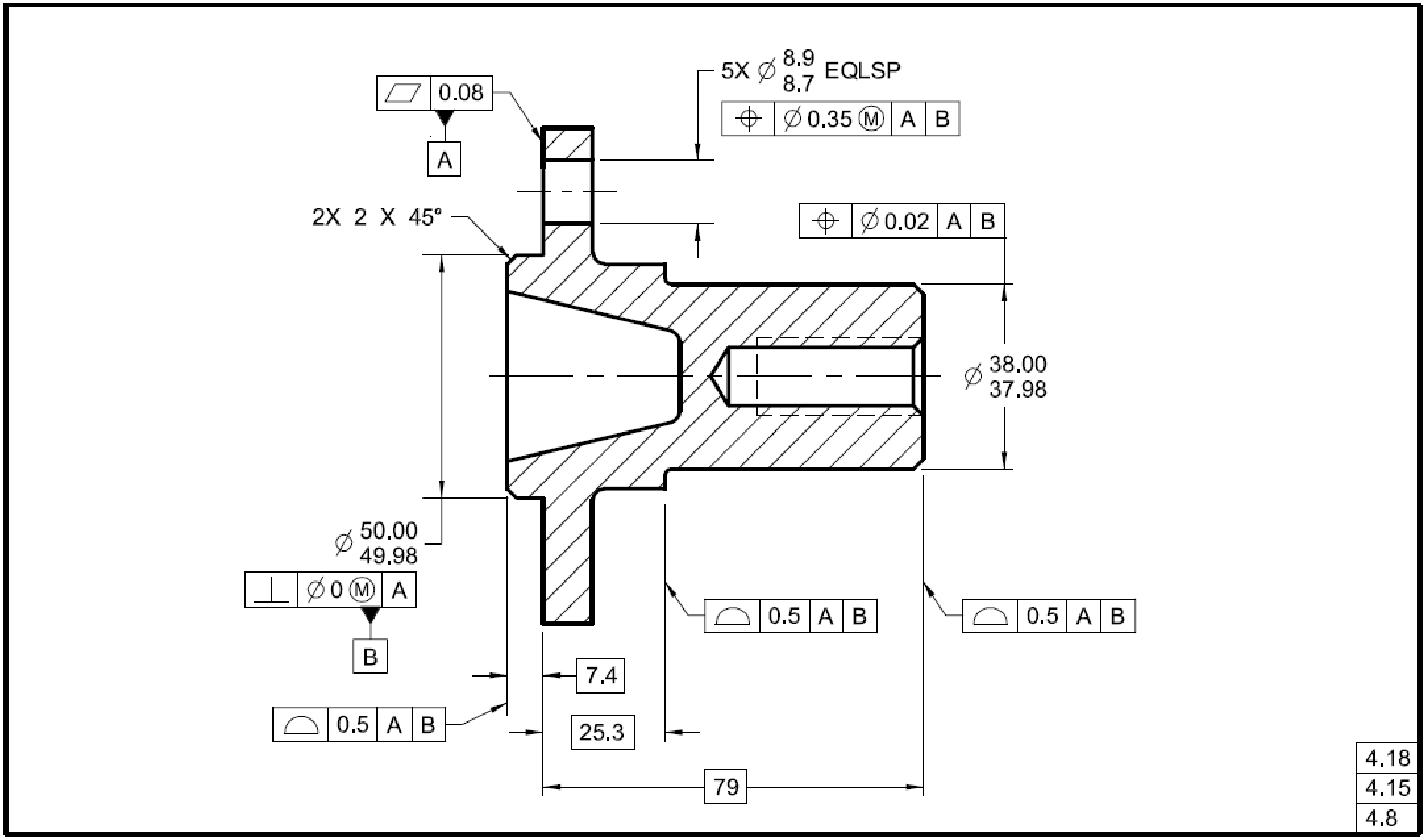


Fig. 4-38 Functional Datum Application — Adapter



4.16 ROTATIONAL CONSTRAINT ABOUT A DATUM AXIS OR POINT

Where a datum reference frame is established from a primary or secondary datum axis or point, a lower precedence datum feature surface or feature of size may be used to constrain rotation. See para. 4.10.4. Depending on functional requirements, there are many ways to constrain the rotational degrees of freedom about the higher precedence datum. Figures 4-8 and 4-29 through 4-32 illustrate the development of a datum reference frame based on the principles outlined in the datum feature simulator requirements. In these figures, datumfeature A establishes an axis. The lower precedence datum feature B is located (positioned or profiled) to datum feature A and is then used to orient the rotational degrees of freedom to establish the datum reference frame that is used to locate the two 6-mm diameter holes. Depending on functional requirements, this lower precedence datum feature may apply at RMB or be modified to apply at MMB or LMB. The datum reference frame is established from the datum feature simulators and not the datum features.

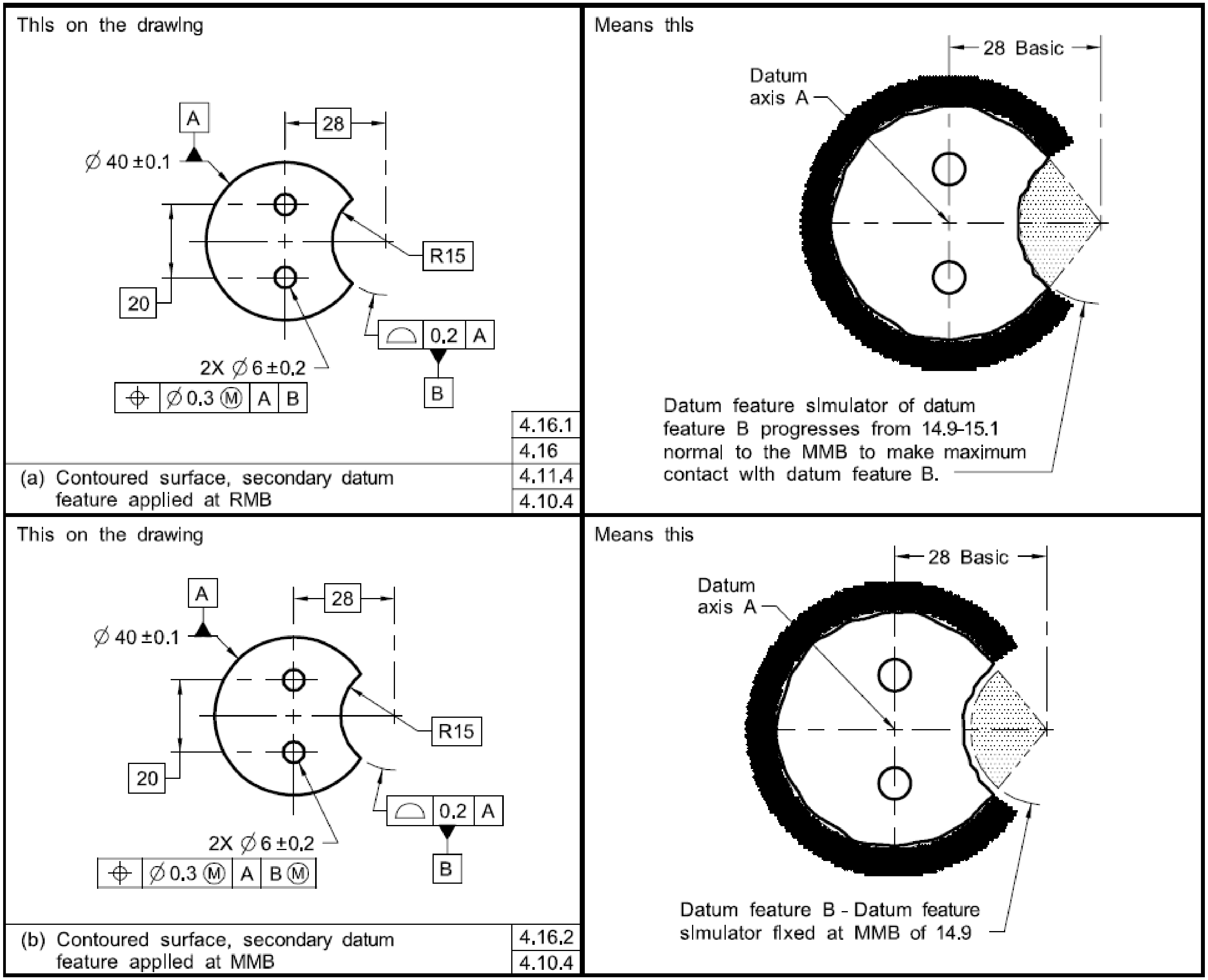
4.16.1 Contoured Datum Feature at RMB Constraining a Rotational Degree of Freedom

In Fig. 4-29, illustration (a), datum feature B applies at RMB. This requires the datum feature simulator geometry to originate at the MMB of R14.9 mm and progress through the profile tolerance zone toward the LMB of R15.1 mm until it makes maximum contact with datum feature B and constrains the rotational degree of freedom of the part around the axis of the datum feature simulator from datum feature A.

4.16.2 Contoured Datum Feature at MMB Constraining a Rotational Degree of Freedom

In Fig. 4-29, illustration (b), datum feature B is modified to apply at MMB. This requires the datum feature simulator to be fixed at the MMB of R14.9 mm and thus orients the two planes that originate at the axis of the datum feature simulator of datum feature A. Datum feature B may rotate within the confines created by its departure from MMB and might not remain in contact with the datum feature simulator.

Fig. 4-29 Contoured Datum Feature Constraining a Rotational Degree of Freedom



(a) Contoured surface, secondary datum feature applied at RMB

(b) Contoured surface, secondary datum feature applied at MMB

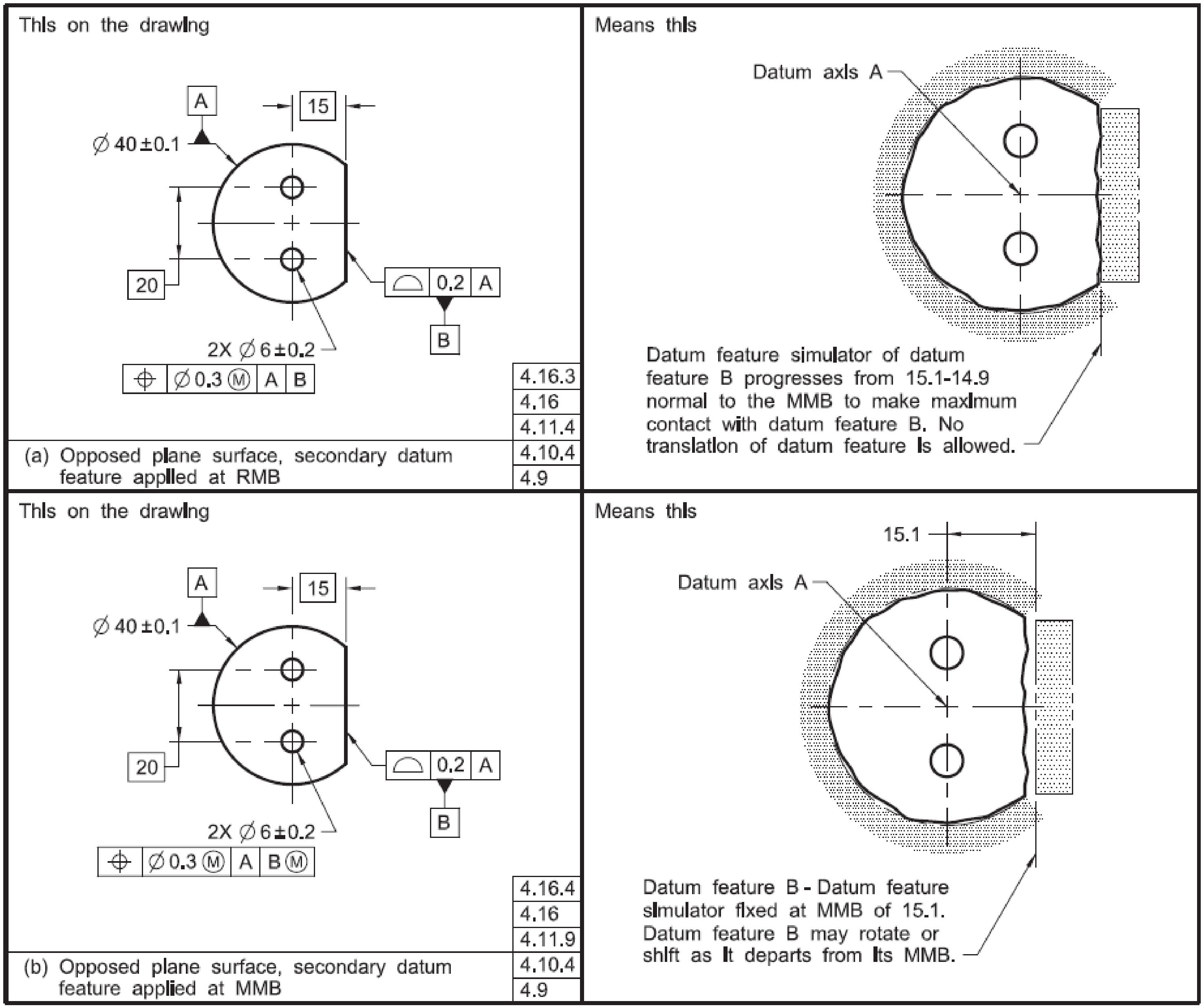
4.16.3 Planar Datum Feature at RMB Constraining a Rotational Degree of Freedom

In Fig. 4-30, illustration (a), datum feature B applies at RMB. This requires the datum feature simulator geometry to originate at MMB of 15.1 mm and progress through the profile tolerance zone toward the LMB of 14.9 mm until it makes maximum contact with datum feature B and constrains the rotational degree of freedom of the part around the axis of the datum feature simulator of datum feature A.

4.16.4 Planar Datum Feature at MMB Constraining a Rotational Degree of Freedom

In Fig. 4-30, illustration (b), datum feature B is modified to apply at MMB. This requires the datum feature simulator to be fixed at the MMB of 15.1 mm and thus orients the two planes that originate at the axis of the datum feature simulator of datum feature A. Datum feature B may rotate within the confines created by its departure from MMB and might not remain in contact with the datum feature simulator.

Fig. 4-30 Planar Datum Feature Constraining a Rotational Degree of Freedom



4.16.5 Offset Planar Datum Feature at RMB Constraining a Rotational Degree of Freedom

In Fig. 4-31, illustration (a), datum feature B is offset relative to datum axis A and applies at RMB. This requires the datum feature simulator geometry to originate at MMB of 5.1 mm and progress through the profile tolerance zone toward the LMB of 4.9 mm until it makes maximum contact with datum feature B (possible two point contact) and constrains the rotational degree of freedom of the two planes of the datum reference frame around the axis of the true geometric counterpart of datum feature A.

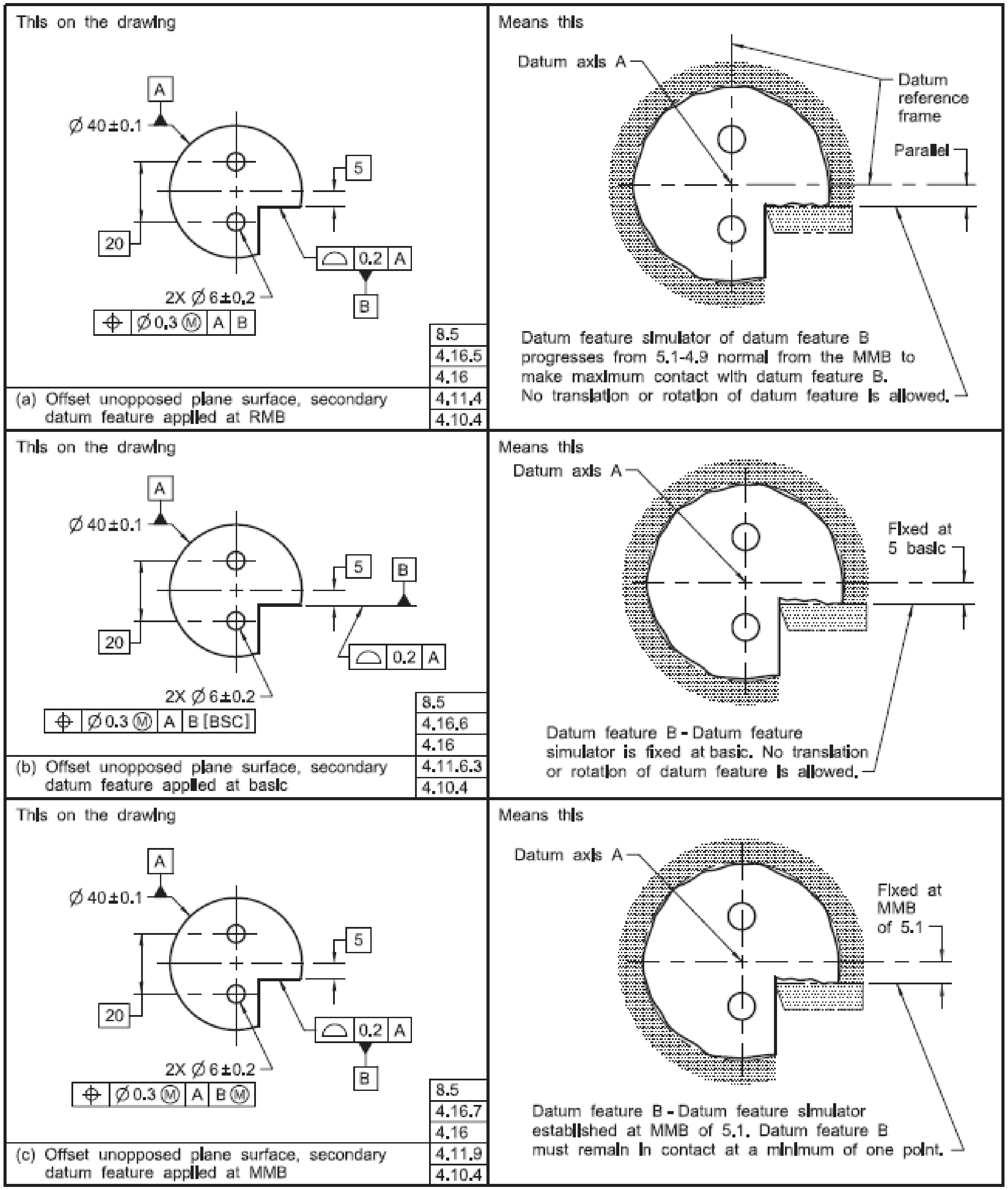
4.16.6 Offset Planar Datum Feature Set at Basic Constraining a Rotational Degree of Freedom

In Fig. 4-31, illustration (b), datum feature B is offset 5 mm relative to datum axis A. RMB does not apply as it is overridden in the feature control frame for the two holes by the abbreviation BSC in brackets following the reference to datum feature B. See para. 4.11.6.3. This requires the datum feature simulator to be fixed at 5 mm basic and constrains the rotational degree of freedom of the two planes of the datum reference frame around the axis of the datum feature simulator from datum feature A.

4.16.7 Offset Planar Datum Feature at MMB Constraining a Rotational Degree of Freedom

In Fig. 4-31, illustration (c), datum feature B is offset relative to datum axis A and modified to apply at MMB. This requires the datum feature simulator to be fixed at the MMB of 5.1 mm and constrains the rotational degree of freedom of the two planes of the datum reference frame that originate at the datum feature simulator of datum feature A. Where the datum feature simulator and the higher precedence datum axis do not limit rotation in both directions about the datum axis, the datum feature must always contact the datum feature simulator.

Fig. 4-31 Datum Modifier Effects — Plane Surface



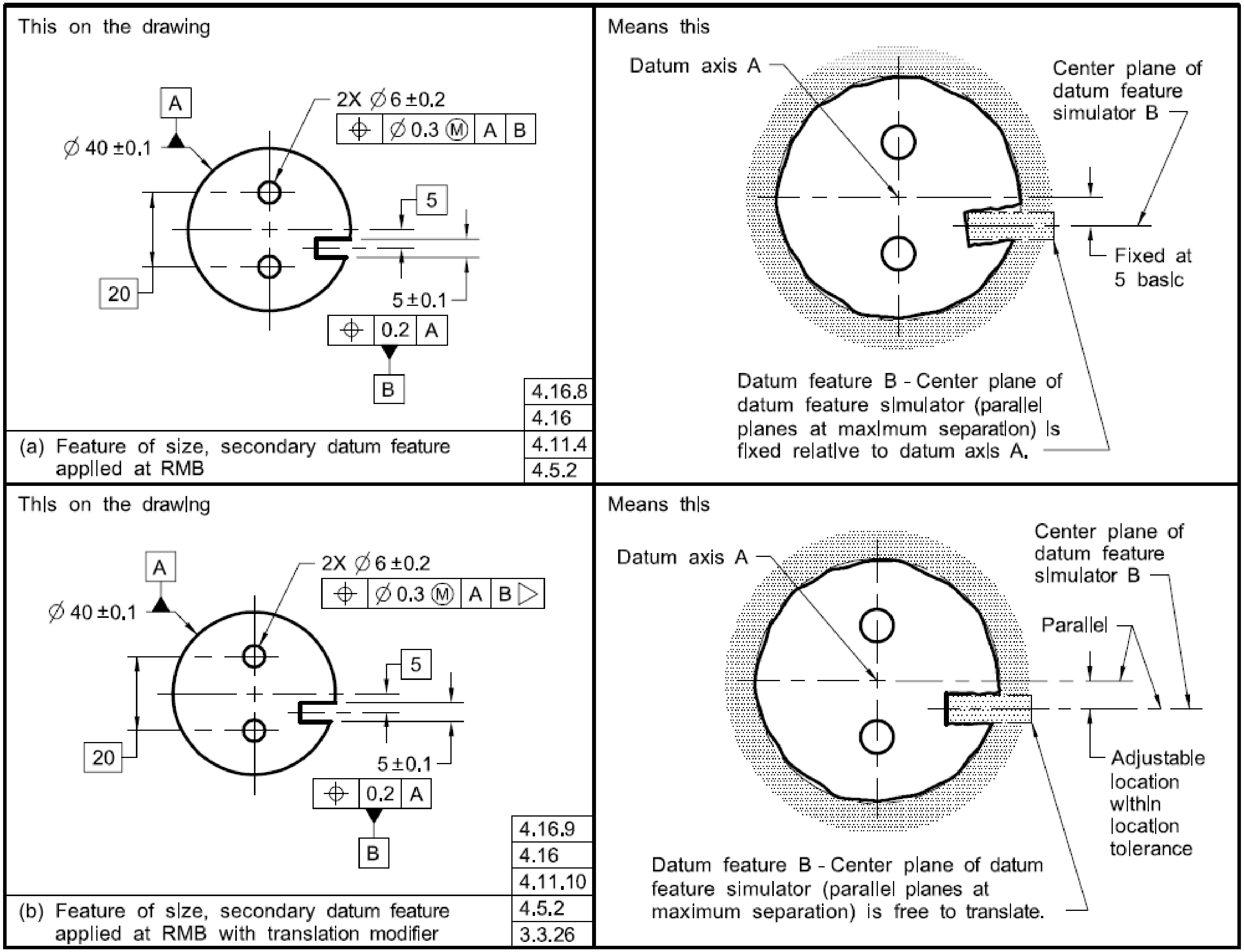
4.16.8 Datum Feature of Size at RMB Constraining a Rotational Degree of Freedom

In Fig. 4-32, illustration (a), datum feature B applies at RMB and is located relative to datum axis A. This requires the center plane of the datum feature simulator geometry to be fixed at the basic 5 mm dimension and the datum feature simulator geometry to expand until it makes maximum contact with datum feature B. This constrains the rotational degree of freedom of the two planes of the datum reference frame around the axis of the datum feature simulator of datum feature A.

4.16.9 Datum Feature of Size at RMB With Translation Modifier Constraining Rotational Degrees of Freedom

In Fig. 4-32, illustration (b), datum feature B applies at RMB with a translation modifier. This allows the center plane of the datum feature simulator to translate while maintaining its orientation to higher precedence datums. The parallel planes of the datum feature simulator expand to make maximum contact with the datum feature.

Fig. 4-32 Datum Modifier Effects — Size Feature

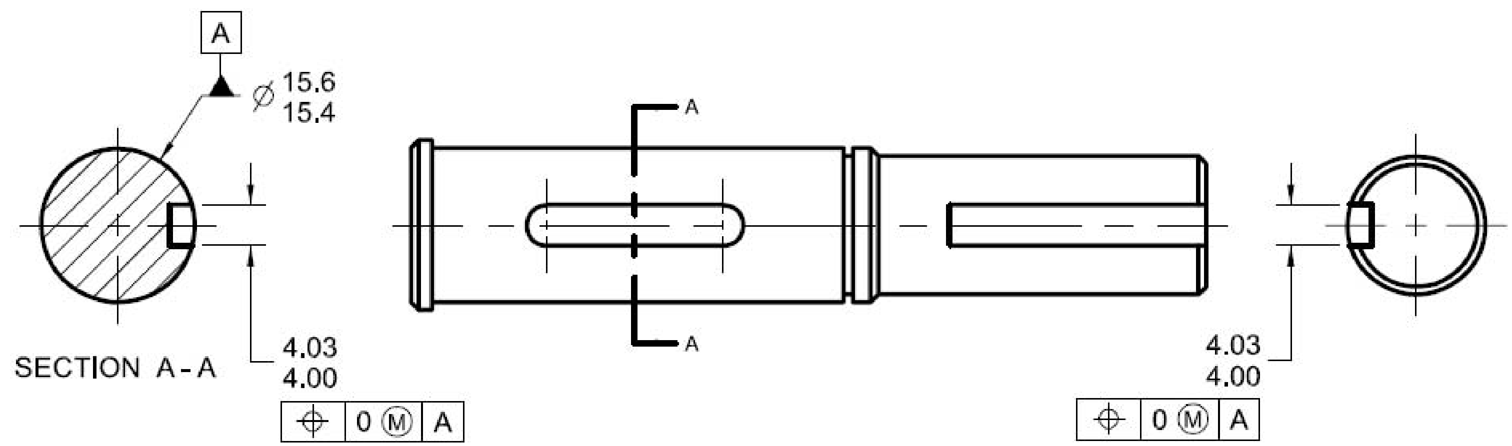


4.19 SIMULTANEOUS REQUIREMENTS

A simultaneous requirement is where two or more geometric tolerances apply as a single pattern or part requirement. A simultaneous requirement applies to position and profile tolerances that are located by basic dimensions, related to common datum features referenced in the same order of precedence at the same boundary conditions. In a simultaneous requirement there is no translation or rotation between the datum reference frames of the included geometric tolerances, thus creating a single pattern. Figures 4-39 and 4-40 show examples of simultaneous requirements. If such interrelationship is not required, a notation such as SEP REQT is placed adjacent to each applicable feature control frame. See Figs. 4-41 and 7-54 and para. 7.5.4.2. This principle does not apply to the lower segments of composite feature control frames. See para. 7.5.4.2. If a simultaneous requirement is desired for the lower segments of two or more composite feature control frames, a notation such as SIM REQT shall be placed adjacent to each applicable lower segment of the feature control frames.

Fig. 4-40 Aligned Features — Simultaneous Requirement

This on the drawing



Means this

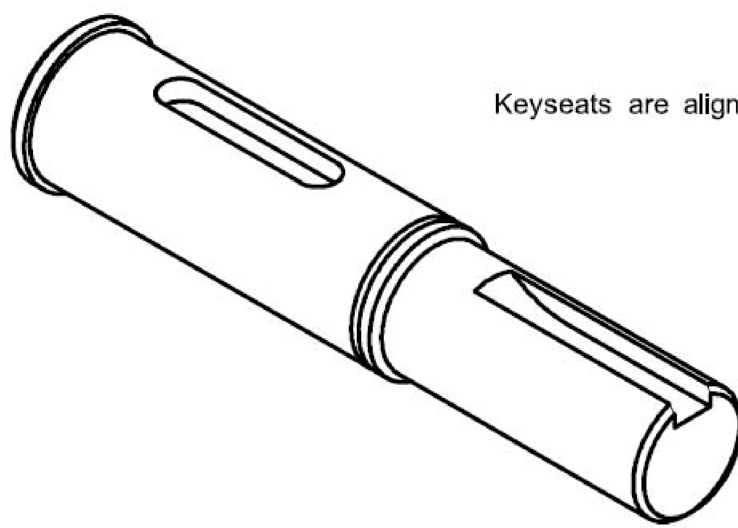
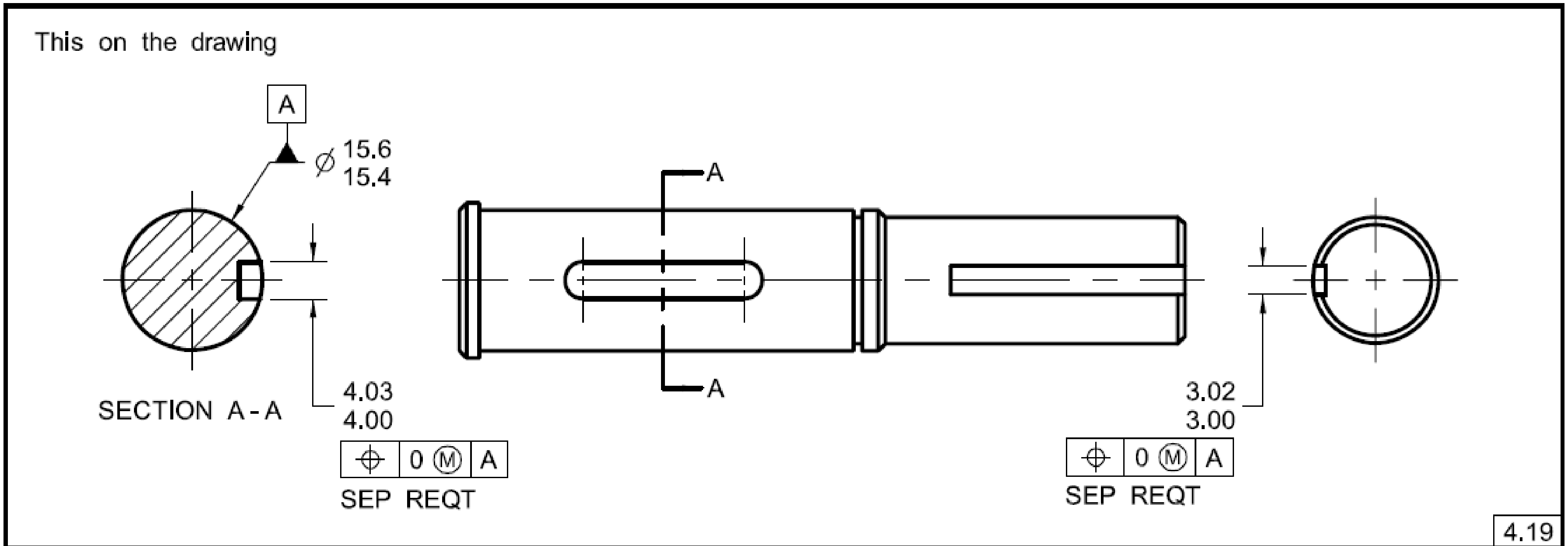
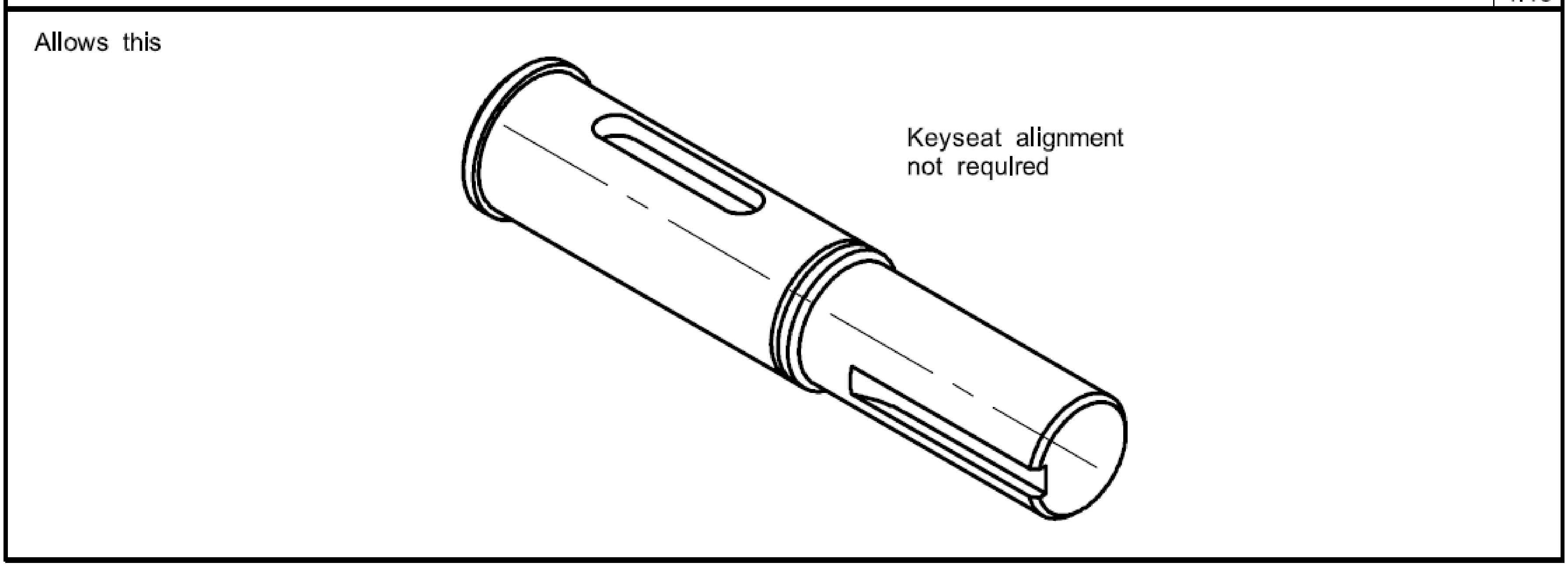


Fig. 4-41 Pattern of Features Not Aligned — Specified Separate Requirements





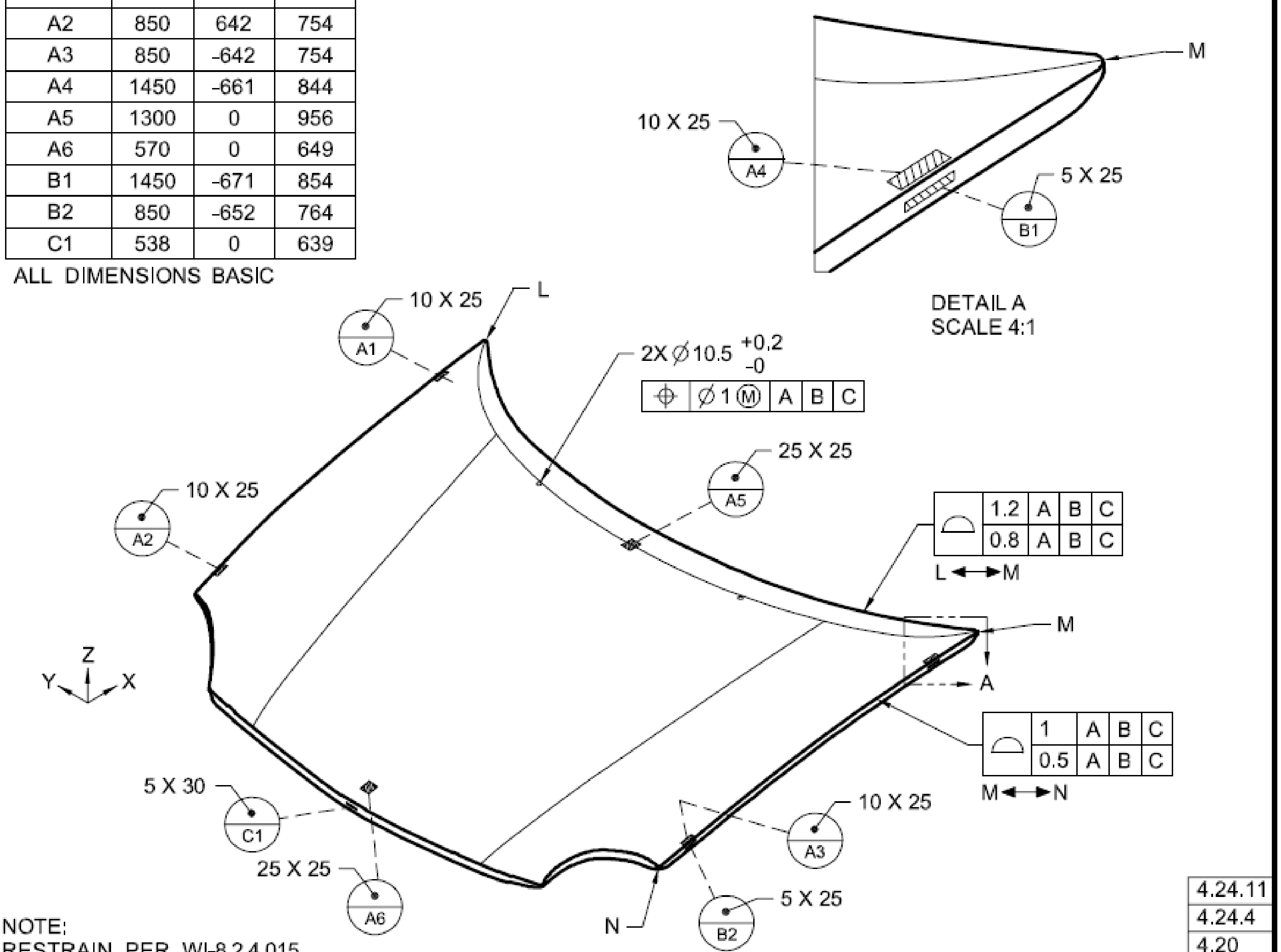
4.20 RESTRAINED CONDITION

Unless otherwise specified, all tolerances apply in a free-state condition. In some cases, it may be desirable to restrain a part on its datum features to simulate their function or interaction with other features or parts. To invoke a restrained condition, a note is specified or referenced on the drawing defining the specific requirements. See Fig. 4-42. This figure illustrates a part that should be restrained until sufficient reinforcement is added to retain its design shape. In this illustration, the restraint must be per a document referenced on the drawing. In a restrained application, it is permissible to use as many datum targets as necessary to establish the datum features.

Fig. 4-42 Restrained Condition Application

DATUM TARGET LOCATIONS			
TARGET	X	Y	Z
A1	1450	661	844
A2	850	642	754
A3	850	-642	754
A4	1450	-661	844
A5	1300	0	956
A6	570	0	649
B1	1450	-671	854
B2	850	-652	764
C1	538	0	639

ALL DIMENSIONS BASIC



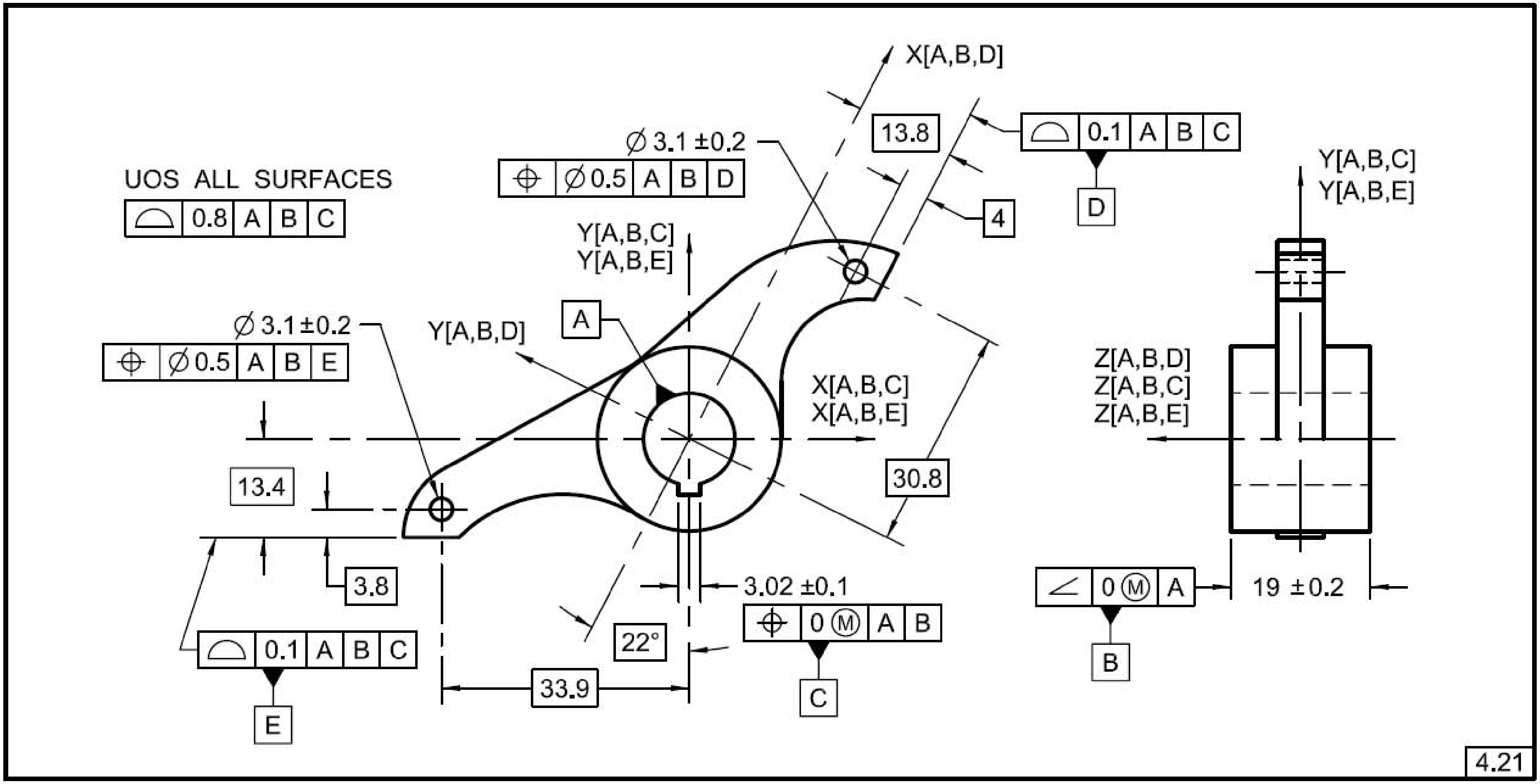
NOTE:
RESTRAIN PER WI-8.2.4.015

4.24.11
4.24.4
4.20
3.3.3

4.21 DATUM REFERENCE FRAME IDENTIFICATION

Where a datum reference frame has been properly established and it is considered necessary to illustrate the axes of a datum reference frame on the drawing, the axes or center planes may be labeled to determine the translational degrees of freedom X, Y, and Z. See Figs. 4-2, 4-7, 4-8, and 4-54. Where multiple datum reference frames exist, and it is desirable to label the axes (X, Y, and Z), any labeled axes shall include a reference to the associated datum reference frame. In Fig. 4-43 the X, Y, and Z axes for the three datum reference frames are identified by the notation [A, B, C], [A, B, D], and [A, B, E]. These labels represent the datum features (without modifiers) for each datum reference frame and follow the X, Y, and Z identification letters.

Fig. 4-43 Datum Reference Frame Identification



ASME

SETTING THE STANDARD

Section 4
Datum Reference Frames

Don Day

President, Tec-Ease, Inc.

Section 4 Work Group, Member

ASME Y14.8, Chair

- Irregular features of Size as Datum Features
- Customizing the Datum Reference Frame
- Movable Datum Targets

ASME
SETTING THE STANDARD

Irregular Features of Size as Datum Features

Features and collections of features that act like features of size, but Rule #1 does not apply.

Regular Features of Size

From ASME Y14.5M-1994:

- One cylindrical or spherical surface, or a set of two opposed elements or opposed parallel surfaces, associated with a size dimension.
  - Sphere
  - Cylinder
  - Width (slot or tab)
- Rule #1
  - The limits of size control the form of the feature
  - Feature had to be within the actual local size at each cross section

Regular Features of Size

From ASME Y14.5M-1994:

- Follow Rule #1
- Have a derived center, axis or center plane
- May be referenced at ⇦ or ⇨

1.3.32.2 Irregular Feature of Size.

- ~~Follow Rule #1~~
- May have a derived center, axis or center plane
- May be referenced at ⇦ or ⇨

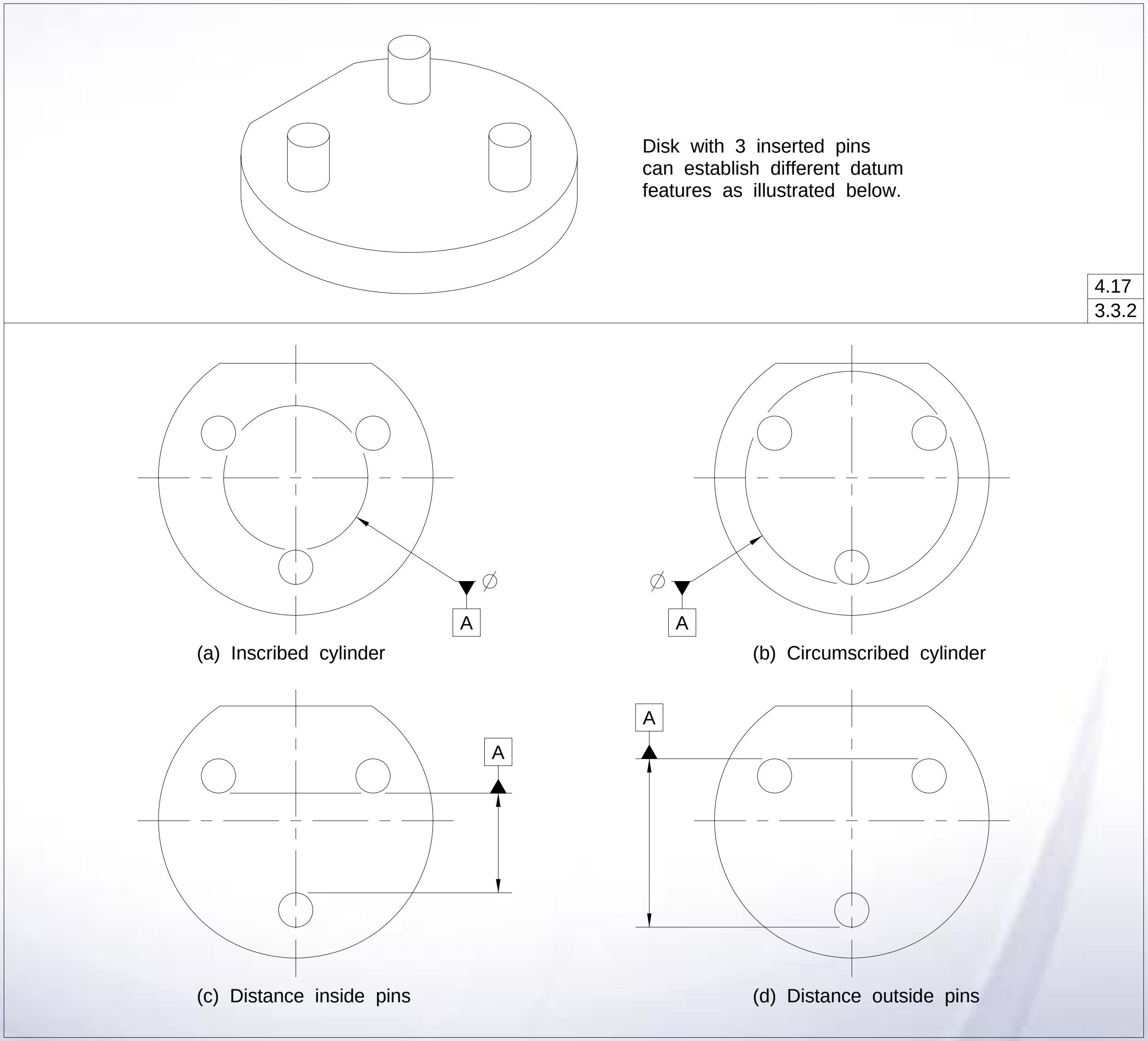
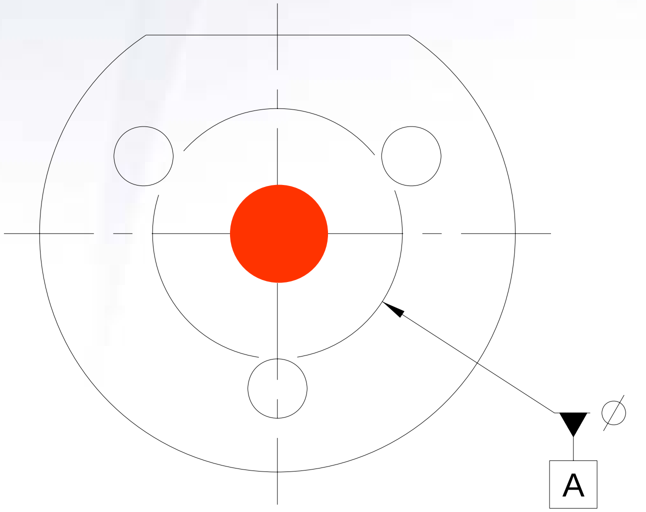
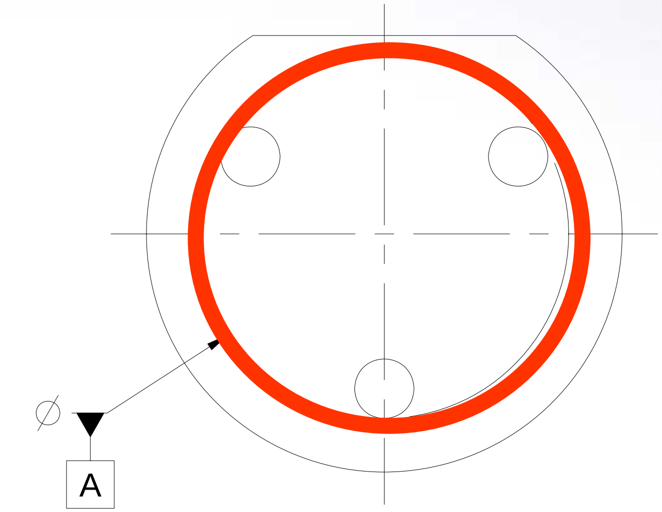


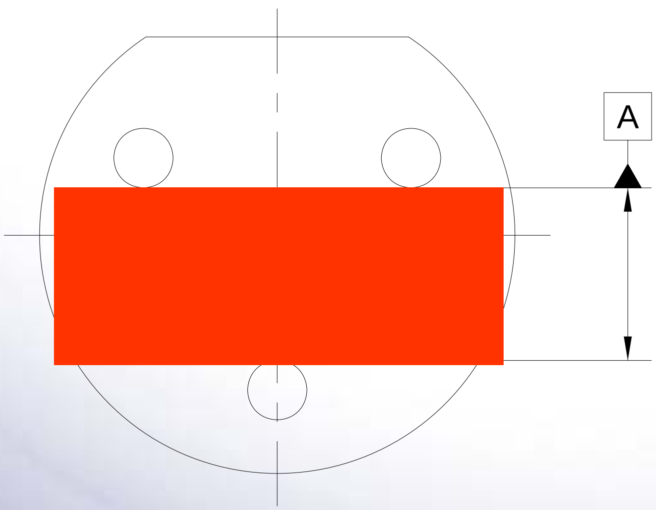
FIG. 4-35 DATUM POSSIBILITIES FROM THREE PINS FOR AN IRREGULAR FEATURE OF SIZE



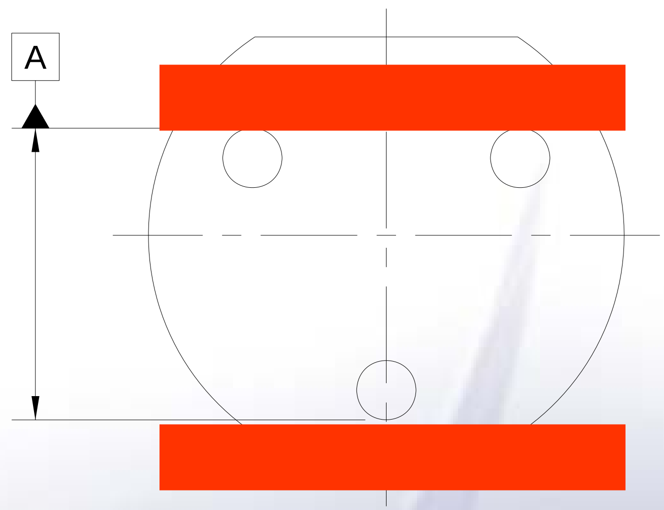
(a) Inscribed cylinder



(b) Circumscribed cylinder

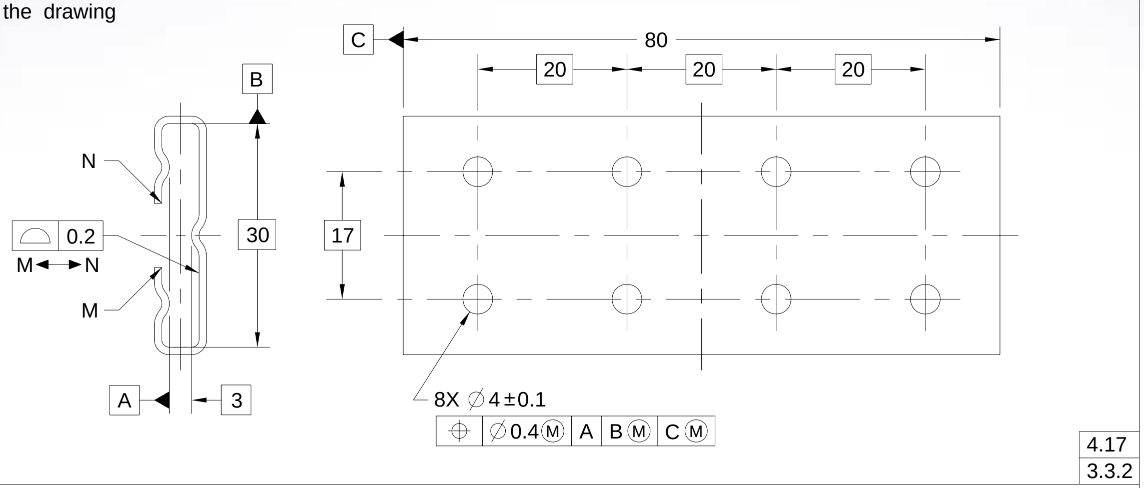


(c) Distance inside pins



(d) Distance outside pins

This on the drawing



Means this

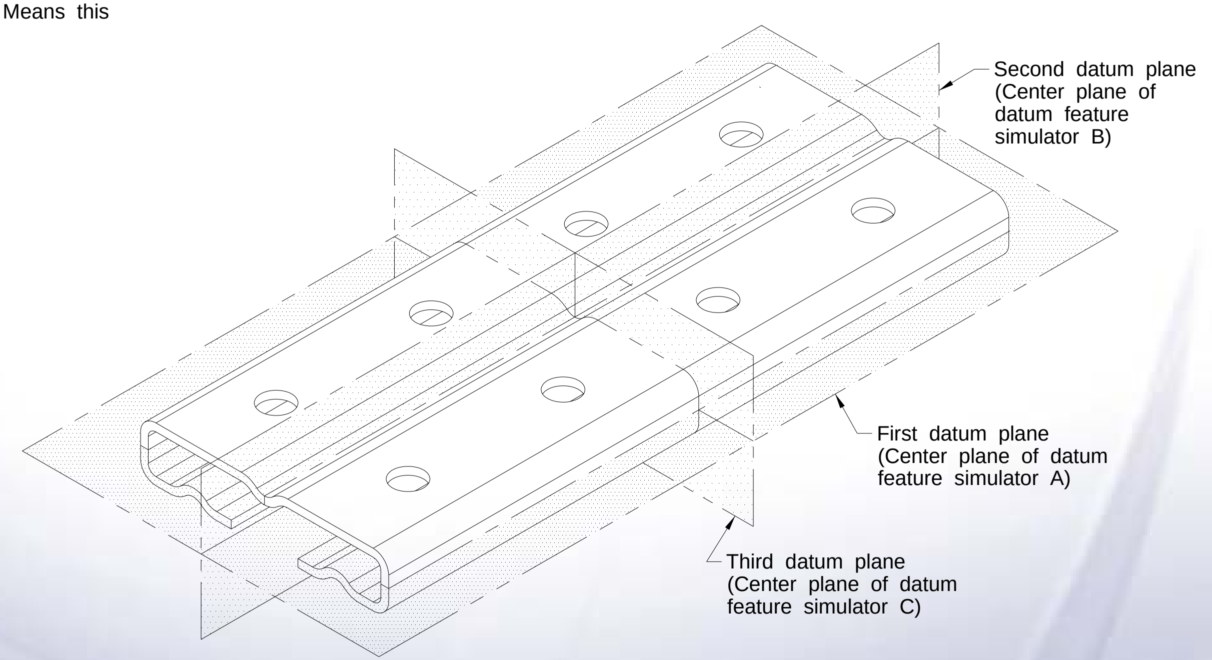
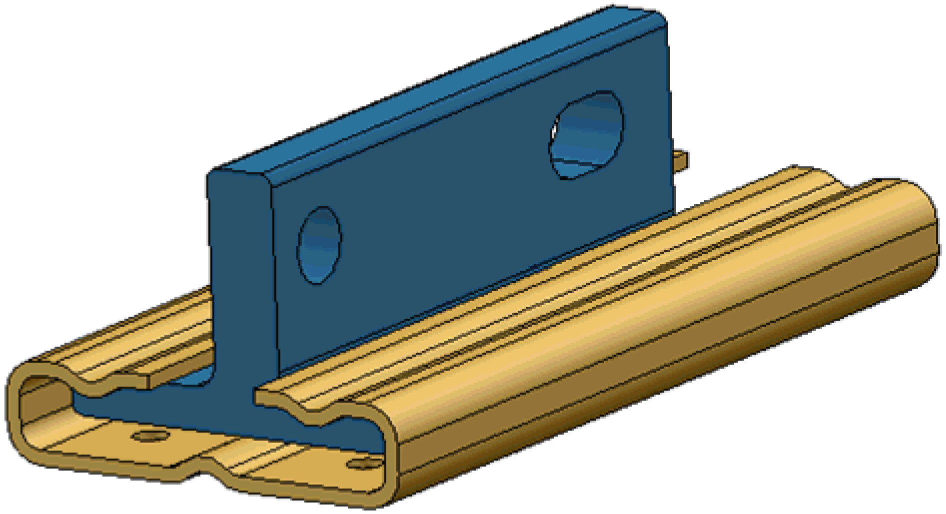
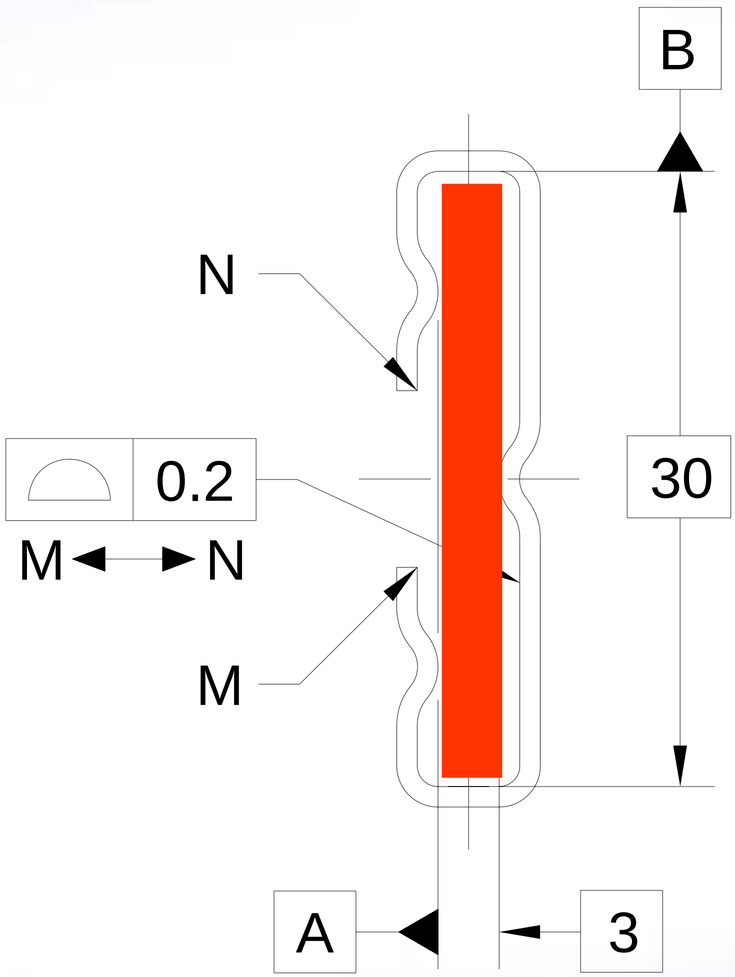


FIG. 4-33 NON SIZE DATUM FEATURE





B
N
0.2
30
M ◄—► N
M
A
3

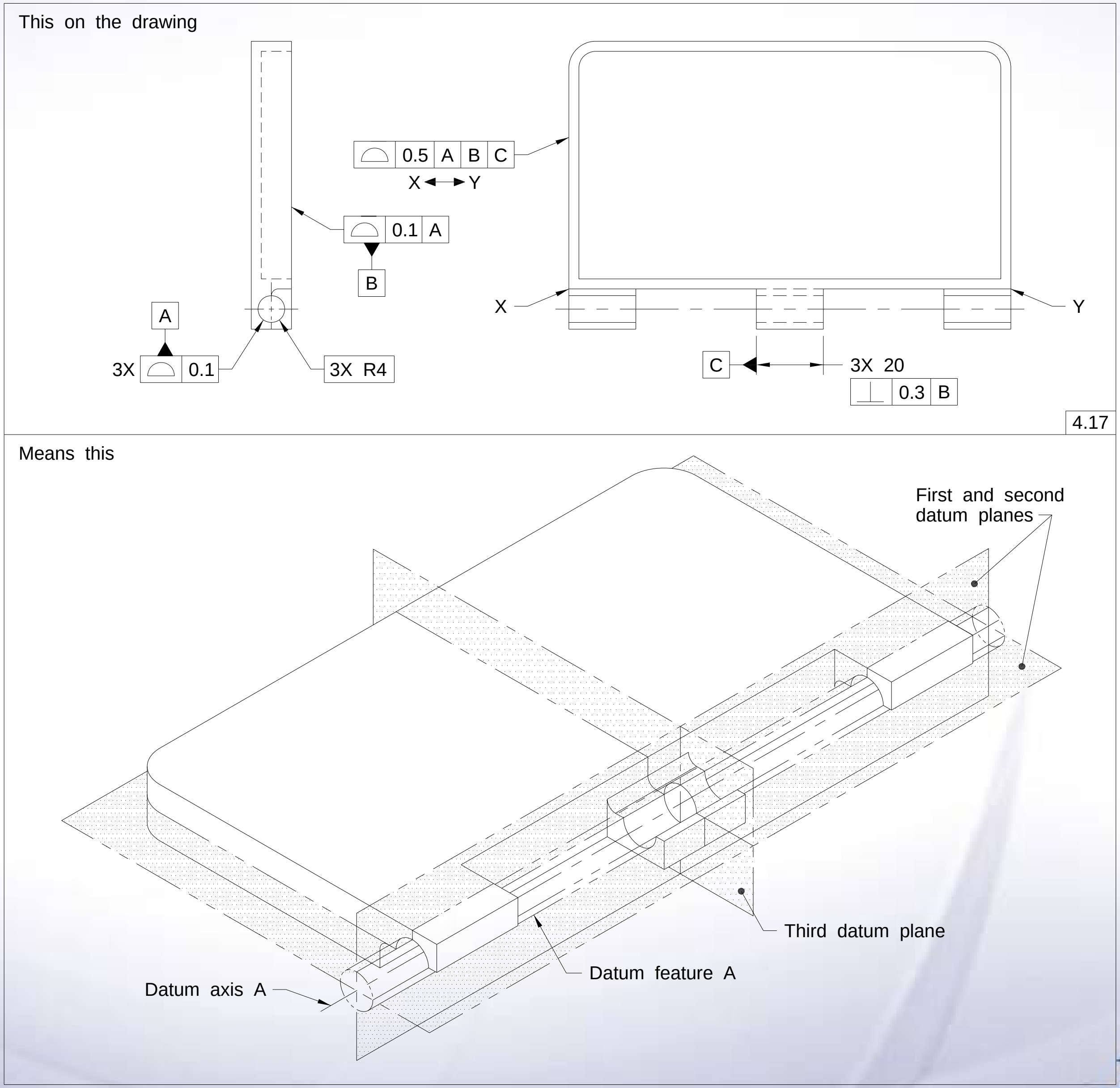
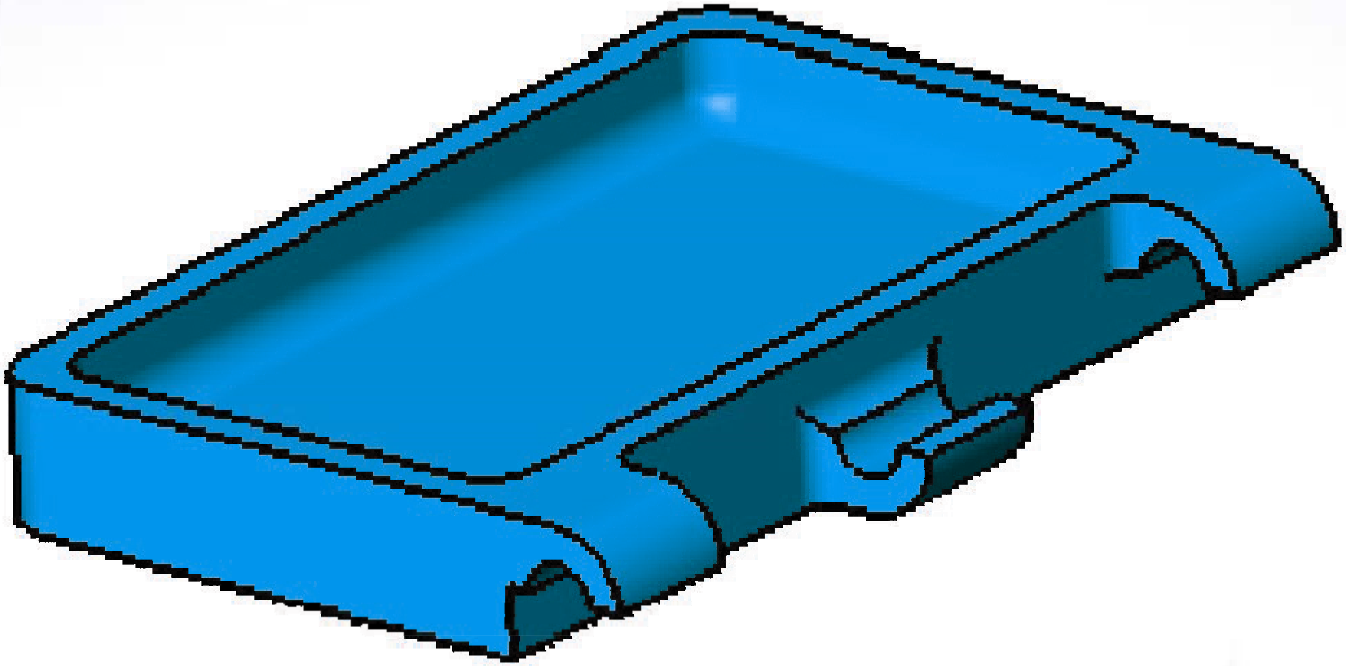


FIG. 4-34 COAXIAL IRREGULAR NON SIZE DATUM FEATURE



ASME
SETTING THE STANDARD

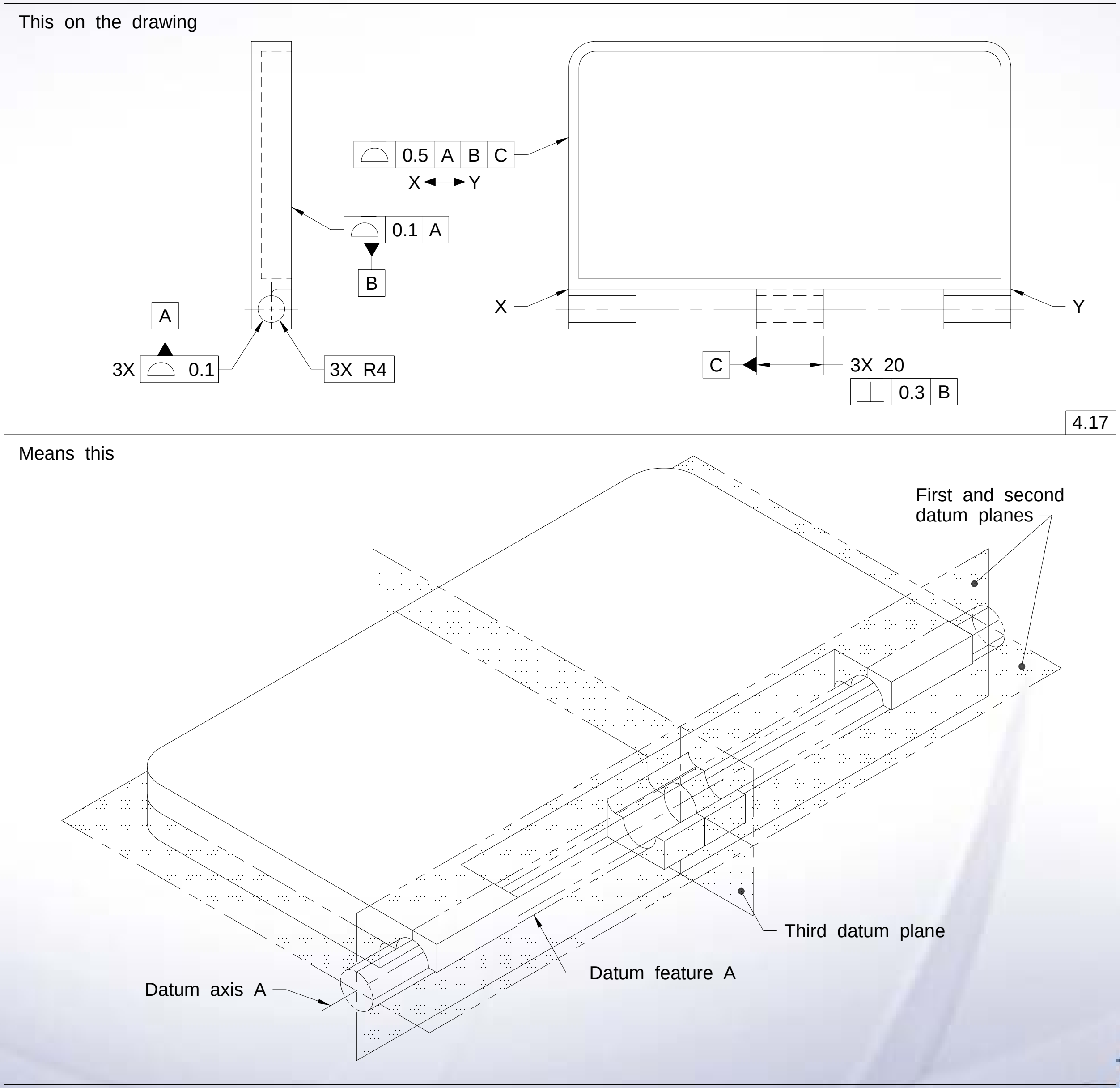
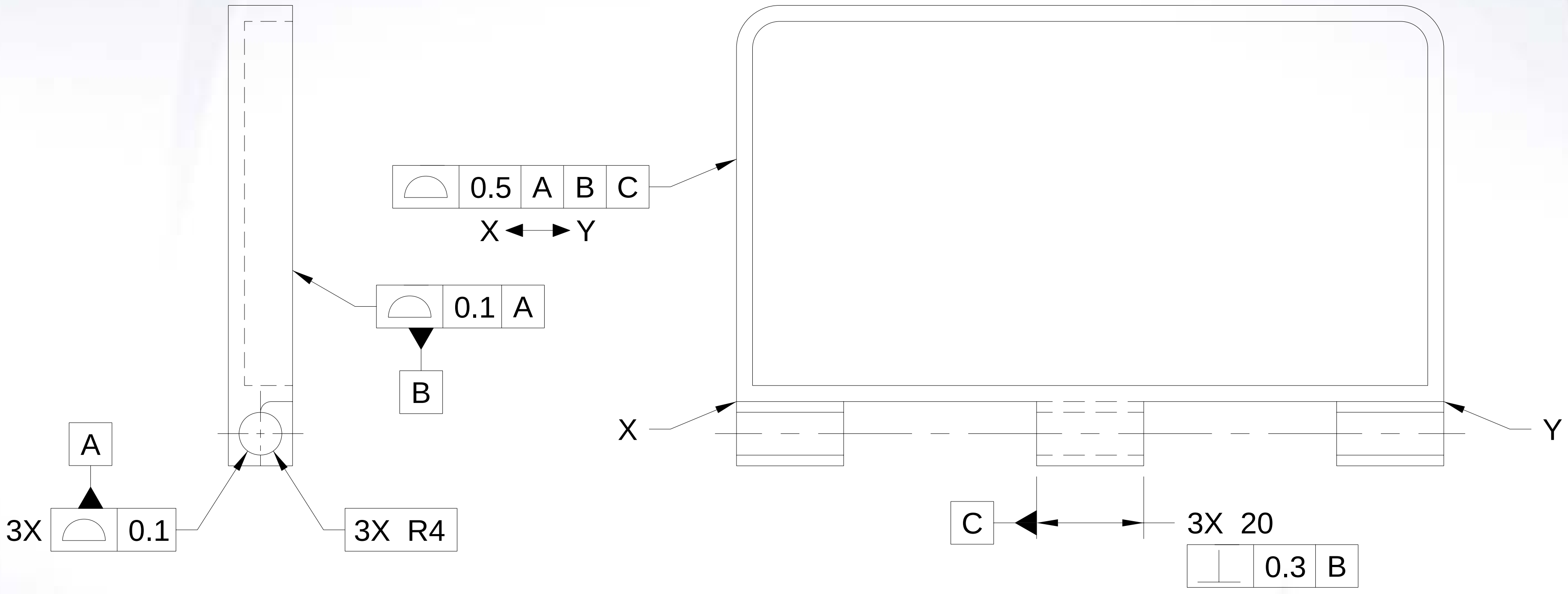


FIG. 4-34 COAXIAL IRREGULAR NON SIZE DATUM FEATURE

the drawing



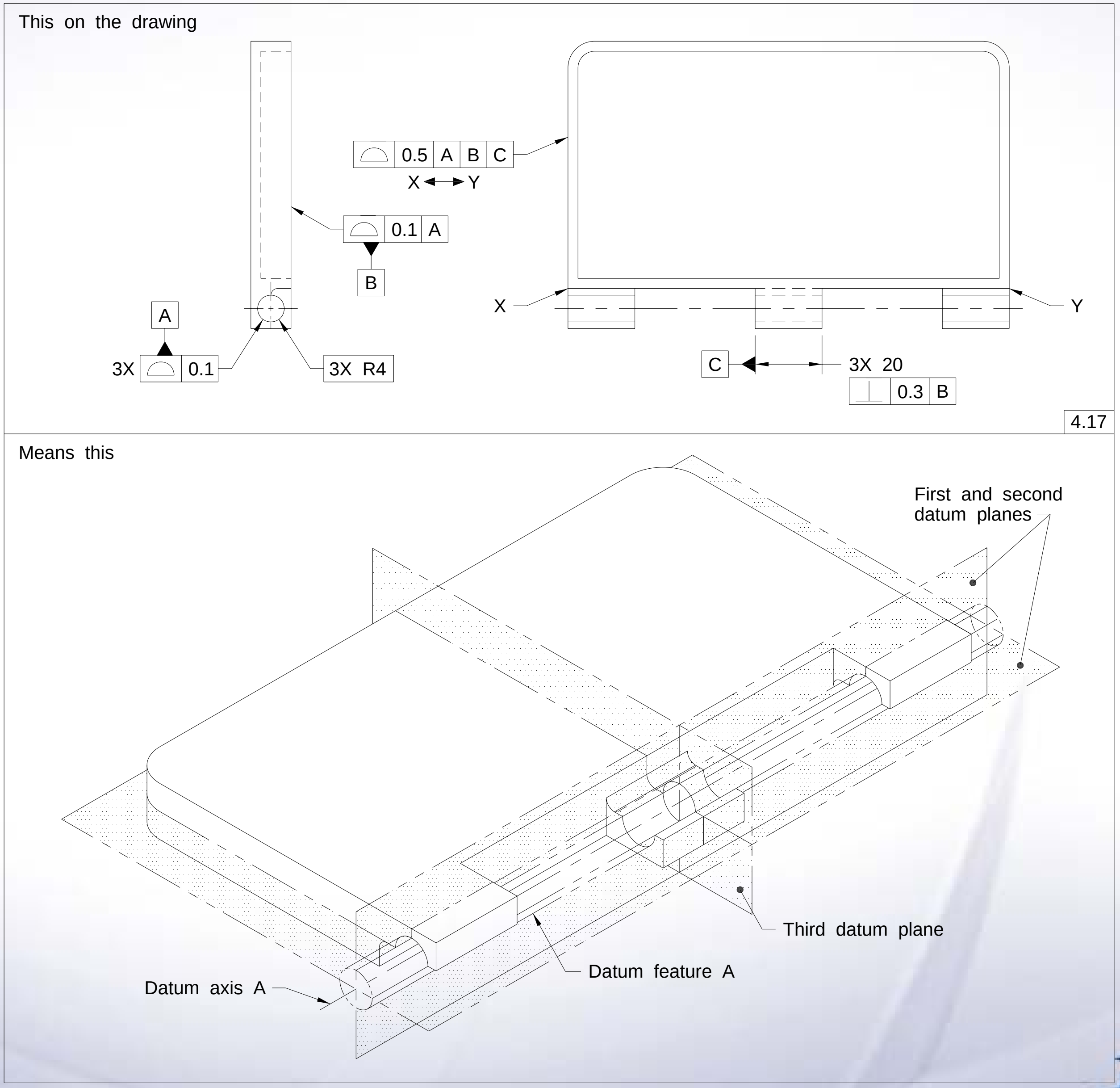


FIG. 4-34 COAXIAL IRREGULAR NON SIZE DATUM FEATURE

Constraining Degrees of Freedom

From ASME Y14.5M-1994:

4.4 SPECIFYING DATUM FEATURES IN AN ORDER OF PRECEDENCE

Datum features must be specified in an order of precedence to position a part properly on the datum reference frame.

Now datum precedence may be overridden by **customizing** the datum reference frame.

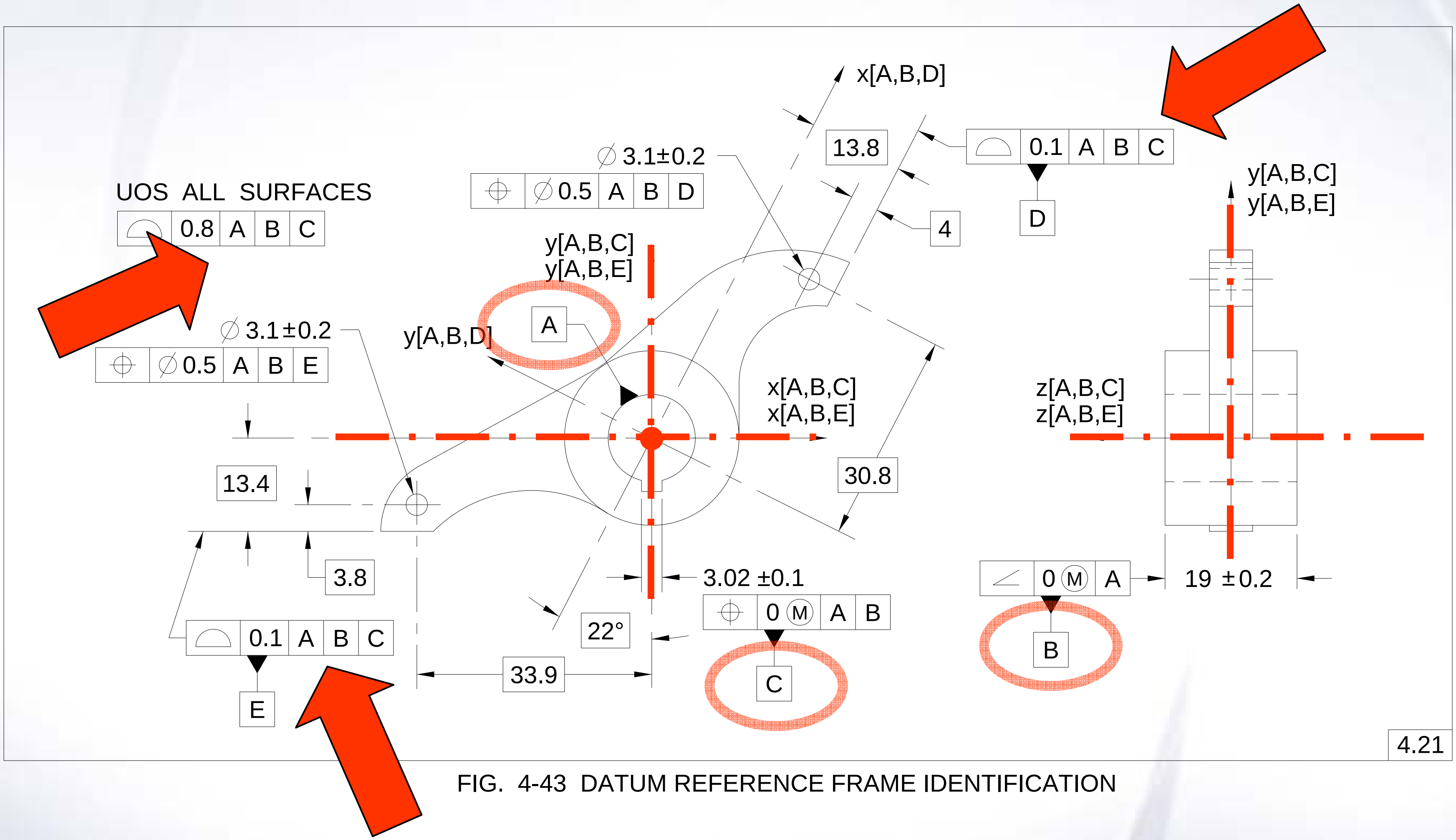


FIG. 4-43 DATUM REFERENCE FRAME IDENTIFICATION

ASME
SETTING THE STANDARD

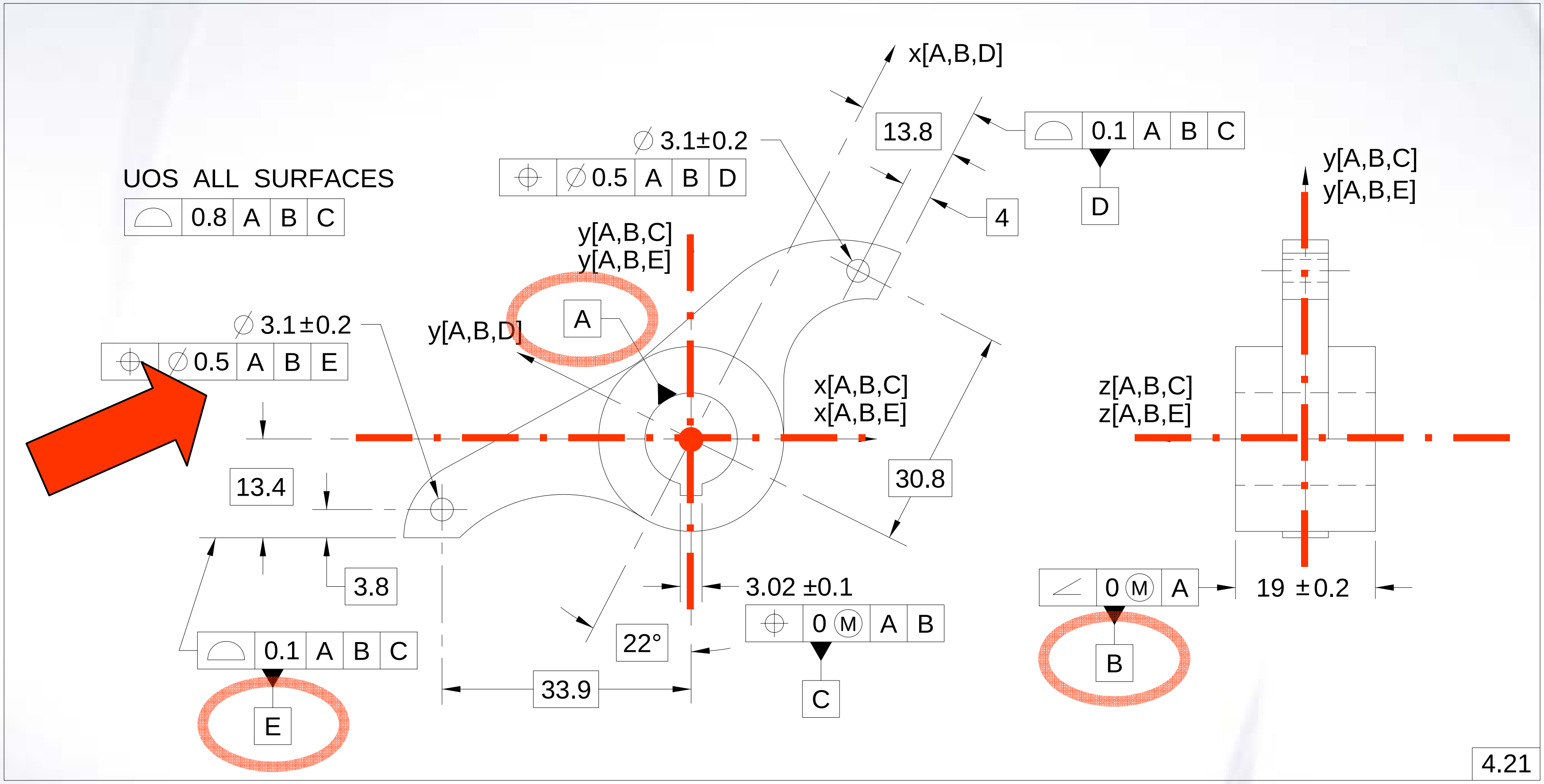


FIG. 4-43 DATUM REFERENCE FRAME IDENTIFICATION

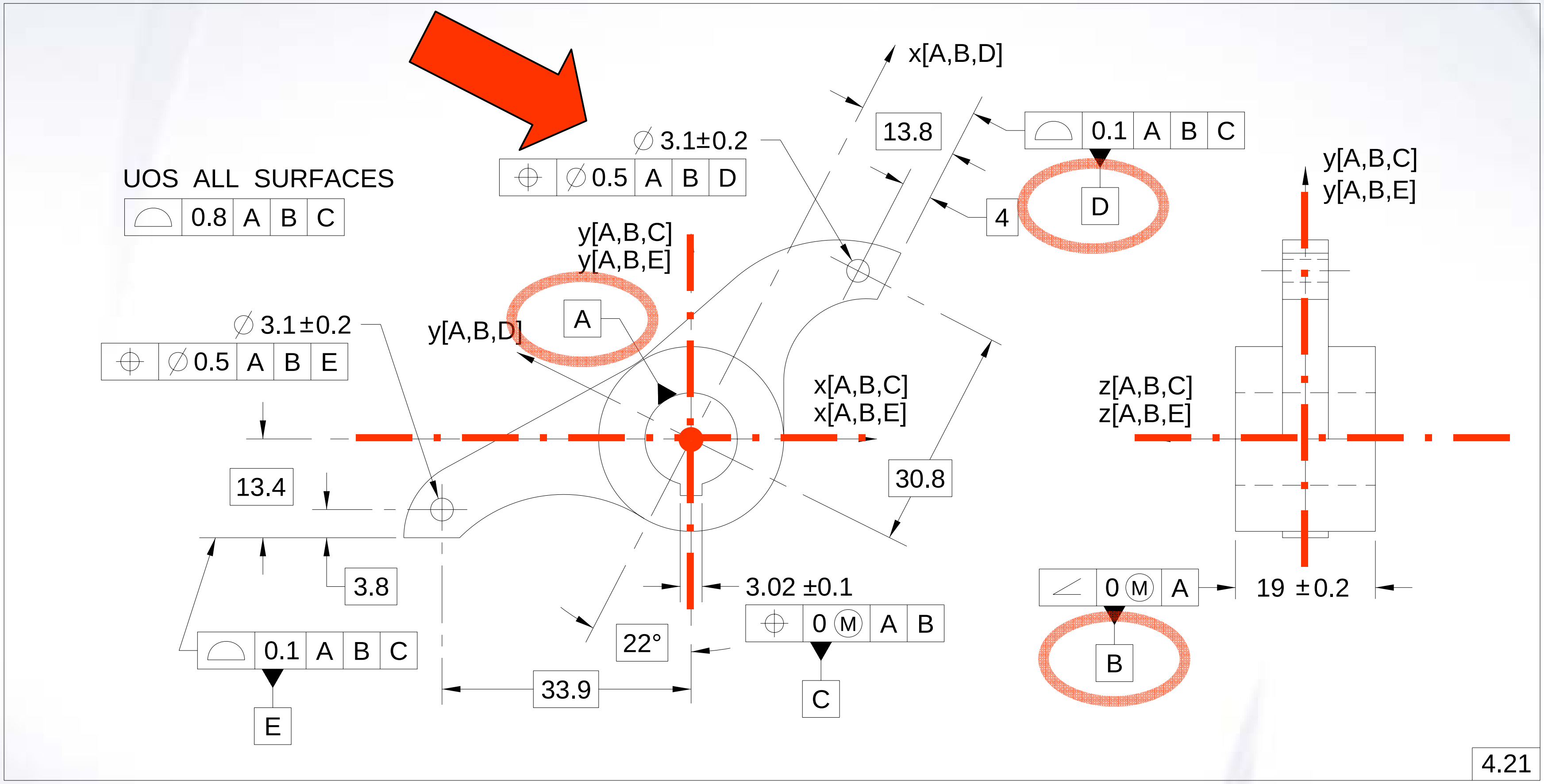


FIG. 4-43 DATUM REFERENCE FRAME IDENTIFICATION

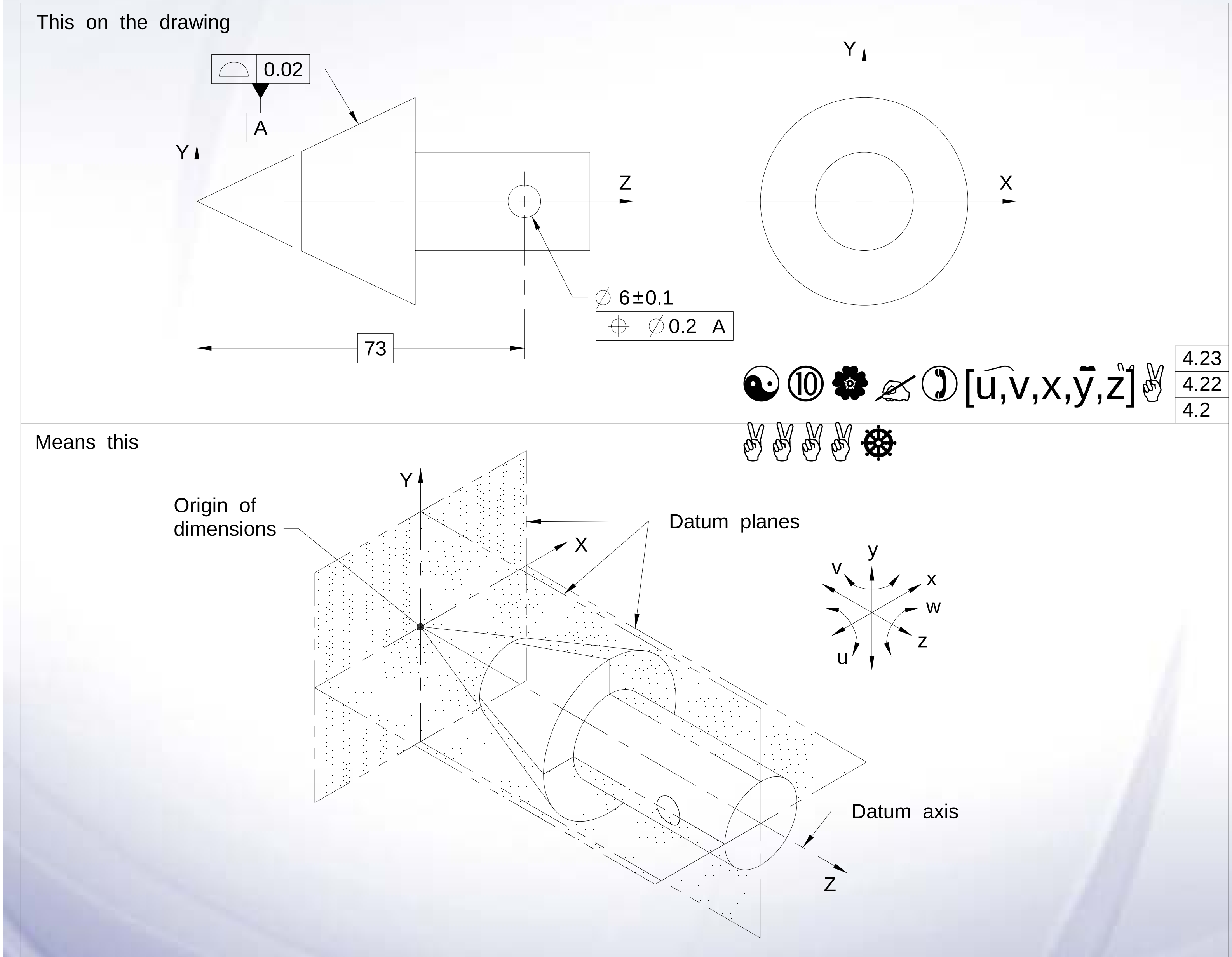


FIG. 4-44 CONICAL DATUM FEATURE

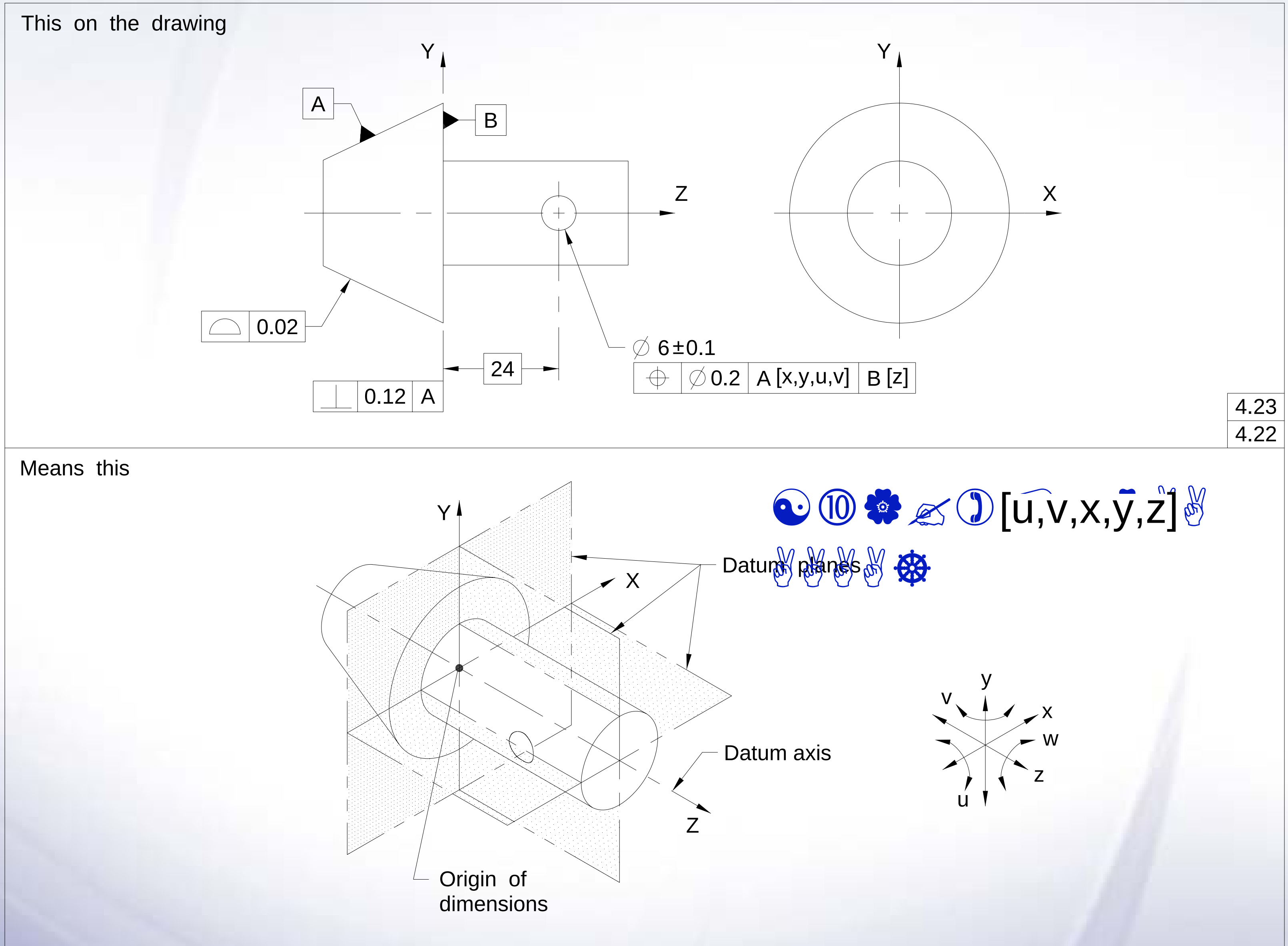
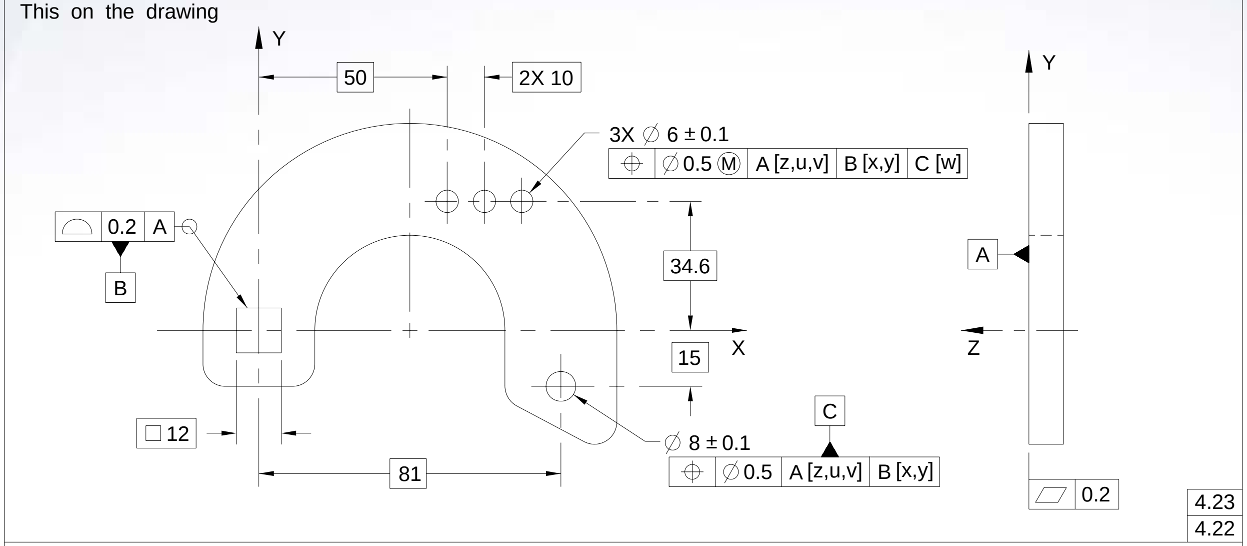


FIG. 4-45 CONICAL DATUM FEATURE

This on the drawing



Means this

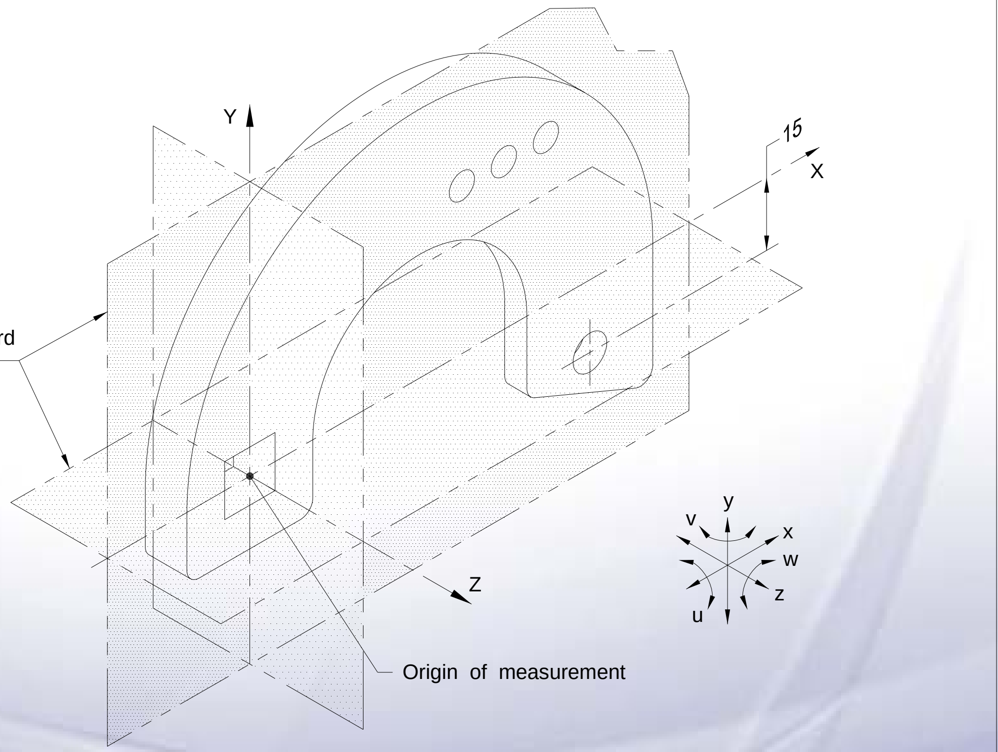
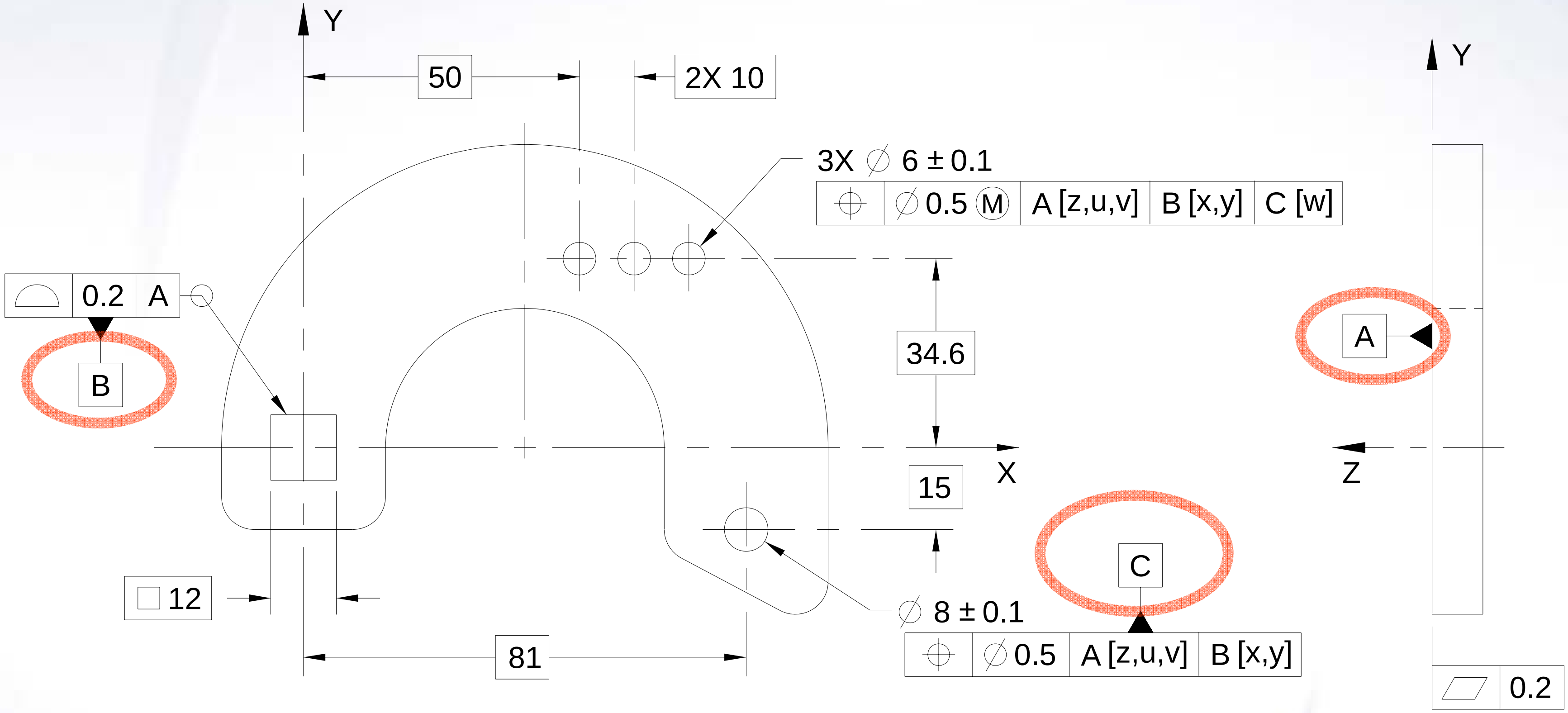


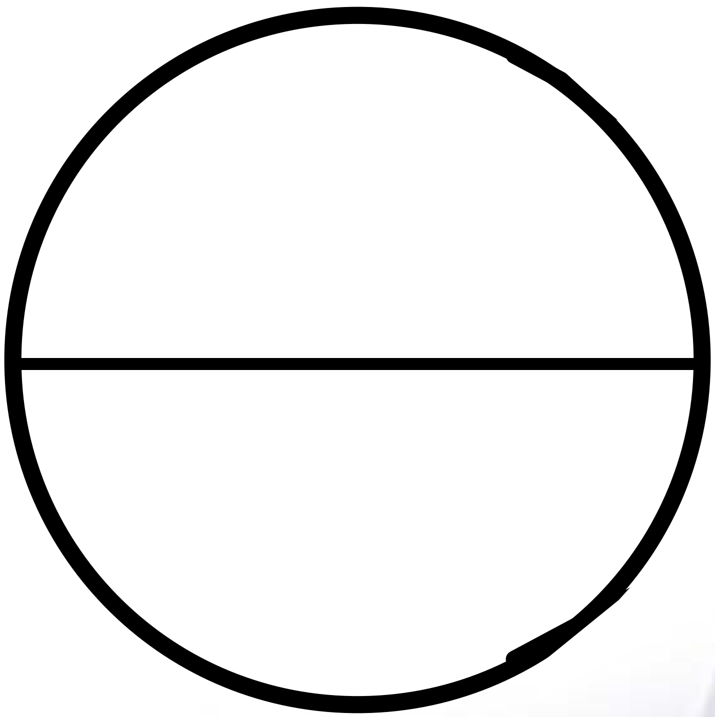
FIG. 4-46 CUSTOMIZED DATUM REFERENCE FRAME



Y
50
2X 10
3X ∅ 6 ± 0.1
⌖ | ∅0.5 Ⓜ | A [z,u,v] | B [x,y] | C [w]
⌓ | 0.2 | A
B
34.6
X
15
□12
81
∅ 8 ± 0.1
⌖ | ∅0.5 | A [z,u,v] | B [x,y]
C
Y
A
Z
⏥ | 0.2

Datum

Target



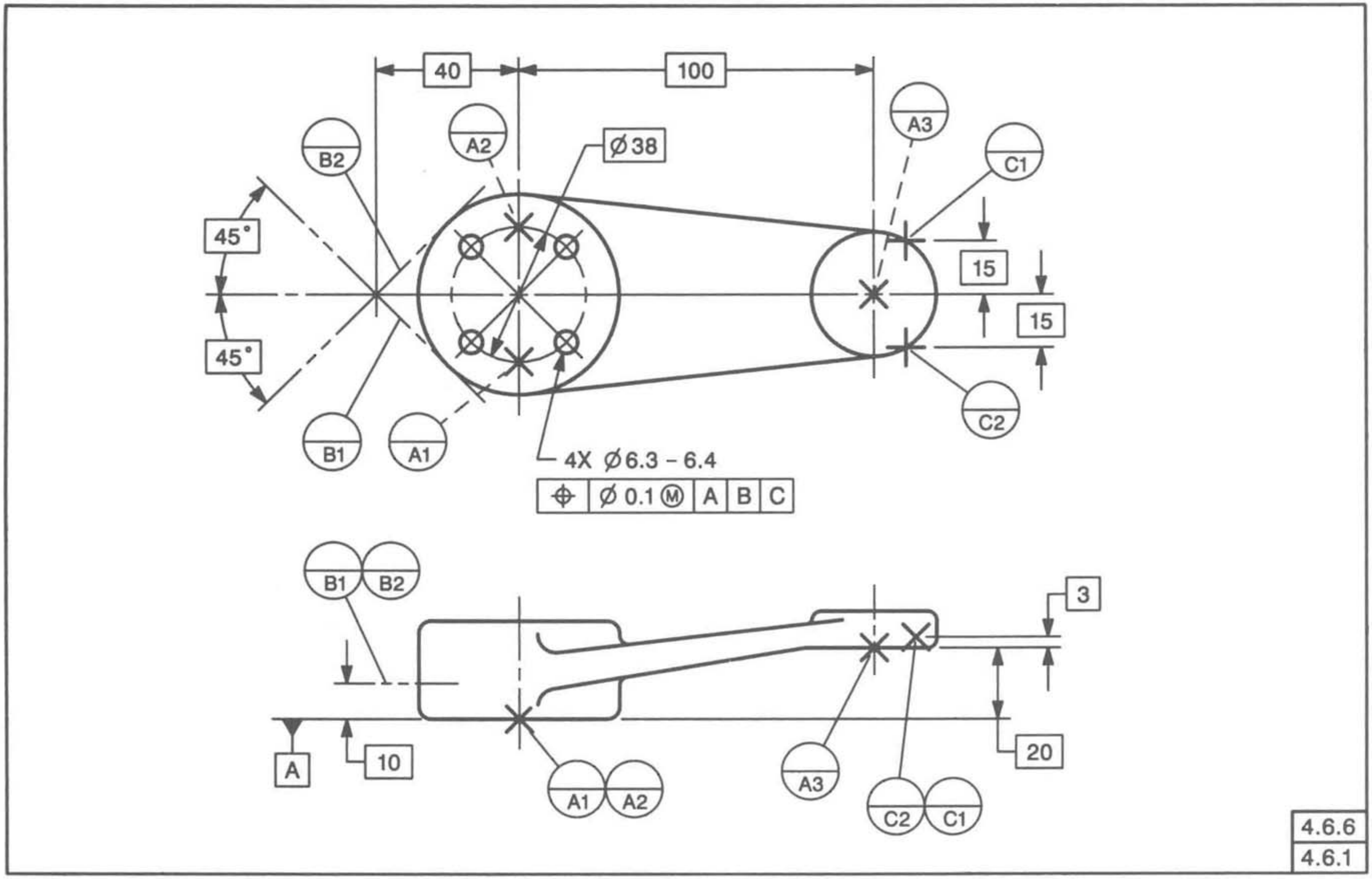
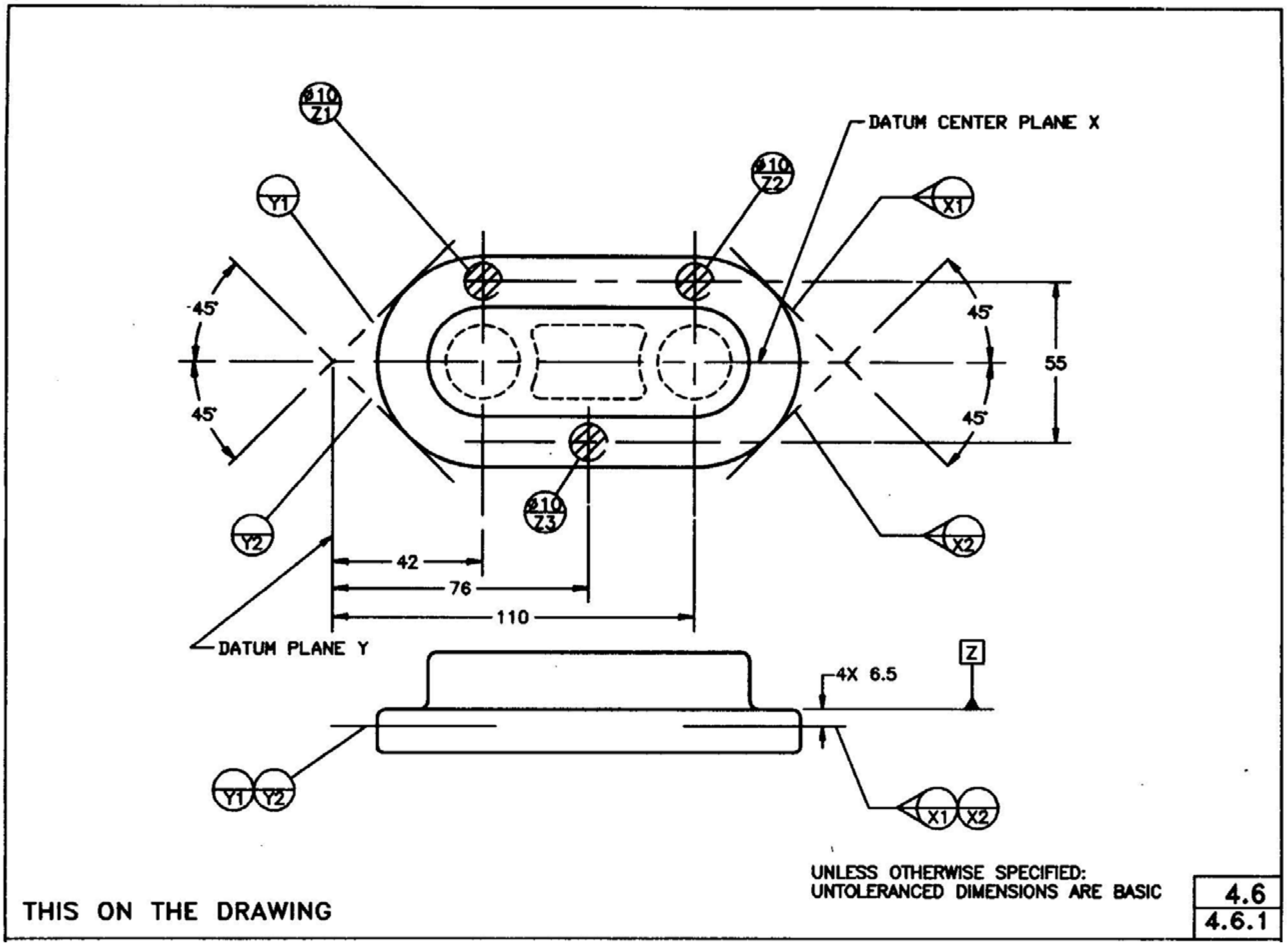


FIG. 4-38 APPLICATIONS OF EQUALIZING DATUMS

From the ASME Y14.5M-1994 standard



From the ASME Y14.8M-1996 standard

From the ASME Y14.8M-1996 standard

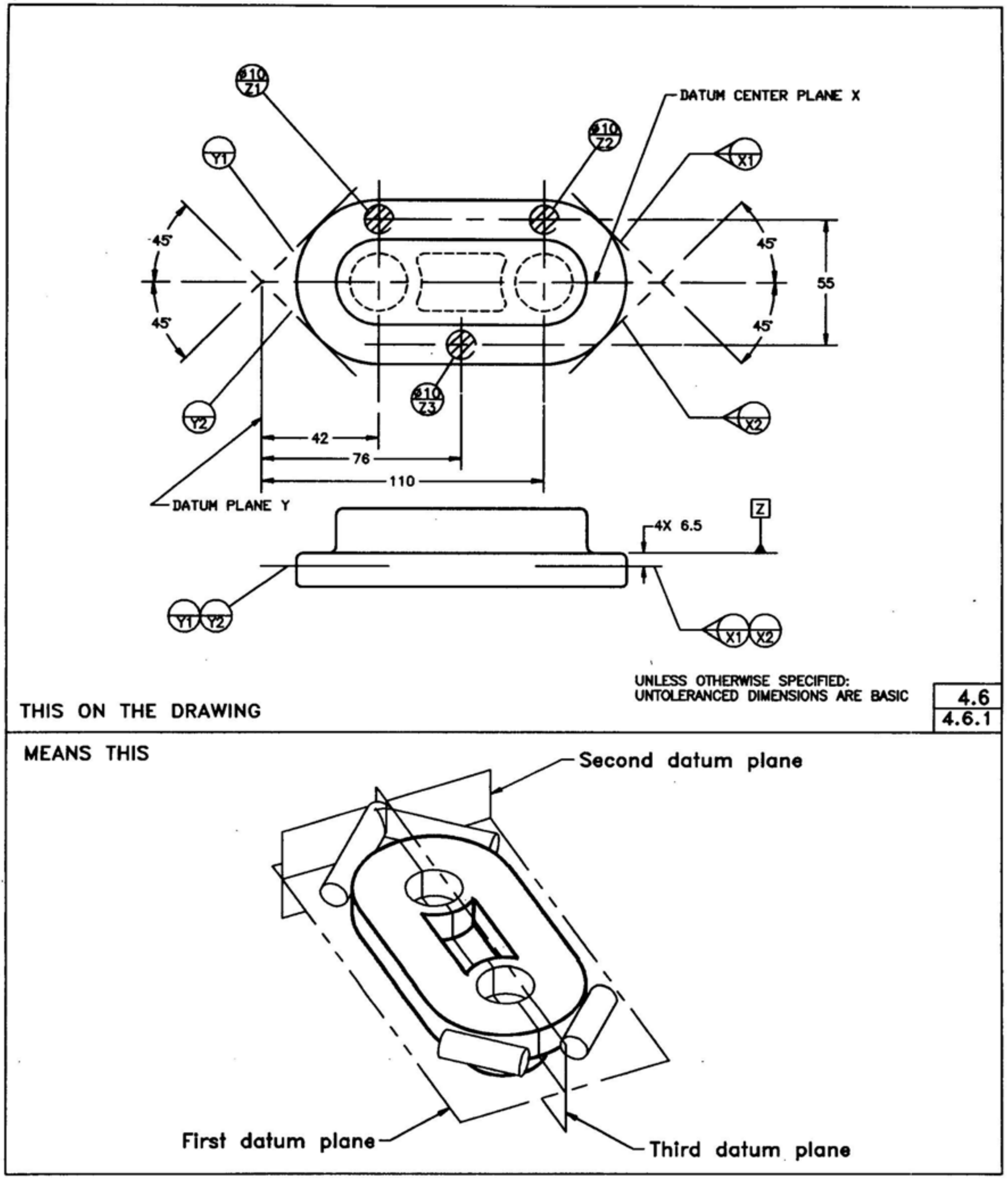
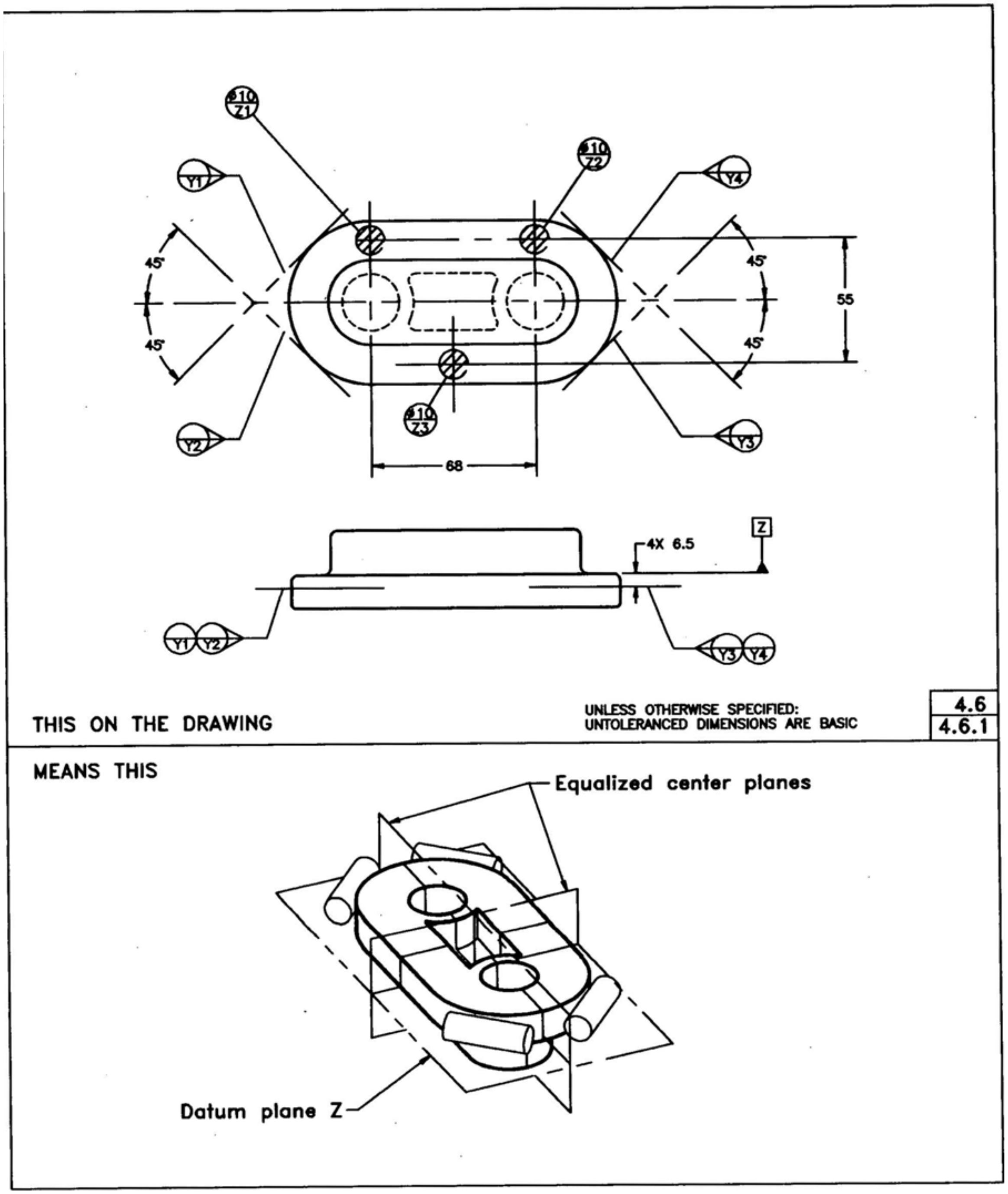


FIG. 4-12 MOVABLE DATUM TARGETS ESTABLISHING A DATUM CENTER PLANE



From the ASME Y14.8M-1996 standard

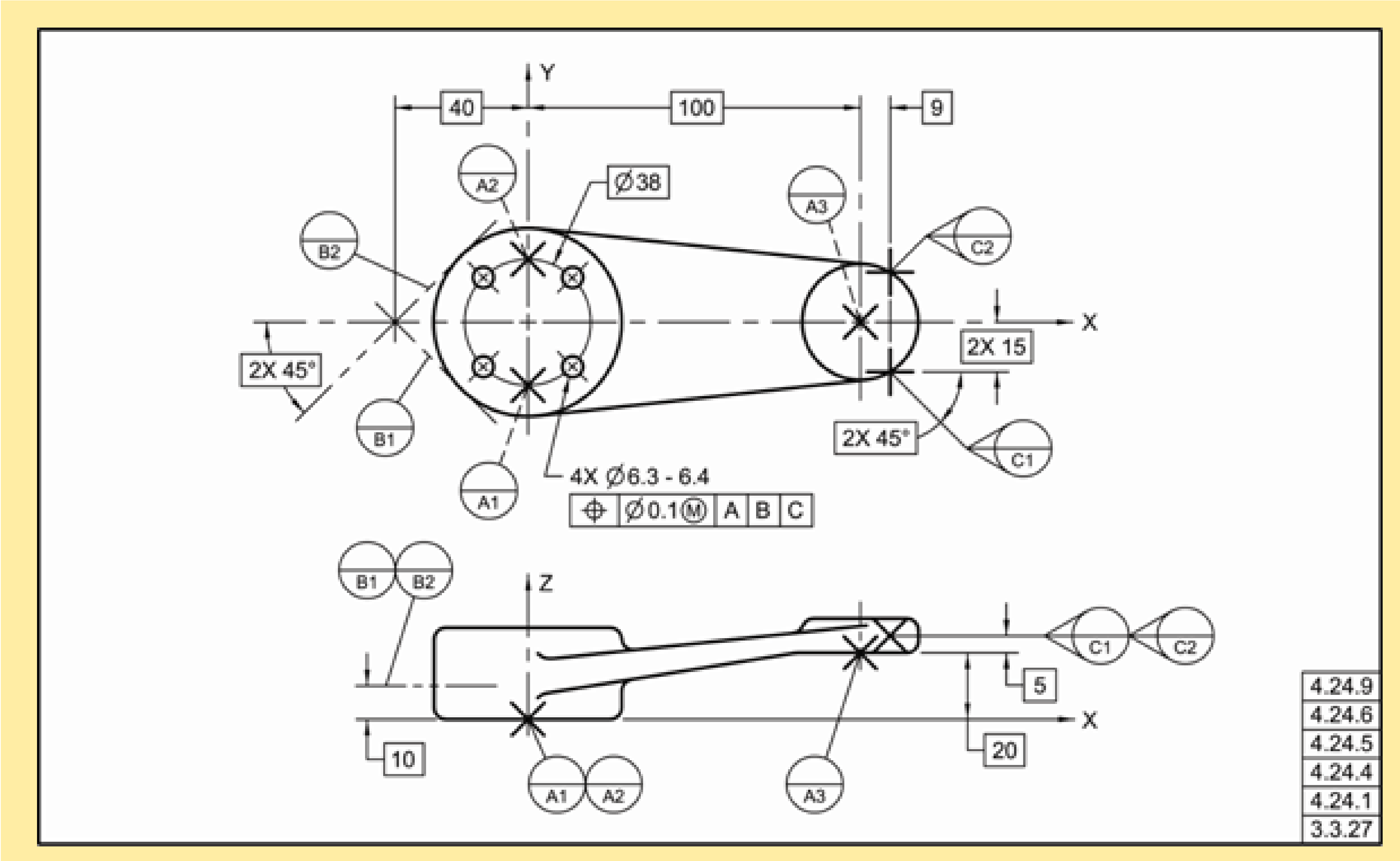


Fig. 4-47 from the ASME Y14.5-2009 standard.

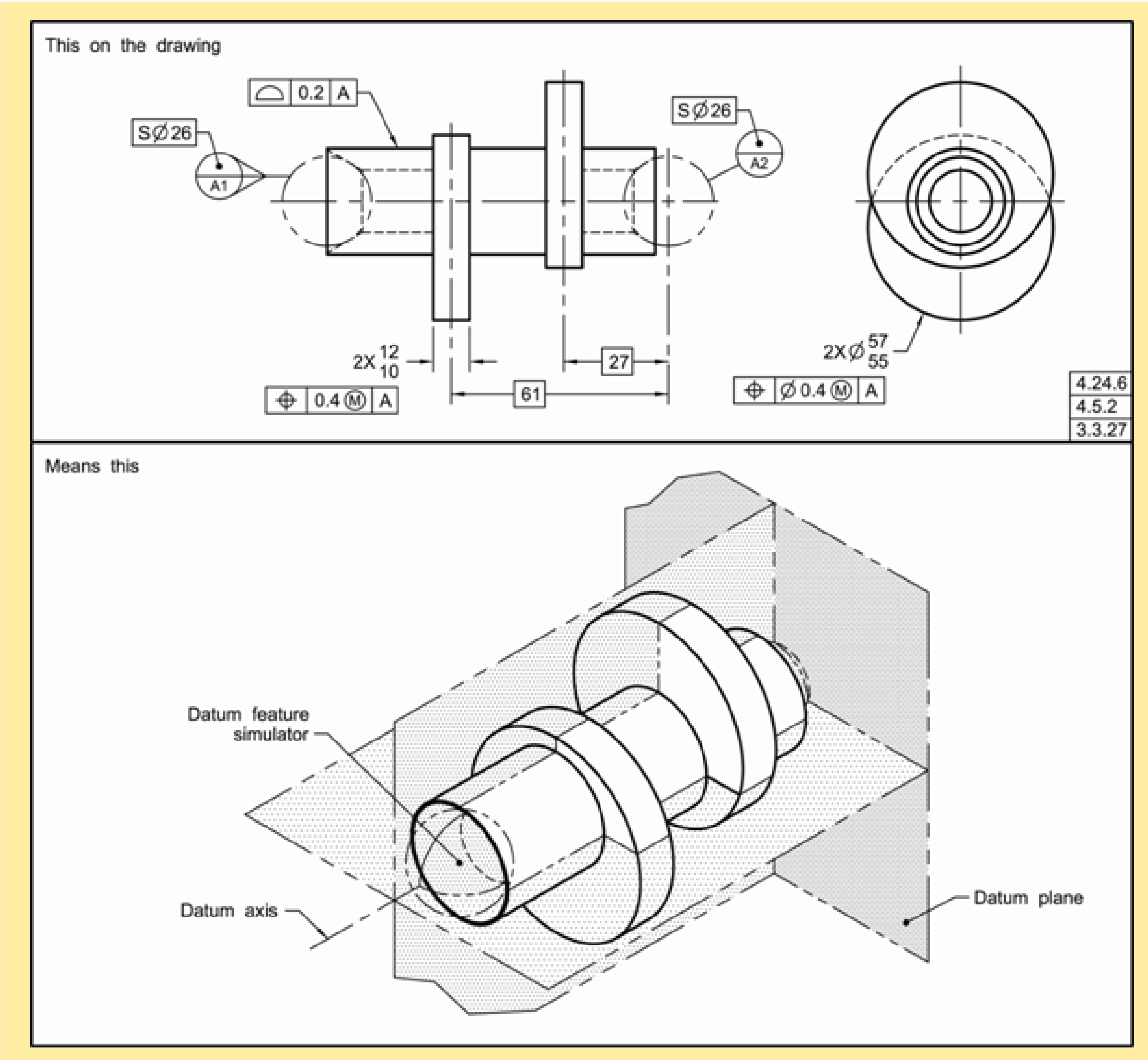


Fig. 4-49 from the ASME Y14.5-2009 standard.

ASME

SETTING THE STANDARD

Section 5
Tolerances of Form

Dr. Don W. Shepherd

Shepherd Industries

Location of Form Tolerances Section

Form tolerances were formerly in Section 6 with profile, orientation, and runout. To better guide the user of the Standard in the application of geometric dimensioning and tolerancing, form tolerances now are in a separate section.

Straightness of a Center Plane

Straightness of a center plane has been revised to flatness of a center plane to better reflect that it is a three-dimensional control, even though it is the same principle as in the previous edition of the Standard.

Flatness of a derived median plane - RFS

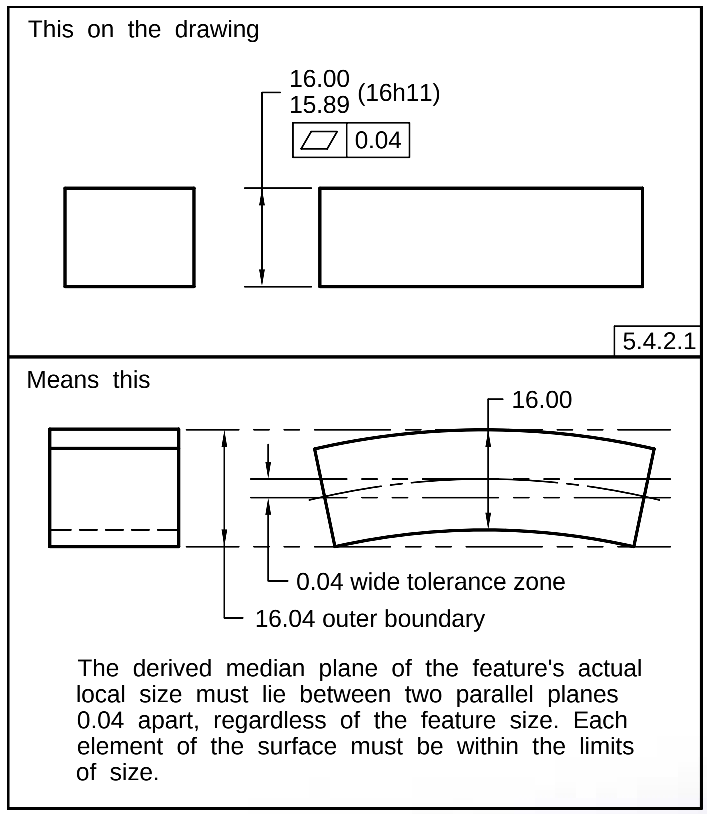
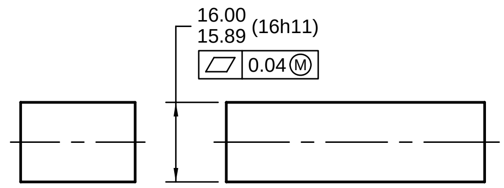


FIG. 5-8 SPECIFYING FLATNESS OF A DERIVED MEDIAN PLANE - RFS
REV A - 2009-04-27

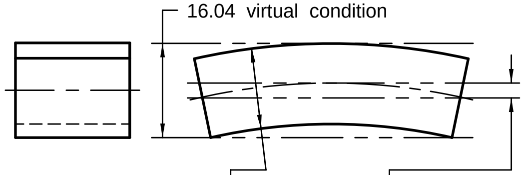
Flatness of a derived median plane at MMC

This on the drawing



5.4.2.1

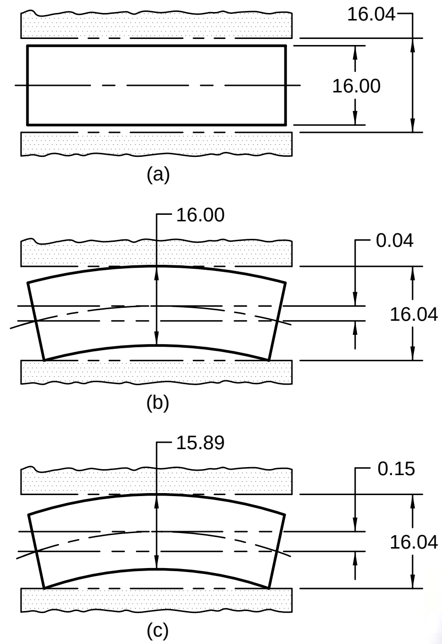
Means this



Feature size	Parallel planes tolerance allowed
16.00	0.04
15.99	0.05
15.98	0.06
↓	↓
15.90	0.14
15.89	0.15

The derived median plane of the feature actual local sizes must lie between two parallel planes 0.04 apart at MMC. As each actual local size departs from MMC, an increase in the local width of the tolerance zone is allowed which is equal to the amount of such departure. Each element of the surface must be within the specified limit of size.

Acceptance boundary



Meanings:

(a) The maximum thickness of the part with perfect form is shown in a simulated boundary with a 16.04 wide slot;

(b) with the part at maximum thickness (16.00), the simulated boundary will accept the part with up to 0.04 variation in flatness;

(c) with the part at minimum thickness (15.89), the simulated boundary will accept the part with up to 0.15 variation in flatness.

FIG. 5-9 SPECIFYING FLATNESS OF A DERIVED MEDIAN PLANE AT MMC

ASME

SETTING THE STANDARD

Section 6
Tolerances of Orientation

Dr. Don W. Shepherd

Shepherd Industries

Location of Orientation Tolerances Section

Orientation tolerances were formerly in Section 6 with form, profile, and runout. To better guide the user of the Standard in the application of geometric dimensioning and tolerancing, orientation tolerances now are in a separate section.

Definitions

The definitions of Angularity, Parallelism, and Perpendicularity are unchanged from ASME Y14.5M-1994. In that revision the description of the three orientation tolerances and their tolerance zones was repeated for each of the three types of tolerance. The only difference between the three descriptions was the terms angularity, parallelism, or perpendicularity. In this revision the description of the tolerances and their tolerance zones is only stated one time and applies to all three orientation tolerances.

Orientation Definition as Condensed

6.4.2 Orientation Tolerance

An orientation tolerance specifies one of the following:

(a) a tolerance zone defined by two parallel planes at the specified basic angle from, parallel to, or perpendicular to one or more datum planes or a datum axis, within which the surface or center plane of the considered feature must lie. See Figs. 6-1 through 6-5.

(b) a tolerance zone defined by two parallel planes at the specified basic angle from, parallel to, or perpendicular to one or more datum planes or a datum axis, within which the axis of the considered feature must lie. See Figs. 6-6 and 6-7.

(c) a cylindrical tolerance zone at the specified basic angle from, parallel to, or perpendicular to one or more datum planes or a datum axis, within which the axis of the considered feature must lie. See Figs. 6-8 through 6-15.

(d) a tolerance zone defined by two parallel lines at the specified basic angle from, parallel to, or perpendicular to a datum plane or axis, within which the line element of the surface must lie. See Figs. 6-16 and 6-17.

ASME SETTING THE STANDARD

Angularity Symbol

The use of the angularity symbol meaning orientation is introduced as an alternative practice to using perpendicularity or parallelism.

Specifying Orientation for a plane surface relative to two datums

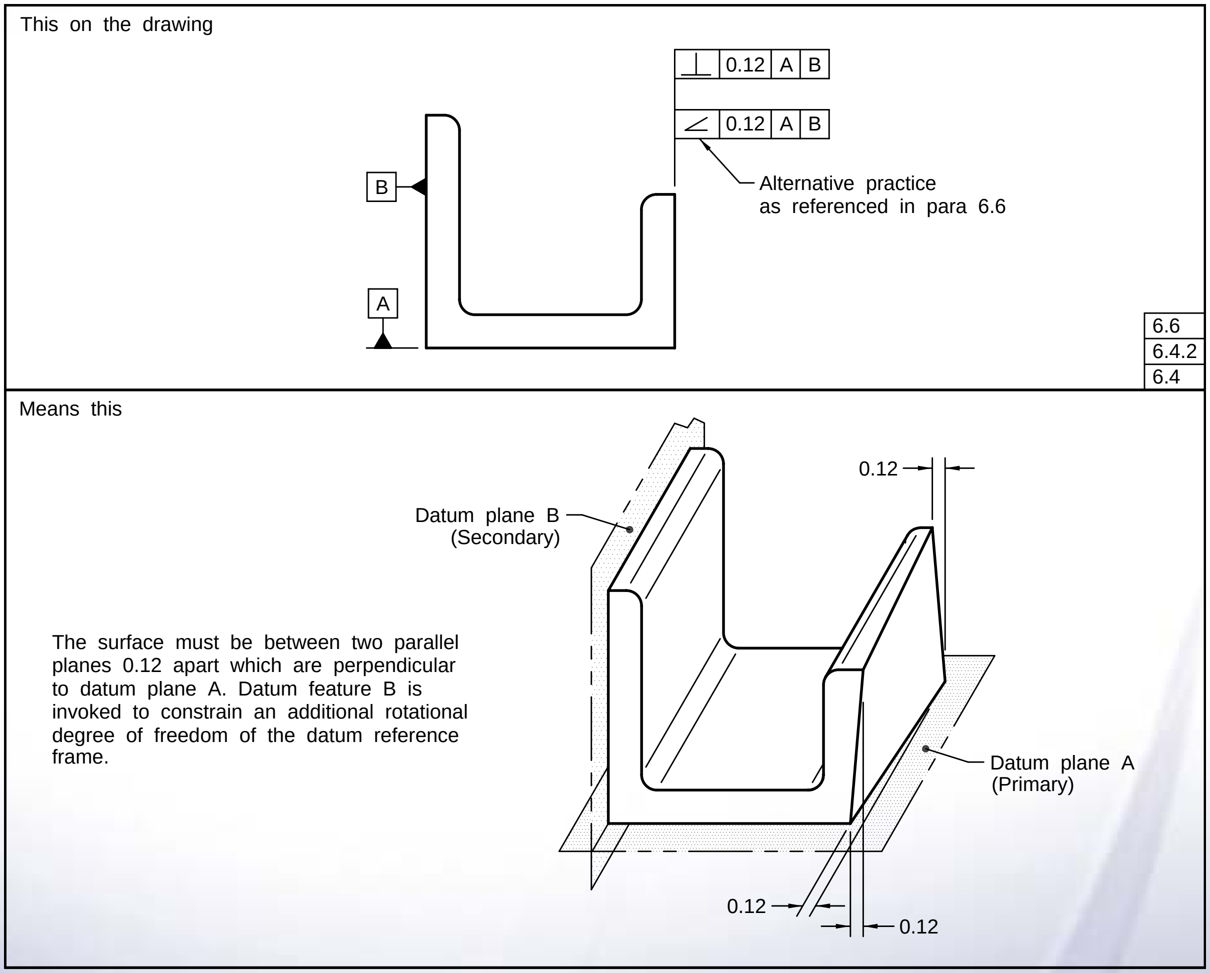


FIG. 6-4 SPECIFYING ORIENTATION FOR A PLANE SURFACE RELATIVE TO TWO DATUMS

Section 7
Tolerances Of Location

SETTING THE STANDARD

Bruce A. Wilson
10/8/2009

New Section Location

- Tolerances of position were formerly in Section 5. In order to better guide the user of the Standard in the application of geometric dimensioning and tolerancing, tolerances of position have been moved to the new Section 7. (A9.1)

Reorganized Content

- Tolerances of position have been reformatted so the material flows from the basic to more complex principles. (A9.2)
  - 7.3 Positional Tolerancing Fundamentals: I
  - 7.4 Positional Tolerancing Fundamentals: II
  - 7.5 Pattern Location
  - 7.6 Coaxial Feature Controls
  - 7.7 Tolerancing For Symmetrical Relationships

Position Between Features and Position To Datums

- Para. 7.2 defines positional tolerances to include their relationship to other features and relative to datums. (A9.3)

7.2 POSITIONAL TOLERANCING

Position is the location of one or more features of size relative to one another or to one or more datums. A positional tolerance defines either of the following:

(a) a zone within which the center, axis, or center

- This does not say that implied datums are allowed.

Applicability to Digital Data Files (CAD Models)

- Para. 7.2.1.1 added basic dimensions defined by CAD data. (A9.4)
  - Ensures applicability to digital data files (CAD models) without displayed basic dimensions
  - Comply with Y14.41

7.2.1.1 Dimensions for True Position. Dimensions used to locate true position shall be basic and defined in accordance with para. 2.1.1.2. See Fig. 7-1. For applicable notes in digital data files, see ASME Y14.41.

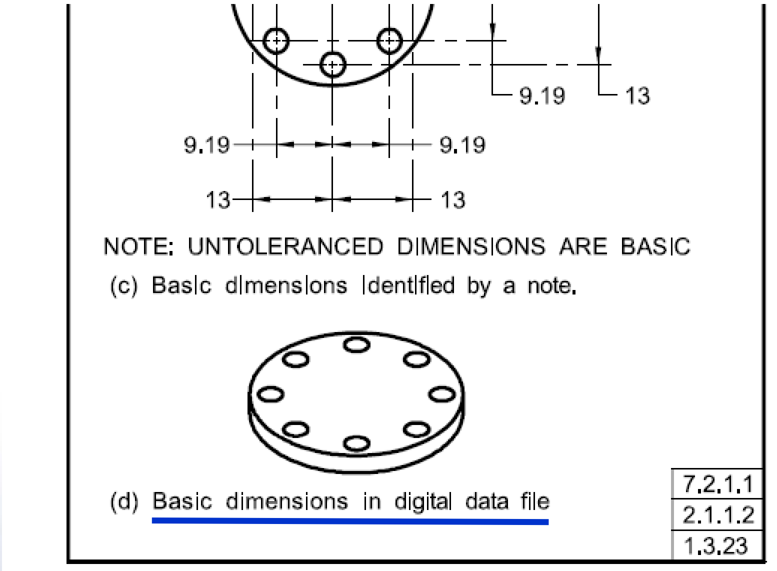


Figure 7-1.

Surface Interpretation Precedence Emphasized

- Para. 7.3.3.1 Previously a note had indicated that the surface interpretation takes precedence over the axis interpretation of positional tolerance. This was made part of Para. 7.3.3.1. (A9.5)
  - Text in three locations and a figure emphasize the boundary approach.

Para. 2.8.

specified on the drawing where it is required.

NOTES:

(1) The following paragraphs describe the principles based on an axis interpretation for RFS, MMC, and LMC. In certain cases of surface deviation of the feature, the tolerance in terms of the feature axis or feature center plane may not be exactly equivalent to the tolerance in terms of the surface limited by a boundary. In such cases, the surface interpretation shall take precedence. See para. 7.3.3.1(a) and Fig. 7-6.

7.3.3.1 Explanation of Positional Tolerance at MMC. A positional tolerance applied at MMC may be explained in terms of the surface or the axis of the feature of size. In certain cases of extreme form deviation (within limits of size) or orientation deviation of the hole, the tolerance in terms of the axis may not be exactly equivalent to the tolerance in terms of the surface. See Fig. 7-6. In such cases, the surface interpretation shall take precedence. In some instances, the additional tolerance may indirectly benefit features other than the one that departed from MMC.

6.4.5 Explanation of Orientation Tolerance at MMC

An orientation tolerance applied at MMC may be explained in terms of the surface or the feature axis. In certain cases of extreme form deviation (within limits of size) of the hole, the tolerance in terms of the feature axis may not be exactly equivalent to the tolerance in terms of the surface. In such cases, the surface interpretation shall take precedence as in Fig. 7-6.

(a) *In Terms of the Surface of a Hole.* While maintaining

ASME SETTING THE STANDARD

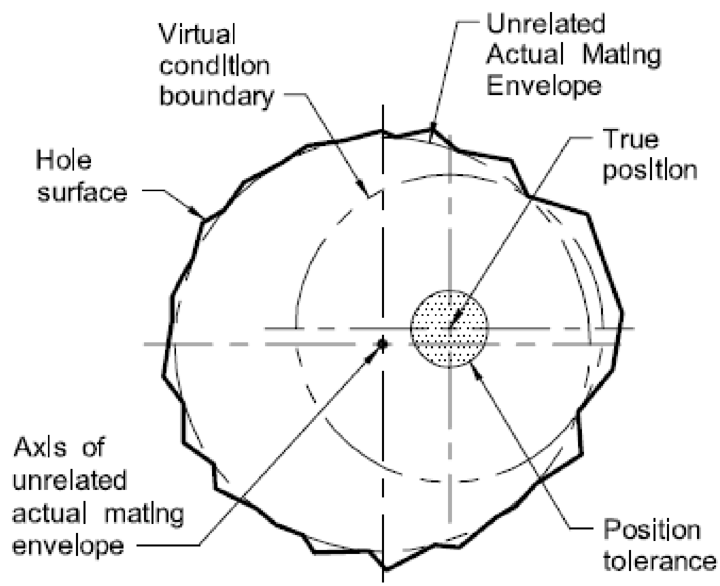
Surface Interpretation Precedence Emphasized

- the difference at a conceptual level as shown in the standard.

7.3.3.1 Explanation of Positional Tolerance at MMC. A positional tolerance applied at MMC may be explained in terms of the surface or the axis of the feature of size. In certain cases of extreme form deviation (within limits of size) or orientation deviation of the hole, the tolerance in terms of the axis may not be exactly equivalent to the tolerance in terms of the surface. See Fig. 7-6. In such cases, the surface interpretation shall take precedence. In some instances, the additional tolerance may indirectly benefit features other than the one that departed from MMC.

(*a*) *Surface Interpretation*. While maintaining the specified size limits of the feature, no element of the surface

Fig. 7-6 Illustration of Difference Between Surface and Axis Interpretations of Position Tolerancing for a Cylindrical Hole



If size requirements are met, and the virtual condition is not violated, the feature is acceptable even if the axis of the unrelated actual mating envelope is outside the positional tolerance zone.

7.3.5
7.3.3.1
6.4.5
2.8

Zero Position at LMC

- Para. 7.3.5.3 Zero positional tolerance at LMC has been added [illustrated]. (A9.6)

requirements (such as wall thickness) while specifying a zero positional tolerance at LMC. When this is done, the positional tolerance allowed is totally dependent on the actual minimum material size of the considered feature of size. Figure 7-14 shows the same drawing as Fig. 7-13, except the tolerances have been changed to show zero positional tolerance at LMC. Note that the minimum size limit of the hole remains the same, but the maximum was adjusted to correspond with a 20.25-diameter virtual condition. This results in an

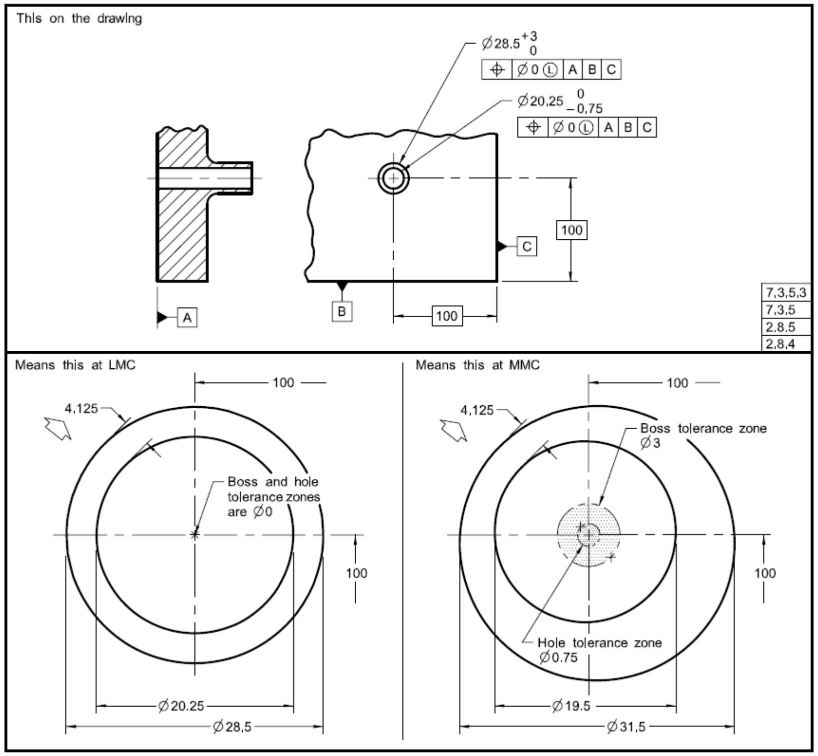


Figure 7-14

Degrees of Freedom Terminology

- Effects of datum references in positional tolerances explained in degrees of freedom. (A9.7)
  - Degrees of freedom explain the tolerance zone framework relationship to the datum reference frame
  - Separates terminology for constraints (translation and rotations) on the framework from the terminology for the toleranced features (location and orientation)

Degrees of Freedom Terminology & Variation

- Effects of datum references in positional tolerances explained in degrees of freedom. (A9.7)

(a) Pattern-Locating Tolerance Zone Framework (PLTZF). (The acronym is pronounced "Plahtz.") Where composite controls are used, the uppermost segment is the pattern-locating control. The PLTZF is constrained in rotation and translation relative to the specified datums. It specifies the larger positional tolerance for the location of the pattern of features of size as a group. Applicable datum features are referenced in a desired order of precedence, and serve to relate the PLTZF to the datum reference frame. See Fig. 7-38, illustration (a).

(b) Feature-Relating Tolerance Zone Framework (FRTZF). (The acronym is pronounced "Fritz.") Each lower segment is a feature-relating control. They govern the smaller positional tolerance for each feature of size within the pattern (feature-to-feature relationship). Basic dimensions used to relate the PLTZF to specified datums are not applicable to the location of the FRTZF. See Fig. 7-38, illustration (b).

(1) Where datum references are not specified in a

Fig. 7-33 Tolerance Zone for Center Plane of Slot at MMC

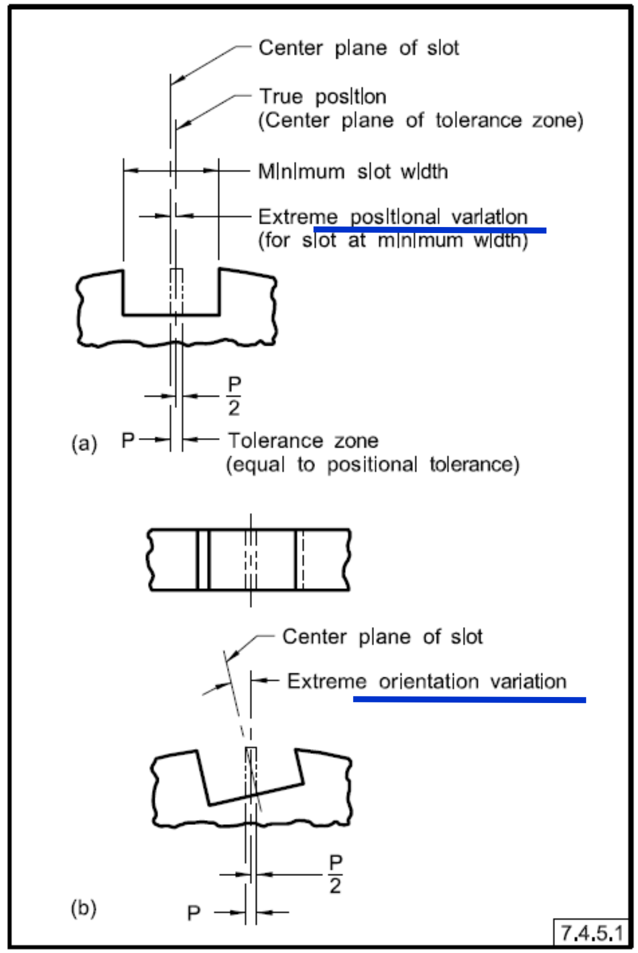


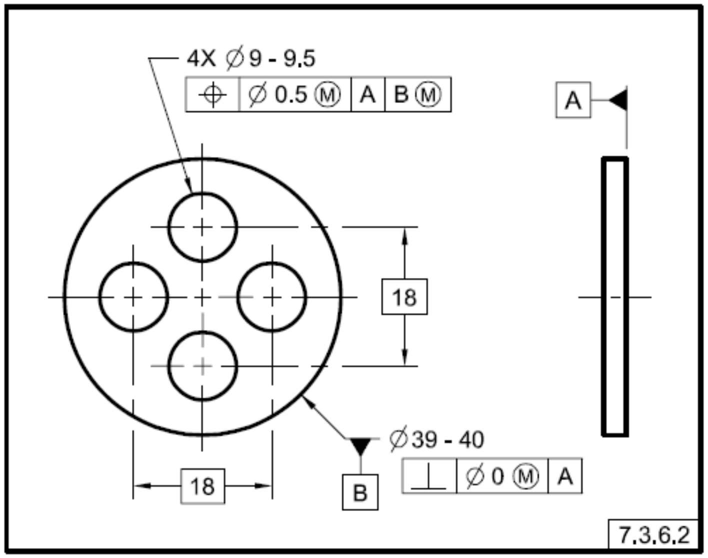
Figure 7.5.1

MMB Departure Effects on Position

- Para. 7.3.6.2.2 Effects on position tolerances caused by datum feature departure from MMB explained. (A9.8)
  - Relative movement between the datum features and the datums

7.3.6.2 Displacement Allowed by Datum Features at MMB. For some applications, a feature or group of features (such as a group of mounting holes) may be positioned relative to a datum feature(s) of size at MMB. See Fig. 7-18. In the given figure, displacement is allowed when the datum feature departs from MMB.

Fig. 7-18 Datum Feature Referenced at MMB



MMB Departure Effects on Position (cont.)

- Para. 7.3.6.2.2 Effects on position tolerances caused by datum feature departure from MMB explained. (A9.8)

Figure 7-18(a)

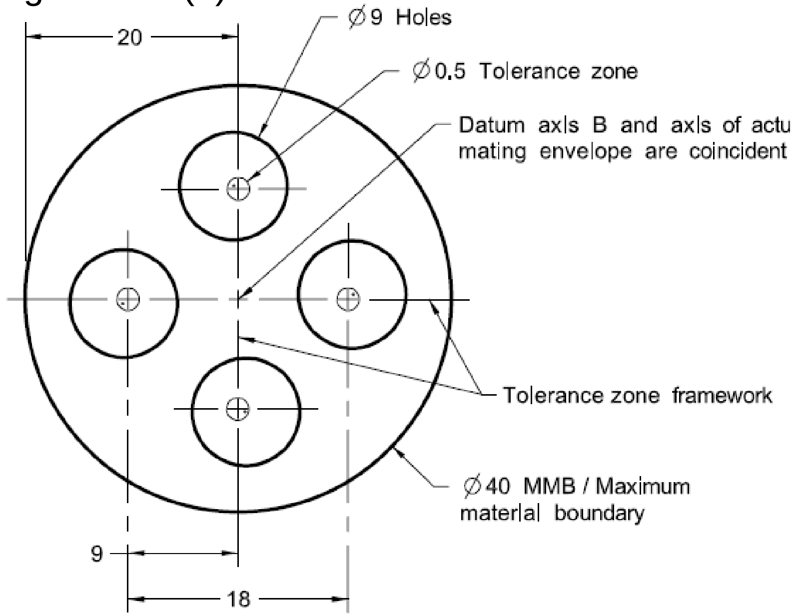
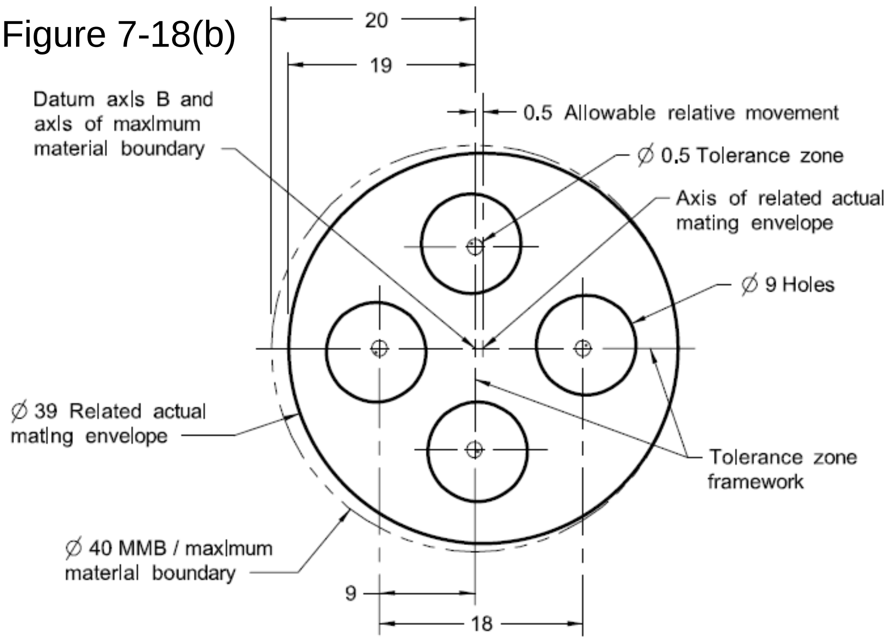


Figure 7-18(b)



ASME
SETTING THE STANDARD

Repetitive Patterns

- Para. 7.4.8 Positional tolerances on repetitive patterns of features. (A9.9)
  - Expanded explanation

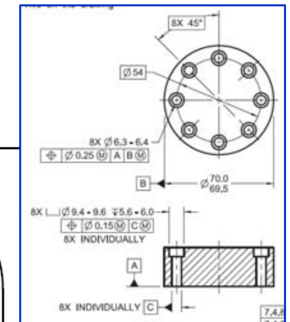
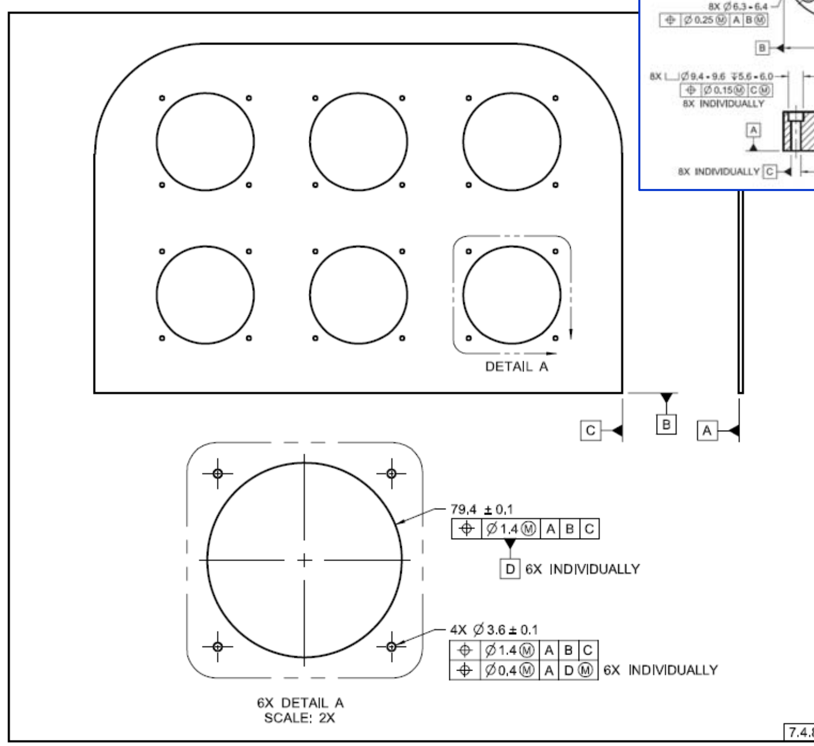


Fig. 7-37 **Multiple Patterns of Features**



7.4.8 Repetitive Pattern of Features of Size Related to a Repeated Datum Reference Frame

Where positional tolerances are used to locate patterns of features of size relative to repetitive datums, the feature control frames and datums are specified as shown in Figs. 7-26 and 7-37. A note is placed beneath or adja-

ual requirements in a detail view. When a detail view includes a notation of the number of occurrences of that detail view, then the 6X on the INDIVIDUALLY notation may be omitted. The 6X INDIVIDUALLY notation beside the datum feature D symbol indicates that each of the six occurrences of the 79.4-diameter hole acts as a separate datum feature and establishes a separate datum D. The 6X INDIVIDUALLY notation associated with the second segment of the positional tolerances on the 4X 3.6-diameter holes indicates that each pattern of four holes has a tolerance zone framework that is located relative to the specified datums.

Simplified Figures - Repetition Minimized

- Sequential figures for composite positional tolerances were simplified so that each figure contains only the information to build on the previously shown figure. This reduced the amount of information that was repeated and makes finding the relevant information much easier. (A9.10)
  - 1994 repeated the entire series of figure segments for each explanation
  - 2009 relies on the previous figure so that only the relevant information is shown
    - Figure 7-38(a) through (f) develops the complete set of requirements
    - Figure 7-39(a) builds on 7-38

Project Zones – Composite Position

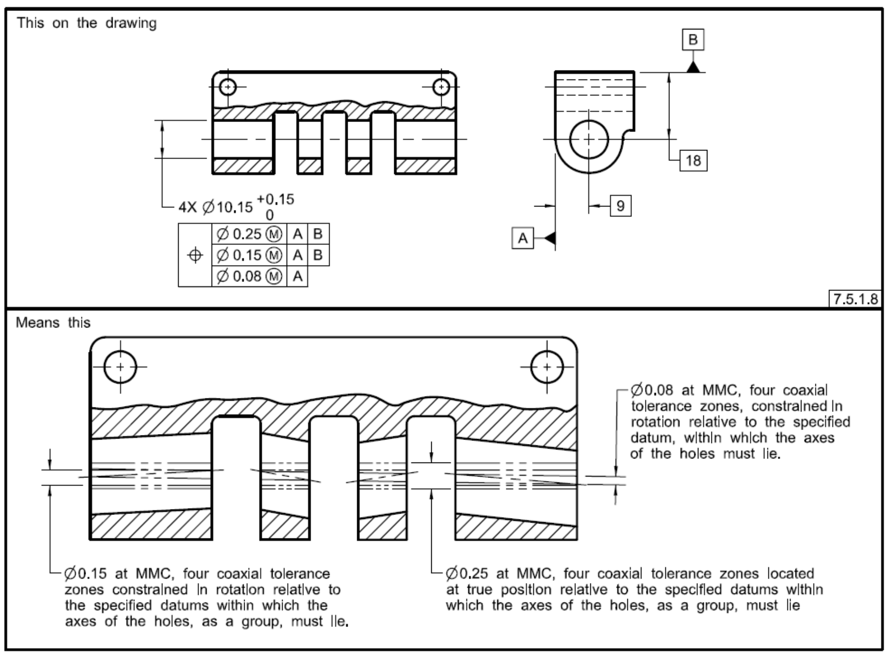
- Para. 7.5.1.7 Projected tolerance zones for composite tolerances are shown and explained. (A9.11)
  - Projected zones were not previously explained for composite

7.5.1.7 Projected Tolerance Zones for Composite Positional Tolerancing. Where the design dictates the use of a projected tolerance zone for composite positional tolerancing, the projected tolerance zone symbol is placed in the applicable segment(s) of the composite feature control frame as required. The projected tolerance zone applies only to the segment in which the symbol is shown. Where a projected tolerance zone is specified, the feature axes shall simultaneously lie within both the pattern and feature locating tolerance zones.

Triple Segment Composite Illustrated

- Para. 7.5.1.8 Addition of multiple segments to a composite tolerance feature control frame are illustrated and explained. (A9.12)
  - Three segments

Fig. 7-44 Positional Tolerancing for Coaxial Holes of Same Size, Partial (Parallelism) Refinement of Feature-Relating Axis Relative to Datums A and B With Further Refinement of Parallelism to Datum A



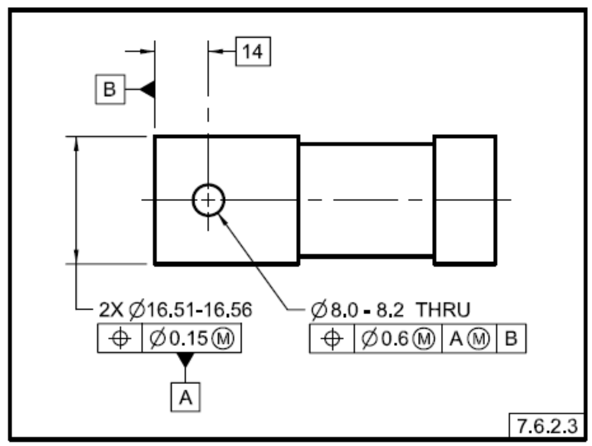
7.5.1.8 Composite Positional Tolerances: Multiple Segments. Composite tolerances have two or more segments. Each segment establishes tolerance zones and constraints to any referenced datums shown in the segment. Datum references in the first segment establish all applicable rotational and translational constraints relative to the referenced datums. Datum references in the second and subsequent segments establish only rotational constraints relative to the referenced datums. See Fig. 7-44. Absence of datum references in a segment indicate that no rotational or translation constraints are established by that segment. For a pattern of features

Position Without Datum Feature References

- Para. 7.6.2.3 Position tolerance is shown without a datum reference to control coaxial features. (A9.13)
  - The pattern of features serves as a datum feature – probably the primary datum feature for other tolerances.
  - Assumed or implied datum feature references **not** permitted

7.6.2.3 Coaxial Features Without Datum References. A coaxial relationship may be controlled by specifying a positional tolerance without datum references, as shown in Fig. 7-59. This method allows specific control of feature-to-feature coaxiality. Where features are specified with different sizes, a single feature control frame, supplemented by a notation, such as TWO COAXIAL FEATURES, is used. A positional tolerance specification with no datum reference creates a relationship between the toleranced features, but implies no relationship to any other features. The toleranced features may be identified as a single datum feature that may then

Fig. 7-59 Two Datum Features, Single Datum Axis



ASME
SETTING THE STANDARD

Profile for Coaxiality

- Para. 7.6.5 Profile tolerancing is added as a method to control coaxiality. (A9.14)
  - Profile may be used and is explained in Section 8

7.6.5 Profile of a Surface Tolerance Control

For information on controlling the coaxiality of a surface of revolution relative to a datum axis with profile of a surface tolerance, see para. 8.4.2.

Other Small Additions

- Additional tolerance may come from other features in a pattern
- Additional tolerance based on surface interpretation may be different than axis interpretation

7.3.3.1 Explanation of Positional Tolerance at MMC. A positional tolerance applied at MMC may be explained in terms of the surface or the axis of the feature of size. In certain cases of extreme form deviation (within limits of size) or orientation deviation of the hole, the tolerance in terms of the axis may not be exactly equivalent to the tolerance in terms of the surface. See Fig. 7-6. In such cases, the surface interpretation shall take precedence. In some instances, the additional tolerance may indirectly benefit features other than the one that departed from MMC.

(a) *Surface Interpretation.* While maintaining the speci-

ASME
SETTING THE STANDARD

Questions

Answers represent the opinion of the respondent and do not represent ASME or the respondent's employer.

ASME

SETTING THE STANDARD

ASME

SETTING THE STANDARD

Section 6
Tolerances of Orientation

Dr. Don W. Shepherd

Shepherd Industries

Location of Orientation Tolerances Section

Orientation tolerances were formerly in Section 6 with form, profile, and runout. To better guide the user of the Standard in the application of geometric dimensioning and tolerancing, orientation tolerances now are in a separate section.

Definitions

The definitions of Angularity, Parallelism, and Perpendicularity are unchanged from ASME Y14.5M-1994. In that revision the description of the three orientation tolerances and their tolerance zones was repeated for each of the three types of tolerance. The only difference between the three descriptions was the terms angularity, parallelism, or perpendicularity. In this revision the description of the tolerances and their tolerance zones is only stated one time and applies to all three orientation tolerances.

Orientation Definition as Condensed

6.4.2 Orientation Tolerance

An orientation tolerance specifies one of the following:

(a) a tolerance zone defined by two parallel planes at the specified basic angle from, parallel to, or perpendicular to one or more datum planes or a datum axis, within which the surface or center plane of the considered feature must lie. See Figs. 6-1 through 6-5.

(b) a tolerance zone defined by two parallel planes at the specified basic angle from, parallel to, or perpendicular to one or more datum planes or a datum axis, within which the axis of the considered feature must lie. See Figs. 6-6 and 6-7.

(c) a cylindrical tolerance zone at the specified basic angle from, parallel to, or perpendicular to one or more datum planes or a datum axis, within which the axis of the considered feature must lie. See Figs. 6-8 through 6-15.

(d) a tolerance zone defined by two parallel lines at the specified basic angle from, parallel to, or perpendicular to a datum plane or axis, within which the line element of the surface must lie. See Figs. 6-16 and 6-17.

ASME SETTING THE STANDARD

Angularity Symbol

The use of the angularity symbol meaning orientation is introduced as an alternative practice to using perpendicularity or parallelism.

Specifying Orientation for a plane surface relative to two datums

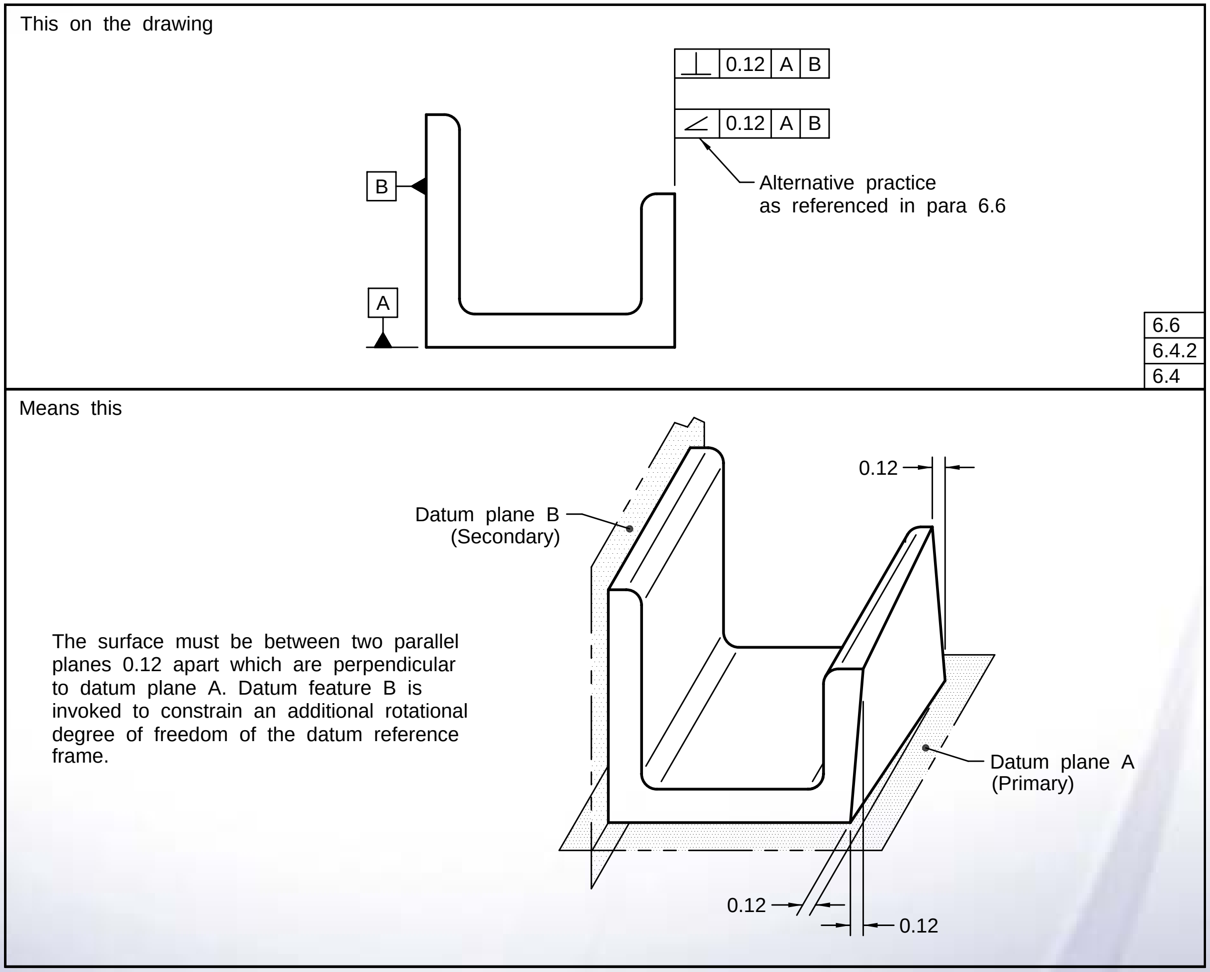


FIG. 6-4 SPECIFYING ORIENTATION FOR A PLANE SURFACE RELATIVE TO TWO DATUMS

Form Tolerances at MMC

Para 6.4.5

A form tolerance applied at MMC may be explained in terms of either the surface or the derived median line/plane. In certain cases of extreme form deviation, the tolerance in terms of the feature’s derived median line/plane may not be exactly equivalent to the tolerance in terms of the surface. In such cases, the surface interpretation shall take precedence.

ASME

SETTING THE STANDARD

Section 9
Tolerances of Runout

Dr. Don W. Shepherd
Shepherd Industries

Location of Runout Tolerances Section

Runout tolerances were formerly in Section 6 with form, profile, and orientation. To better guide the user of the Standard in the application of geometric dimensioning and tolerancing, runout tolerances now are in a separate section.