THIRD EDITION

GEOMETRIC DIMENSIONING AND TOLERANCING

FOR MECHANICAL DESIGN

GENE R. COGORNO

Geometric Dimensioning and Tolerancing for Mechanical Design

Instructors' Guide

Contents

Course Calendar

Week	Date	1st Weekly Meeting	2nd Weekly Meeting	3rd Weekly Meeting
1		Admin. & Overview	Lecture 1 Ch. 1	Lecture 2 Ch. 2
2		Lecture 3 Ch. 3	Lecture 4 Ch. 3	Lecture 5 Ch. 3
3		Lecture 6 Ch. 4	Lecture 7 Ch. 4	Lecture 8 Ch. 4
4		Lecture 9 Ch. 5	Lecture 10 Ch. 5	Lecture 11 Ch. 5
5		Lecture 12 Ch. 5	Lecture 13 Ch. 5	Lecture 14 Ch. 6
6		Lecture 15 Ch. 6	Lecture 16 Ch. 6	**First Midterm Exam**
7		**Midterm Review**	Lecture 17 Ch. 7	Lecture 18 Ch. 7
8		Lecture 19 Ch. 7	Lecture 20 Ch. 7	Lecture 21 Ch. 8
9		Lecture 22 Ch. 8	Lecture 23 Ch. 8	Lecture 24 Ch. 8
10		Lecture 25 Ch. 9	Lecture 26 Ch. 9	Lecture 27 Ch. 9
11		**Second Midterm Exam**	**Midterm Review**	Lecture 28 Ch. 10
12		Lecture 29 Ch. 10	Lecture 30 Ch. 11	Lecture 31 Ch. 11
13		Lecture 32 Ch. 11	Lecture 33 Ch. 12	Lecture 34 Ch. 12
14		Lecture 35 Ch. 12	Lecture 36 Append. A	Lecture 37 Append. A
15		**Final Review**	**Final Review**	**Final Exam**

Week	Date	1st Weekly Meeting	2nd Weekly Meeting	3rd Weekly Meeting
1		Admin. & Overview	Lecture 1 Ch. 1	Lecture 2 Ch. 3
2		Lecture 3 Ch. 3	Lecture 4 Ch. 3	Lecture 5 Ch. 4
3		Lecture 6 Ch. 4	Lecture 7 Ch. 5	Lecture 8 Ch. 5
4		Lecture 9 Ch. 6	Lecture 10 Ch. 6	Lecture 11 Ch. 6
5		**Midterm Exam**	**Midterm Review**	Lecture 12 Ch. 7
6		Lecture 13 Ch. 7	Lecture 14 Ch. 7	Lecture 15 Ch. 8
7		Lecture 16 Ch. 8	Lecture 17 Ch. 8	Lecture 18 Ch. 8
8		Lecture 19 Ch. 9	Lecture 20 Ch.10	Lecture 21 Ch.11
9		Lecture 22 Ch.11	Lecture 23 Ch.12	Lecture 24 Ch.12
10		Lecture 25 Ch. 12	**Final Review**	**Final Exam**

Lecture Topics

No.	Topics
	1. Introduction
1	A. What is Geometric Dimensioning and Tolerancing
	B. When should GD&T be used?
	C. Advantages of GD&T over coordinate dimensioning and tolerancing
	1. The cylindrical tolerance zone
	2. The maximum material condition
	3. Datums specified in order of precedence
	2. Dimensioning and Tolerancing Fundamentals
2	A. Fundamental drawing rules
	B. Units of linear measurement
	C. Specifying linear dimensions
	D. Specifying linear tolerances
	E. Interpreting dimensional limits
	F. Specifying angular dimensions
	G. Specifying angular tolerances
	H. Dimensioning and Tolerancing for CAD/CAM database models
	3. Symbols, Terms, and Rules
3	A. Geometric characteristic symbols
	B. The datum feature symbol
	C. The feature control frame
	D. Reading the feature control frame
	E. Other symbols used with geometric tolerancing
4	F. Terms
5	G. General rules
	1. Rule #1
	2. Rule #2
	3. Pitch diameter rule
	4. Datums
6	A. Definition
	B. Application of datums
	C. Immobilization of a part
	D. Datum feature selection
	E. Datum feature identification
	F. Inclined datum features
7	G. Cylindrical datum features
	H. Establishing datum features
	I. Irregular features of size
	J. Common datum features
8	K. Partial datum features
	L. Datum targets

	5. Form controls
9	A. Flatness
	1. Definition
	2. Specifying flatness tolerance
	3. Specifying flatness of a median plane
	4. Unit flatness
10	B. Straightness
	1. Definition
	2. Specifying straightness of a surface
	3. Specifying straightness of a median line
	4. Unit straightness
11	C. Circularity
	1. Definition
	2. Specifying Circularity Tolerance
12	D. Cylindricity
	1. Definition
	2. Specifying Cylindricity Tolerance
13	E. Free state variation
	1. Average diameter
	2. Free state
	3. Restrained condition
	6. Orientation
14	A. Perpendicularity
	1. Definition
	2. Specifying perpendicularity of a flat surface
	3. Tangent plane
	4. Specifying perpendicularity of an axis
	B. Parallelism
15	1. Definition
	2. Specifying parallelism of a plane surface
	3. Specifying parallelism of an axis
16	C. Angularity
	1. Definition
	2. Specifying angularity of a plane flat surface
	3. Specifying angularity of an axis
	7. Position, General
	A. Definition
	B. The tolerance of position
17	C. Specifying the position tolerance at
	1. Regardless of feature size (RFS)
	2. Maximum material condition (MMC)
	D. Datum features of size at regardless of material boundary (RMB)
18	E. Datum features of size at maximum material boundary (MMB)
	F. MMB explained in more detail
19	G. Least material condition

20	H. Boundary conditions
21	I. "0" positional tolerancing at MMC
	8. Position, Location
22	A. Floating and fixed fasteners
23	B. Projected tolerance zones
	C. Multiple patterns of features
24	D. Composite positional tolerancing
	E. Multiple single-segment positional tolerancing
	F. Nonparallel holes
25	G. Counterbored holes
	H. Noncircular features
	I. Symmetrical features
	9. Position, Coaxiality
	A. Definition
26	B. Comparison between coaxiality controls
	C. Specifying coaxiality at MMC
	D. Composite control of coaxial features
27	E. Coaxial features controlled without datum references
	F. Tolerancing a plug and socket
	10. Runout
	A. Definition
	B. Circular runout
28	C. Total runout
	D. Specifying runout and partial runout
	E. Multiple datum features
	F. Face and diameter datum features
29	G. Geometric control to refine datum features
	H. The relationship between feature surfaces
	I. Inspecting runout
	11. Profile
	A. Definition
	B. Specifying profile tolerance
30	C. The application of datum features
	D. A radius refinement with profile
	E. Combining profile tolerances with other geometric controls
	F. Coplanarity
31	G. Profile of a conical feature
	H. Composite profile tolerancing
	I. Multiple single-segment profile tolerancing
	J Inspection
	12. A Strategy for Tolerancing Parts
32	A. Feature of sizes located to plane surface features
33	B. Feature of sizes located to feature of sizes
34	C. A pattern of features located to a second pattern of features

	13. Graphic Analysis
35	A. Advantages of graphic analysis
	B. The accuracy of graphic analysis
	C. Analysis of a composite geometric tolerance
36	D. Analysis of a pattern of features controlled to a datum feature of size
	Appendix A. Concentricity & Symmetry
37	A. Concentricity
	1. The definition of concentricity
	2. Specifying concentricity
	3. Applications of concentricity
	B. Symmetry
	1. The definition of symmetry
	2. Specifying symmetry
	3. Applications of symmetry

Chapter 1

Introduction to Geometric Dimensioning and Tolerancing

Chapter Review

Page 7

1. ___ASME Y14.5–2018___ is the current, authoritative reference document that specifies the proper application of GD&T.
2. GD&T is a symbolic language used to specify the

   ___size___ , ___shape___ , ___form___ , ___orientation___

   and ___location___ of features on a part.
3. Features toleranced with GD&T reflect the ___actual relationship___

   between mating parts.
4. GD&T was designed to insure the proper assembly of

   ___mating parts___ , to improve ___quality___ , and reduce ___cost___ .
5. Geometric tolerancing allows the maximum available ___tolerance___ and,

   consequently, the most ___economical___ parts.
6. Plus or minus tolerancing generates ___rectangular___ tolerance zone.
7. ___GD&T___ generates a cylindrical shaped tolerance zone to control an axis.
8. If the distance across a square tolerance zone is ± .005 or a total of .010, what is the approximate

   distance across the diagonal? ___±.007 or .014___

9. The *cylindrical tolerance zone* defines a uniform distance from true position to the tolerance zone boundary.
10. Bonus tolerance equals the difference between the actual mating envelope size and the *maximum material condition*.
11. While processing, a rectangular part usually rests against a *datum reference frame* consisting of three mutually perpendicular intersecting planes.

Chapter 2

Dimensioning and Tolerancing Fundamentals

Chapter Review

Page 15

1. Each feature must be a *toleranced*. Those dimensions specifically identified as reference, maximum, minimum, or stock do not require the application of a tolerance.
2. Dimensioning and tolerancing must be *complete* so there is a full understanding of the *characteristics* of each feature.
3. Dimensions must be selected and arranged to suite the function and mating relationship of a part and must not be subject to more than one *interpretation*.
4. The drawing should define a part without specifying *manufacturing* methods.
5. An *implied 90° angle* always applies where centerlines and lines depicting features are shown on orthographic views at right angles and no angle is specified.
6. An *implied 90° basic angle* always applies where center lines of features or surfaces shown at right angles on an orthographic view are located or defined by basic dimensions and no angle is specified.
7. Unless otherwise specified, all dimensions and tolerances are applicable at *68°F (20°C)*. Compensation may be made for measurements made at other temperatures.
8. Unless otherwise specified, all dimensions and tolerances apply in a *free state condition* except for restrained non-rigid parts.

9. Unless otherwise specified, all tolerances and datum features apply for the full ___*depth*___ ___*length*___ and ___*width*___ of the feature.

10. Dimensions and tolerances apply only at the ___*drawing level*___ where they are specified.

11. Units of linear measurement are typically expressed either in the ___*inch*___ system or the ___*metric*___ system.

12. Where specifying decimal inch dimensions, a ___*zero*___ is never placed before the decimal point for values less than one inch.

13. What are the three types of direct tolerancing methods?

*Limit, Plus and Minus, and Geometric tolerancing*___

14. Where inch tolerances are used, a dimension is specified with the same number of decimal places as its *tolerance*___.

15. Where a unilateral tolerance is specified and either the plus or minus limit is zero, its dimension and zero value must have the ___*same number of decimal places*___ as the other limit and the 0 value must have the ___*opposite sign*___ of the nonzero value.

16. For decimal inch tolerances, where bilateral tolerancing or limit dimensioning and tolerancing is used, both values ___*have the same number of decimal places.*___

17. Dimensional limits are used as if an ___*infinite number of zeros*___ followed the last digit after the decimal point.

18. Angular units of measurement are specified either in ___*degrees and decimal parts of a degree*___ or ___*degrees, minutes, and seconds*___.

19. What two dimensions are not placed on the field of the drawing?

___*90° angles and zero dimensions*___

20. Dimensioning and Tolerancing for CAD/CAM Database Models. Dimensioning and tolerancing shall be complete so there is full understanding of the characteristics of each feature. Values may be expressed in an engineering drawing or in a ___*CAD product definition data set specified in ASME Y14.41.*___

Chapter 3

Symbols, Terms, and Rules

Chapter Review

Page 46

1. What type of geometric tolerances has no datum features? *form*
2. What is the name of the symbol used to identify physical features of a part as a datum feature and must not be applied to centerlines, center planes, or axes? *datum feature symbol*
3. Datum feature identifying letters may be any letter of the alphabet except? *I, O, & Q*
4. If the datum feature symbol is placed in line with a dimension line or on a feature control frame associated with a feature of size, the datum feature is what kind of feature?

   a feature of size
5. One of the 12 geometric characteristic symbols always appears in the *first* compartment of the feature control frame.
6. The second compartment of the feature control frame is the *tolerance* section.
7. The tolerance is preceded by a diameter symbol only if the tolerance zone is *cylindrical*.
8. Datum features are arranged in order of *precedence*.

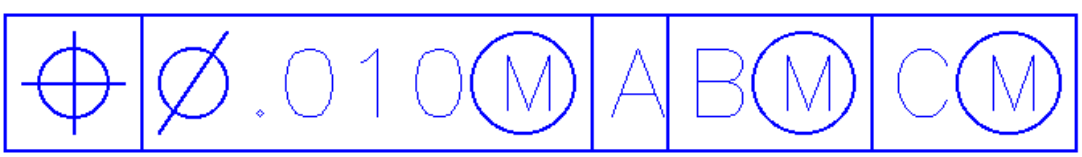


Figure 3-36 Position feature control frame

9. Read the feature control frame in Fig. 3-36

   *1. The **position** tolerance requires that*

   2. The axis of the controlled feature

3. Must lie within a cylindrical tolerance zone

4. 010 in diameter

5. At maximum material condition (MMC)

6. Oriented and located with basic dimensions to a datum reference frame established by datum feature A, datum feature B at its maximum material boundary, and datum feature C at its maximum material boundary.

10. The all around and between symbols are used with what control? *the profile control*
11. The all over symbol consists of two small *concentric circles* placed at the joint of the leader connecting the feature control frame to the feature.
12. The *continuous feature* symbol specifies that a group of two or more interrupted features of size are to be considered one single feature of size.
13. If no depth or remaining thickness is specified, the spotface is the *minimum* depth necessary to clean up the surface of the specified diameter.
14. The *independency* symbol indicates that perfect form of a feature of size at MMC or LMC is not required.
15. The *unequally disposed* symbol indicates that the profile tolerance is unilateral or unequally disposed about the true profile.
16. The *datum translation* symbol indicates that a datum feature simulator is not fixed and is free to translate within the specified geometric tolerance.
17. The *actual mating envelope* is a similar, perfect, feature(s) counterpart of smallest size that can be contracted about an external feature(s) or largest size that can be expanded within an internal feature(s) so that it coincides with the surface(s) at its highest points.
18. A theoretically exact dimension is called a *basic dimension*.
19. What is the theoretically exact point, axis, line, plane, or combination thereof derived from the theoretical datum feature simulator called? *a datum*

20. A *datum feature* is a feature that is identified with either a datum feature symbol or a datum target symbol.

21. A *datum feature simulator* is the physical boundary used to establish a simulated datum from a specified datum feature.

22. A *datum reference frame* consists of three mutually perpendicular intersecting datum planes and three mutually perpendicular axes at the intersection of those planes.

23. What is the name of a physical portion of a part, such as a surface, pin, hole, tab, or slot?

    A feature

24. A regular feature of size is a feature, which is associated with a directly toleranced dimension and takes one of the following forms:

    a) A cylindrical surface

    b) A set of two opposed parallel surfaces

    c) A spherical surface

    d) A circular element

    e) *A set of two opposed parallel elements*

Pertains to	Type of Tolerance	Geometric Characteristics	Symbol
Individual Feature Only	Form	STRAIGHTNESS	⏤
		FLATNESS	⏥
		CIRCULARITY	○
		CYLINDRICITY	⌭
Individual Feature or Related Features	Profile	PROFILE OF A LINE	⌒
		PROFILE OF A SURFACE	⌓
Related Features	Orientation	PERPENDICULARITY	⊥
		PARALLELISM	∥
		ANGULARITY	∠
	Location	POSITION	⌖
	Runout	CIRCULAR RUNOUT	↗
		TOTAL RUNOUT	⌰

Figure 3-37 Geometric characteristic symbols

25. Write the names and geometric characteristic symbols where indicated in Fig. 3-37.

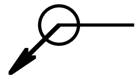
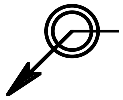
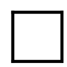
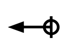
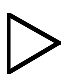
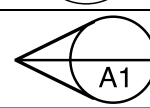
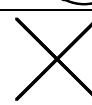
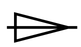
All Around		Free State	Ⓕ
All Over		Independency	Ⓘ
Between		Projected Tolerance Zone	Ⓟ
Number of Places	X	Tangent Plane	Ⓣ
Continuous Feature	⟨CF⟩	Unequally Disposed Profile	Ⓤ
Counterbore		Spotface	SF
Countersink		Radius	R
Depth/Deep		Radius, Controlled	CR
Diameter	Ø	Spherical Radius	SR
Dimension, Basic	1.000	Spherical Diameter	SØ
Dimension, Reference	(60)	Square	
Dimension Not To Scale	15	Statistical Tolerance	⟨ST⟩
Dimension Origin		Datum Target	Ø.500 A1
Datum Translation		Movable Datum Target	
Arc Length	110	Target Point	
Conical Taper		Dynamic Profile	
Slope		From - To	

Figure 3-38 Geometric tolerancing symbols

26. Draw the indicated geometric tolerancing symbols in the spaces on Fig. 3-38.

27. The *maximum material condition* is the condition in which a feature of size contains the maximum amount of material within the stated limits of size.

28. The *least material condition* is the condition in which a feature of size contains the least amount of material within the stated limits of size.

29. What kind of feature always applies at MMC/MMB, LMC/LMB, or RFS/RMB?

A feature of size

30. The maximum material condition modifier specifies that the tolerance applies at the *maximum material condition* size of the feature.

31. The MMC modifier indicates that the tolerance applies at the MMC size of the feature and that a *bonus* tolerance is available as the size of the feature departs from MMC towards LMC.

32. *Bonus* tolerance is the positive difference or the absolute value between the actual mating envelope and the MMC.

33. The total positional tolerance equals the sum of the *bonus* tolerance and the *positional* tolerance.

Refer to Fig. 3-39 to answer questions 34 through 41.

		Hole	Pin
34.	What is the MMC?	*.515*	*.500*
35.	What is the LMC?	*.540*	*.495*
36.	What is the geometric tolerance?	*.010*	*.005*
37.	What material condition modifier is specified?	*MMC*	*MMC*
38.	What datum feature(s) control(s) perpendicularity?	*A*	*A*
39.	What datum feature(s) control(s) location?	*B & C*	*B & C*

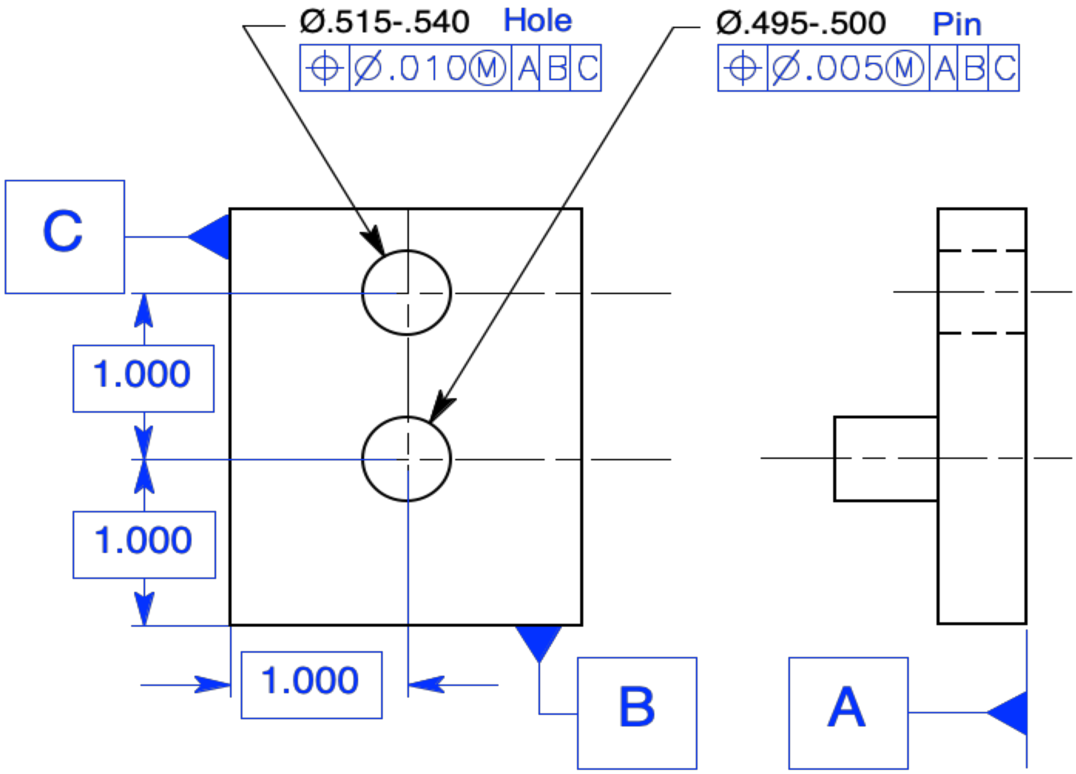


Figure 3-39 Refer to this drawing for questions 34 through 41

Internal Feature (Hole)

Actual Feature Size	MMC	Bonus	Geometric Tolerance	Total Positional Tolerance
MMC.515	*.515*	*.000*	*.010*	*.010*
.520	*.515*	*.005*	*.010*	*.015*
.525	*.515*	*.010*	*.010*	*.020*
.530	*.515*	*.015*	*.010*	*.025*
.535	*.515*	*.020*	*.010*	*.030*
LMC .540	*.515*	*.025*	*.010*	*.035*

Table 3-3 Total positional tolerance for holes

40. Using the drawing in Fig. 3-39, complete Table 3-3.

External Feature (Pin)

Actual Feature Size	MMC	Bonus	Geometric Tolerance	Total Positional Tolerance
MMC .500	*.500*	*.000*	*.005*	*.005*
.499	*.500*	*.001*	*.005*	*.006*
.498	*.500*	*.002*	*.005*	*.007*
.497	*.500*	*.003*	*.005*	*.008*
.496	*.500*	*.004*	*.005*	*.009*
LMC .495	*.500*	*.005*	*.005*	*.010*

Table 3-4 Total positional tolerance for pins

41. Using the drawing in Fig. 3-39, complete Table 3-4.
42. A ___*material condition modifier*___ is specified in a feature control frame when it is associated with the geometric tolerance of a feature of size or a datum feature of size.
43. The ___*Regardless of feature size*___ modifier indicates that the specified geometric tolerance applies at any increment of size of the actual mating envelope of the unrelated feature of size.
44. The ___*resultant condition*___ of a feature of size specified with an MMC modifier is the single worst-case boundary generated by the collective effects of the LMC limit of size, the specified geometric tolerance, and the size tolerance.
45. ___*True position*___ is the theoretically exact location of a feature of size, as established by basic dimensions.
46. ___*True profile*___ is the theoretically exact profile on a drawing defined by basic dimensions or a digital data file.
47. The___*Virtual condition*___of a feature of size specified with an MMC modifier is a constant boundary generated by the collective effects of the considered feature's MMC limit of size and the specified geometric tolerance.
48. For an individual regular feature of size, no element of the feature shall extend beyond the maximum material condition boundary (envelope) of perfect form.

    This statement is the essence of ___*Rule #1*___

49. The local form tolerance increases as the actual local size of the feature departs from MMC toward *LMC*

50. If features on a drawing are shown coaxial, or symmetrical to each other and not controlled for *location* or *orientation*, the drawing is incomplete.

51. If there is no orientation control specified for a rectangle on a drawing, the perpendicularity is controlled, not by the size *tolerance*, but by the *angularity* tolerance.

52. Rule #2 states that *RFS* automatically applies, to individual tolerances of feature of size and *RMB* automatically applies to datum features of size.

53. Each tolerance of orientation or position and datum reference specified for a screw thread applies to the axis of the *pitch diameter*.

54. Each geometric tolerance or datum reference specified for gears and splines must designate the specific feature at which each applies such as *MAJOR DIA, PITCH DIA, or MINOR DIA*

Problems

Page 55

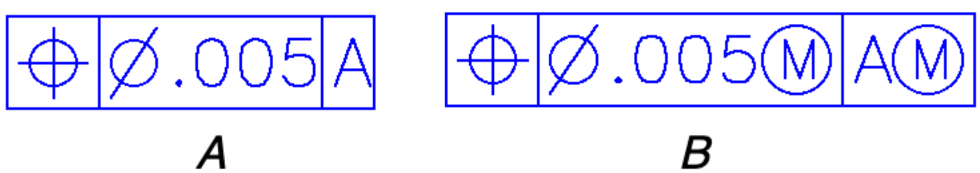


Figure 3-40 Feature control frames with material condition symbols: Prob. 1.

1. Read the complete tolerance in each feature control frame in Fig. 3-40, and write it below (Datum feature A is a feature of size).

A. *1. The* ***position*** *tolerance requires that*
2. The axis of the controlled feature
3. Must lie within a cylindrical tolerance zone
4. .005 in diameter
5. At RFS
6. Oriented and located with basic dimensions to datum feature A at RMB

B. *1. The* ***position*** *tolerance requires that*

2. The axis of the controlled feature

3. Must lie within a cylindrical tolerance zone

4. .005 in diameter

5. *At MMC*

6. *Oriented and located with basic dimensions to datum feature A at MMB*

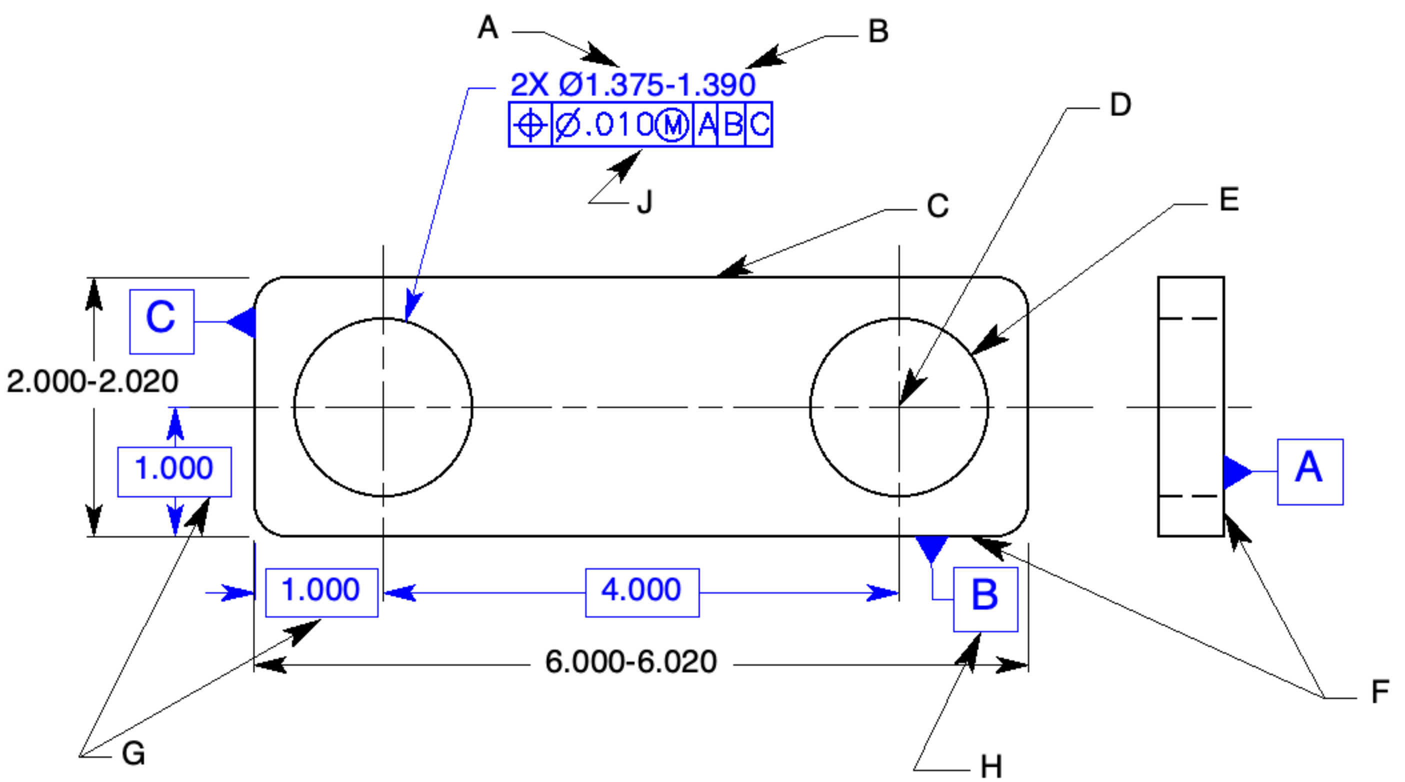


Figure 3-41 Geometric dimensioning and tolerancing terms: Prob. 2.

2. Place each letter of the items on the drawing in Fig. 3-41 next to the most correct term below.

C	Feature	*G*	Basic Dimension	*J*	Feature control frame
A	MMC	*F*	Datum Feature	*D*	True Position
B	LMC	*E*	Feature of Size	*H*	Datum feature symbol

Chapter 4

Datums

Chapter Review

Page 72

1. Datums are theoretically perfect *points, axes, lines, planes, or a combination thereof*.
2. Datums establish the *origin* from which the location or geometric characteristic of features of a part are established.
3. Datums are assumed to exist in and be simulated by datum *feature simulators.*
4. A datum reference frame consists of three mutually perpendicular *planes* and three mutually perpendicular *axes* at the intersection of those planes.
5. A part is oriented and immobilized relative to the three mutually perpendicular intersecting datum planes of the datum reference frame in a selected order of *precedence*.
6. The primary datum feature contacts the datum reference frame with a minimum of *three* points of contact that are not in a straight line.
7. Datum features are specified in order of precedence as they appear from left to right in the *feature control frame*.
8. Datum feature letters need not be in *alphabetical* order.
9. The primary datum feature controls *the orientation of the part*.
10. When selecting datum features, the designer should consider features that are: *Functional surfaces, mating surfaces, readily accessible surfaces, and surfaces that allow repeatable measurements*.
11. Datum features must be identified with *datum feature symbols* or *datum targets* and specified in a feature control frame.
12. Datum feature symbols must not be applied to *centerlines, center planes, or axes*.

13. One method of tolerancing datum features at an angle to the datum reference frame is to place a datum feature symbol on the *inclined surface* and control that surface with an angularity tolerance and a basic angle.

14. A cylindrical datum feature is always associated with two *theoretical planes* meeting at right angles at its datum axis.

15. The two kinds of features specified as datum features are:
Plane flat surfaces and features of size

16. Datum features of sizes may apply at *regardless of material boundary, maximum material boundary, or least material boundary*

17. Where datum features of sizes are specified at RMB, the processing equipment must make *physical contact* with the datum features.

18. Where features of sizes are specified at MMB, the size of the processing equipment has is equal to its *virtual condition with respect to the previous datum feature.*

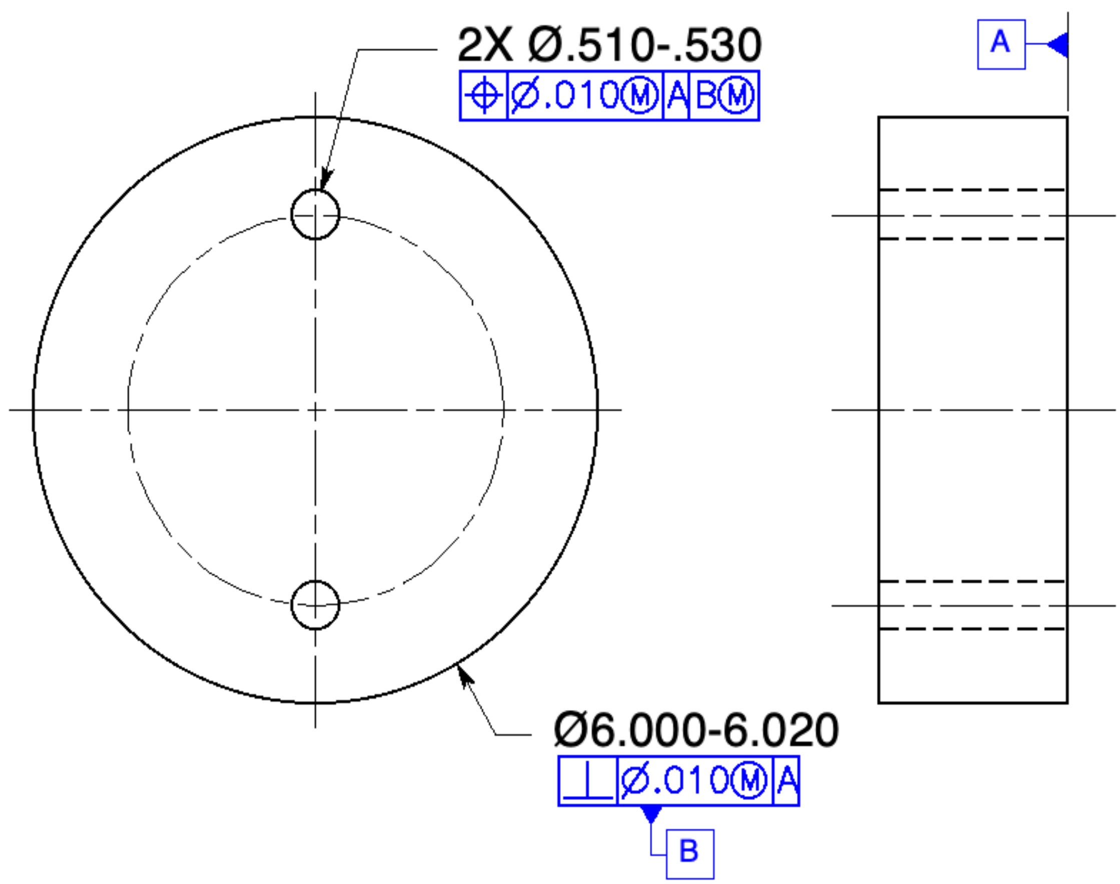


Figure 4-20 Datum feature of size: Questions 19 through 24.

(Refer to Fig. 4-20 to answer questions 19 through 24.)

19. The 2-hole pattern is perpendicular to what datum feature? *Datum feature A*

20, The 2-hole pattern is located to what datum feature? *Datum feature B*

21. If inspected with a gage, what is the diameter of datum feature B on the gage? *Ø 6.030*

22. If inspected with a gage, what is the diameter of the 2 pins on the gage? *Ø.500*

23. If datum feature B had been specified at RFS, explain how the gage would be different.

    Datum feature B would have to be a variable diameter like a chuck to make physical contact with the outside diameter.

24. If datum feature B had been specified as the primary datum at RFS, explain how the gage would be different.

    Datum feature B would not only have to be a variable diameter, such as a chuck, to make physical contact with the outside diameter, but the outside diameter of the part, datum feature B, must align with the gage as well.

25. If a datum feature symbol is in line with a dimension line, the datum feature is the *feature of size* measured by the dimension.

26. Where more than one datum feature is used to establish a single datum, the *datum reference letters* and appropriate *modifiers* are separated by a dash and specified in one compartment of the feature control frame.

27. Where cylinders are specified as datum features, the entire surface of the feature is considered to be the *datum feature*.

28. If only a part of a feature is required to be the datum feature, a *heavy chain line* is drawn adjacent to the surface profile and dimensioned with basic dimensions.

29. Datum targets may be used to immobilize parts with *uneven or irregular surfaces*

30. Costly manufacturing and inspection *tooling* is required to process a part with datum targets.

Problems

Page 76

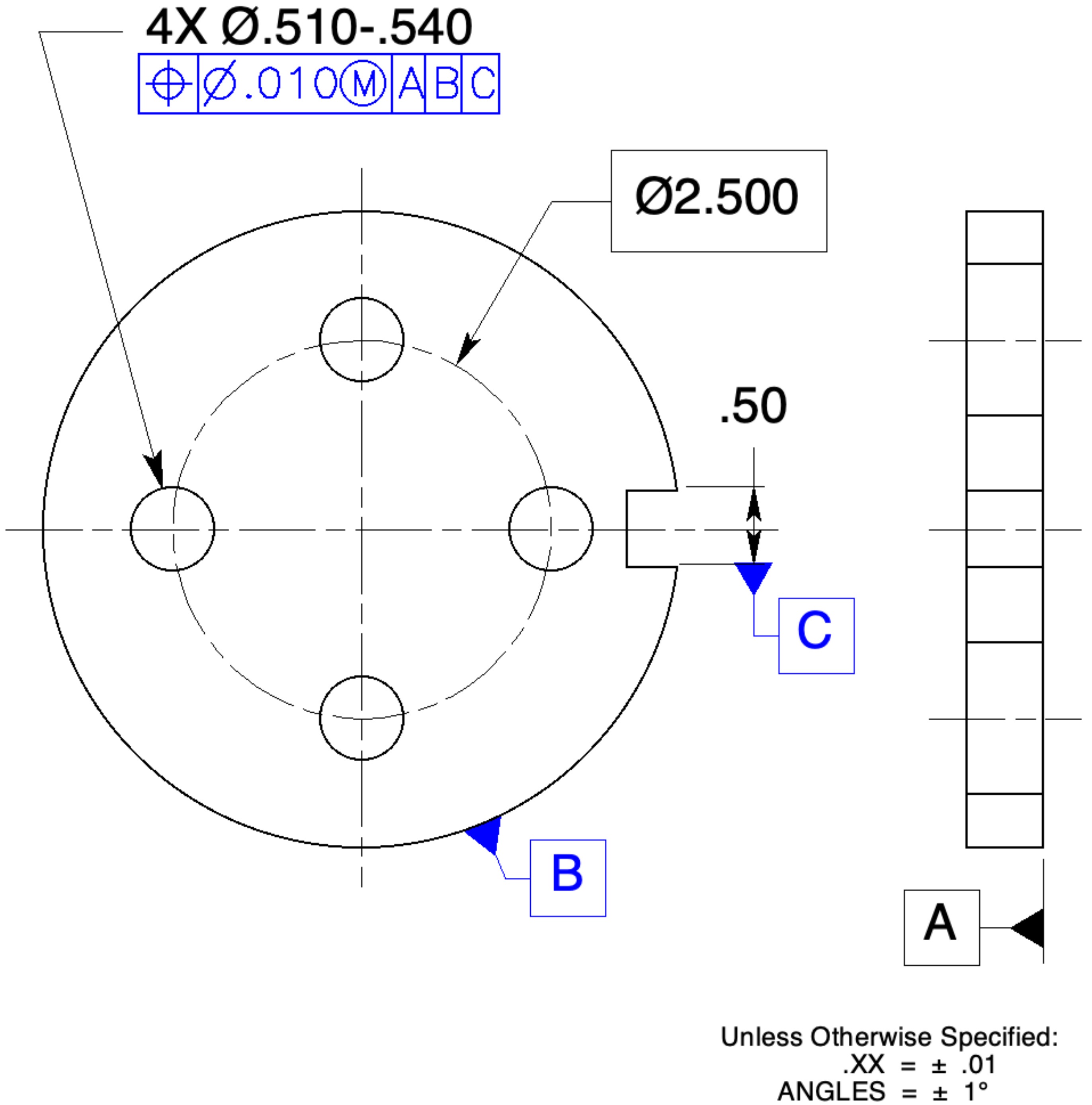


Figure 4-21 Placement of datum feature symbols: Prob. 1.

1. Attach the appropriate datum feature symbols on the drawing in Fig. 4-21.

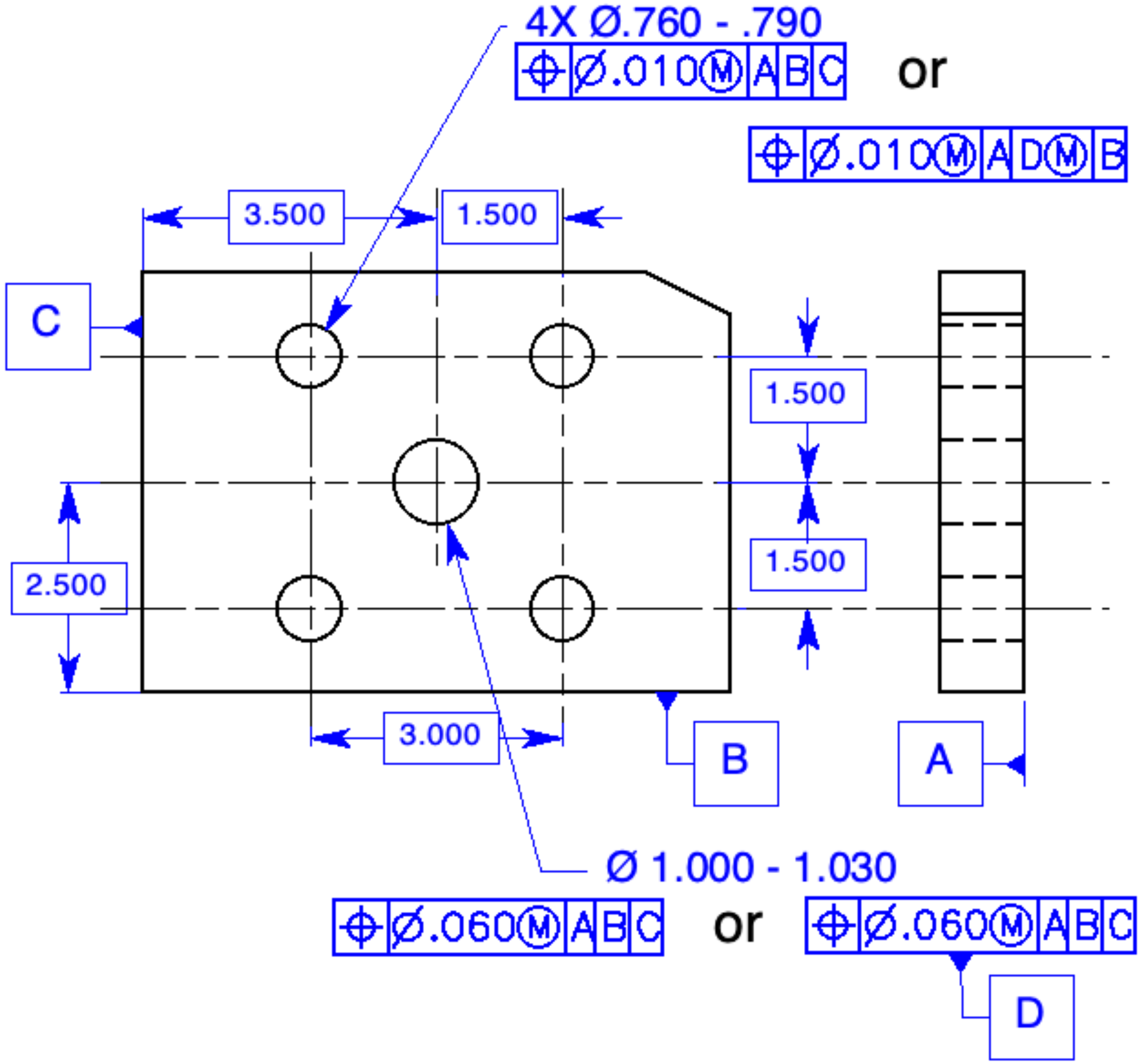


Figure 4-22 Placement of datum references in a feature control frames: Prob. 2.

2. Provide the appropriate datum feature symbols and complete the feature control frames on the drawing in Fig. 4-22.

(Two solutions suggested.)

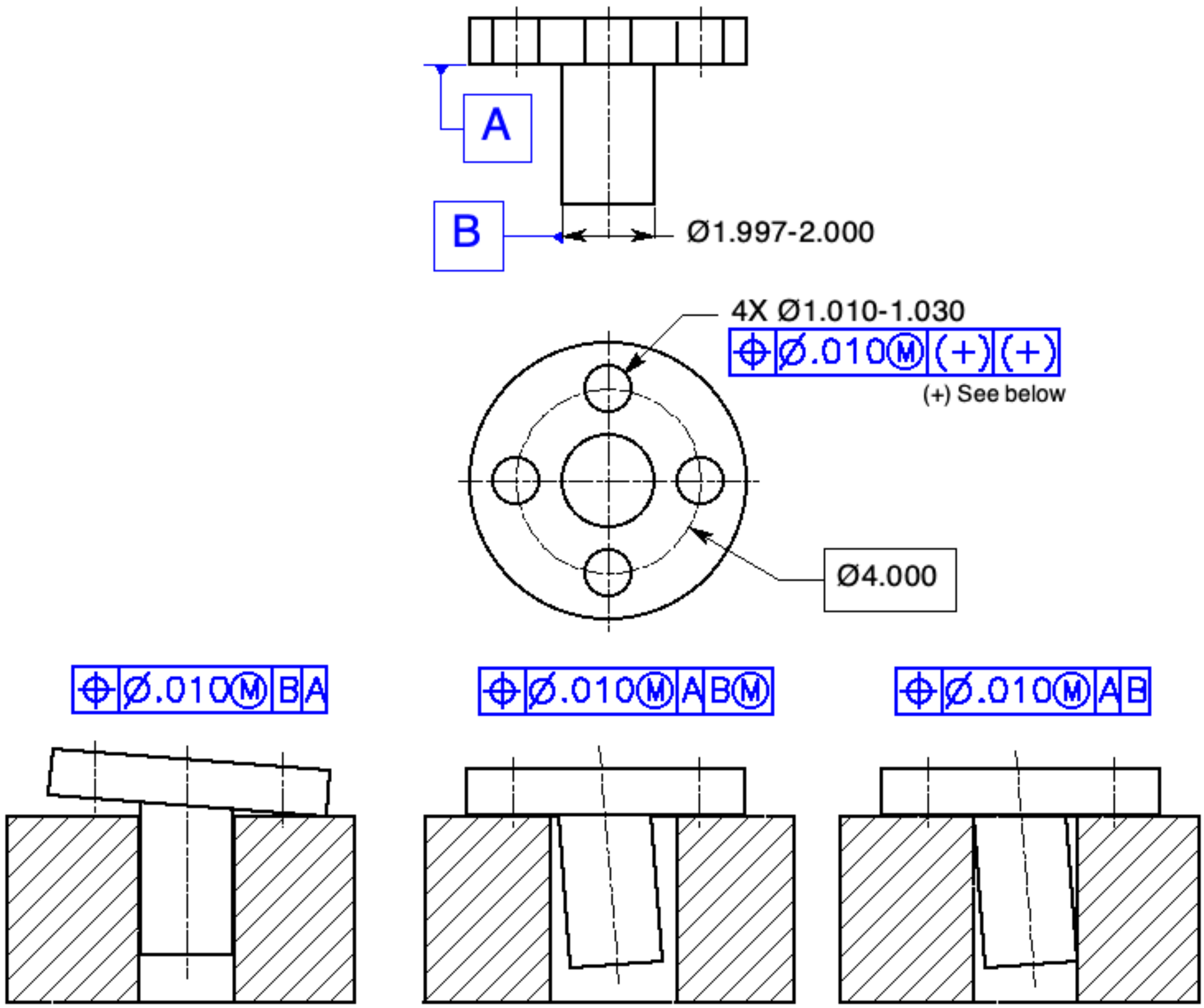


Figure 4-23 Datum features of size at MMB and RMB: Prob. 3.

3. Complete the feature control frames with datum references and material condition modifiers to reflect the drawings in Fig. 4-23.

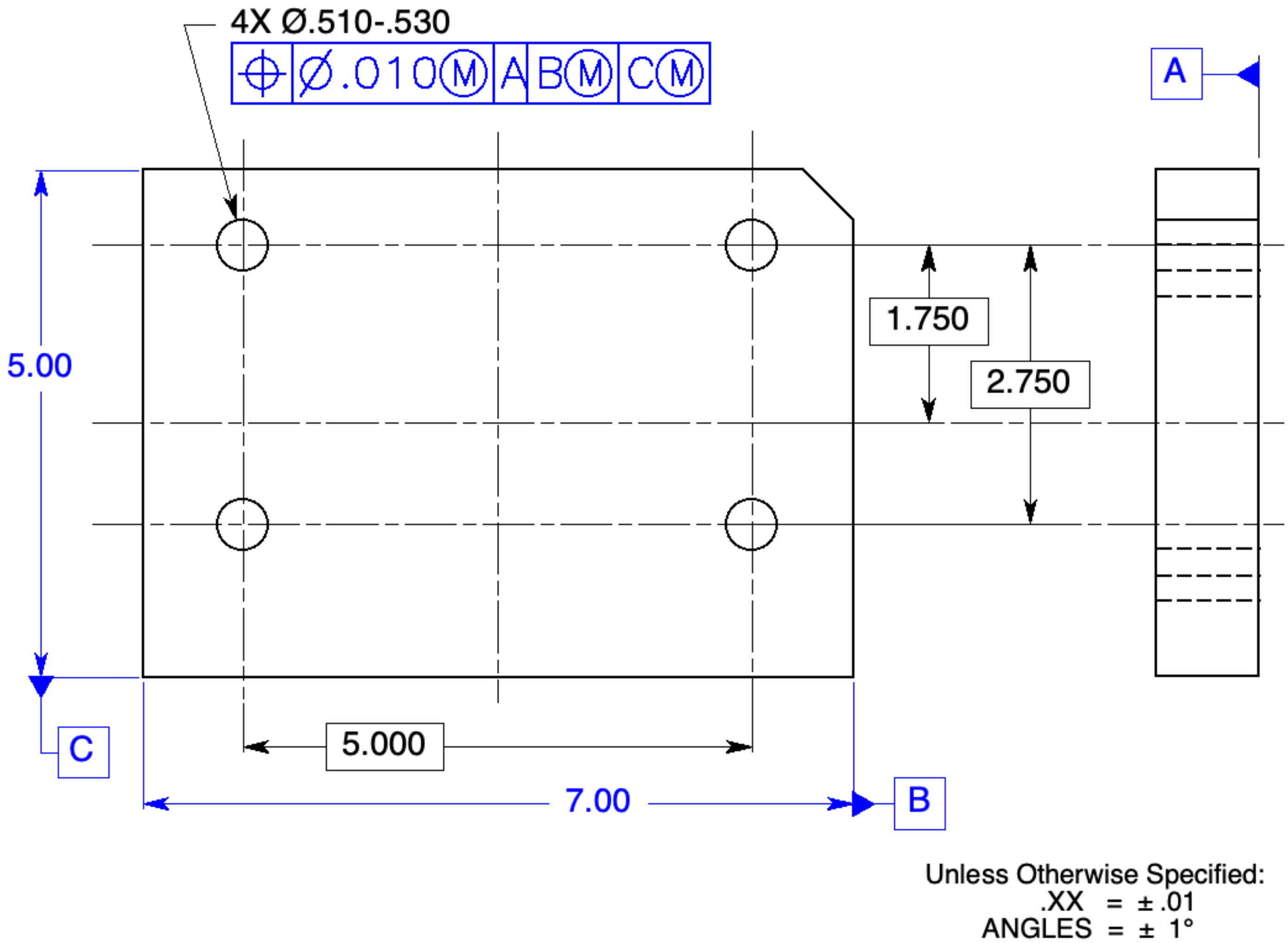


Figure 4-24 Datum features located to the center planes of the drawing: Prob. 4.

4. Specify the appropriate datum feature symbols to locate the four-hole pattern to the center planes of the drawing in Fig 4-24.

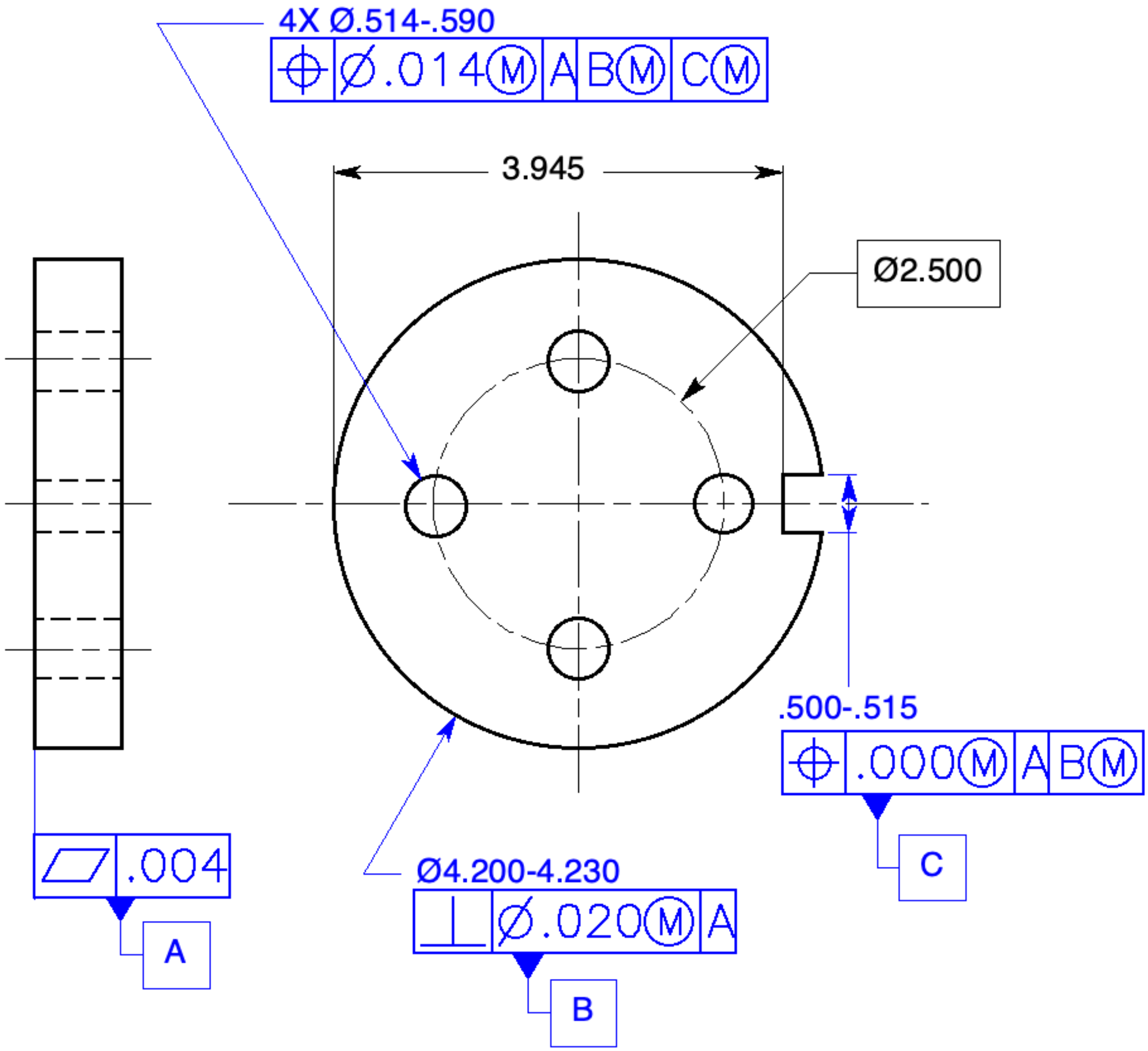


Figure 4-25 Specifying datum references and datum feature symbols: Probs. 5 and 6.

5. Specify the appropriate datum feature symbols and complete the feature control frames in the datum exercise in Fig. 4-25.

(One solution. Explore other possibilities.)

Chapter 5

Form

Chapter Review

Page 93

1. Form tolerances are independent of all other *features*.
2. No *datums* apply to form tolerances.
3. The form of individual features of size is automatically controlled by the *size tolerance, Rule #1*.
4. Where the size tolerance does not sufficiently control the form of a feature, a form tolerance may be specified as a *refinement*.
5. All form tolerances are surface controls except for *flatness of a median plane and straightness of a median line*.
6. No *cylindrical tolerance zones* or *material condition symbols* are appropriate for surface controls.
7. Flatness of a surface or derived median plane is a condition where all line elements of that surface are in one *plane*.
8. In a view where the surface to be controlled with a flatness tolerance appears as a *line*, a feature control frame is attached to the surface with a *leader or extension line*.
9. The feature control frame controlling flatness contains a *flatness symbol* and a *numerical tolerance*.
10. The surface being controlled for flatness must lie between *two parallel planes* separated by the flatness tolerance. In addition, the feature must fall within the *size tolerance*.
11. The flatness tolerance zone does not need to be *parallel* to any other surface.

12. The feature of size may not exceed the *boundary of perfect form at MMC*

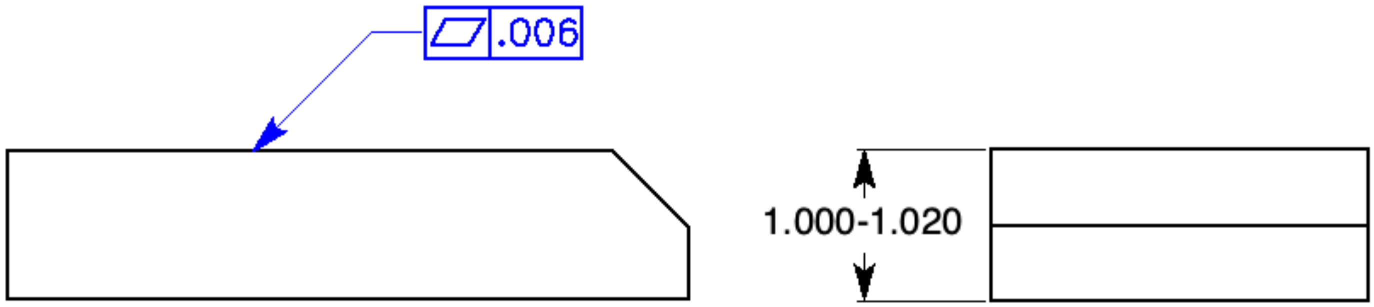
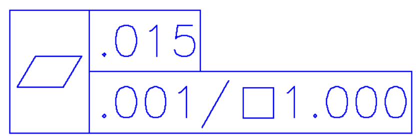


Figure 5-15 Specifying flatness: Question 13.

13. Specify a flatness tolerance of .006 in a feature control frame to the top surface of the part in Fig. 5-15.

14. Draw a feature control frame below with an overall flatness of .015 and a unit flatness of .001 per square inch.



15. When verifying flatness, the feature of size is first measured to verify that it falls within the

size limits.

16. The surface is adjusted with jackscrews to remove any *parallelism* error.

17. Flatness verification is achieved by measuring the surface in all directions with a

dial indicator

18. Straightness is a condition where *an element of a surface, or a derived median line,*

is a straight line.

19. In a view where the line elements to be controlled appear as a *line*,

a feature control frame is attached to the surface with a *leader or an extension line*

20. Straightness tolerance is a refinement of the *size tolerance, Rule #1*,

and must be less than the *size tolerance*.

Actual Part Size	Straightness Tolerance	Controlled By
1.020	*.000*	*Rule #1*
1.018	*.002*	
1.016	*.004*	
1.014	*.004*	*Straightness Tolerance*
1.010	*.004*	
1.005	*.004*	
1.000	*.004*	

Table 5-6 Review question 21

21. Complete Table 5-6 specifying the straightness tolerance and what controls it for the drawing in Fig. 5-6.
22. The measurement of surface variation for straightness is performed similar to the measurement for *flatness*.
23. Each line element is *independent* of every other line element.
24. When a feature control frame with a straightness tolerance is associated with a size dimension, the straightness tolerance applies to *the median line*.
25. While each actual local size must fall within the *size tolerance*, the feature controlled with straightness of a median line may exceed the *boundary of perfect form* at maximum material condition.
26. A straightness control of a median line will allow the feature to violate *Rule #1*.
27. If specified at MMC, the total straightness tolerance of a median line equals the tolerance in the feature control frame plus any *bonus tolerance*.

Feature Size	Cylindrical Feature (Straightness of a Median Line)	
	— ⌀.006	— ⌀.006 Ⓜ
1.020 MMC	*.006*	*.006*
1.015	*.006*	*.011*
1.010	*.006*	*.016*
1.005	*.006*	*.021*
1.000 LMC	*.006*	*.026*

Table 5-7 Straightness Tolerance: Question 28.

28. Complete Table 5-7 specifying the appropriate tolerances for the sizes given.

29. Straightness verification of a feature of size specified at MMC can be achieved by *placing the part in a full form functional gage*.

30. Straightness verification of a feature of size specified at *RFS* cannot be achieved by placing the part in a full form functional gage.

31. Circular elements must lie between two, *concentric circles* in which the *radial distance* between them is equal to the tolerance specified in the circularity feature control frame.

32. When verifying circularity, the feature of size is first measured to verify that it falls within the *limits of size* and *Rule #1*.

33. Circularity can be accurately inspected on a *circularity inspection machine*.

34. Cylindricity is a condition of the surface of a cylinder where all points of the surface are *equidistant from the axis*.

35. The cylindricity tolerance consists of two *coaxial cylinders* in which the *radial distance* between them is equal to the tolerance specified in the *feature control frame*.

36. Cylindricity is a *composite* form tolerance that simultaneously controls *circularity, straightness of a surface, and taper* of cylindrical features.

	⏥	—	Feature of Size ⏥	Feature of Size —	○	⌭
1. For these controls, datums do not apply	X	X	X	X	X	X
2. For these controls, rule #1 applies	X	X			X	X
3. These are surface controls	X	X			X	X
4. These controls may be specified with a leader	X	X			X	X
5. These are refinements of the size tolerance	X	X			X	X
6. These tolerances violate rule #1			X	X		
7. These controls apply to features of size			X	X		
8. These controls are associated with the dimension			X	X		
9. These controls may exceed the size tolerance			X	X		
10. The Ø symbol may be used				X		
11. The MMC (circle M) symbol may be used			X	X		

Table 5-8 The Application of Form Controls: Question 37

37. In Table 5-8, place an X under the control that agrees with the statement.

38. Except for restrained flexible parts, all dimensions and tolerances apply in a *free state condition*.

39. A minimum of *four measurements* must be taken to insure the accuracy of an average diameter.

40. Restraint may be applied to flexible parts dude to the distortion of a part after the removal of forces applied during the *manufacturing process*.

41. The restrained condition should simulate *actual assembly conditions*.

Problems

Page 98

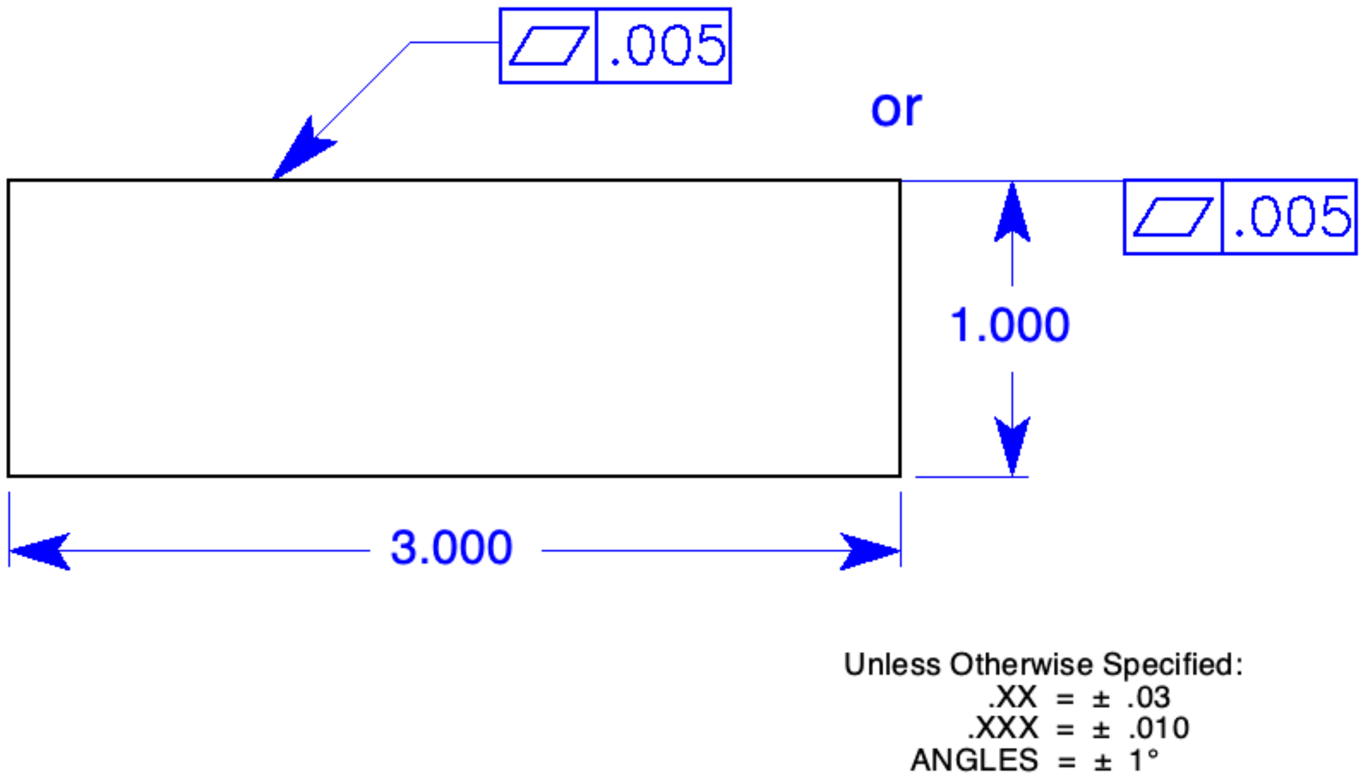
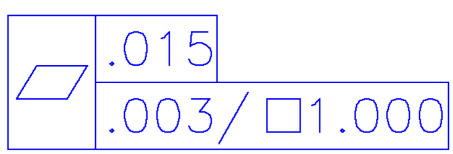


Figure 5-16 Flatness: Prob. 1.

1. Specify a flatness control of .005 for the top surface of the part in Fig. 5-16.

(Either a leader or an extension line can be used)

2. Below, specify a feature control frame with a unit flatness of .003 per square inch and an overall flatness of .015.



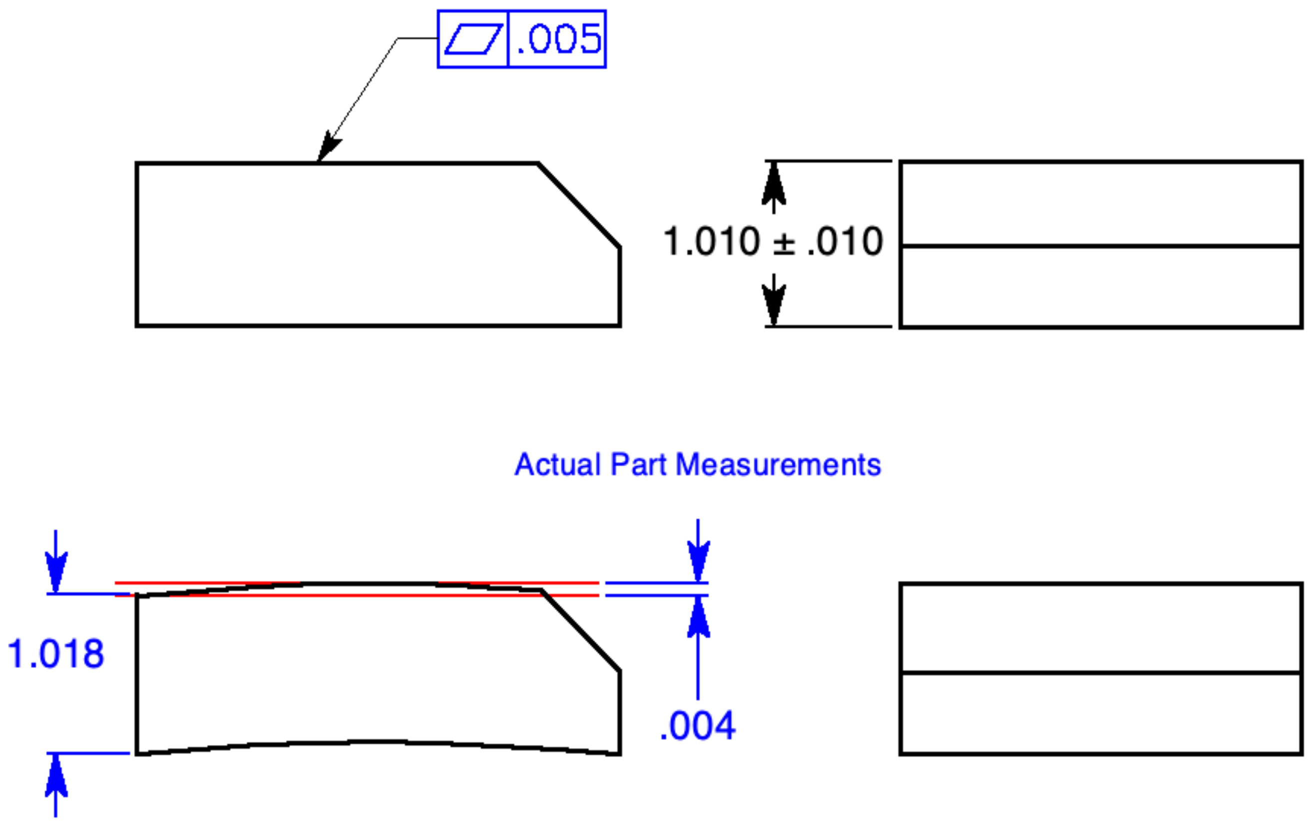


Figure 5-17 Flatness check: Prob. 3.

3. Is the part in Fig. 5-17 an acceptable part? Why or why not?

No, this part is not acceptable. It is 1.018 thick and bowed .004, a total of 1.022. The part exceeds the 1.020 boundary of perfect form at MMC.

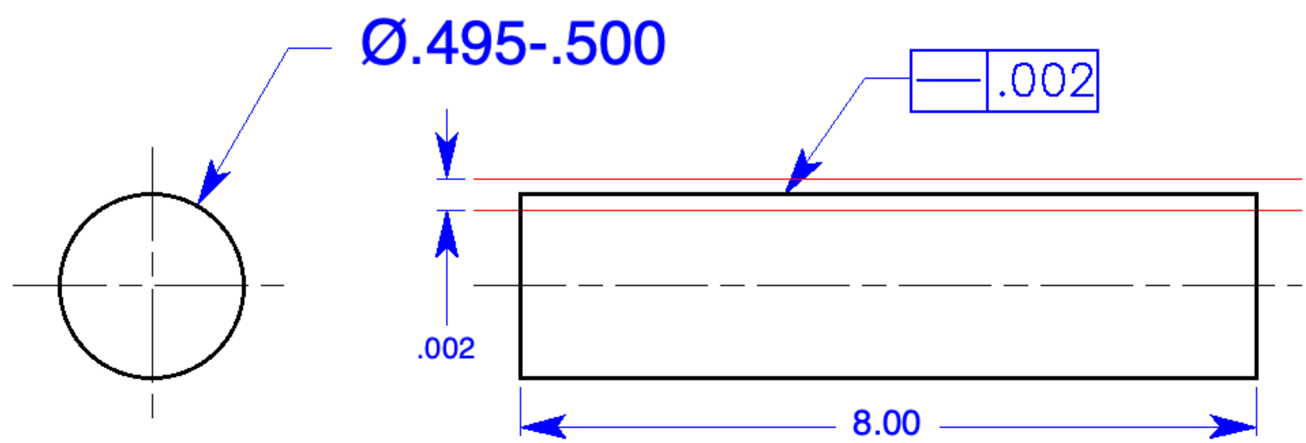


Figure 5-18 Straightness of a surface: Prob. 4.

4. Specify straightness of a surface of .002 on the cylinder in Fig. 5-18. Draw and dimension the tolerance zone on the drawing

(Either a leader or an extension line can be used)

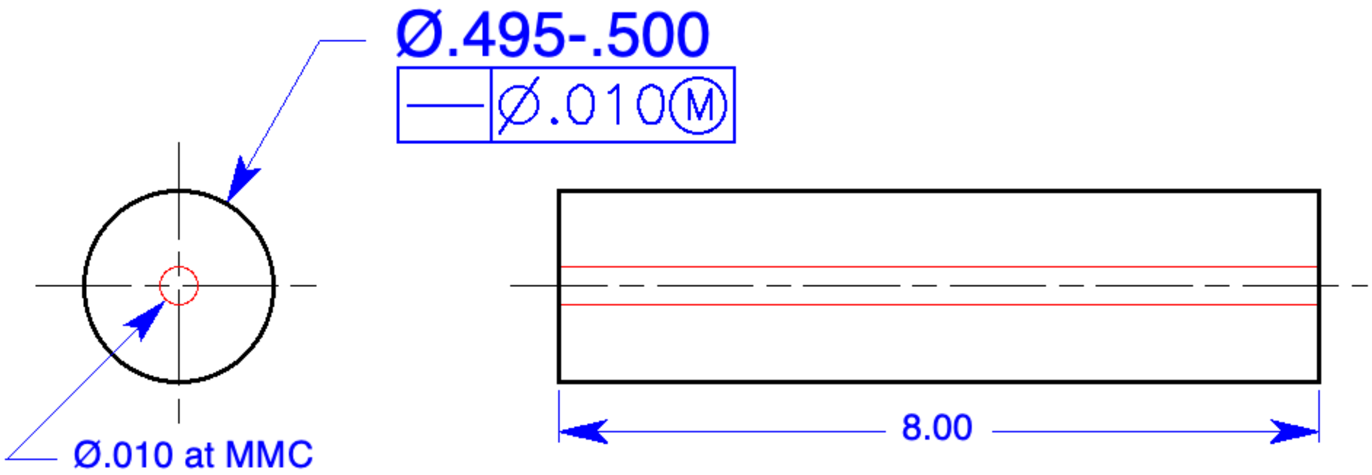


Figure 5-19 Straightness of a median line: Prob. 5.

4. On the cylinder in Fig. 5-19, specify straightness of a median line of .010 at MMC. Draw and dimension the tolerance zone on the drawing.

(The feature control frame must be associated with the dimension.)

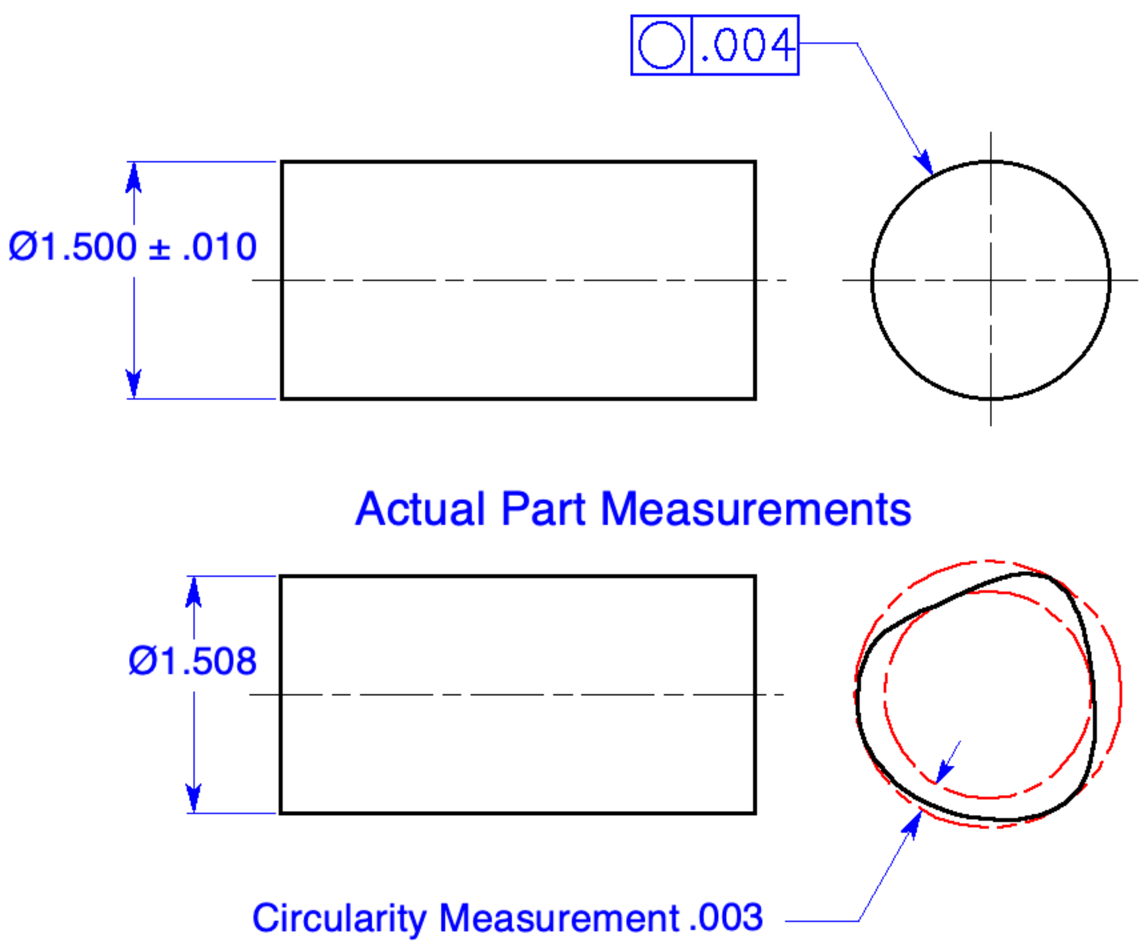


Figure 5-20 Circularity: Prob. 6.

6. Is the part in Fig. 5-19 an acceptable part – why or why not?

Undetermined, further inspection is necessary to reject this part. If the outer diameter of the circularity tolerance zone is within 1.510, the MMC of the cylinder, it is a good part. The cylinder may not exceed the boundary of perfect form at MMC.

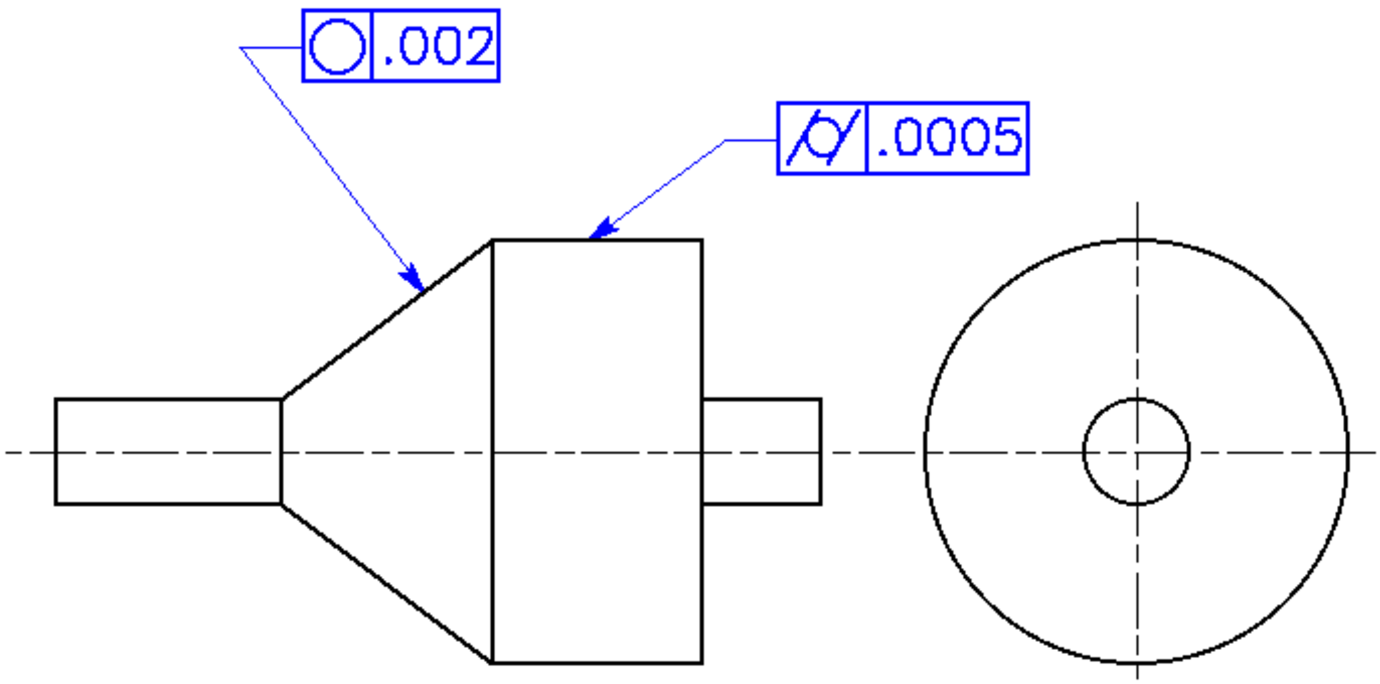


Figure 5-21 Circularity and cylindricity: Probs. 7 and 8.

7. Specify a circularity tolerance of .002 on the cone in the drawing in Fig. 5-21.

8. Specify a cylindricity tolerance of .0005 on the large cylinder on the drawing in Fig. 5-21.

Chapter 6

Orientation

Chapter Review

Page 110

1. Orientation is the general term used to describe the ___*angular*___ relationship between features.
2. Orientation controls include ___*parallelism, perpendicularity, angularity, and in some cases, profile*___
3. All orientation controls must have ___*datum features*___.
4. In a view where the surface to be controlled appears as a line, the perpendicularity feature control frame is attached to the surface with a ___*leader or extension line*___.
5. The datum feature is identified with a ___*datum feature symbol*___.
6. A surface being controlled with a perpendicularity tolerance must lie between ___*two parallel planes*___ separated by the perpendicularity tolerance specified in the feature control frame. The tolerance zone must also be ___*perpendicular*___ to the datum plane.
7. A tangent plane symbol, circle T, in the feature control frame specifies that the tolerance applies to a precision plane contacting the ___*high points*___ of the surface.
8. Where controlling the perpendicularity of a feature of size, the feature control frame is associated with the ___*size dimension*___ of the feature being controlled.
9. If the tolerance in the feature control frame applies to a feature of size and no material condition symbol is specified, ___*RFS*___ automatically applies.
10. If the tolerance applies at MMC then a possible ___*bonus*___ tolerance exists.

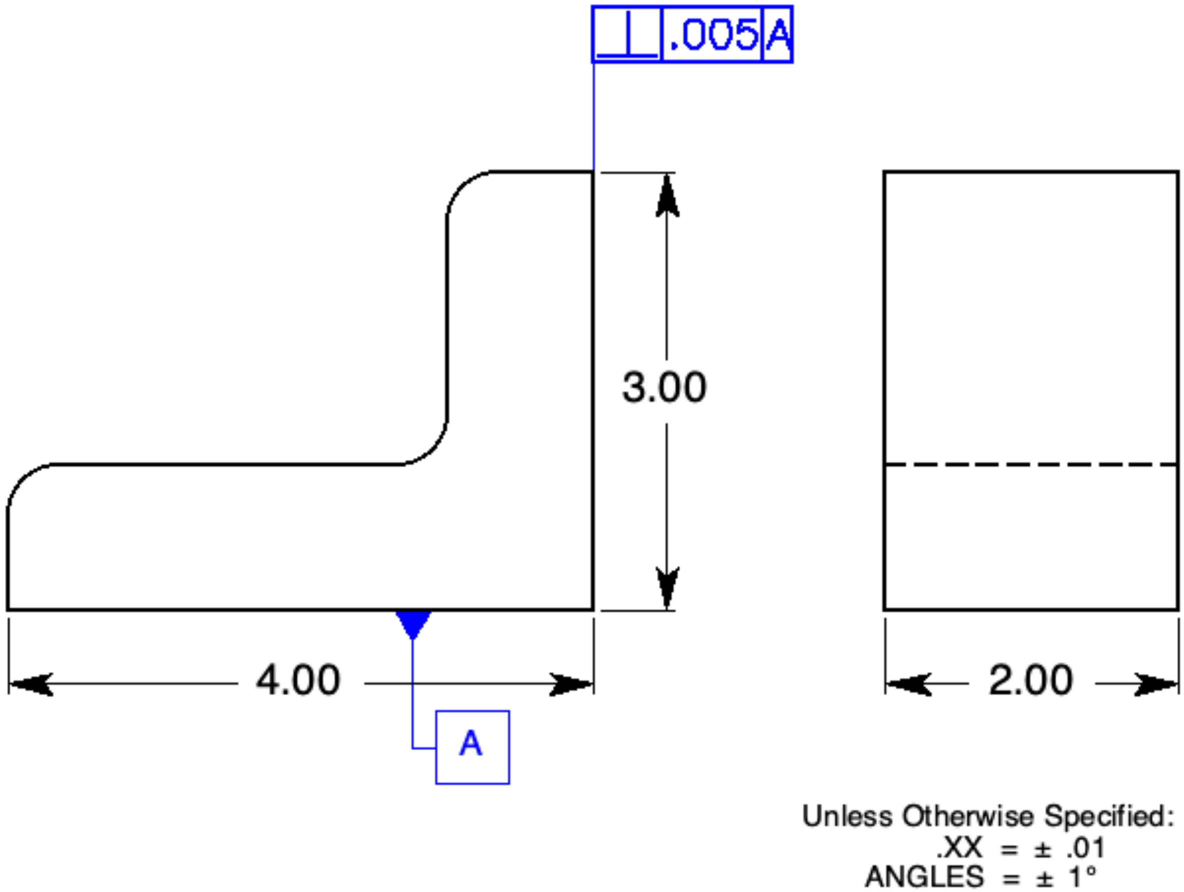


Figure 6-14 Specifying perpendicularity of a plane surface: Question 11.

11. Supply the appropriate geometric tolerance on the drawing in Fig. 6-14 to control the 3.00-inch vertical surface of the part perpendicular to the bottom surface within .005.

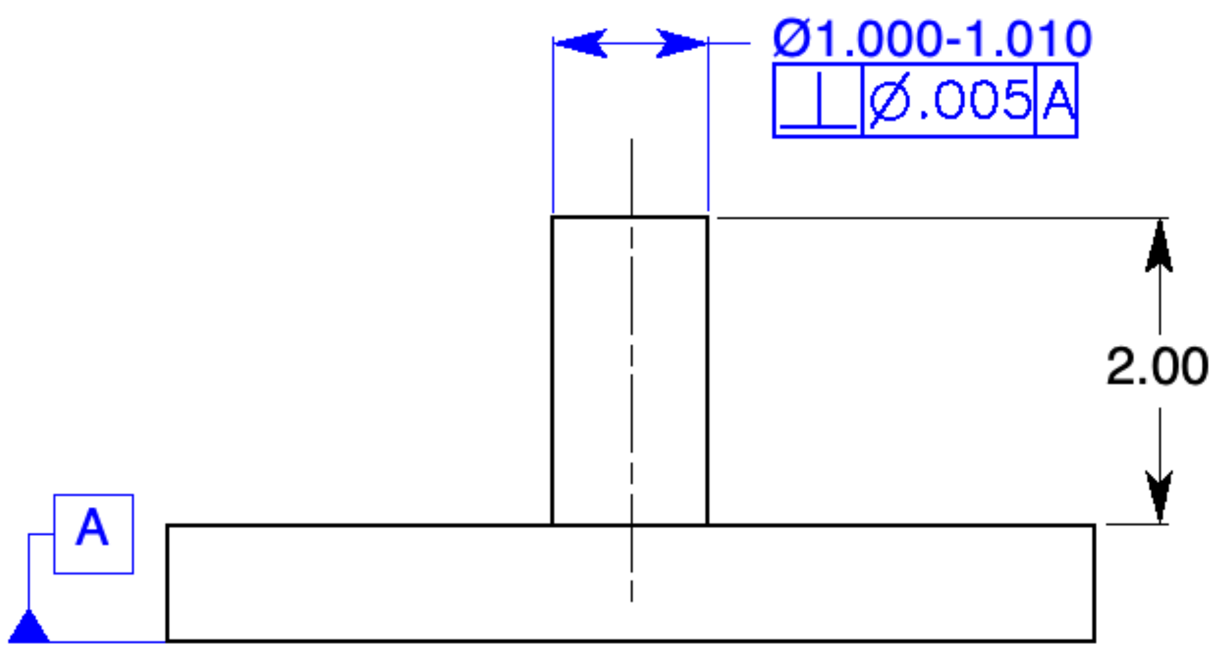


Figure 6-15 Specifying perpendicularity of a feature of size: Question 12.

12. Supply the appropriate geometric tolerance on the drawing in Fig. 6-15 to control the 1.00-inch diameter vertical pin perpendicular to the bottom surface of the plate within .005 at RFS.

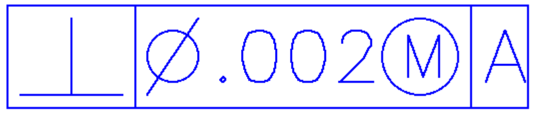


Figure 6-16 Perpendicularity specified at MMC: Question 13.

13. If the pin in Fig. 6-15 were produced at a diameter of 1.004 and toleranced with the feature control frame in Fig. 6-16, what would the total perpendicularity tolerance be? *.008*

14. The feature control frame contains a parallelism symbol, a numerical tolerance, and at least one *datum feature*.

15. Parallelism tolerance of a flat surface is a refinement of the size tolerance and must be less than the *size tolerance*.

16. A surface being controlled with a parallelism tolerance must lie between *two parallel planes* separated by the parallelism tolerance specified in the feature control frame. The tolerance zone must also be *parallel* to the datum feature.

17. The controlled surface may not exceed the *boundary of perfect form at MMC*

18. Where applied to a flat surface, parallelism is the only orientation control that requires perfect orientation (Parallelism is a 0° angle.) at *MMC*.

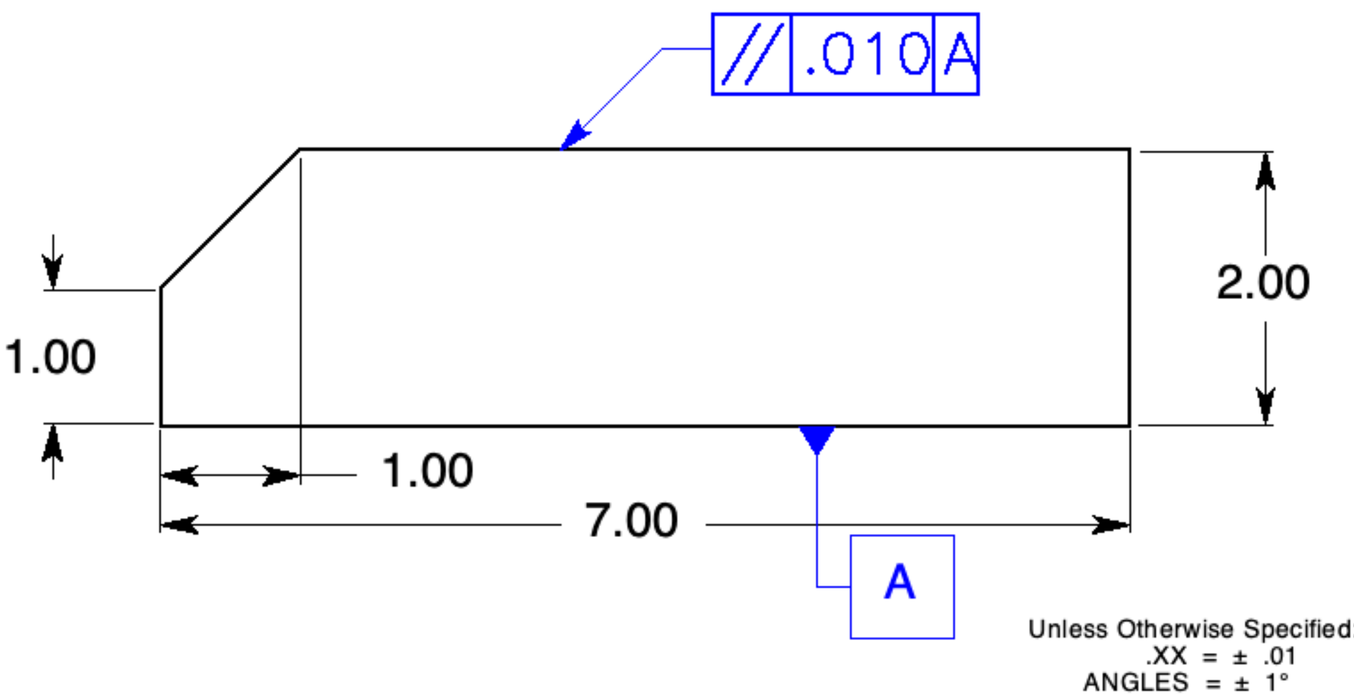


Figure 6-17 Specifying parallelism: Question 19.

19. Supply the appropriate geometric tolerance on the drawing to control the top surface of the part in Fig. 6-17 parallel to the bottom surface within .010.

(Either a leader or an extension line can be used)

20. Where controlling the parallelism of a feature of size, the feature control frame is associated with the *size dimension* of the feature being controlled.

21. If the element being controlled is an axis, the numerical tolerance is usually preceded by a diameter symbol.

22. The numerical tolerance for angularity of a surface is specified as a linear dimension because it generates a *uniform* shaped tolerance zone.

23. A plus or minus angularity tolerance is not used because it generates a *nonuniform, fan* shaped tolerance zone.

24. Where controlling the angularity of a feature of size, the feature control frame is associated with the *size dimension* of the feature being controlled.

25. If the element being controlled is an axis, the numerical tolerance is usually preceded by a *diameter* symbol.

26. The angularity control is a refinement of the orientation of an axis at a basic angle to a *datum feature*.

	Plane Surfaces			**Axes & Ctr. Planes**		
	⊥	//	∠	⊥	//	∠
Datum features are required	**X**	**X**	**X**	**X**	**X**	**X**
Controls flatness if flatness is not specified	**X**	**X**	**X**			
Circle T modifier can apply	**X**	**X**	**X**			
Tolerance specified with a leader or extension line	**X**	**X**	**X**			
Tolerance associated with a dimension				**X**	**X**	**X**
Material condition modifiers apply				**X**	**X**	**X**
A virtual condition applies				**X**	**X**	**X**

Table 6-2 The Application of Orientation Controls: Question 27

27. In Table 6-2, mark an **X** in the box that indicates the control applies to the statement at the left.

Problems

Page 114

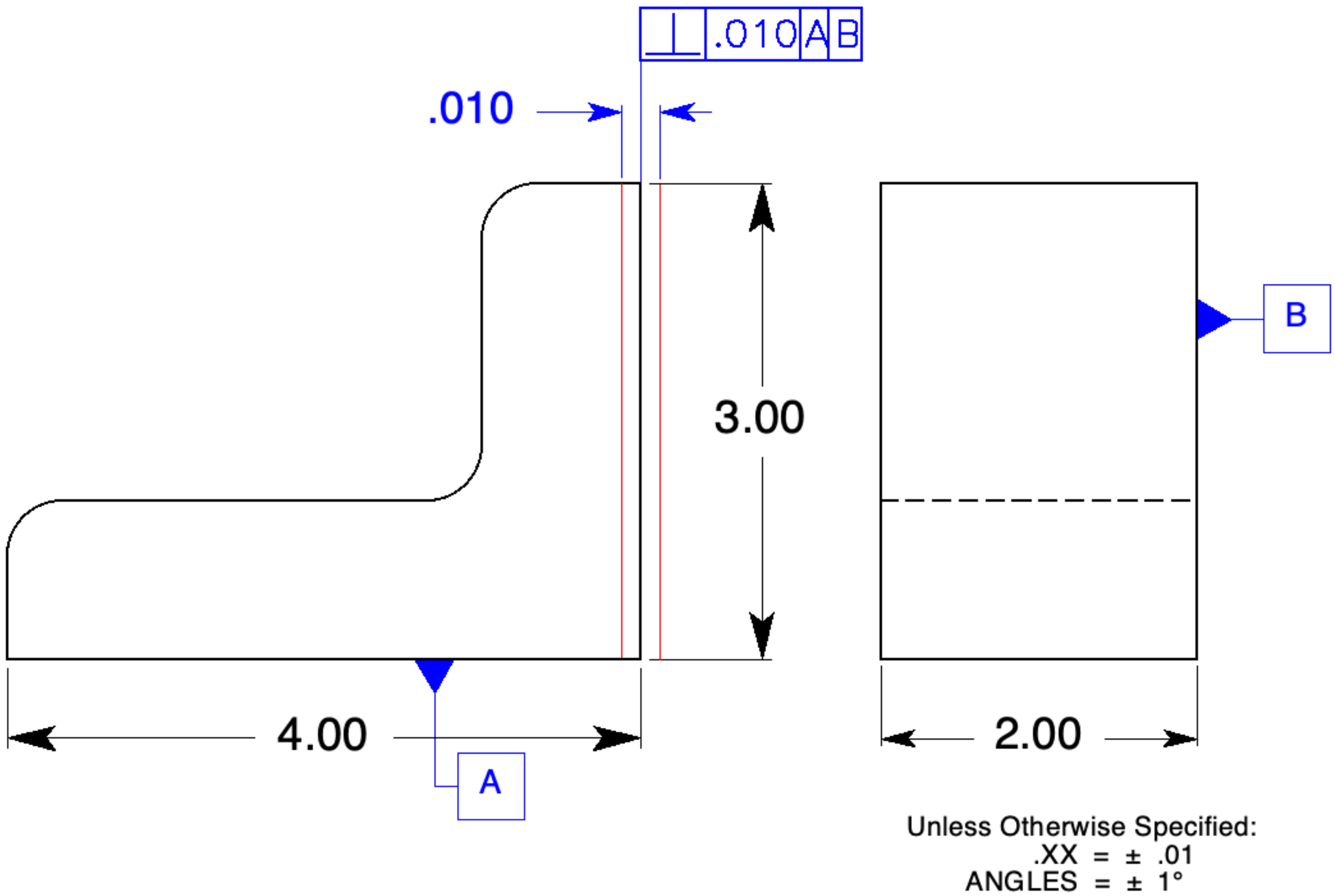
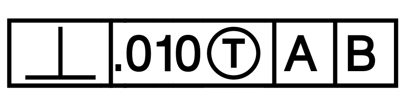


Figure 6-18 Perpendicularity of a plane surface: Probs.1 and 2.

1. Specify the 3.00-inch surface of the part in Fig. 6-18 to be perpendicular to the bottom and back surfaces within a tolerance of .010. Draw and dimension the tolerance zone.

2. Specify a feature control frame that would require a precision plane surface placed against the 3.00-inch surface of the part in Fig. 6-18 to be perpendicular to the bottom and back surfaces within a tolerance of .010.



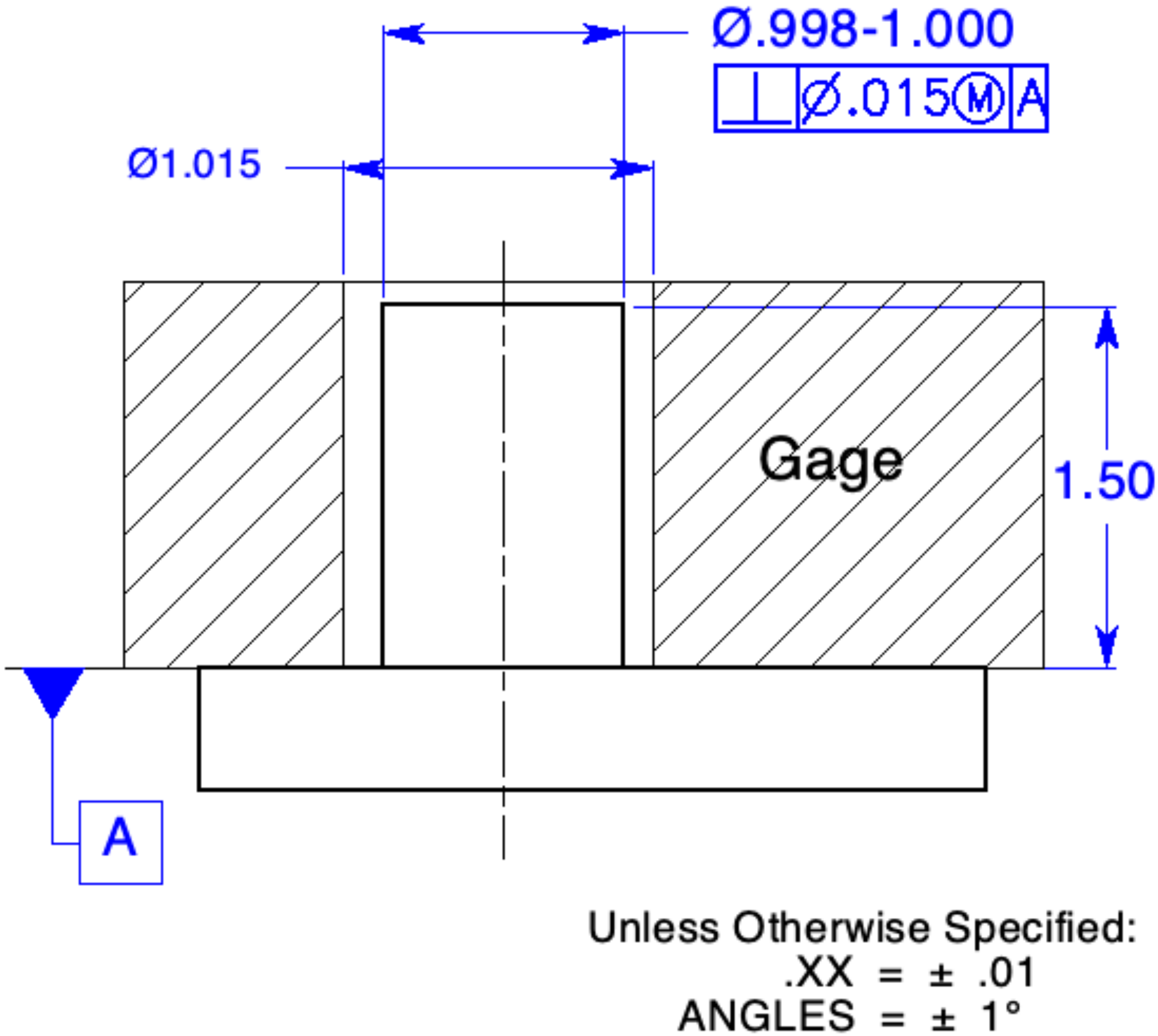


Figure 6-19 Perpendicularity of a pin to a plane surface: Prob. 3.

3. Specify the one-inch diameter pin perpendicular to the top surface of the horizontal plate in Fig. 6-19 within a tolerance of .015 at MMC. On the drawing, sketch and dimension a gage used to inspect this part.

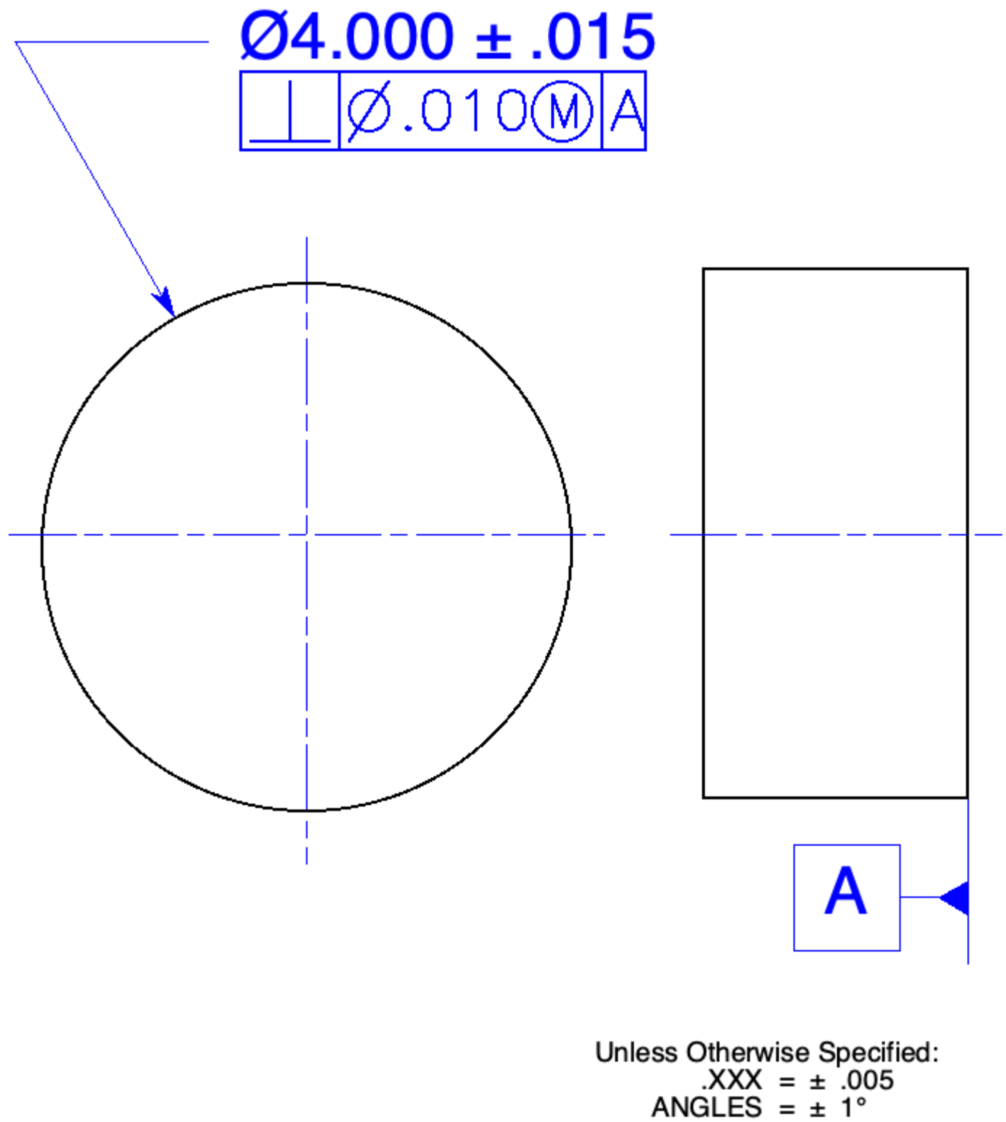


Figure 6-20 Perpendicularity of a cylinder to a plane surface: Prob. 4.

4. Specify the Ø4.000 cylinder perpendicular to its back surface, datum feature A, within a tolerance of .010 at MMC in Fig. 6-20.

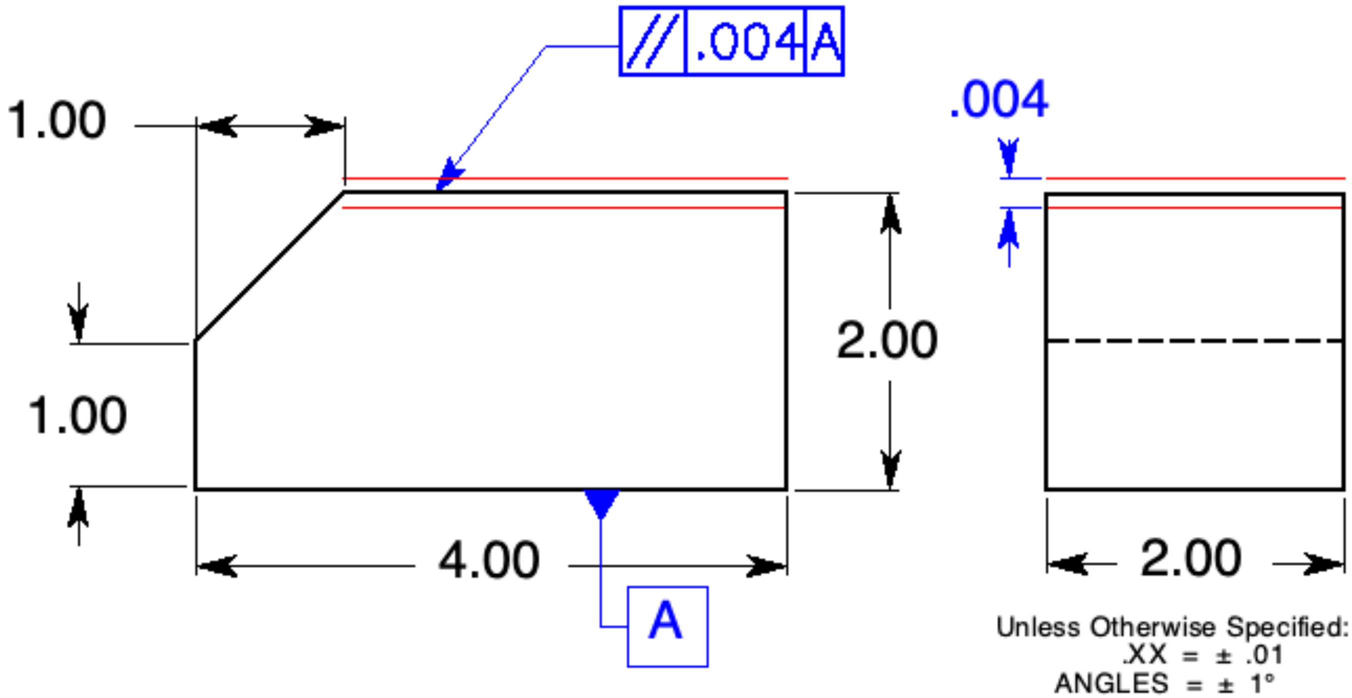


Figure 6-21 Parallelism of a plane surface: Prob. 5.

5. Specify the top surface of the part in Fig. 6-21 parallel to the bottom surface within a tolerance of .004. Draw and dimension the tolerance zone.

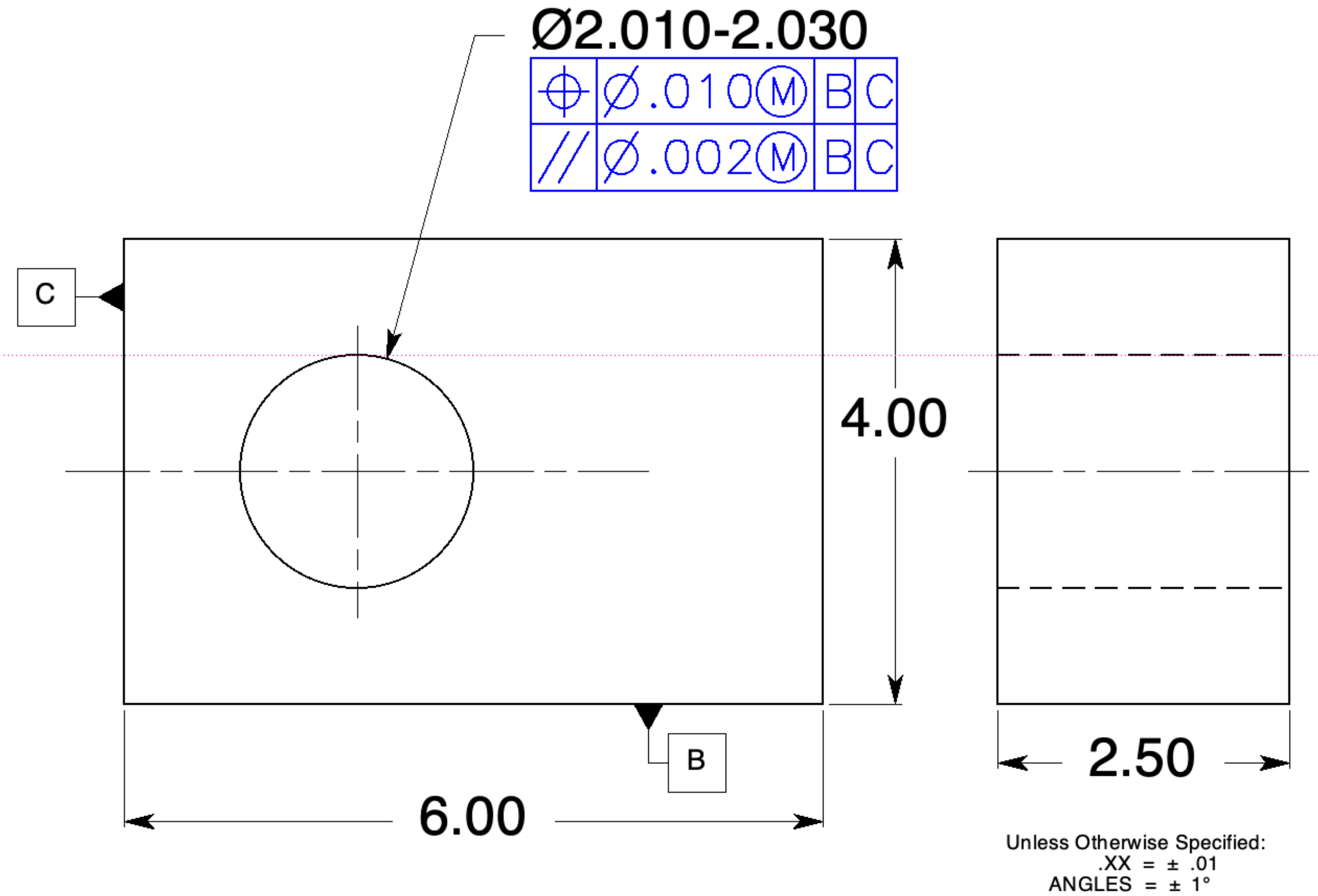


Figure 6-22 Parallelism of a cylindrical feature of size: Prob. 6.

6. Specify the hole in the part in Fig. 6-22 parallel to datum features B and C within a tolerance of .002 at MMC.

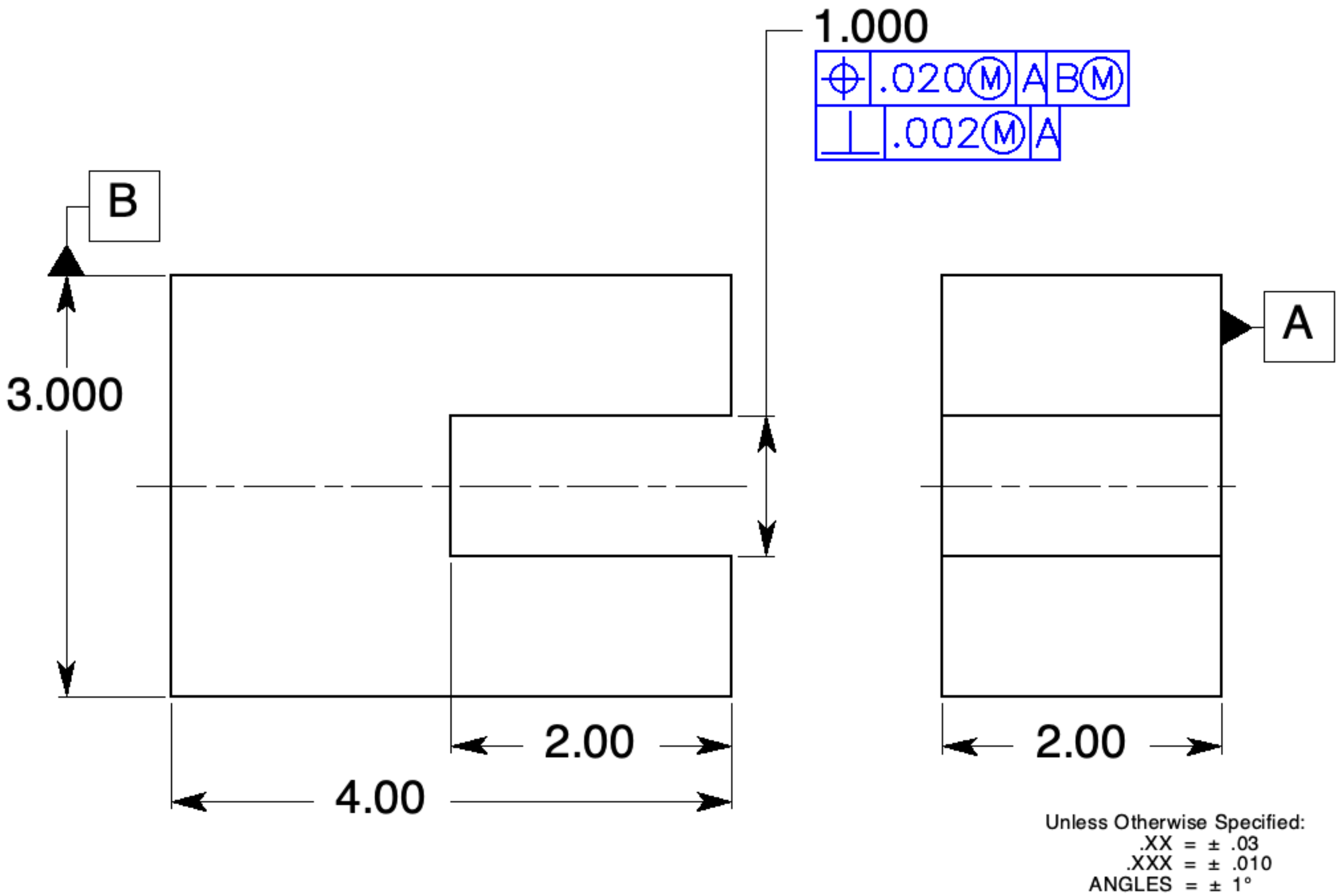


Figure 6-23 Perpendicularity of a noncylindrical feature of size: Prob. 7.

7. For the slot in Fig. 6-23, refine the perpendicularity to datum feature A within a tolerance of .002 at MMC.

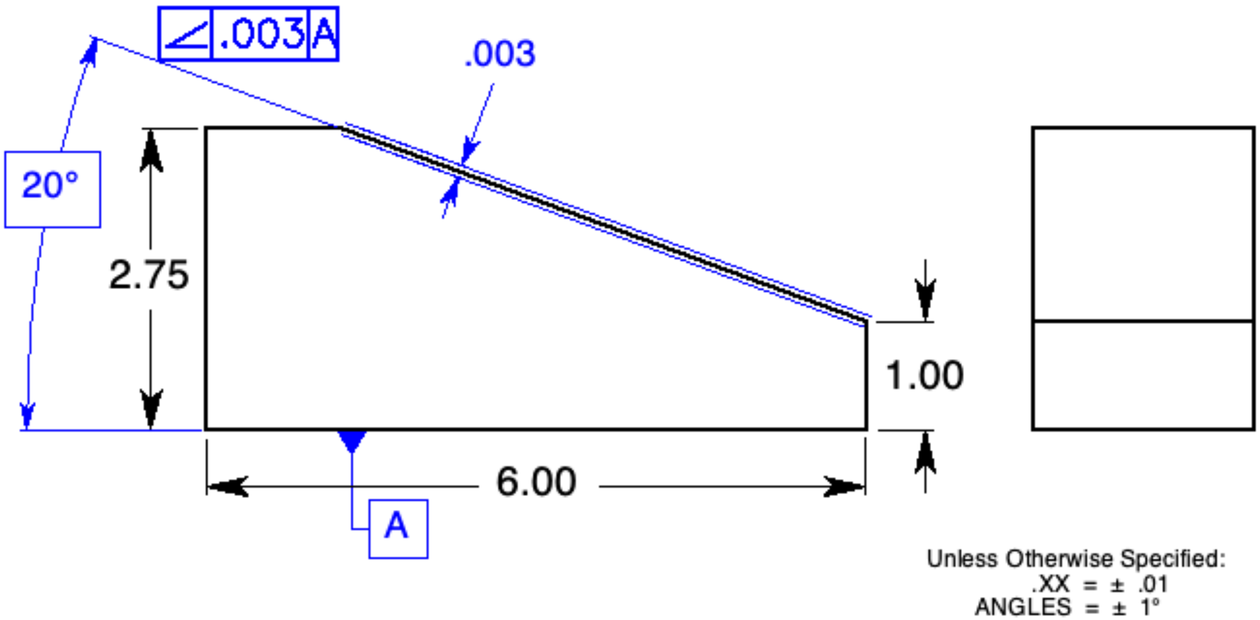


Figure 6-24 Angularity of a plane surface: Prob. 8.

8. Specify the top surface of the part in Fig. 6-24 to be at an angle of 20° to the bottom surface within a tolerance of .003. Draw and dimension the tolerance zone.

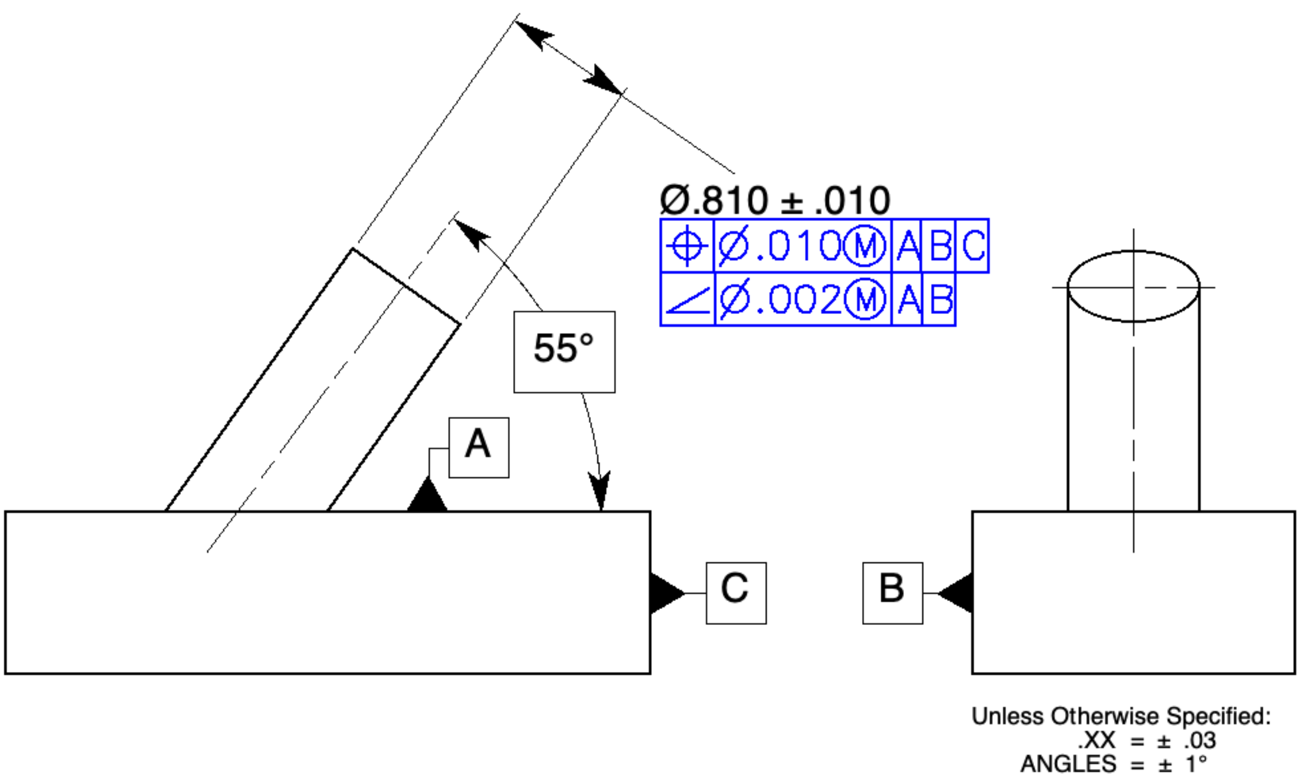
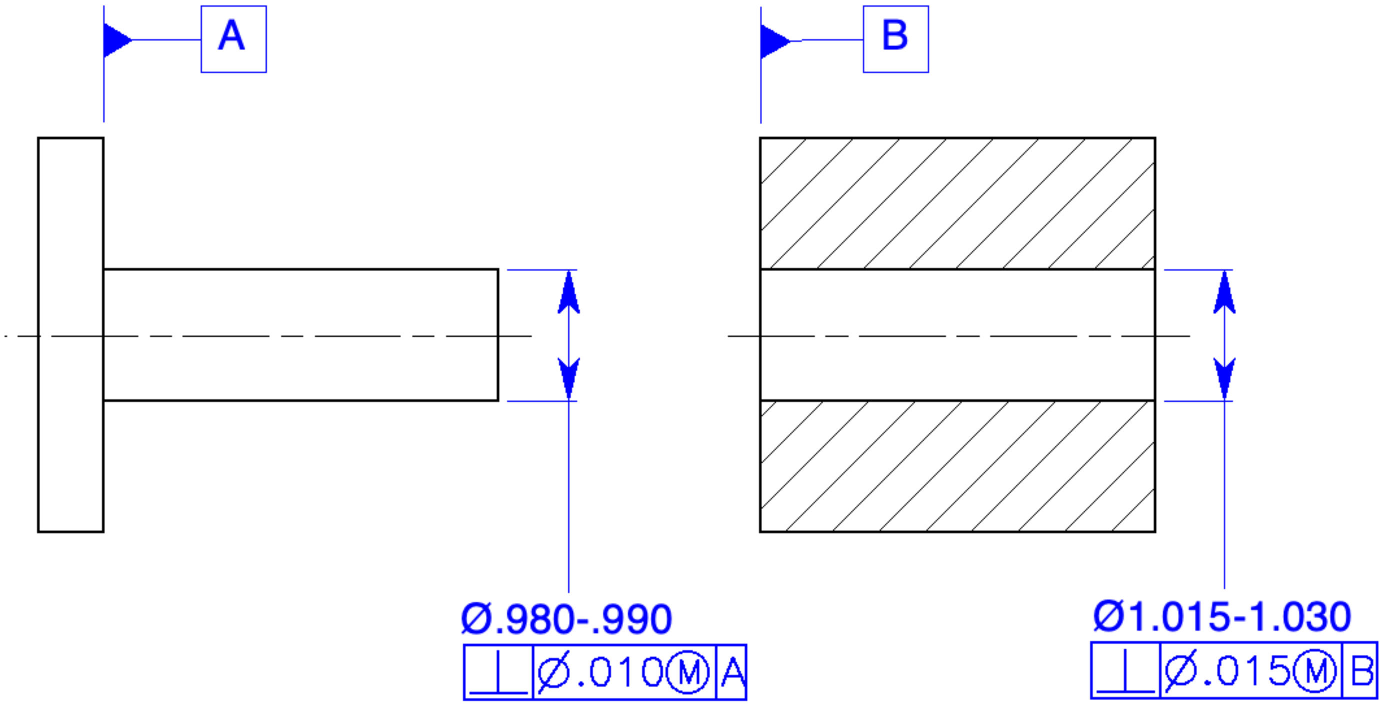


Figure 6-25 Angularity of a feature of size: Prob. 9.

9. For the pin in Fig. 6-25, refine the angularity to datum feature A and parallelism to datum feature B within a tolerance of .002 at MMC.



MMC	*.990*	*1.015*
Geometric Tolerance	*+.010*	*–.015*
Virtual Condition	*1.000*	*1.000*

Figure 6-26 Orientation: problem 10

10. Complete the feature control frames in Fig. 6-26 so that the two parts will always assemble, datum features A and B will meet, and the part can be produced using the most cost-effective design. The pin is machined in a lathe and the hole is drilled.

    (There are several possible solutions to this problem. The virtual conditions should be equal to insure assembly and to provide maximum tolerance. Typically, for this method of manufacturing, more tolerance is given to the hole.)

Chapter 7

Position, General

Chapter Review

Page 139

1. Position is a composite tolerance that controls both the *location and the orientation* of features of size at the same time.
2. The tolerance of position may be viewed in either of two ways:
   - *A **theoretical tolerance zone** located at true position of the toleranced feature within which the center point, axis, or center plane of the feature may vary from true position*.
   - *A **virtual condition boundary** of the toleranced feature, when specified at MMC or LMC and located at true position, which may not be violated by its surface or surfaces*.
3. A feature of size has four geometric characteristics that must be controlled. These characteristics are *size, form, orientation, and location*.
4. Since the position tolerance controls only features of sizes such as pins, holes, tabs, and slots, the feature control frame is always associated with a *size dimension*.
5. The location of true position, the theoretically perfect location of an axis, is specified with *basic dimensions* from the datum features indicated.
6. Once the feature control frame is assigned, an imaginary *tolerance zone* is defined and located about true position.
7. Datum features are identified with *datum feature symbols*.
8. Datum features A, B, and C identify a *datum reference frame*; consequently, they describe how the part is to be positioned for *processing*.

9. If no material condition modifier is specified in the feature control frame, the *RFS* modifier automatically applies to the tolerance of the feature.

10. To inspect a hole, the largest pin gage to fit inside the hole is used to simulate the *actual mating envelope*.

11. The measurement from the surface plate to the top edge of the pin gage minus half of the diameter of the pin gage equals the distance from *datum feature B to the actual axis of the hole.*

12. Where the maximum material condition symbol is specified to modify the tolerance of a feature of size, the following two requirements apply:

- The specified tolerance applies at *the maximum material condition of the feature*.
- As the size of the feature departs from MMC toward LMC, *a bonus tolerance is achieved*.

13. The difference between the actual mating envelope size and the MMC is the *bonus.*

14. The bonus plus the geometric tolerance equals *the total positional tolerance*.

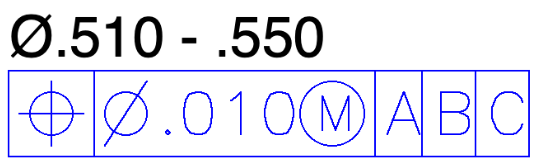


Figure 7-19 Geometric tolerance: Questions 15 through 18.

15. If the tolerance in Fig.7-19 is for a pin .525 in diameter, what is the total positional tolerance? *.035*

16. What would be the size of the hole in a functional gage to inspect the pin above? *.560*

17. If the tolerance in Fig. 7-19 is for a hole .540 in diameter, what is the total positional tolerance? *.040*

18. What would be the size of the pin on a functional gage to inspect the hole above? *.500*

19. Where a datum feature of size is toleranced with a geometric tolerance and is referenced in a feature control frame at MMB, the resulting maximum material boundary for the feature is equal to *its virtual condition with respect to the preceding datum feature.*

20. A zero tolerance is not used where the tolerance applies at *RFS*, or where no bonus tolerance is available as in a tolerance specified for *threads or press fit pins.*

Problems

Page 142

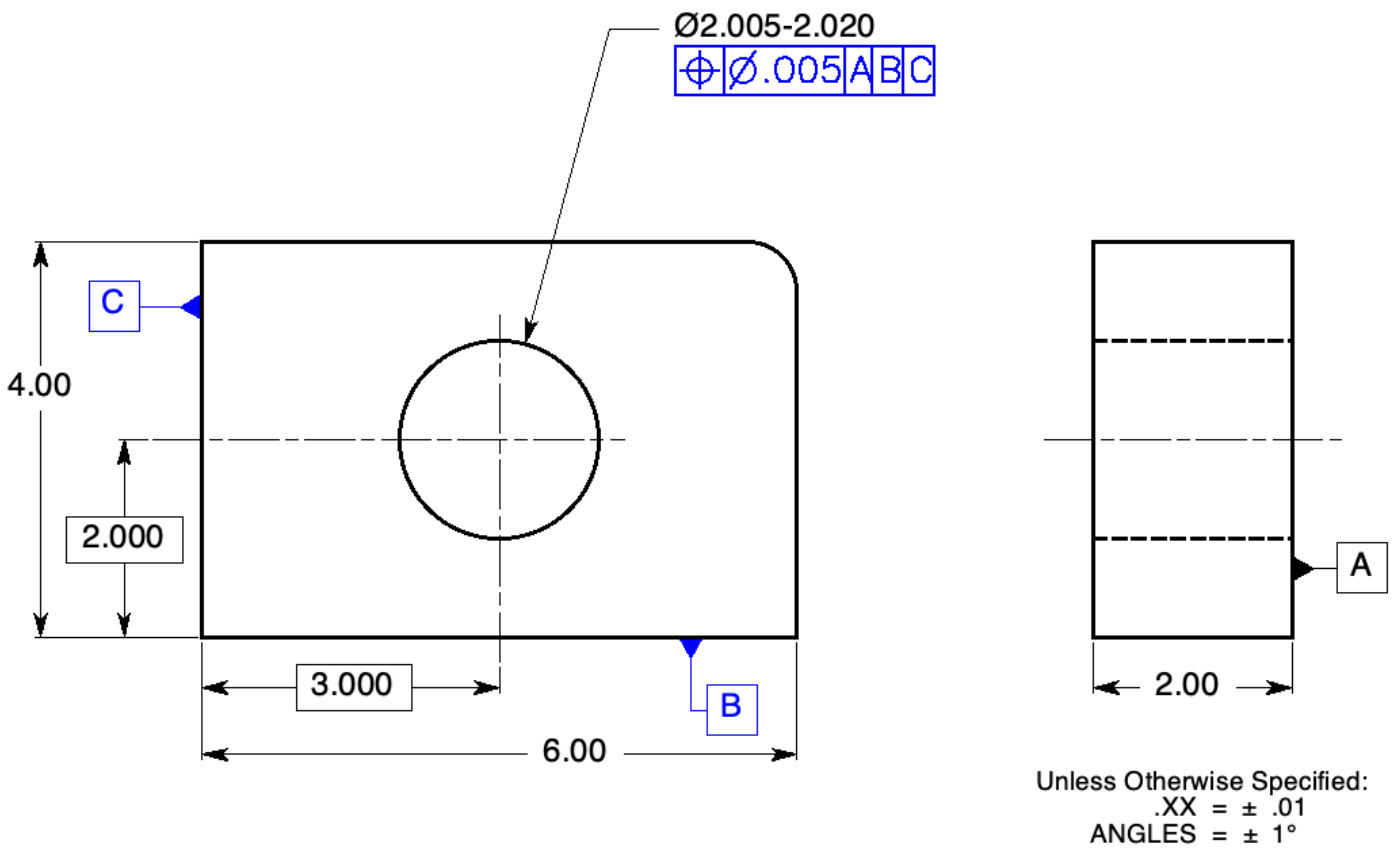
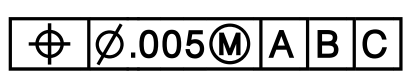


Figure 7-20 Locating a hole with the position control: Probs. 1 through 3.

1. Locate the hole in Fig. 7-20 with a positional tolerance of .005 at RFS.

2. Draw a feature control frame below to locate the hole in Fig. 7-20 with a positional tolerance of .005 at MMC.



3. If the actual mating envelope in the hole in problem 2 is produced at a diameter of 2.010 and the axis is located .003 over and .005 up from true position, is the part within tolerance? If not, can it be reworked to meet specifications?

Bonus = AME – MMC = 2.010 – 2.005 = .005
The Total Positional Tolerance = Geometric Tolerance + Bonus = .005 +.005 = .010
The distance from the axis to true position = the square root of $(.003)^2 + (.005)^2 = .006$.
This distance from the axis to true position requires a cylindrical tolerance zone Ø.012.
The part is not in tolerance. If the size of the hole is increased by at least .002, it would be in tolerance.

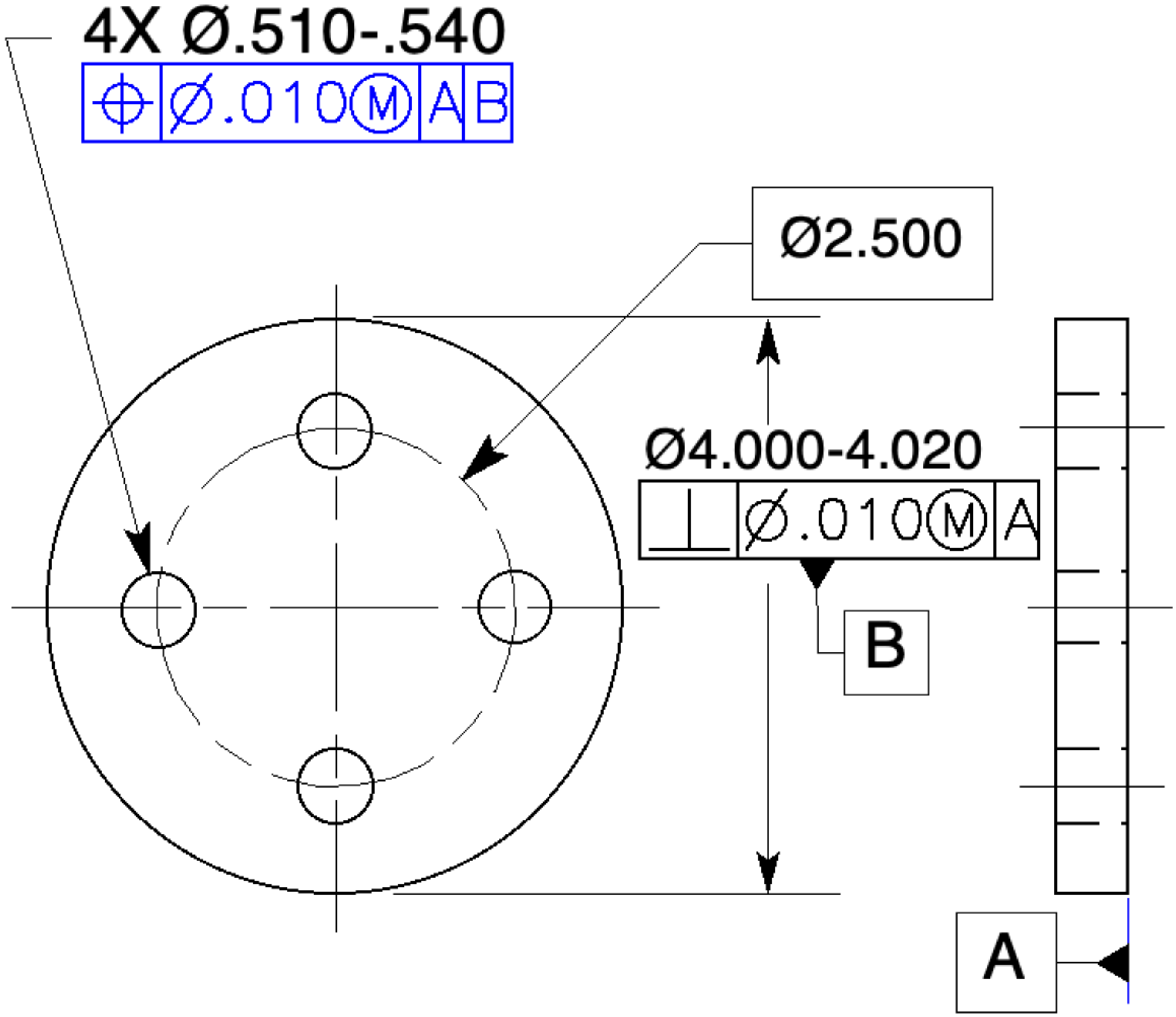
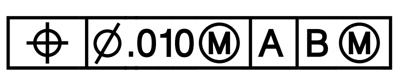


Figure 7-21 Locating features to a datum feature of size: Probs. 4 through 6.

4. Position the four-hole pattern with a tolerance of .010 at MMC in Fig. 7-21 perpendicular to datum feature A and located to datum feature B at RMB. If datum feature B is produced at a diameter of 4.010, how much shift tolerance is available?

None, datum feature is specified at RMB.

5. Draw a feature control frame below to position the four-hole pattern in Fig. 7-21 perpendicular to datum feature A and located to datum feature B at MMB.



6. How much shift tolerance is available in Fig. 7-21 if datum feature B is specified at MMB and is produced at 4.015 in diameter?

MMB = 4.030,   4.030 – 4.015 = .015 Shift Tolerance

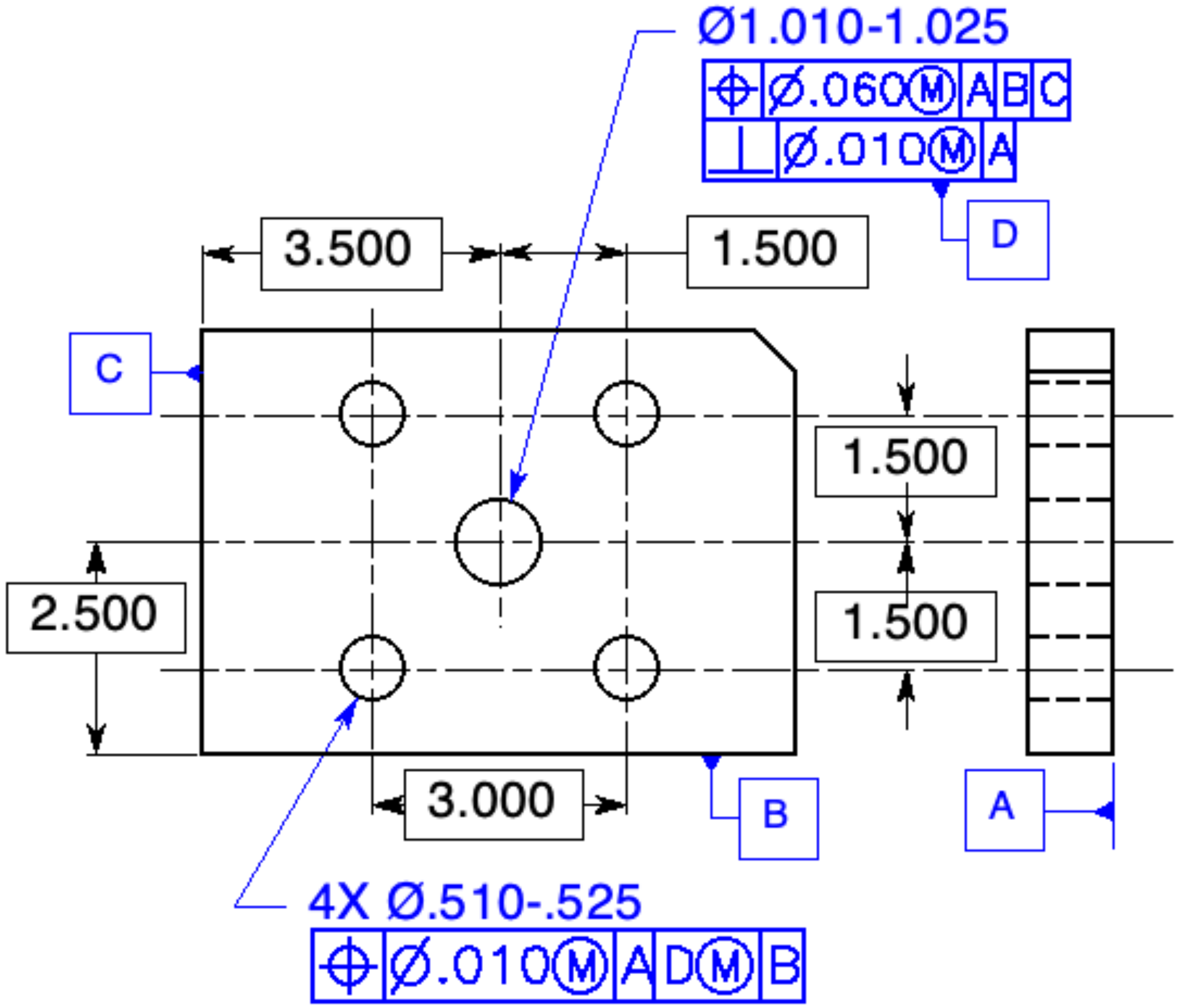


Figure 7-22 Design a gage to inspect for shift tolerance: Probs. 7 and 8.

7. On a gage designed to control the 4-hole pattern in Fig. 7-22, what size pin must be produced to inspect the center hole (datum feature D)? *Ø1.000*

8. On the same gage, what is the diameter of the four pins locating the hole pattern? *Ø.500*

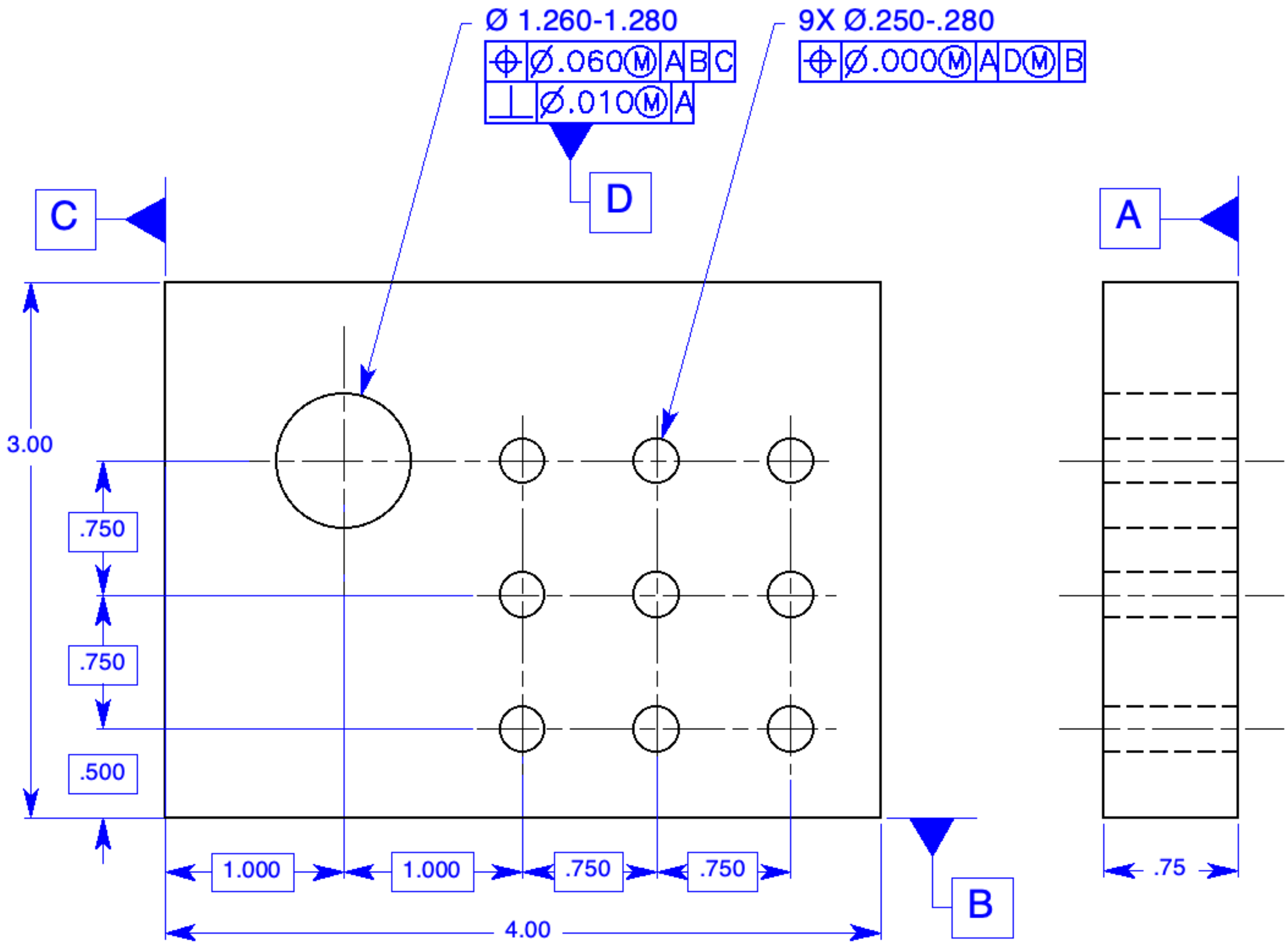


Figure 7-23 A pattern of holes located to a datum feature of size: Prob. 9.

9. In Fig. 7-23, locate the 1¼-inch-diameter hole to the edge datum features within a tolerance of .060 and refine its perpendicularity to datum feature A with a tolerance of .010. Locate the nine-hole pattern to the 1¼-inch-diameter hole and clock it to an edge datum feature with a zero positional tolerance. Use MMC and MMB wherever possible.

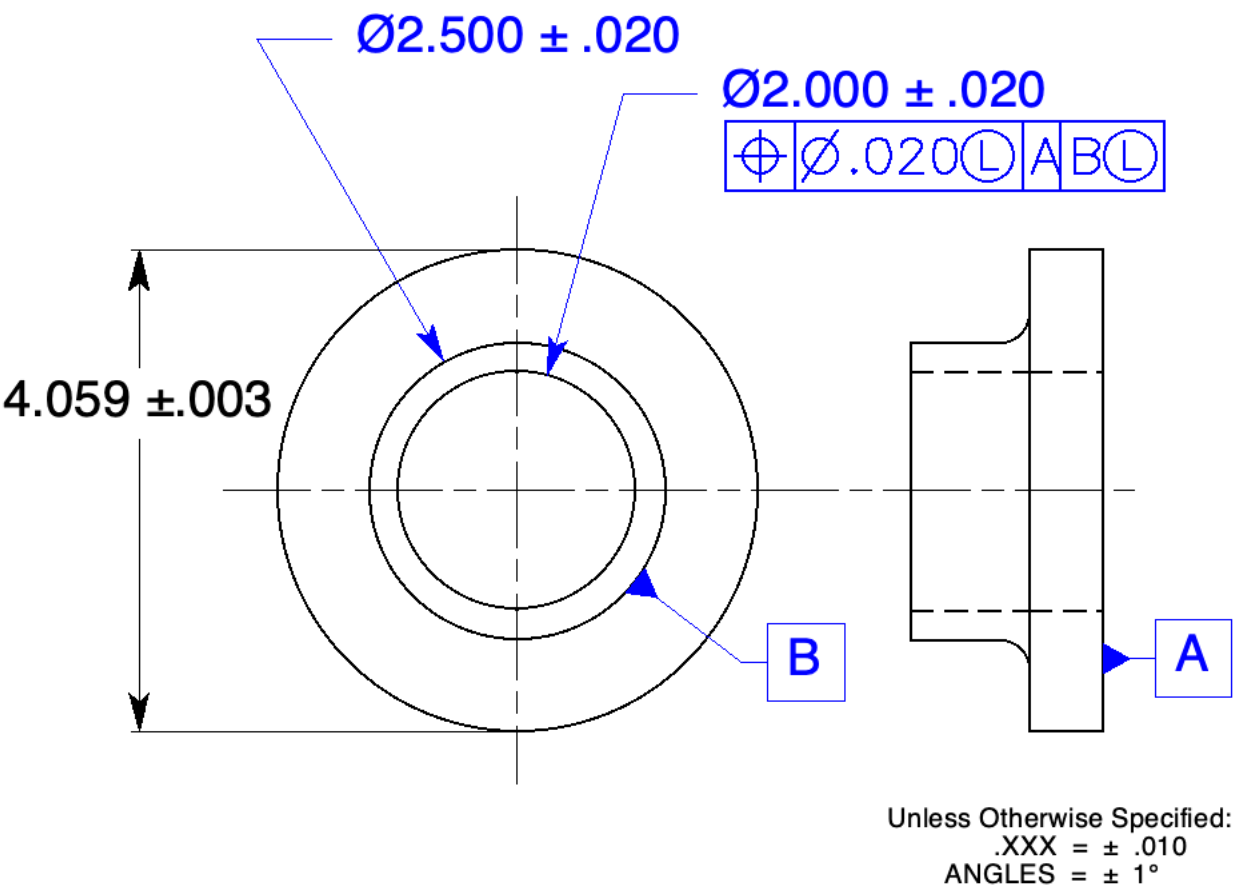


Figure 7-24 A hole specified at LMC: Prob. 10.

10. Calculate the minimum wall thickness between the inside diameter and datum feature B shown in Fig. 7-24.

Datum B @ LMC	*Ø 2.480*
I. D. @ LMC	*– Ø 2.020*
Tolerance @ LMC	*– Ø .020*
	.440

The wall thickness equals half of the differences in diameters or .220.

(Calculating diameters and diving the final diameter in half minimize Errors.)

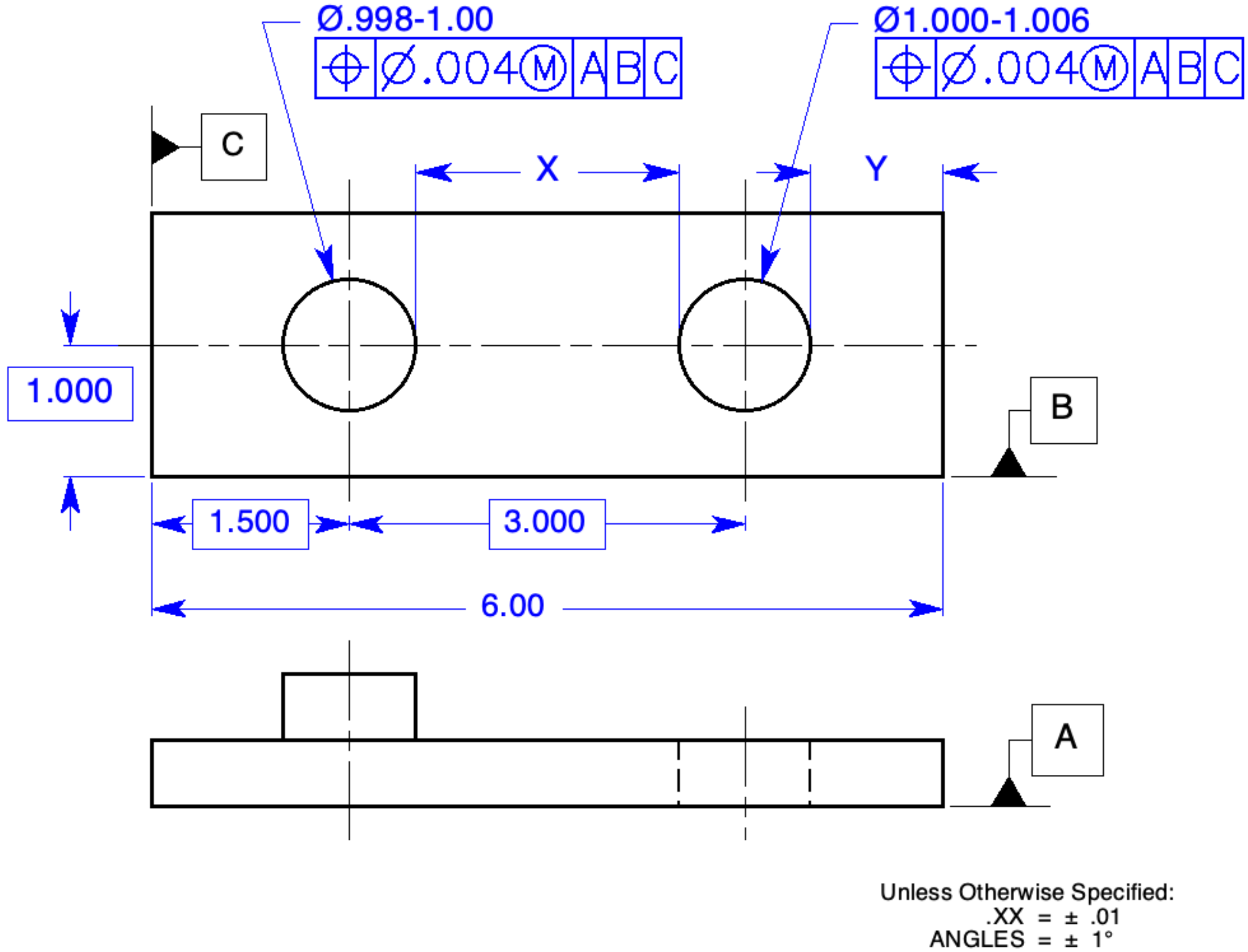


Figure 7-25 Boundary conditions: Prob. 11.

11. First calculate the virtual conditions and resultant conditions for the pin and hole in Fig.7-25. Then calculate the maximum and minimum distances for dimensions X and Y.

The **Virtual Condition** of the **PIN**.

$VC_p = MMC + Geo.\ Tol.$
$VC_p = 1.000 + .004 = 1.004$
$VC_p/2 = .502$

The **Virtual Condition** of the **HOLE**.

$VC_h = MMC - Geo.\ Tol.$
$VC_h = 1.000 - .004 = .996$
$VC_h/2 = .498$

Resultant Condition of the **PIN**.

$RC_p = LMC - Geo.\ Tol. - Bonus$
$RC_p = .998 - .004 - .002 = .992$
$RC_p/2 = .496$

Resultant Condition of the **HOLE**.

$RC_h = LMC + Geo.\ Tol. + Bonus$
$RC_h = 1.006 + .004 + .006 = 1.016$
$RC_h/2 = .508$

The maximum and minimum distances for dimension **X**:

$X_{Max} = Dist. - RC_p/2 - VC_h/2 =$
$X_{Max} = 3.000 - .496 - .498 =$
$X_{Max} = 2.006$

$X_{Min} = Dist.\ \ VC_p/2 - RC_h/2 =$
$X_{Min} = 3.000 - .502 - .508$
$X_{Min} = 1.990$

The maximum and minimum distances for dimension **Y**:

$Y_{Max} = Length\ @\ MMC - Dist. - VC_h/2 =$
$Y_{Max} = 6.010 - 4.500 - .498 =$
$Y_{Max} = 1.012$

$Y_{Min} = Length\ @\ LMC - Dist - RC_h/2 =$
$Y_{Min} = 5.990 - 4.500 - .508 =$
$Y_{Min} = .982$

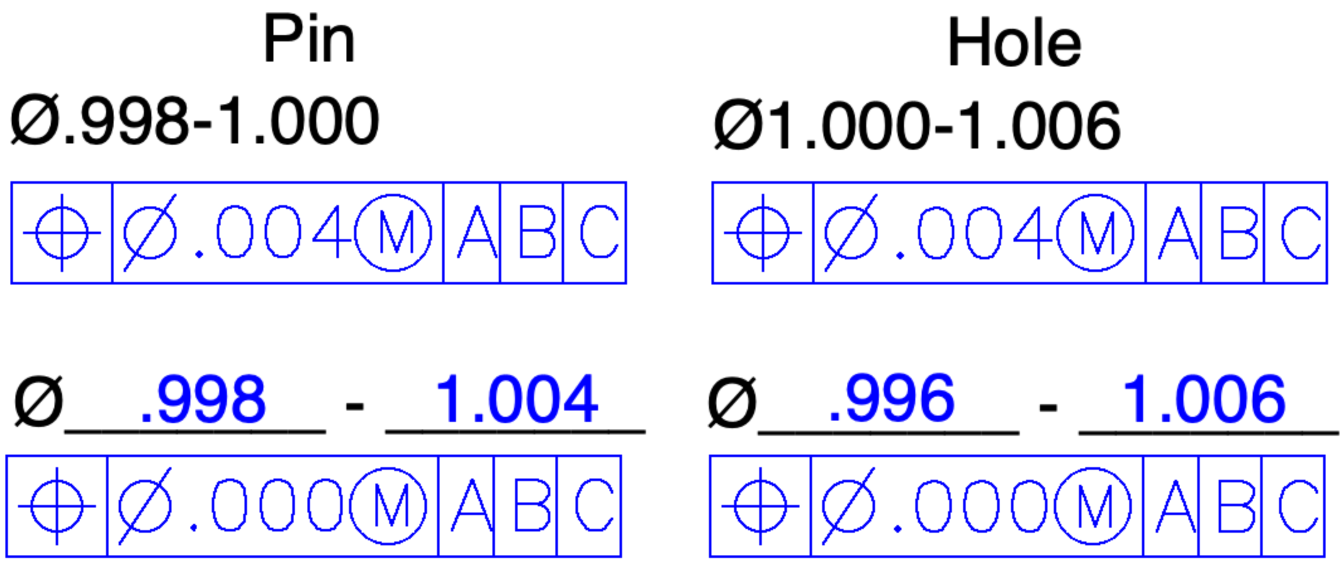


Figure 7-26 Zero positional tolerance conversion: Probs. 12 and 13

12. Convert the tolerance in Fig. 7-26 to zero positional tolerances.

13. Zero tolerance is not used when the tolerance applies at *RFS*, or when no bonus tolerance is available as in a tolerance specified for *threads or press fit pins.*

Chapter 8

Position, Location

Chapter Review

Page 176

1. The floating fastener formula is:

   $T = H - F$ *or* $H = F + T$

2. T = *Tol. at MMC* H = *Hole Ø at MMC* F = *Fastener Ø at MMC*

3. The clearance hole LMC diameter formula is *H @ LMC = (F +F head) / 2*.

4. The fixed fastener is fixed by one or more of the *members being fastened*.

5. The formula for fixed fasteners is:

$t_1 + t_2 = H - F$ *or* $H = F + t_1 + t_2$

6. The formula for fixed fasteners is essentially the same as for floating fasteners except that the fixed fastener formula includes the tolerance for *each hole.*

7. It is common practice to assign a larger portion of the location tolerance to the *threaded* hole.

8. A fastener fixed at its head in a countersunk hole and in a threaded hole at the other end is called what?

A double fixed fastener

9. Where specifying a threaded hole or a hole for a press fit pin, the orientation of the *hole* determines the orientation of the mating pin.

10. The most convenient way to control the orientation of a pin outside the hole is to *project* the tolerance zone into the mating part.

11. The height of the projected tolerance zone is equal to or greater than the thickest *mating part* or tallest *stud or pin* after installation.

12. Two or more patterns of features are considered to be a single pattern of features if they are:

located with basic dimensions, to the same datums features, in the same order of precedence, and at the same material conditions

13. Datum features of size specified at RMB require *physical contact* between the gagging element and the datum feature.

14. If patterns of features controlled to datum features of size specified at MMB have no relationship to each other, a note such as *SEP REQT* may be placed under each feature control frame allowing each pattern to be inspected separately.

15. When locating patterns of features, there are situations where the relationship from *feature to feature* must be kept to a certain tight tolerance and the relationship between the *pattern* and its datum features is not as critical and may be held to a looser tolerance.

16. A composite feature control frame has one *position* symbol

that applies to the two horizontal ___*segments*___ that follow.

17. The upper segment of a composite feature control frame is called the ___*pattern-locating*___ control, it governs the relationship between the datum features and the ___*pattern*___.

18. The lower segment of a composite feature control frame is called the ___*feature-relating*___ control; it governs the relationship from ___*feature to feature*___

19. The primary function of the position control is to control ___*location*___

20. Datum features in the lower segment of a composite feature control frame must satisfy what two conditions? ___*They are required to repeat the datums in the upper segment*___

    ___*They only control orientation*___

(Assume plane surface datum features for Questions 21 and 23)

21. Where the secondary datum feature is included in the lower segment of a composite feature control frame, the tolerance zone framework must remain ___*Parallel*___ to the secondary datum plane.

22. The lower segment of a multiple single-segment feature control frame acts just like any other ___*position control*___.

23. Counterbores that have the same location tolerance as their respective holes are specified by indicating the ___*hole callout and the counterbore callout followed by the geometric tolerance for both*___

24. Counterbores that have a larger location tolerance than their respective holes are specified by ___*separating the hole callout from the counterbore callout*___.

25. When tolerancing elongated holes, no ___*diameter symbol*___ precedes the tolerance in the feature control frame since the tolerance zone is not ___*cylindrical*___.

26. The virtual condition boundary is the ___*exact shape*___ of the noncircular feature and equal in size to its ___*virtual condition*___.

27. The ___*position control*___ is used to locate a feature of size symmetrically at MMC to a datum feature of size specified at MMB.

Problems

Page 179

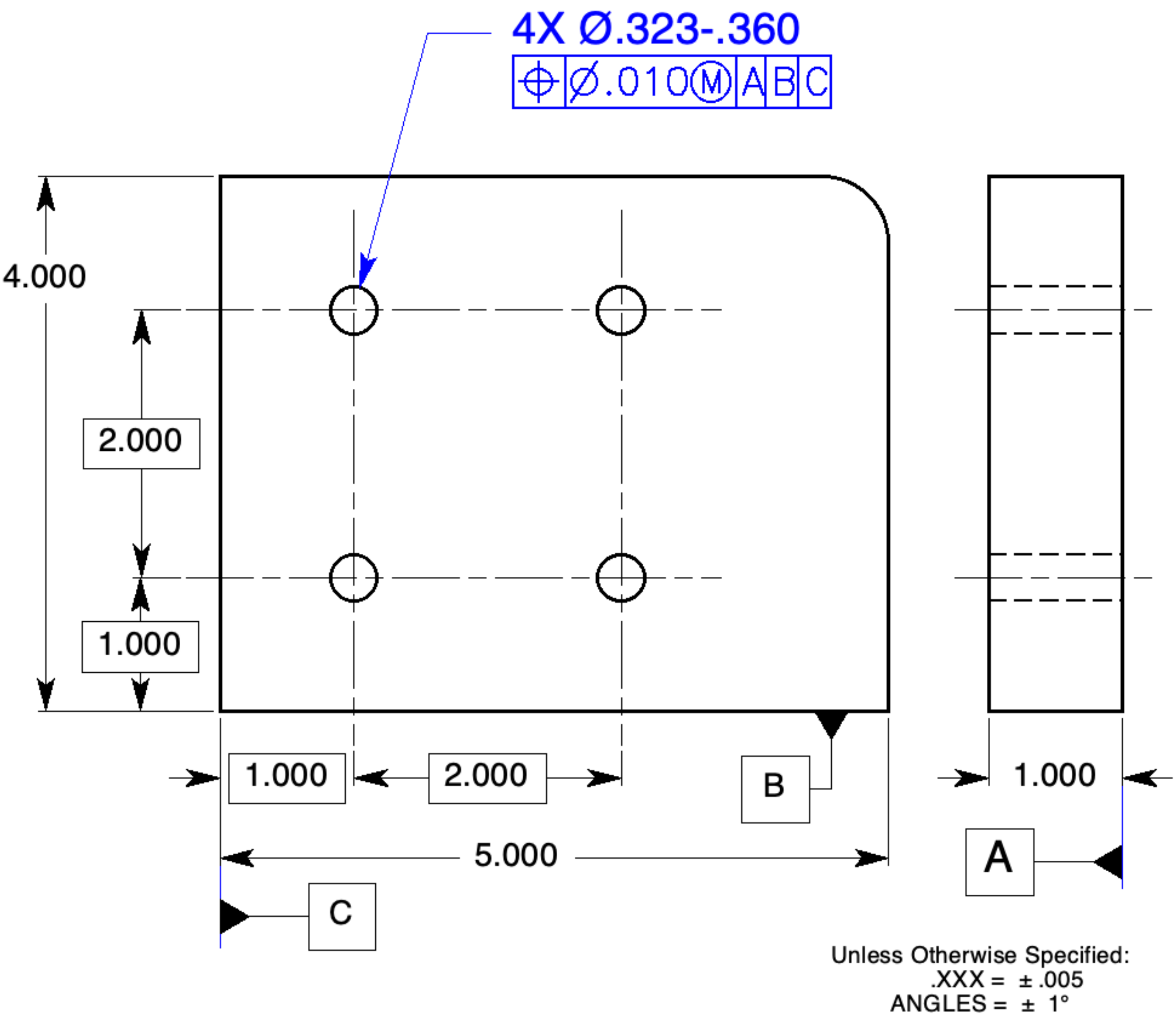


Figure 8-28 Floating fastener tolerance: Prob. 1.

1. Tolerance the clearance holes on the plate in Fig. 8-28 to be fastened with 5/16 – 18 UNC hex head bolts (.313 in diameter) and nuts with a .010-diameter positional tolerance at MMC.

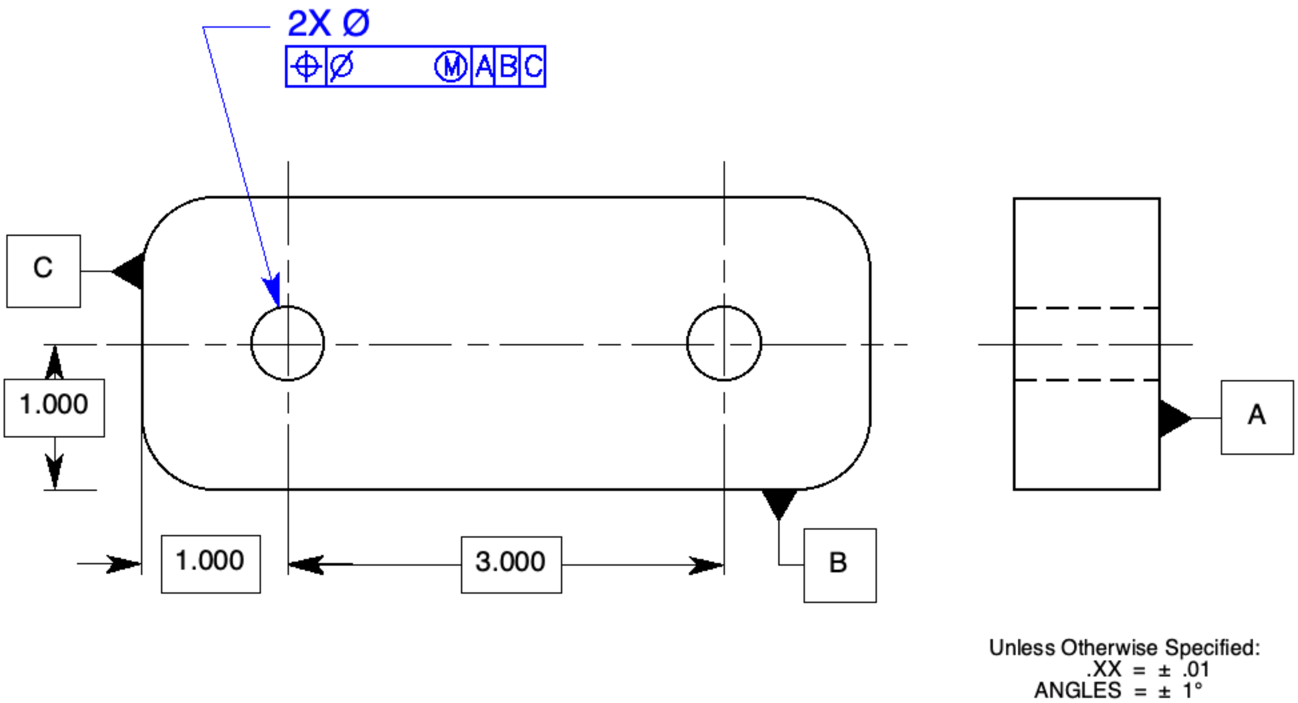


Fig. 8-29 Floating fastener tolerance at MMC: Probs. 2 through 5.

2. Specify the MMC and LMC clearance hole sizes for #10 (Ø.190) socket head cap screws.

(Many other solutions are possible, but they must satisfy the floating fastener formula.)

2X Ø.220-.246 | ⌖ | Ø.030Ⓜ | A | B | C |

2X Ø.200-.246 | ⌖ | Ø.010Ⓜ | A | B | C |

2X Ø.190-.246 | ⌖ | Ø.000Ⓜ | A | B | C |

3. If the actual mating envelope size of the clearance holes in problem 2 are .230 in diameter, calculate the total positional tolerance for each callout.

Actual Size	.230	.230	.230
MMC	– *.220*	– *.200*	– *.190*
Bonus	*.010*	*.030*	*.040*
Geo. Tolerance	+ *.030*	+ *.010*	+ *.000*
Total Tolerance	*.040*	*.040*	*.040*

4. Specify the MMC and LMC clearance hole sizes for 3/8 (Ø.375) hex head bolts.

2X Ø .400-.460 | ⌖ | Ø.025Ⓜ | A | B | C |

2X Ø .390-.460 | ⌖ | Ø.015Ⓜ | A | B | C |

2X Ø .375-.460 | ⌖ | Ø.000Ⓜ | A | B | C |

5. If the actual mating envelope size of the clearance holes in Prob. 4 measure .440 in diameter, calculate the total positional tolerance for each callout.

Actual Size	.440	.440	.440
MMC	*– .400*	*– .390*	*– .375*
Bonus	*.040*	*.050*	*.065*
Geo. Tolerance	*+ .025*	*+ .015*	*+ .000*
Total Tolerance	*.065*	*.065*	*.065*

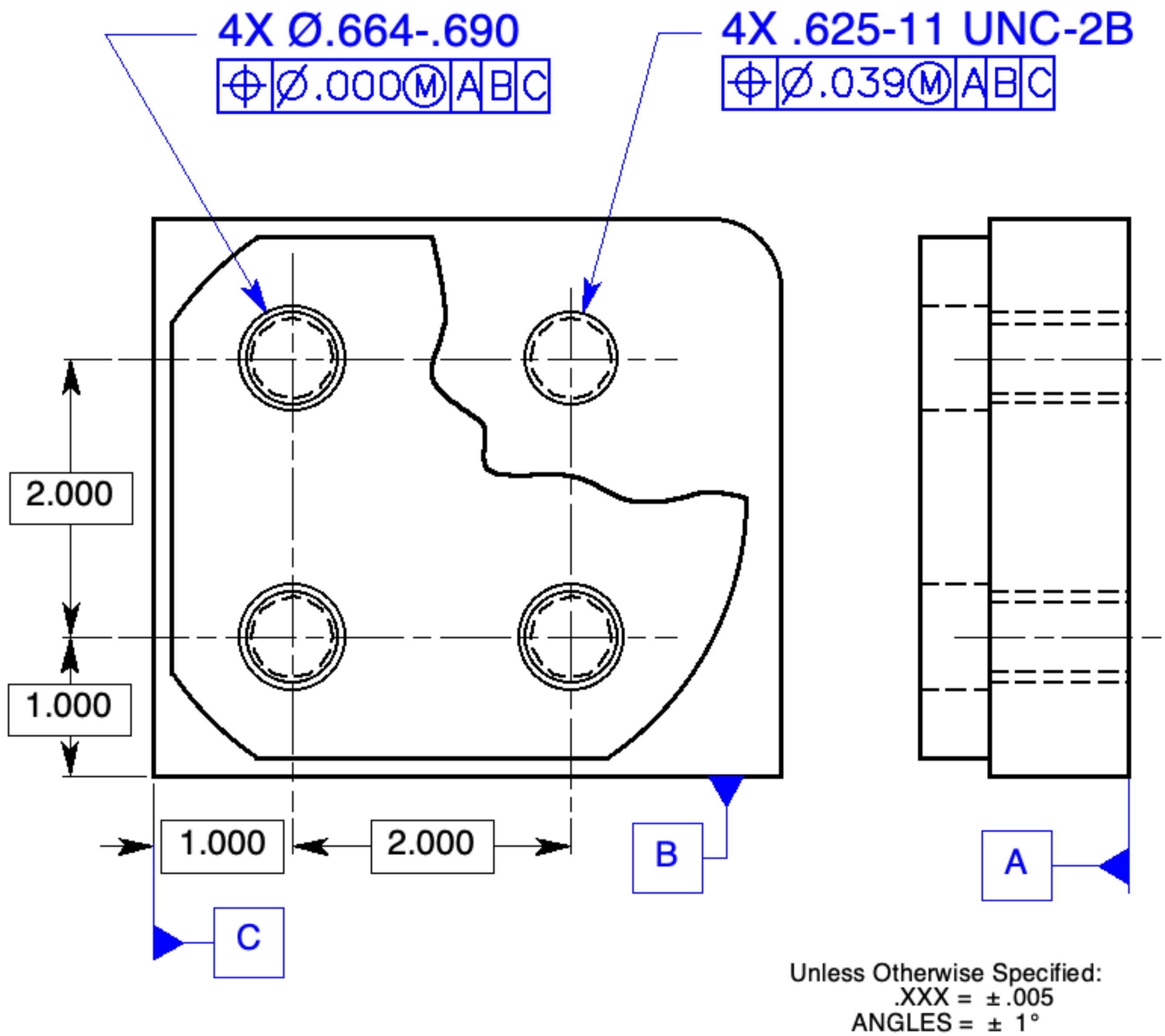


Figure 8-30 Fixed fastener tolerance: Prob. 6.

6. Tolerance the clearance and threaded holes in the plates in Fig. 8-30 to be fastened with 5/8–11 UNC hex head bolts (.625 in diameter). Use a .000 positional tolerance at MMC wherever possible, and calculate a 60% location tolerance for the threaded holes.

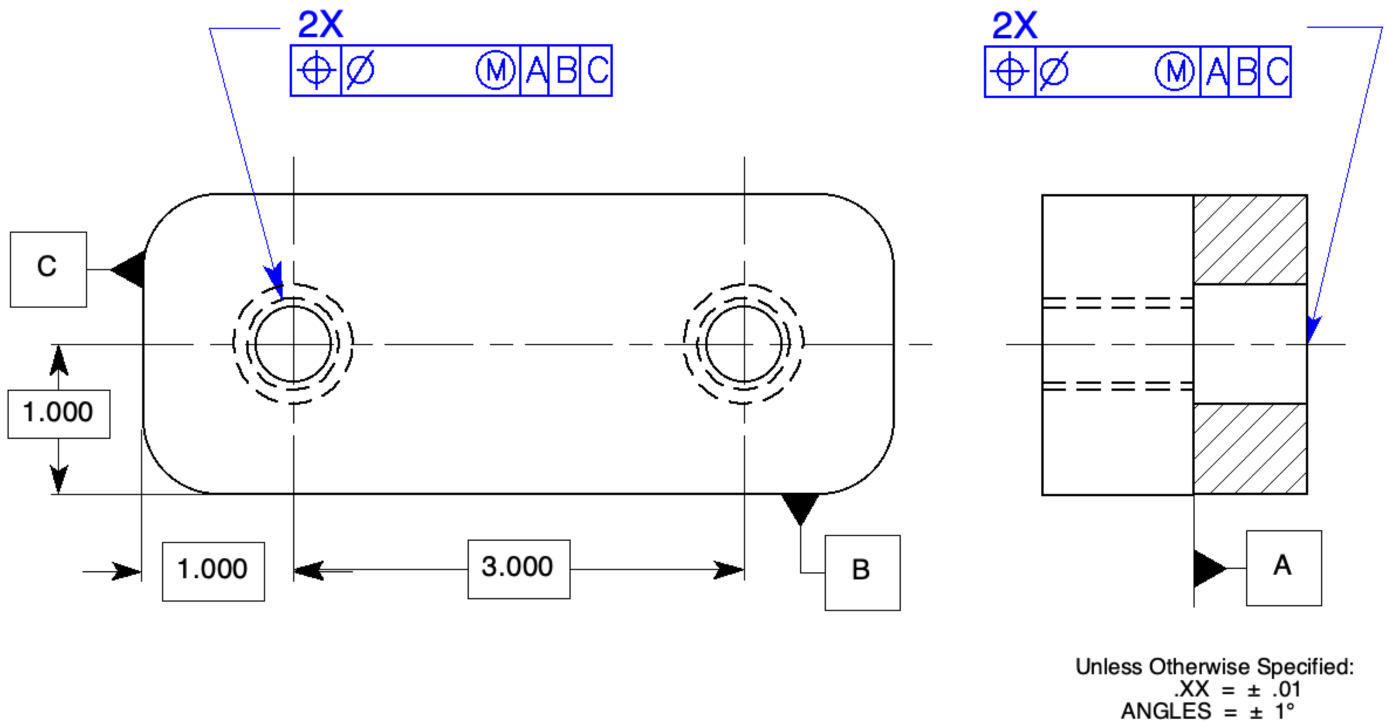


Fig. 8-31 Fixed fastener drawing: Probs. 7 through 10.

7. Specify the MMC and LMC clearance hole sizes for #8 socket head cap screws.

2X Ø.164 (#8)-32 UNF-2B	2X Ø.164 (#8)-32 UNF-2B	2X Ø.164 (#8)-32 UNF-2B
⌖ | Ø.025Ⓜ | A | B | C	⌖ | Ø.025Ⓜ | A | B | C	⌖ | Ø.025Ⓜ | A | B | C
.199 – .213	.194 – .213	.189 – .213
⌖ | Ø.010Ⓜ | A | B | C	⌖ | Ø.005Ⓜ | A | B | C	⌖ | Ø.000Ⓜ | A | B | C

8. If the actual mating envelope size of the clearance holes in problem 7 measure Ø.205, calculate the total positional tolerance for each callout.

Actual Size	.205	.205	.205
MMC	– .199	– .194	– .189
Bonus	*.006*	*.011*	*.016*
Geo. Tolerance	+ *.010*	+ *.005*	+ *.000*
Total Tolerance	*.016*	*.016*	*.016*

9. Specify the MMC and LMC clearance hole sizes for the ½ inch hex head bolts.

2X Ø .500-20 UNF-2B	2X Ø .500-20 UNF-2B	2X Ø .500-20 UNF-2B
⌖ | Ø.060Ⓜ | A | B | C	⌖ | Ø.060Ⓜ | A | B | C	⌖ | Ø.060Ⓜ | A | B | C
2X Ø.580 – .612	2X Ø.570 – .612	2X Ø.560 – .612
⌖ | Ø.020Ⓜ | A | B | C	⌖ | Ø.010Ⓜ | A | B | C	⌖ | Ø.000Ⓜ | A | B | C

10. If the actual mating envelope size of the clearance holes in problem 9 measure Ø.585, calculate the total positional tolerance for each callout.

Actual Size	.585	.585	.585
MMC	*– .580*	*– .570*	*– .560*
Bonus	*.005*	*.015*	*.025*
Geo. Tolerance	*+ .020*	*+ .010*	*+ .000*
Total Tolerance	*.025*	*.025*	*.025*

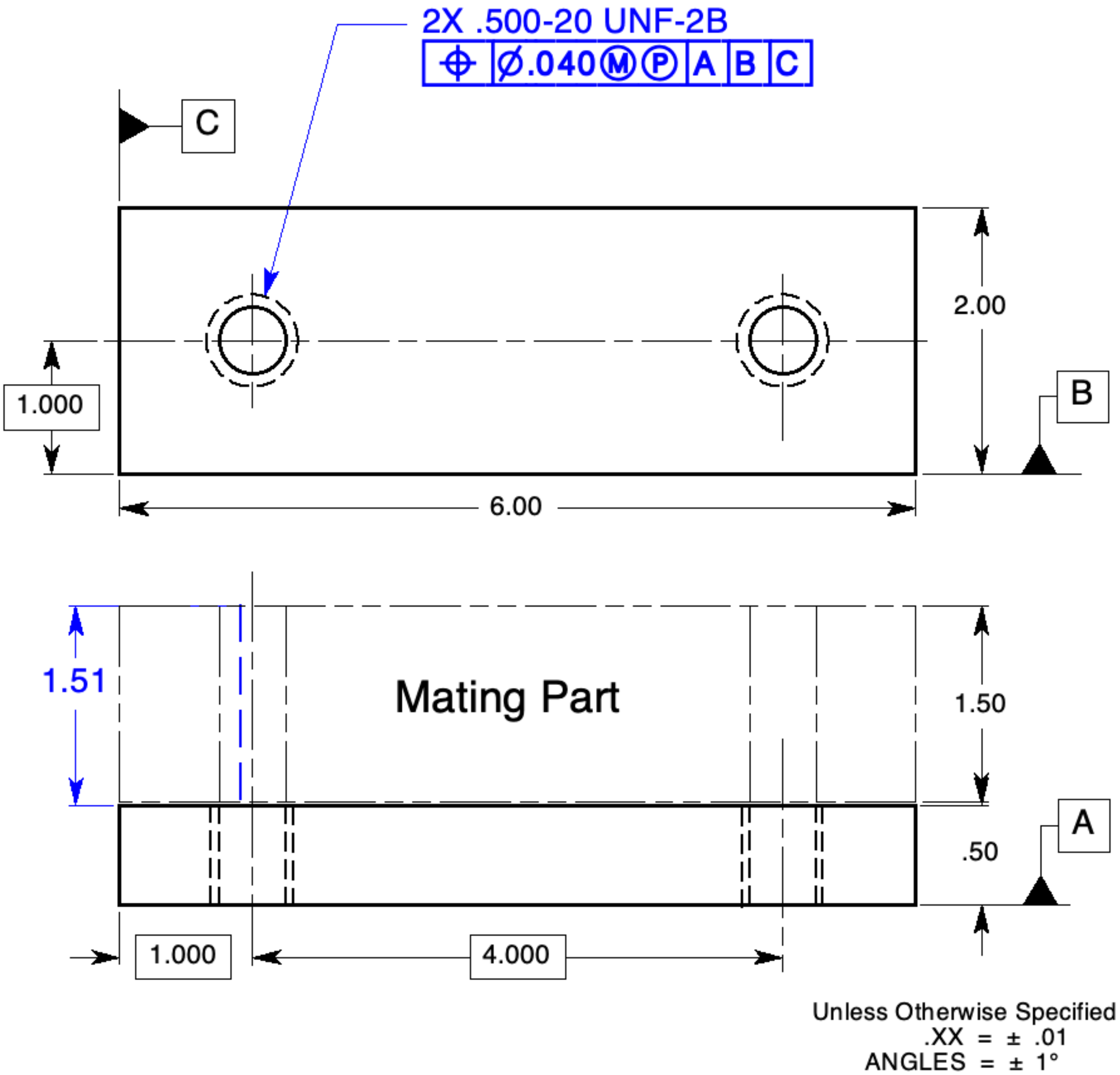


Figure 8-32 Projected tolerance zone: Prob. 11.

11. Complete the drawing in Fig. 8-32. Specify a .040 positional tolerance at MMC with the appropriate projected tolerance.

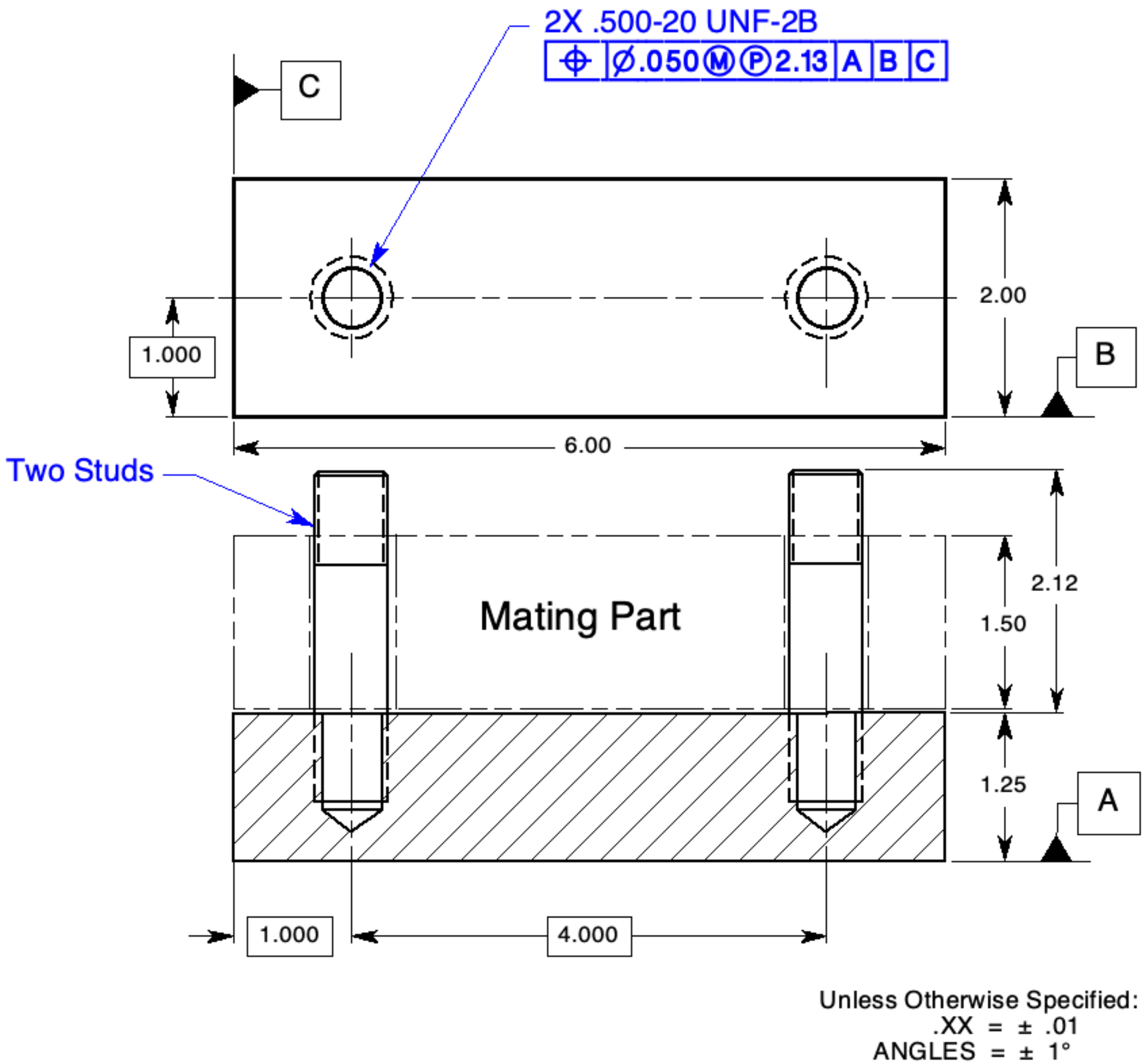


Figure 8-33 Projected tolerance zone for studs: Prob. 12.

12. Complete the drawing in Fig. 8-33. Specify a .050 positional tolerance at MMC with the appropriate projected tolerance.

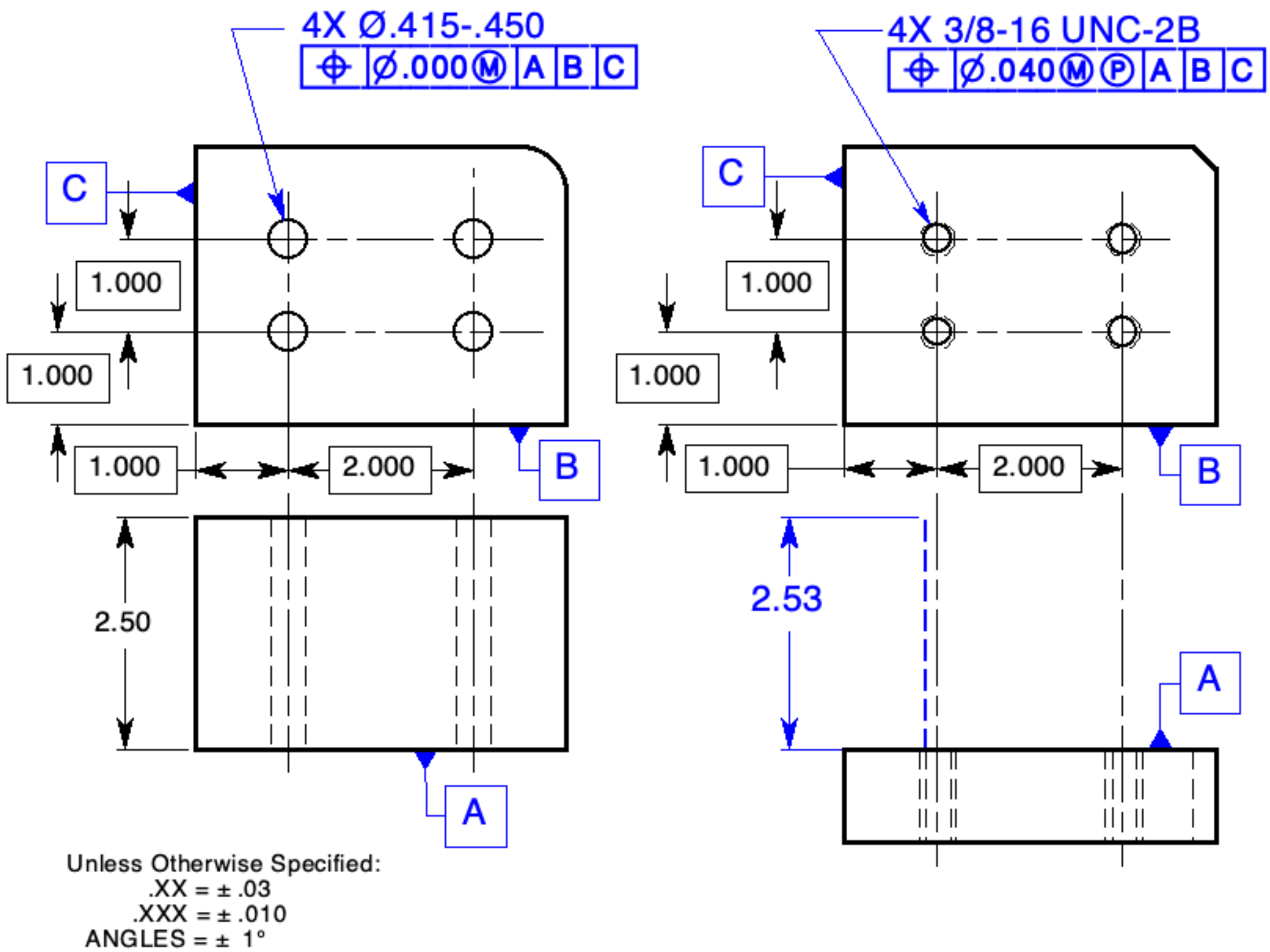


Figure 8-34 Fixed fastener assembly: Prob. 13.

13. The part with clearance holes in Fig. 8-34 assembles on top of the part with threaded holes and is fastened with cap screws. Allow a tolerance of at least .030 on both threaded and clearance holes, use "0" positional tolerance at MMC, and specify projected tolerance zones.

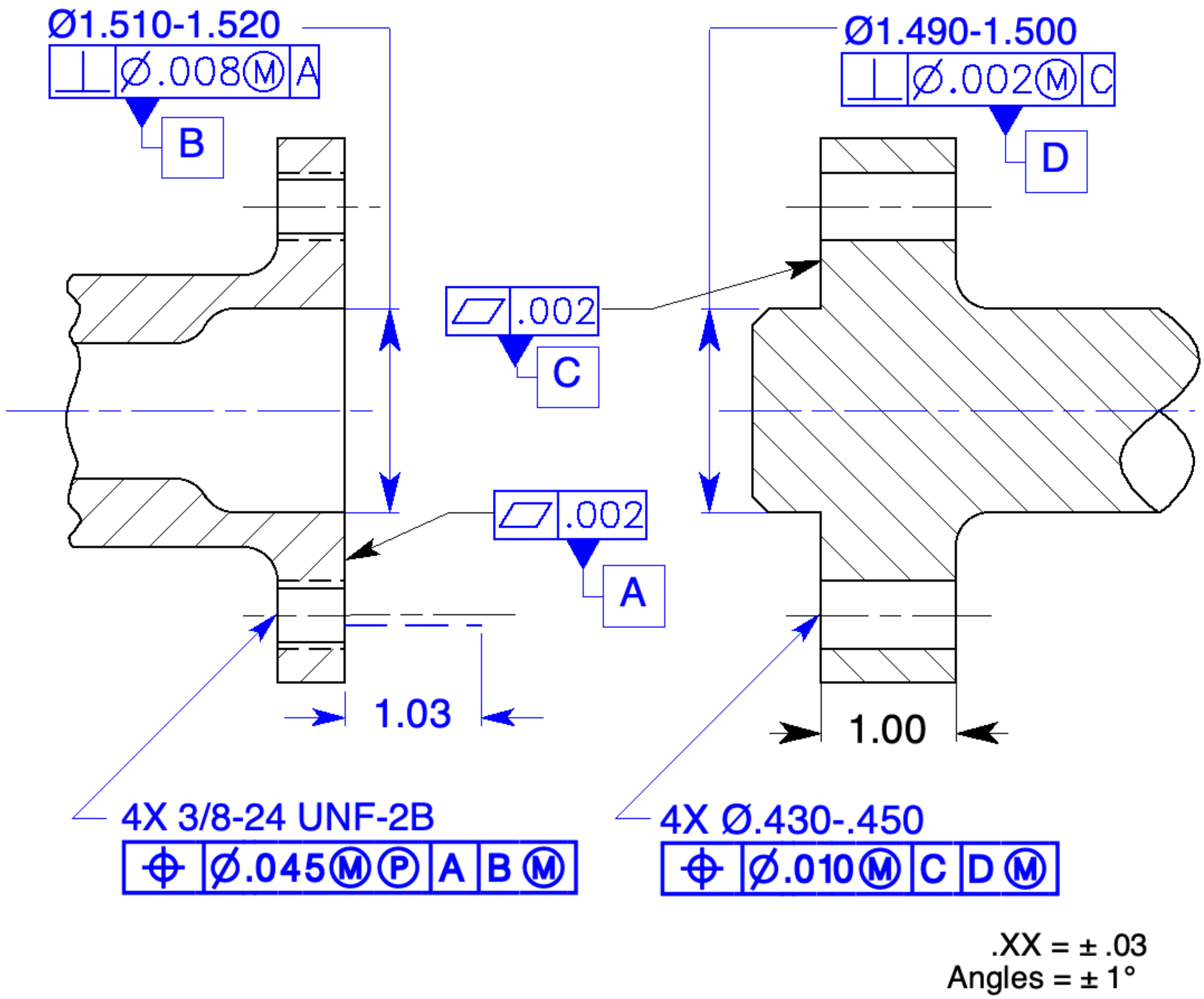


Figure 8-35 A coupling assembly: Prob. 14.

14. Tolerance the two parts in Fig. 8-35. Specify a flatness control of .002 on each of the primary datum features. Specify the appropriate orientation tolerance to control the relationships between the primary and secondary datum features. Finally, complete the location tolerances for the hole patterns for 3/8-inch cap screws using a .010 positional tolerance for the clearance holes. Specify MMC and MMB wherever possible.

There are other possible solutions to this problem.

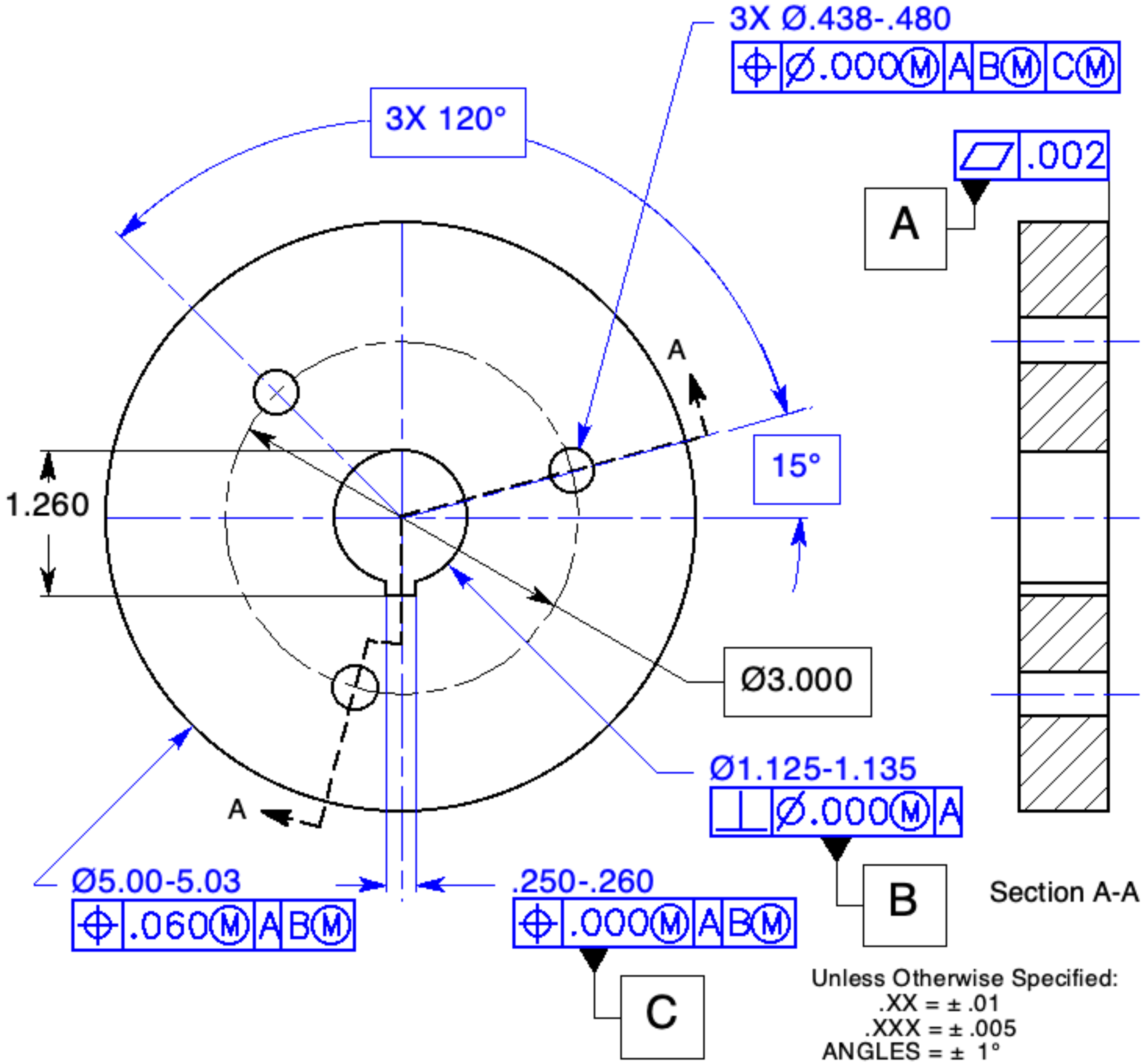


Figure 8-36 A pattern of holes located to a datum feature of size: Prob. 15.

15. In Fig. 8-36, the inside diameter and the back are mating features. Select the primary datum feature. (Consider a form control.) The virtual condition of the mating shaft is 1.125 in diameter. Locate the keyway for a ¼-inch key. Locate the three-hole pattern for 7/16-inch (Ø.438) cap screws as floating fasteners with a zero positional tolerance at MMC. Specify MMC and MMB wherever possible.

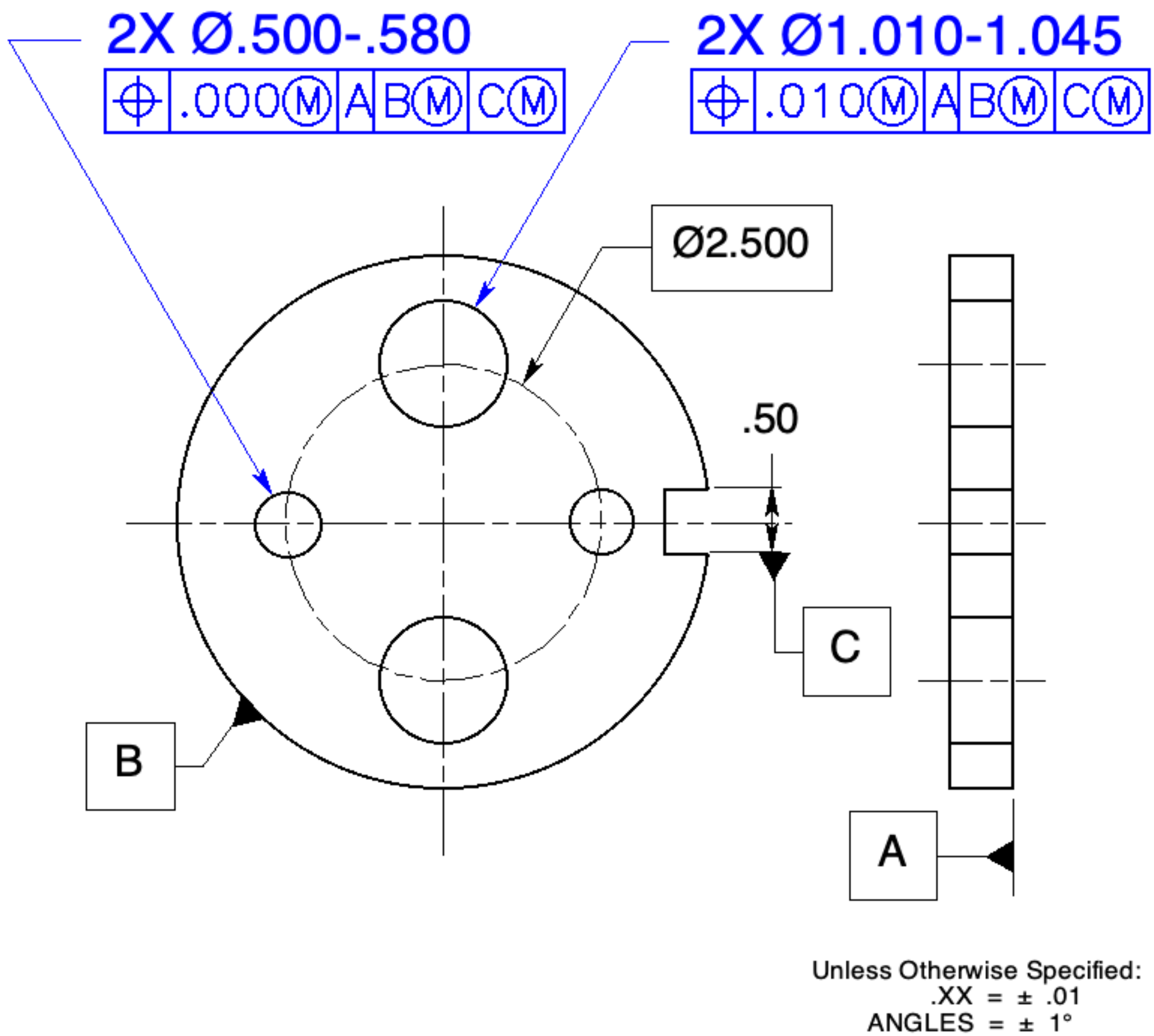


Figure 8-37 Multiple patterns of features: Probs. 16 through 18.

16. In Fig. 8-37, position the small holes with .000 tolerance at MMC and the large holes with .010 tolerance at MMC; locate them to the same datum features and in the same order of precedence. Use MMC and MMB wherever possible.

17. Must the hole patterns be inspected in the same setup or in the same gage? Explain?

Yes, they must be inspected to the same datum features. The large hole and small hole patterns are tied together by their datums features.

18. Can the requirement be changed, how?

Yes, place a note, SEPT REQT, under each feature control frame.

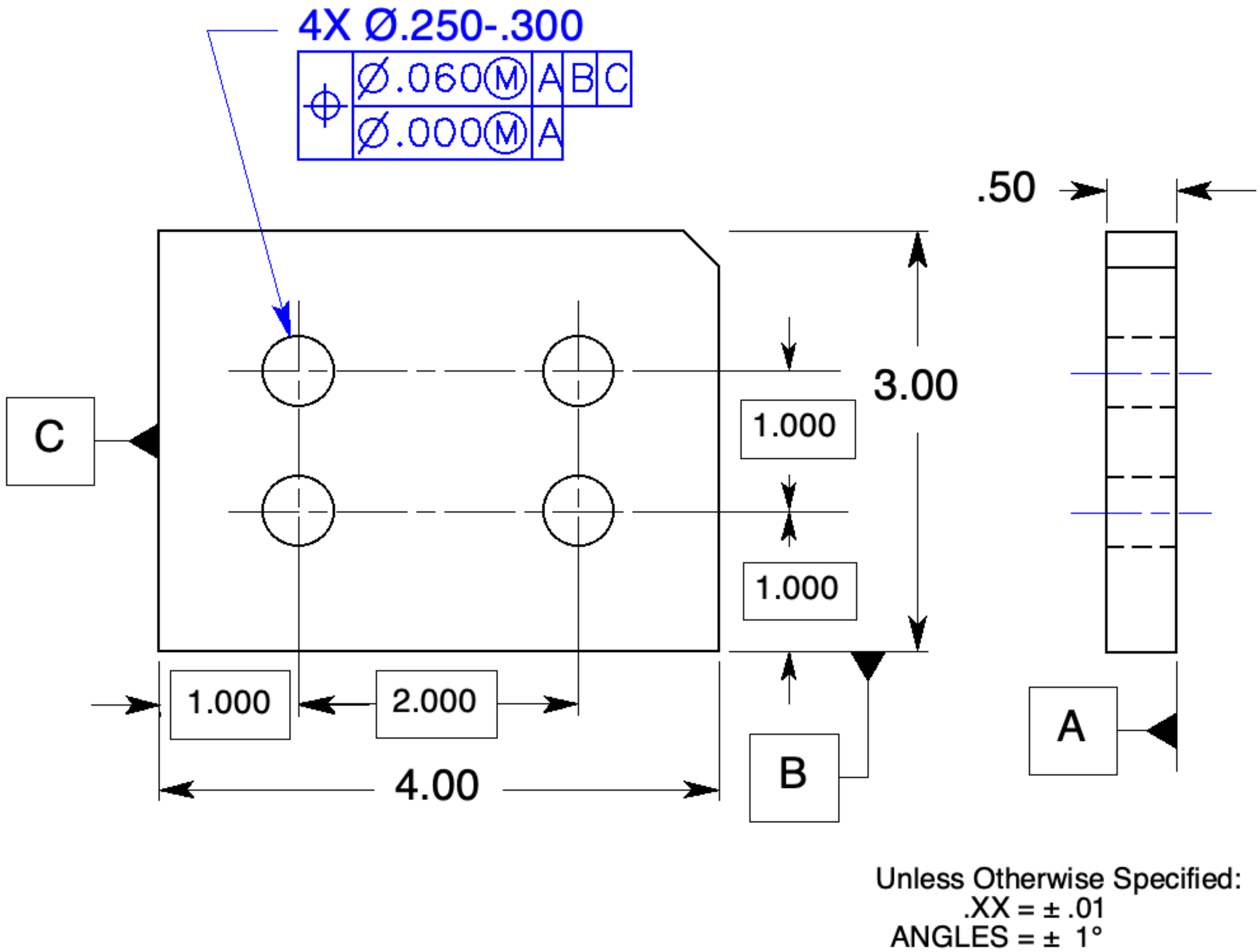
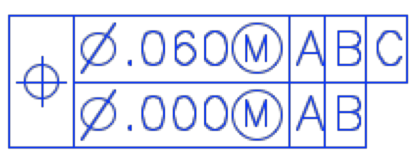


Figure 8-38 Composite positional tolerancing: Probs. 19 and 20.

19. Locate the pattern of clearance holes on the part in Fig. 8-38 with a tolerance of at least .060 in diameter at MMC to the datum features specified. This plate is required to assemble to the mating part with ¼-inch hex bolts as floating fasteners. Complete the geometric tolerance.

20. It has been determined that the hole pattern in Fig. 8-38 is required to remain parallel to datum feature B within the smaller tolerance. Draw the feature control frame that will satisfy this requirement.



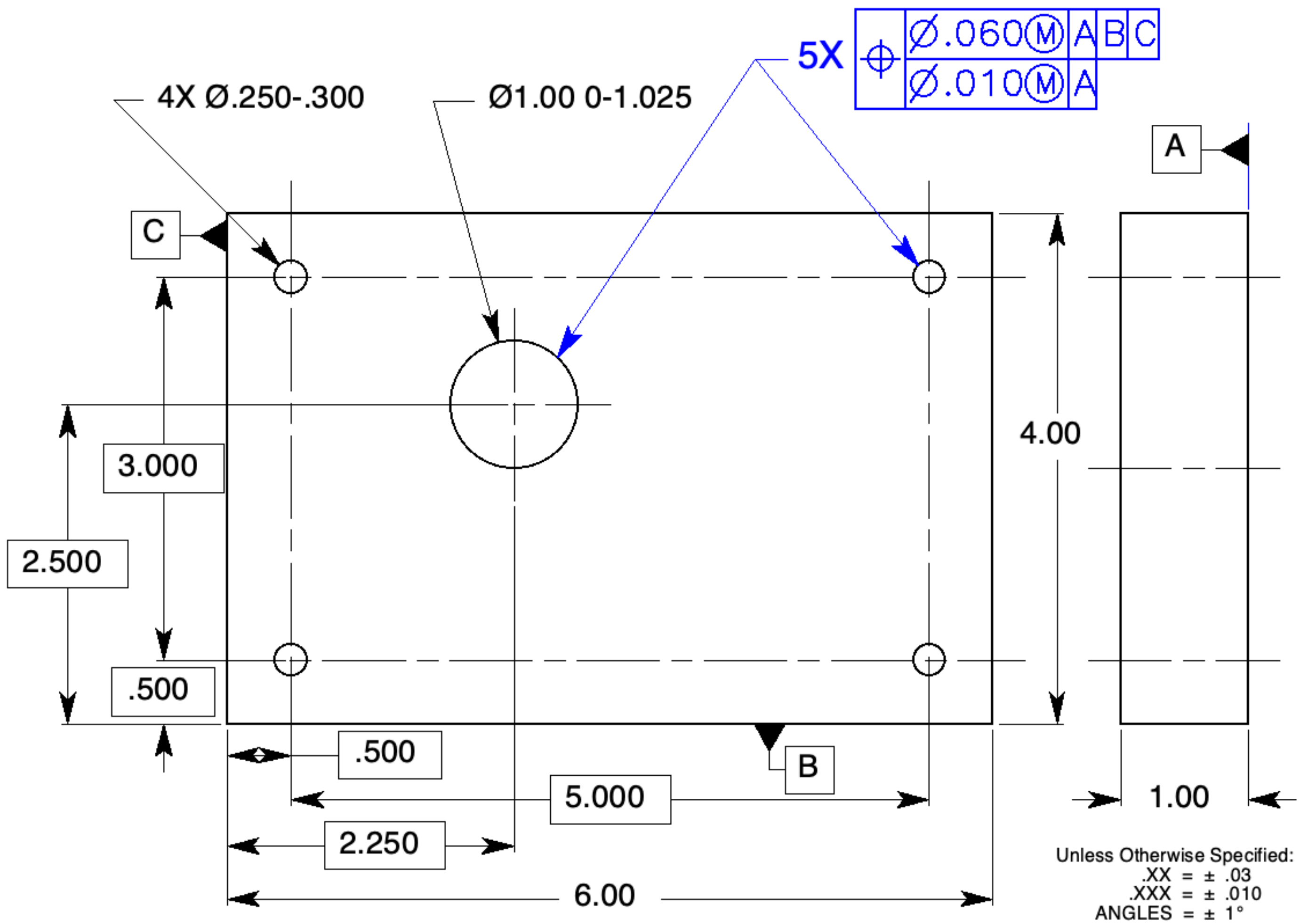


Figure 8-39 Composite positional tolerancing of holes with different sizes: Prob. 21.

21. Locate all five holes in Fig. 8-39 within a tolerance of .060 to the datum features specified. Also, locate all five holes to each other and perpendicular to datum feature A within a positional tolerance of .010. Use MMC and MMB wherever applicable.

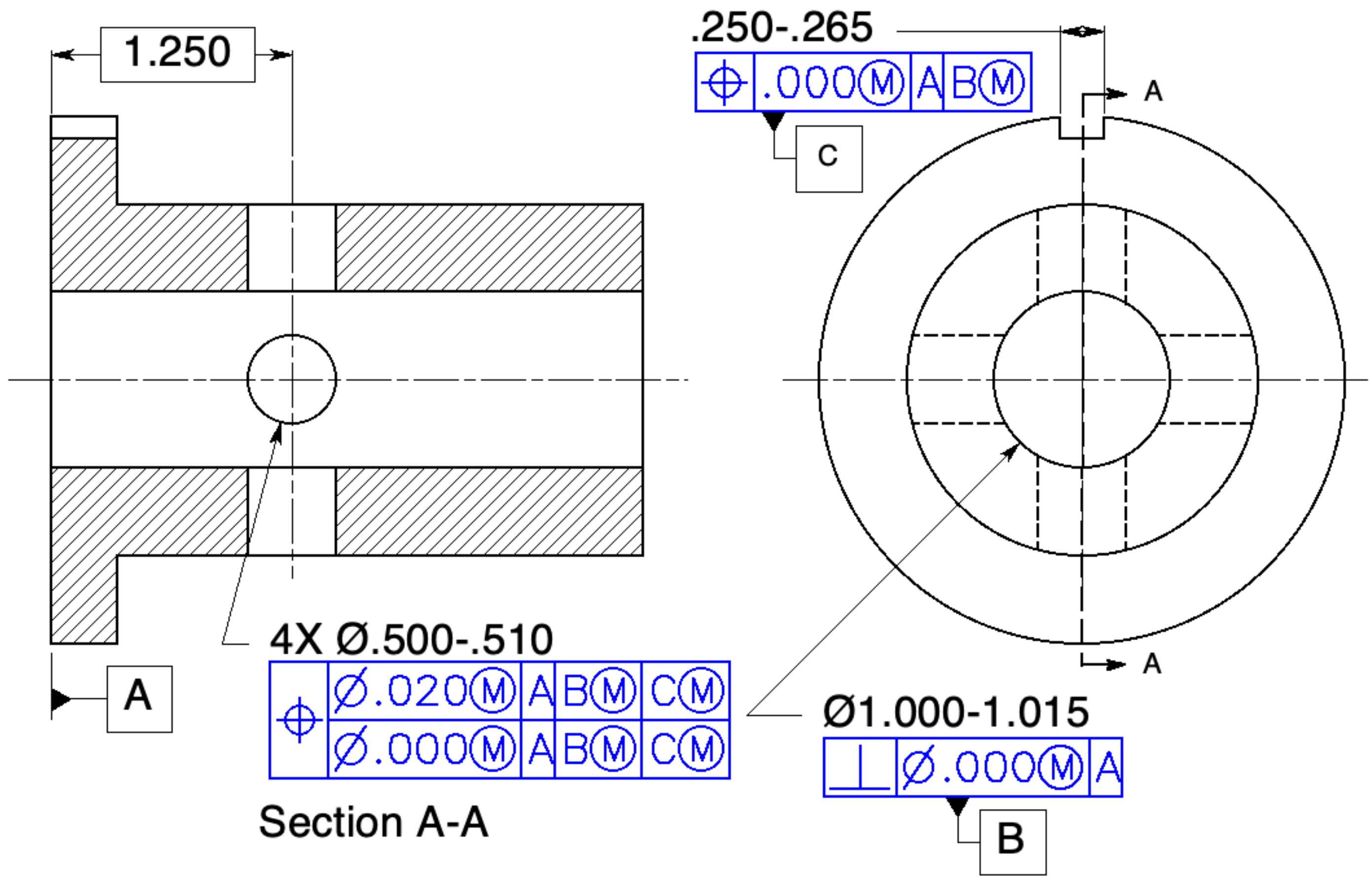


Figure 8-40 Composite positional tolerancing locating a radial hole pattern: Prob. 22.

22. Locate the ¼ inch keyway to datum features A and B on Fig. 8-40. Position the four-hole pattern within a tolerance of .020 to datum features A, B, and C. Refine the orientation of the four-hole pattern parallel to datum feature A, perpendicular to datum feature B at MMB, and parallel and perpendicular to the center plane of the keyseat within zero tolerance at MMC.

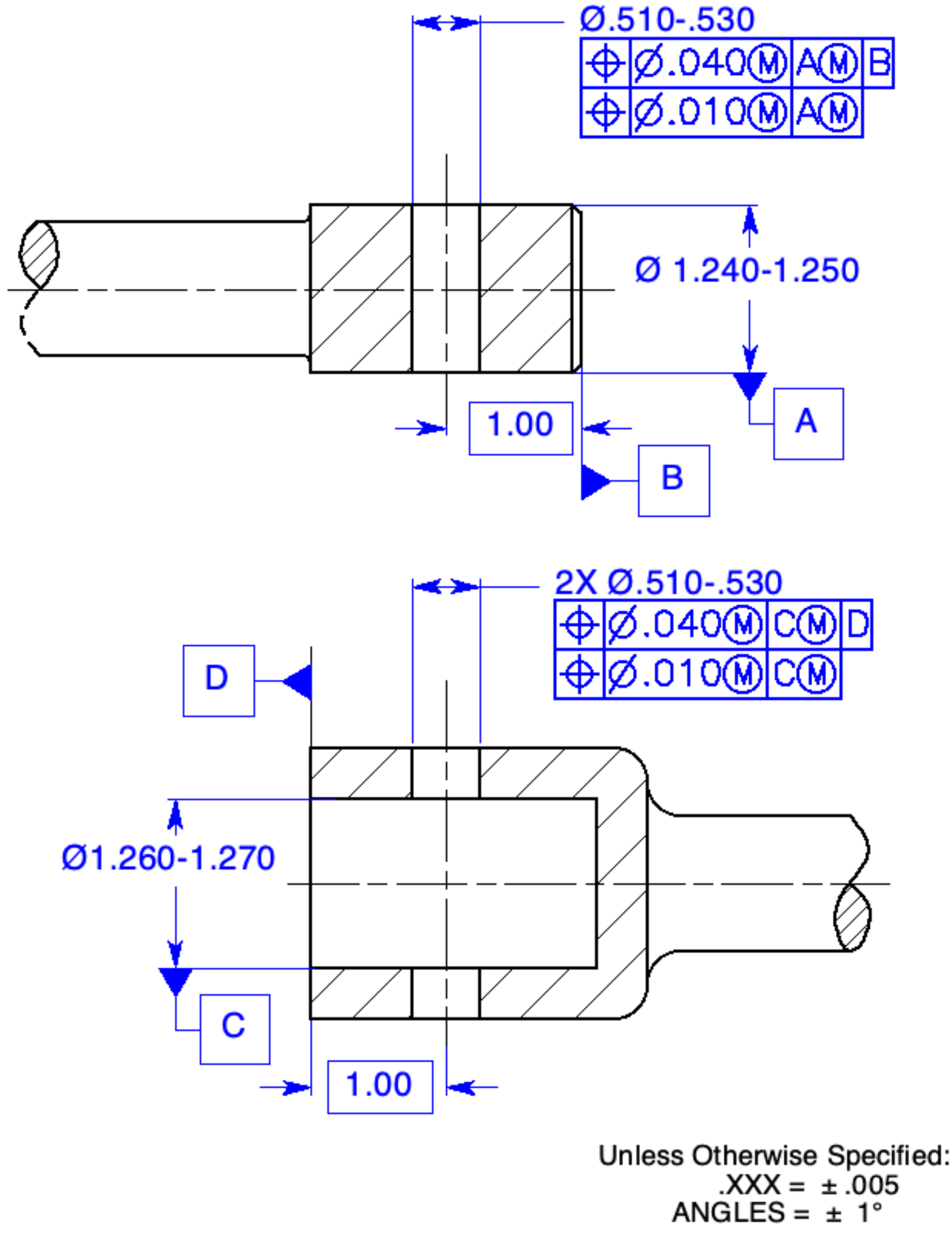


Figure 8-41 Multiple single-segment positional tolerancing to control holes: Prob. 23.

23. The inner and outer shafts in Fig. 8-41 will assemble every time. Control the location of the clearance holes for a ½ inch fastener with a multiple single-segment positional tolerance. Locate the holes to the end of each shaft with a tolerance of .040 Locate the holes to the axis of each shaft using the floating fastener formula. Specify MMC and MMB wherever possible.

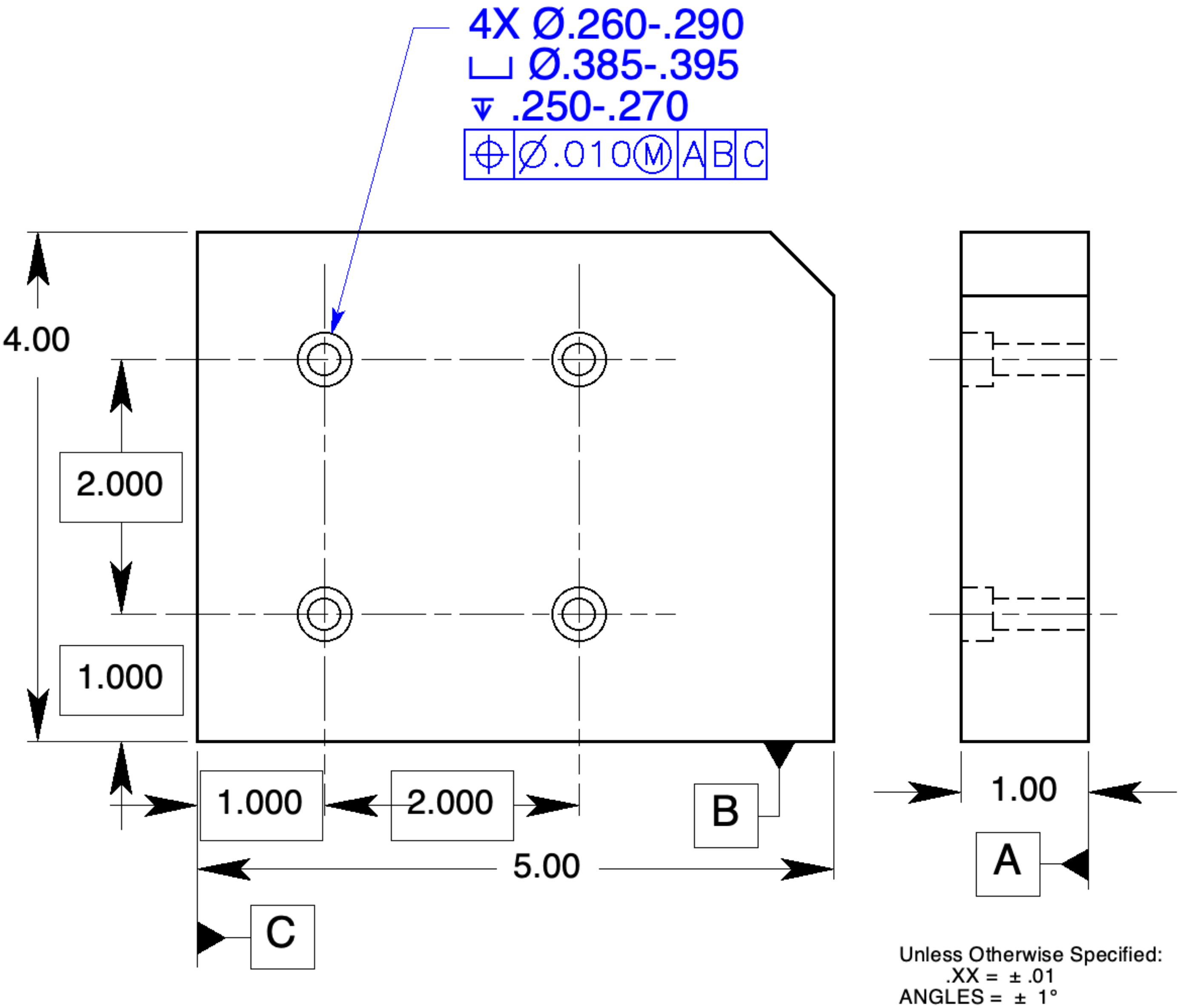
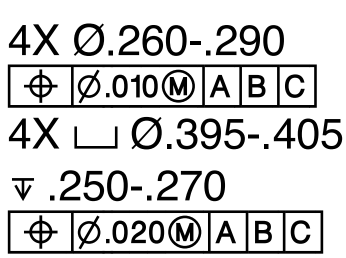


Figure 8-42 Controlling counterbores with positional tolerancing: Probs. 24 and 25.

24. Tolerance the holes and counterbores in Fig. 8-42 for four ¼-inch socket head cap screws. The cap screw head is a diameter of .365-.375, the height is .244-.250. Specify MMC and MMB wherever possible.

25. If the geometric tolerance for just the counterbores in Fig. 8-42 can be loosened to .020 at MMC instead of .010, draw the entire callout below.



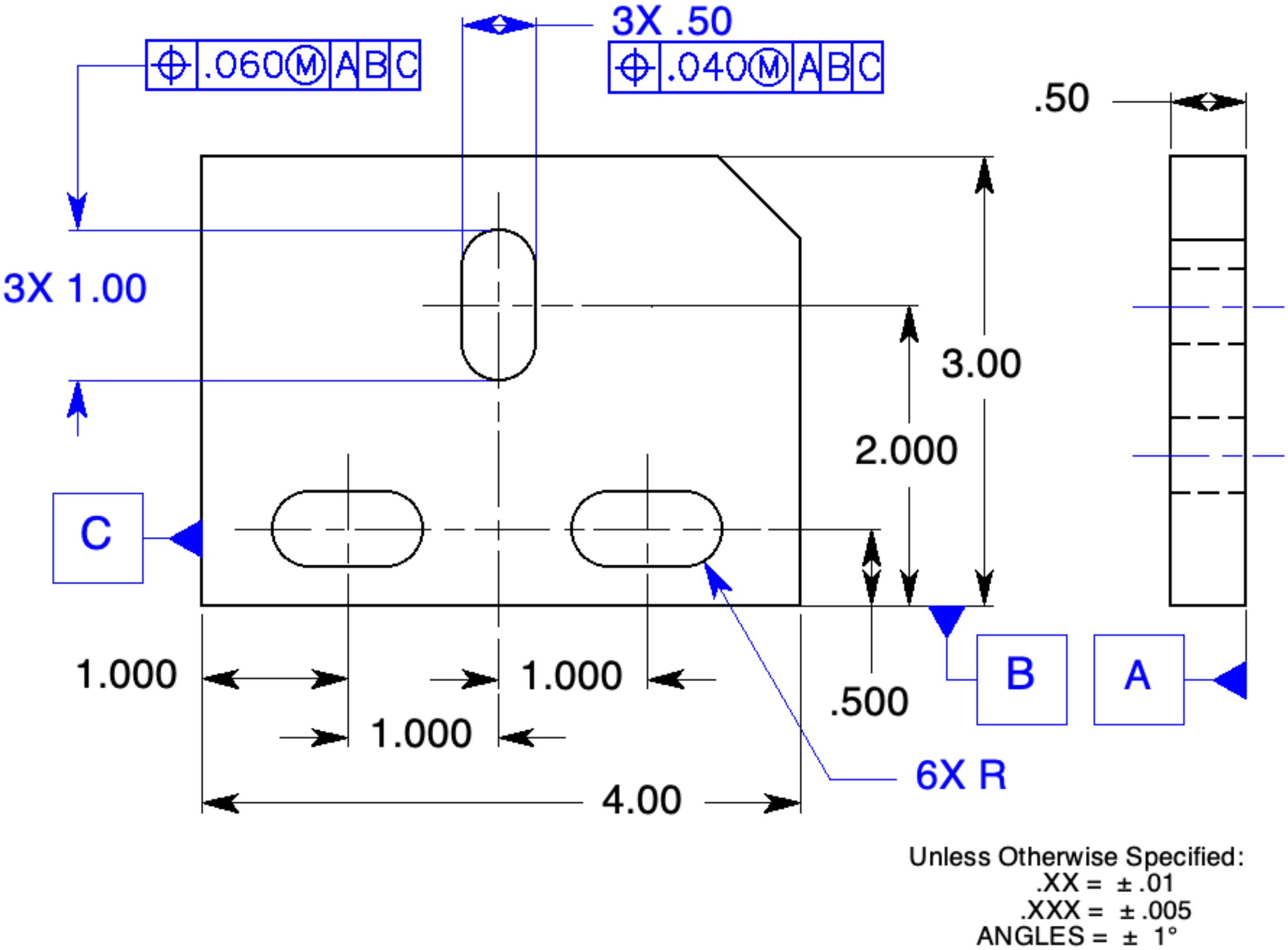


Figure 8-43 Controlling noncircular features with positional tolerancing: Prob. 26.

26. In Fig. 8-43, specify a geometric tolerance of .040 at MMC for the 1/2-inch direction and .060 at MMC for the 1-inch direction for the noncircular features.

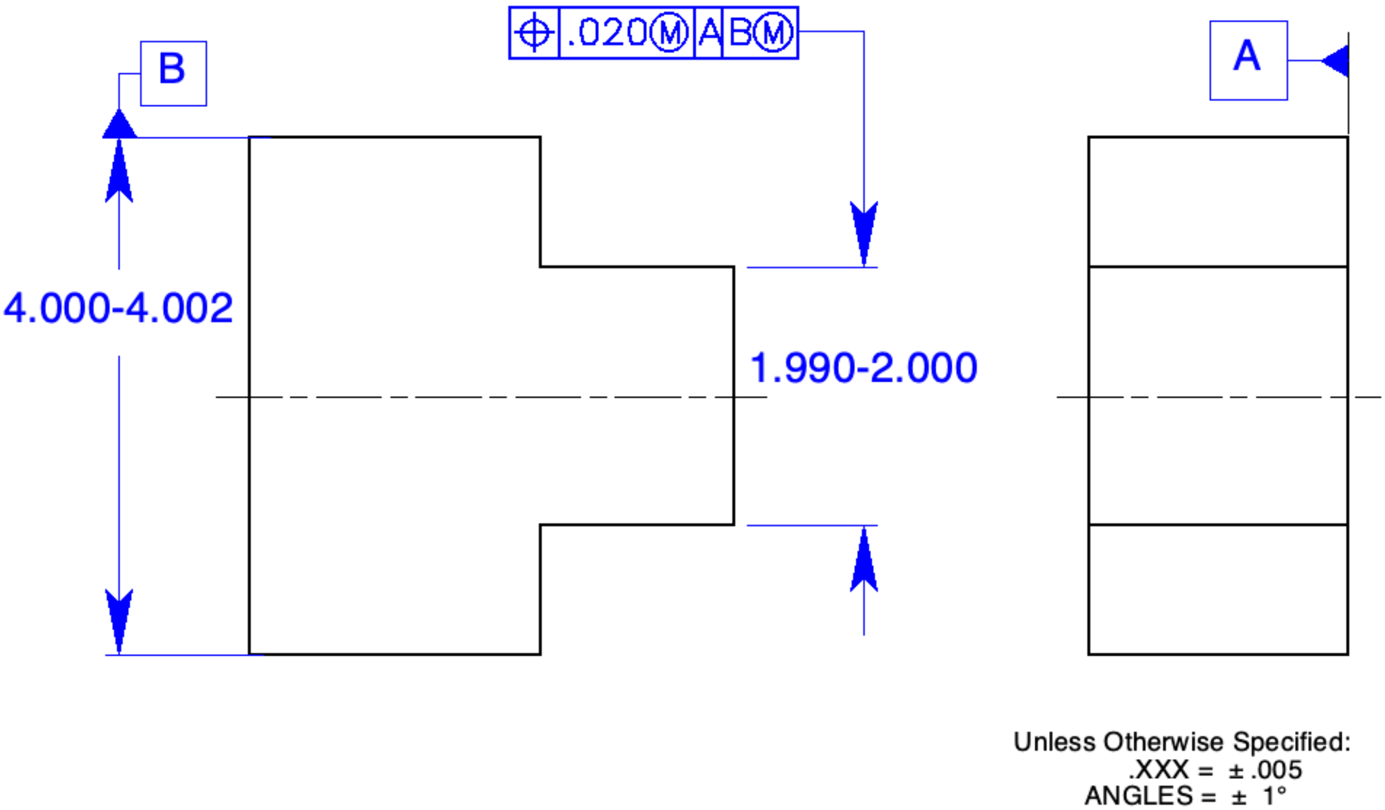


Figure 8-44 Controlling symmetrical features with positional tolerancing: Prob. 27.

27. In Fig. 8-44, control the symmetry of the 2-inch feature with respect to the 4-inch feature and perpendicular to datum feature A within a tolerance of .020. Use MMC and MMB wherever possible.

28. If the controlled feature in Fig. 8-44 happened to be produced at 1.995 and the datum feature produced at 4.000, what would be the total positional tolerance? ________ *.027* ________

Chapter 9

Position, Coaxiality

Chapter Review

Page 204

1. Coaxiality is that condition where the axes of two or more surfaces of revolution are *coincident.*
2. There is a misconception that centerlines or the tolerance block control the *coaxiality* between two cylinders.
3. The *position* control is the appropriate tolerance for coaxial surfaces of revolution that are cylindrical and require an MMC or an LMC.
4. A *cylindrical* tolerance zone is used to control the axis of a feature toleranced with a position control.
5. The tolerance of position to control coaxiality may apply at *MMC, LMC, and RFS* and the datum feature(s) may apply at *MMB, LMB, and RMB*.
6. The upper segment of a composite feature control frame controls the location of the hole pattern to the *location datums*.
7. The lower segment of a composite feature control frame controls the coaxiality of holes to *one another within the tighter tolerance*.
8. The smaller tolerance zone framework of a composite feature control frame with no datums may float *up and down, in and out, and at any angle to the datums within the larger tolerance zone*.
9. The position control, with no datum features, can be applied to two or more coaxial features controlling their *coaxiality* simultaneously within the specified tolerance.
10. A mating plug and socket will assemble every time if they are designed to their *virtual conditions*.

Problems

Page 205

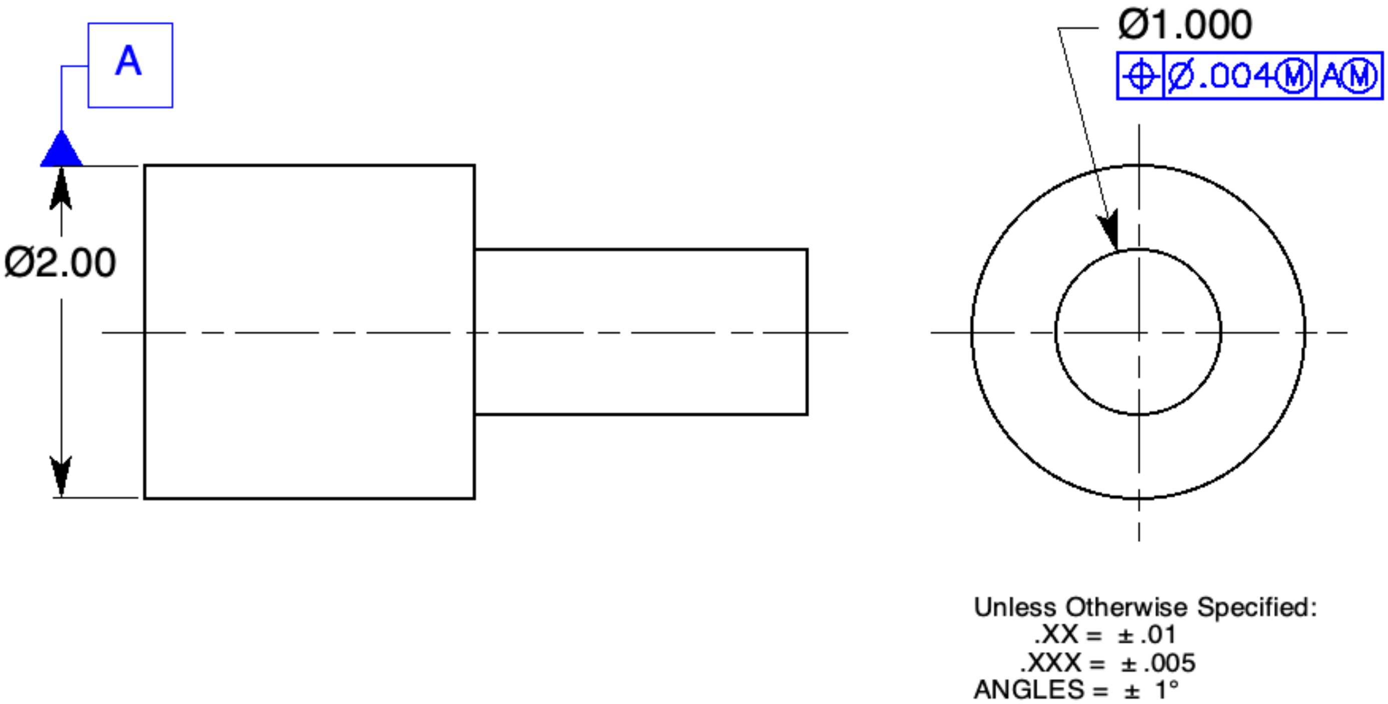


Figure 9-9 Controlling coaxiality with positional tolerancing: Probs. 1 through 3.

1. What controls the coaxiality of the two cylinders on the drawing in Fig. 9-9?
   The way the drawing in Fig. 9-7 is shown, nothing controls coaxiality.

2. On the drawing in Fig. 9-9, specify a coaxiality tolerance to control the 1.000 diameter feature within a cylindrical tolerance zone of .004 to the 2.00-diameter feature. Use MMC and MMB wherever possible.

3. Now that the feature control frame has been added to the drawing in Fig. 9-9, if the larger diameter is produced at 2.00 inches and the smaller diameter is produced at 1.000 inch, how much total coaxiality tolerance applies?

   Ø .019

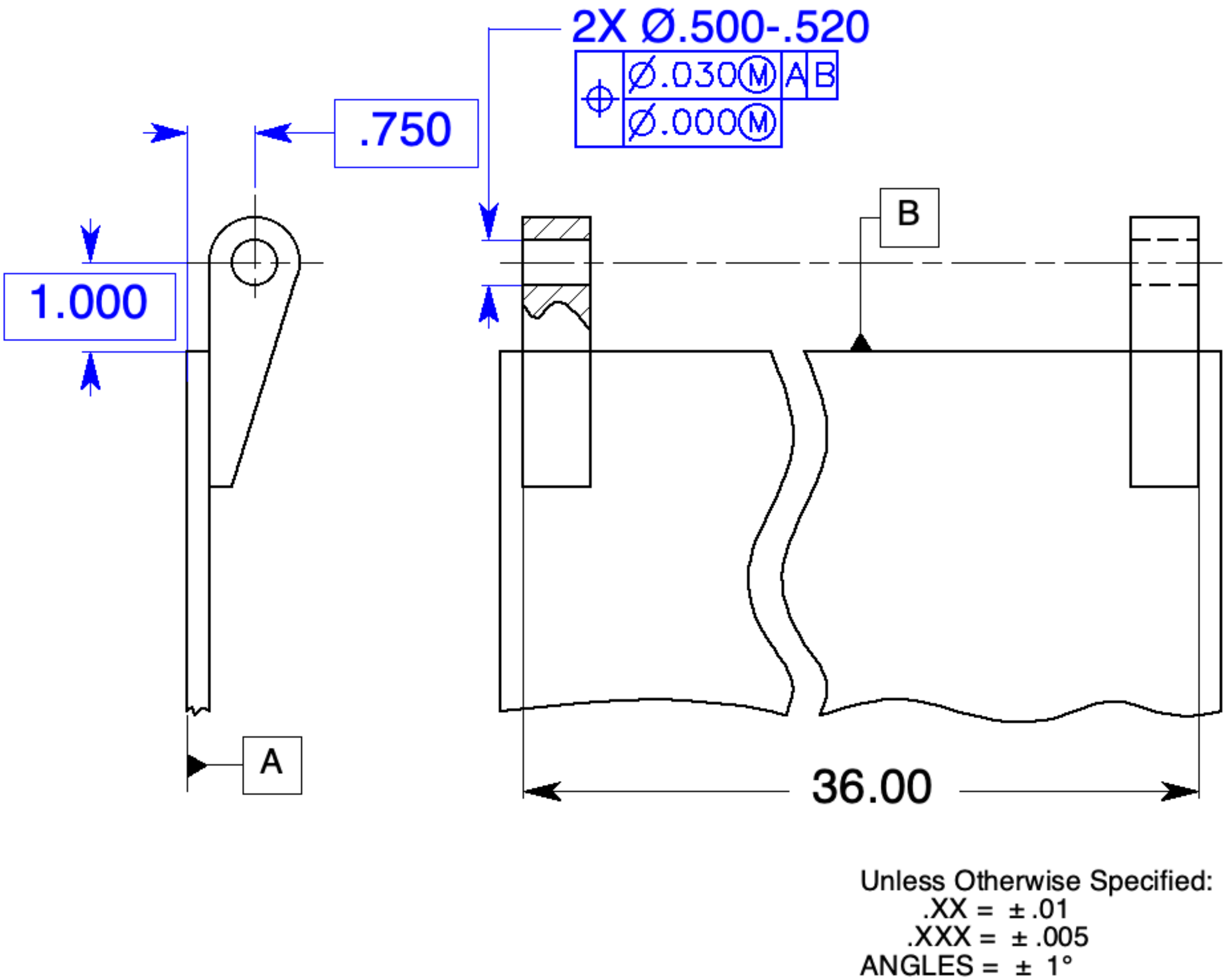


Figure 9-10 Controlling coaxiality with composite positional tolerancing: Prob. 4.

4. Locate the two holes in the hinge brackets within .030 at MMC to the datum features indicated and specify coaxiality to each other. They must be able to accept a .500 diameter hinge pin. Specify MMC and MMB wherever possible.

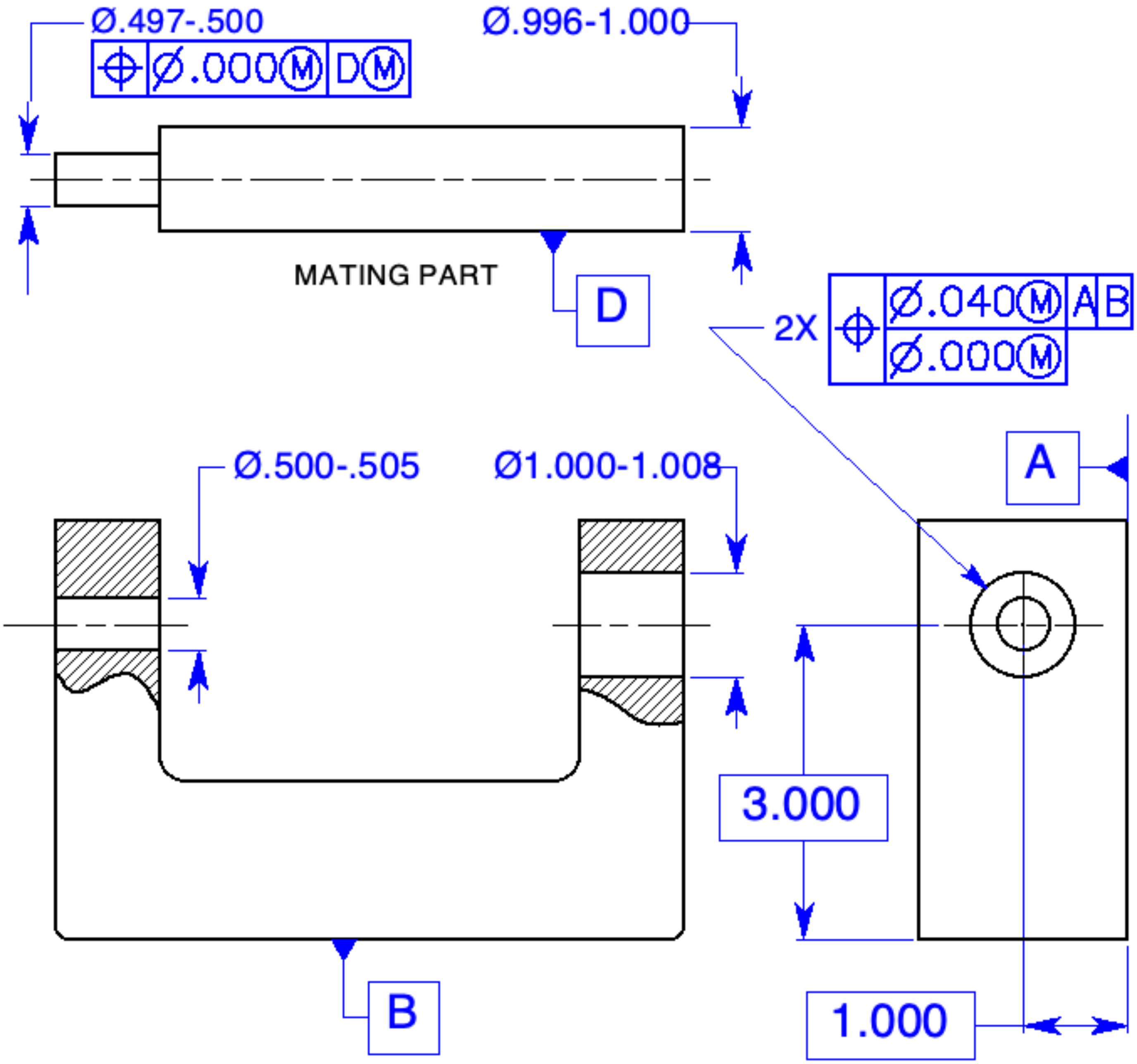


Figure 9-11 Controlling two coaxial holes and mating pin of different diameters: Prob. 5.

5. Locate the two coaxial holes parallel to the back and bottom surfaces of the part within a tolerance of .040 at MMC. Use the appropriate tolerance to control the coaxiality for the two mating parts. Specify MMC and MMB wherever possible.

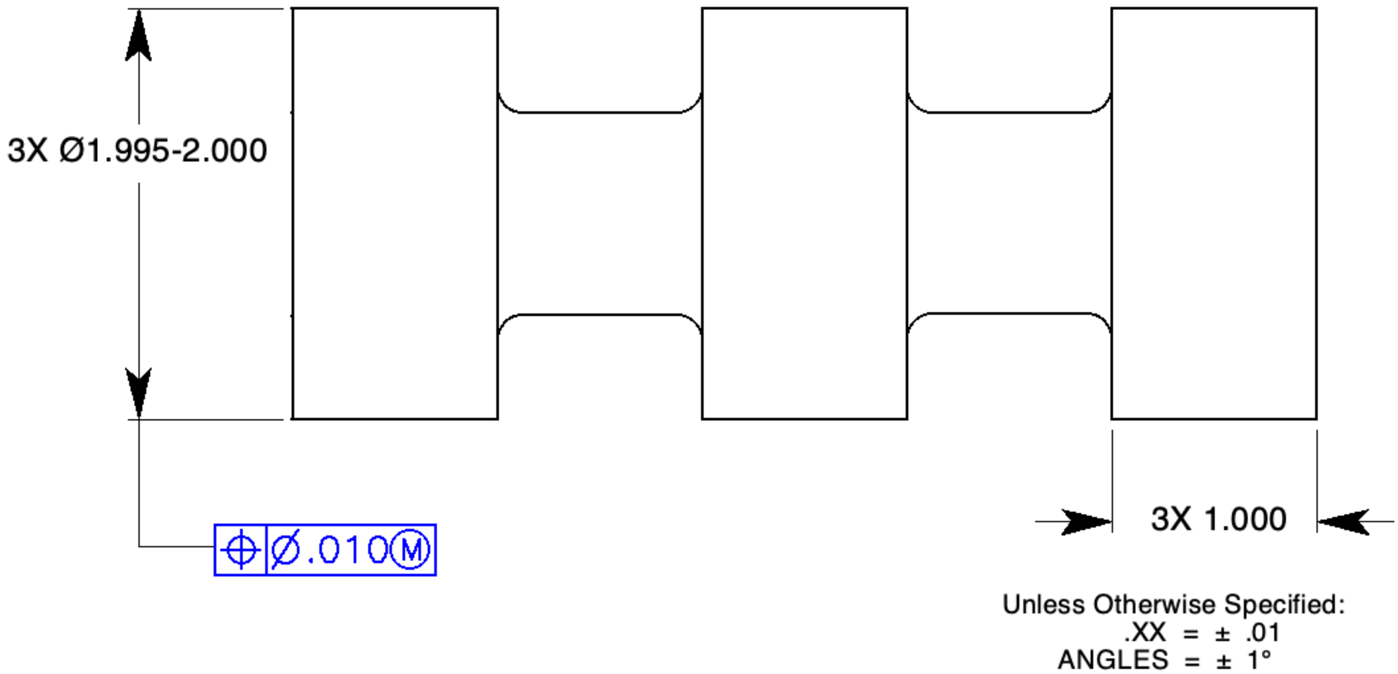


Figure 9-12 Controlling coaxiality with positional tolerancing without datum features: Prob. 6.

6. Control the three, 3-inch diameters coaxial to each other with a tolerance of .010 at MMC using the position control without datum features.

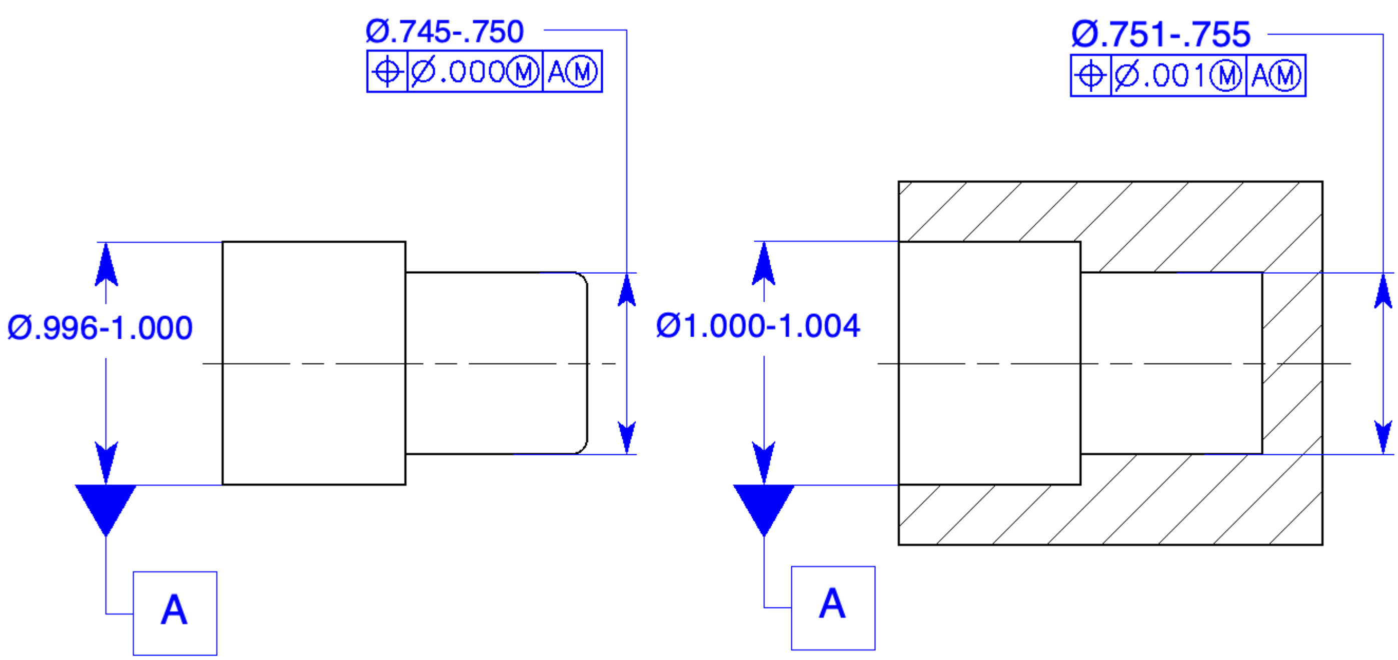


Figure 9-13 Controlling coaxiality for a plug and socket: Prob. 7.

7. Control the coaxiality of both parts in Fig. 9-13 so that they will always assemble. Specify MMC and MMB wherever possible.

	Plug		*Socket*
MMC	*.750*		*.751*
Geo. Tol.	*+ .000*		*– .001*
Virtual Condition	*.750*	=	*.750*

Chapter 10

Runout

Chapter Review

Page 217

1. Circular runout applies independently to each *circular element* on the surface of a feature either constructed around a datum axis or perpendicular to a datum axis as the part is rotated *360°* about its datum axis.
2. Where applied to surfaces of revolution, circular runout controls a combination of variations in *circularity and coaxiality*.
3. Total runout is a compound tolerance that provides control of all *surface elements* of a feature.
4. Total runout tolerance is applied simultaneously to all circular and profile measuring positions either *around* or *perpendicular to its datum axis* to its datum axis as the part is rotated *360°* about that datum axis.
5. Where applied to surfaces constructed around a datum axis, total runout controls a combination of surface variations such as *coaxiality, circularity, straightness, angularity, taper, and profile*
6. Where applied to surfaces at a 90° angle to the datum axis, total runout controls a combination of variations of *perpendicularity to the datum axis and flatness*

7. The runout feature control frame consists of *a runout symbol, the numerical tolerance, and at least one datum feature*.

8. In many cases, two functional *datum features* are used to support a rotating part.

9. The planar and cylindrical datum reference frame is quite a different requirement than the *common datum* reference frame.

10. Design requirements may make it necessary to restrict datum surface variations with respect to (other geometric controls) *straightness, flatness, circularity, cylindricity, and parallelism.*

11. It may be necessary to include a runout control for individual datum features on a *common datum feature reference.*

12. If two or more surfaces are controlled with a runout tolerance to a common datum reference, the worst-case runout between two surfaces is the *sum of the two individual runout tolerances*.

13. If two features have a specific relationship between them, one should be *toleranced directly to the other and not through a common datum axis*.

14. Multiple leaders directed from a runout feature control frame may be specified without *affecting the runout tolerance.*

Problems

Page 218

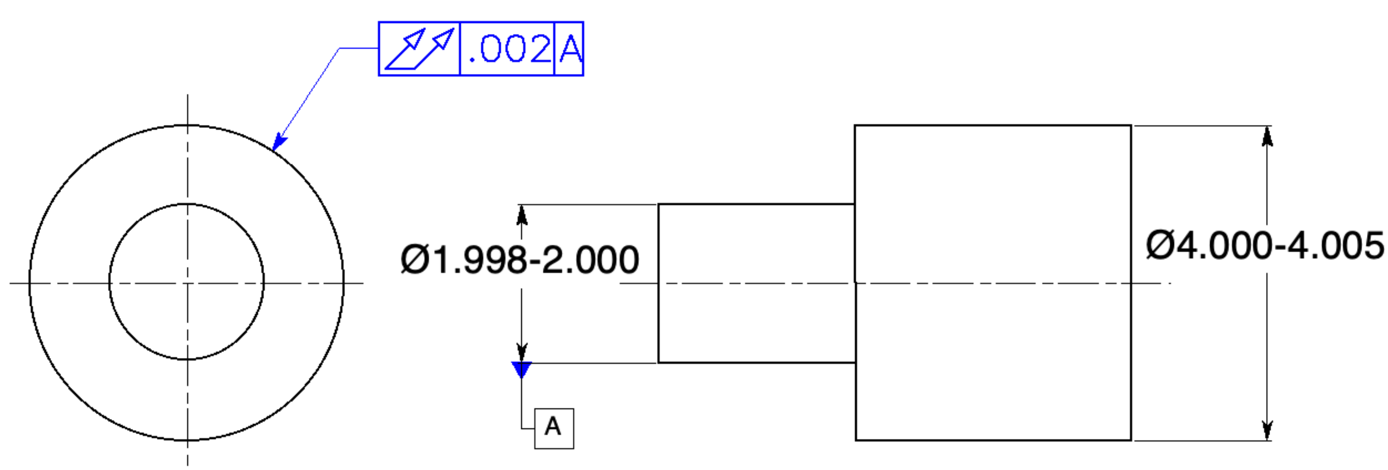


Figure 10-8 Controlling a coaxial feature with a runout control: Prob. 1.

1. On the part in Fig.10-8, control the four-inch diameter with a total runout tolerance of .002 to the 2-inch diameter.

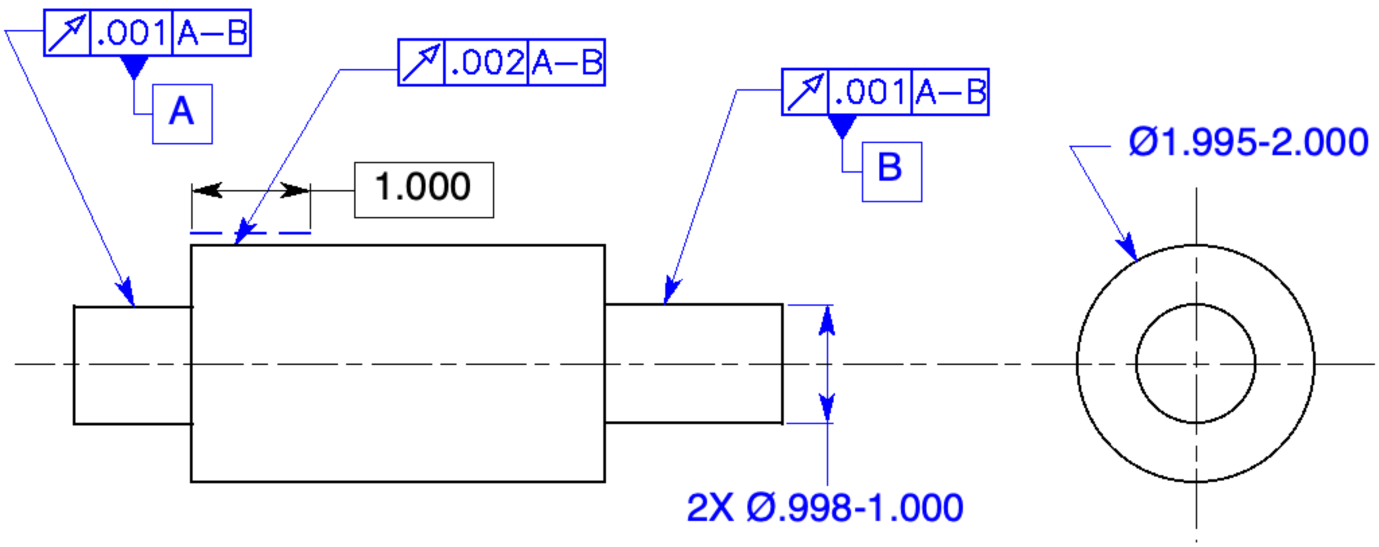


Figure 10-9 Controlling a partial runout: Prob. 2.

2. On the drawing in Fig. 10-9, specify a circular runout tolerance of .002 controlling the two-inch diameter to both of the 1-inch diameters. This control is a partial runout tolerance one inch long, starting from the left end of the two-inch cylinder. Also, specify a circular runout tolerance of .001 for each of the 1-inch diameters.

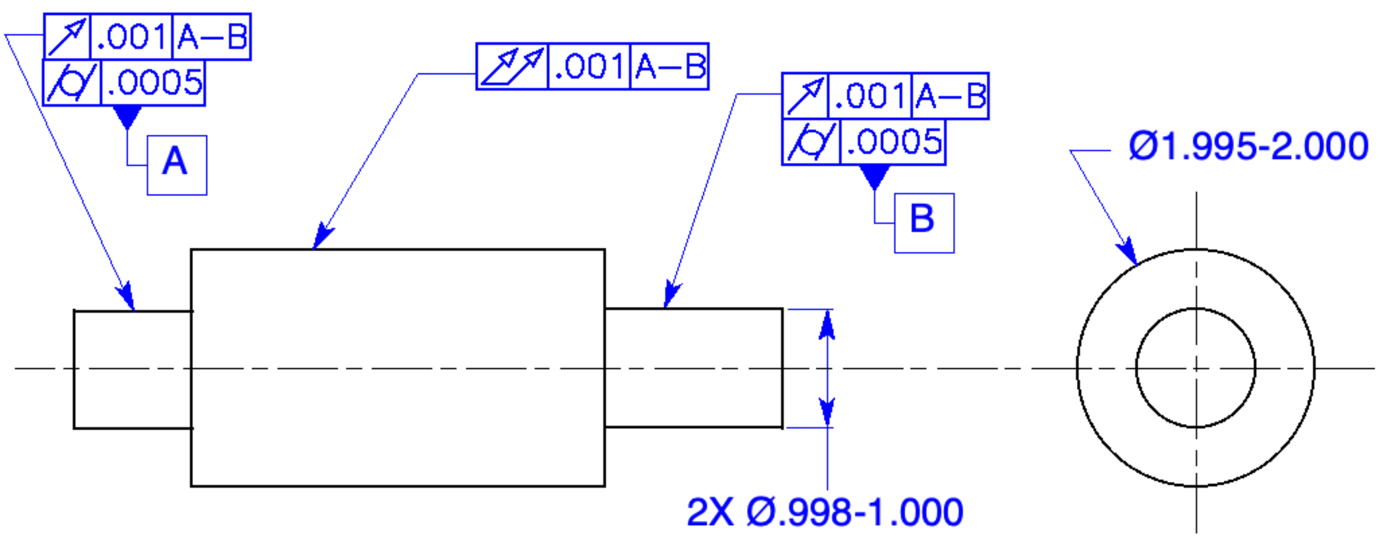


Figure 10-10 Datum features toleranced with a cylindricity tolerance: Prob. 3.

3. Tolerance the two-inch diameter in Fig. 10-10 with a total runout tolerance of .001 to both of the one-inch diameter shafts. Tolerance each one-inch diameter shaft with a circular runout tolerance of .001 and a cylindricity tolerance of .0005.

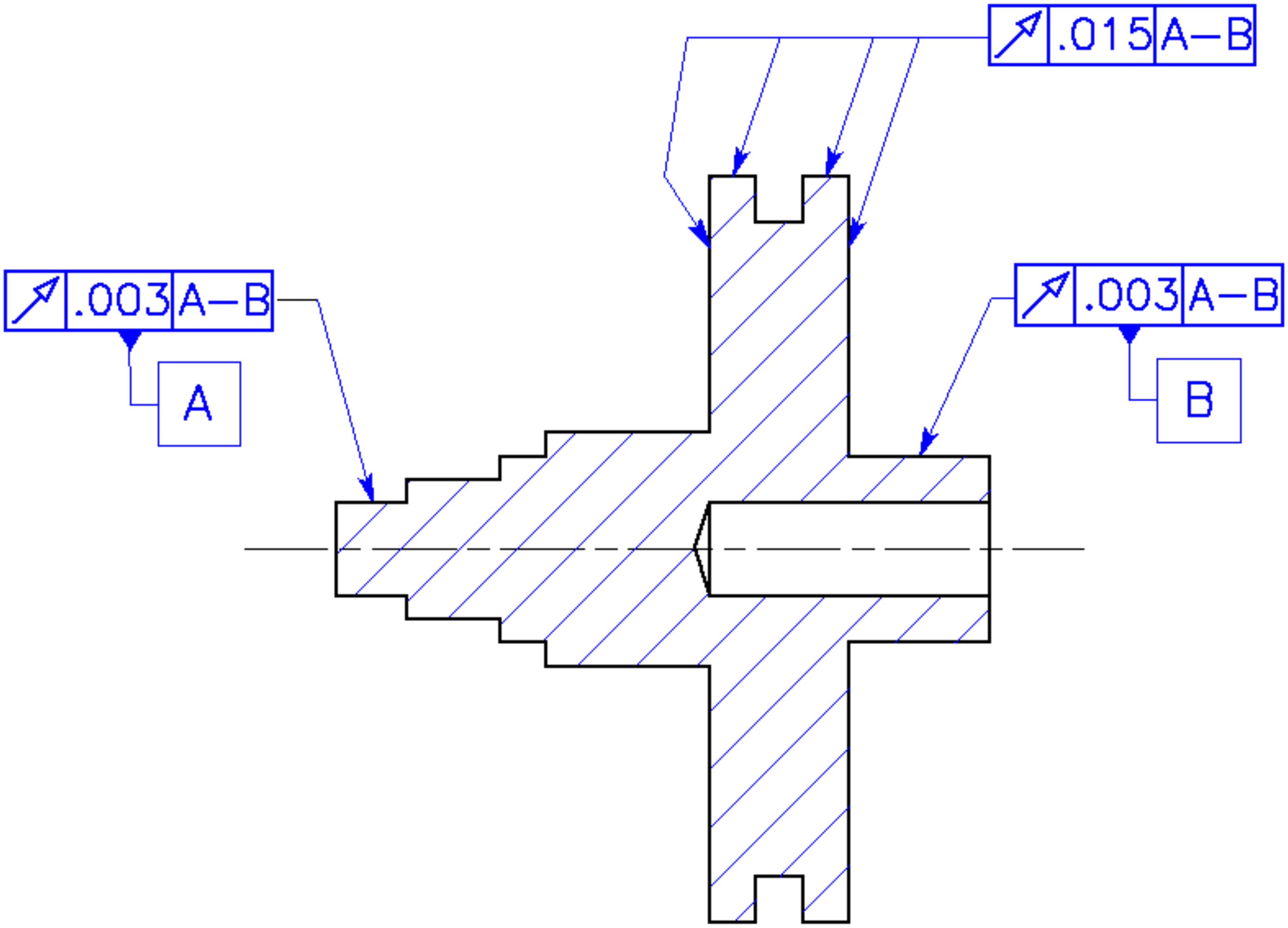


Figure 10-11 Multiple features tolerance with one feature control frame: Probs. 4 and 5.

4. In Fig. 10-11, which datum feature, A or B, takes precedence?

   Datum feature A is no more important than datum feature B, and datum feature B is no more important than datum feature A.

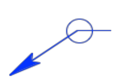
5. What is the worst possible runout tolerance between datum feature A and datum feature B in Fig. 10-11? ______*.006*______

Chapter 11

Profile

Chapter Review

Page 236

1. Profile of a line is the *outline*
   of an object in a plane as the plane passes through the object.
2. Profile of a surface is the result of *projecting the profile of an object on a plane*.
   or taking cross sections through the object at various intervals.
3. The true profile may be dimensioned with what kind of dimensions?
   With basic size dimensions, basic coordinate dimensions, basic radii, basic angular dimensions, formulas, or undimensioned drawings
4. The feature control frame is directed to the profile surface with a leader or an extension line.
5. What symbols do not apply in the tolerance section of profile feature control frames?
   the diameter symbol and material condition modifiers
6. The unequally disposed profile symbol, the circle U, indicates that the profile of a surface tolerance applies *unequally about the true profile.*
7. The tolerance following the circle U indicates the amount of the tolerance zone *allowing material to be added* to the true profile. the true profile.
8. Where a profile tolerance applies all around the profile of a part, the *"all around" symbol* is specified.
9. Draw the "all around" symbol. 
10. If the profile is to extend between two points, the points are *labeled*
    and a note using the *between symbol* is placed beneath the feature control frame.
11. Draw the between symbol. 

12. If a part is to be controlled with a profile tolerance over the entire surface of the part, the *"ALL OVER"* ______ symbol is specified.

13. Profile tolerances ______ *may or may not* ______ have datum features.

14. The profile of a surface control usually requires a datum feature(s) to properly ______ *orient and locate the surface* ______.

15. Datum features are generally ______ *not used* ______ for profile of a line where only ______ *the cross section* ______ is being controlled.

16. If the design requires a smaller radius than the radius allowed by the profile tolerance, a note such as, *"R.015 MAX" or, "ALL CORNERS R.015 MAX,"* ______ is directed to the radius with a ______ *leader* ______.

17. The profile tolerance may be combined with other ______ *geometric tolerances* ______ to ______ *refine* ______ certain aspects of a surface.

18. Coplanarity is the condition ______ *of two or more* ______ surfaces having all ______ *elements in one plane* ______.

19. Coplanarity is toleranced with the profile of a surface feature control frame, connected with a ______ *leader* ______, to an ______ *extension line* ______ connecting the coplanar surfaces.

20. The number of coplanar surfaces followed by an ______ *X* ______ precedes the ______ *feature control frame.*

21. A profile tolerance may be used to control two or more stepped ______ *surfaces* ______ that are required to be flat and parallel within a specific tolerance.

22. Conicity may be controlled with a ______ *profile tolerance* ______.

23. Composite profile tolerancing is very similar to ______ *composite positional tolerancing* ______.

24. The upper segment of a composite profile feature control frame called ______ *the profile locating control* ______ governs the *location relationship between the datum features and the profile* ______.

25. The lower segment, referred to as the ______ *profile refinement control* ______, is a smaller tolerance than the profile locating control and governs ______

the size, form, and orientation relationship of the profile.

26. The feature profile must fall inside *both profile tolerance zones*.

27. Datum features in the lower segment of a composite feature control frame must satisfy two conditions:

    They must repeat the datums in the upper segment.

    And they only control orientation.

28. A second datum feature may be repeated in the lower segment of the composite feature control frame.

    Both datum features only control *orientation*.

29. The lower segment of a multiple single-segment profile feature control frame acts just like

    any other profile control.

30. The upper segment of a multiple single-segment profile feature control frame allows the smaller tolerance zone to *translate* relative to the datum feature not repeated in the lower segment within the larger tolerance.

Problems

Page 238

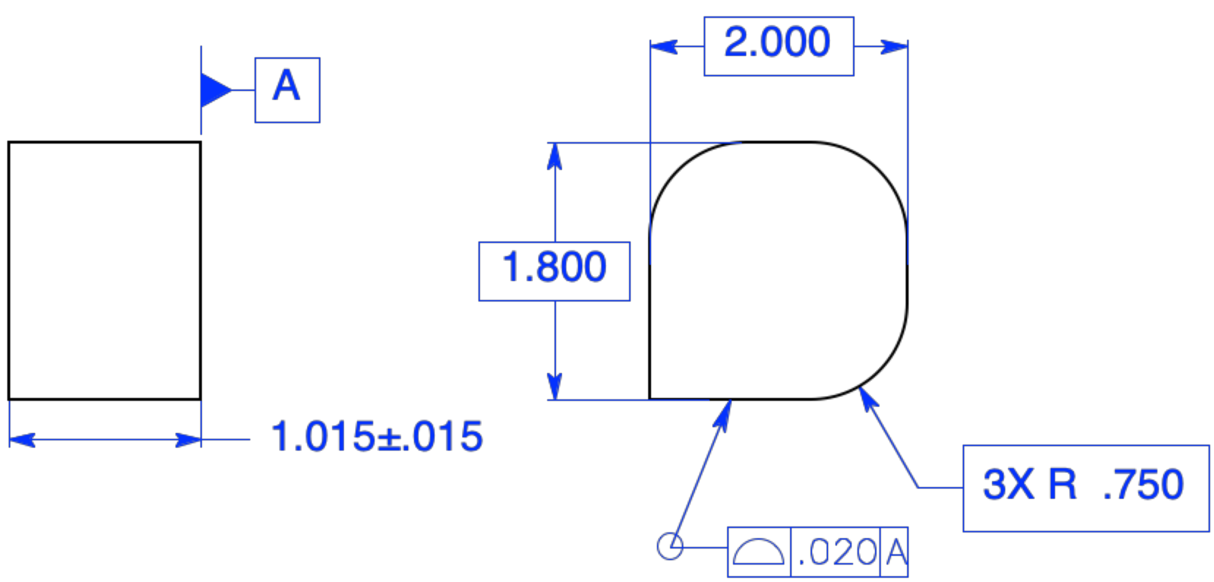


Figure 11-18 Controlling the profile of a surface all around: Prob. 1.

1. Specify a profile of a surface tolerance of .020, perpendicular to datum feature A, and all around the part in Fig. 11-18.

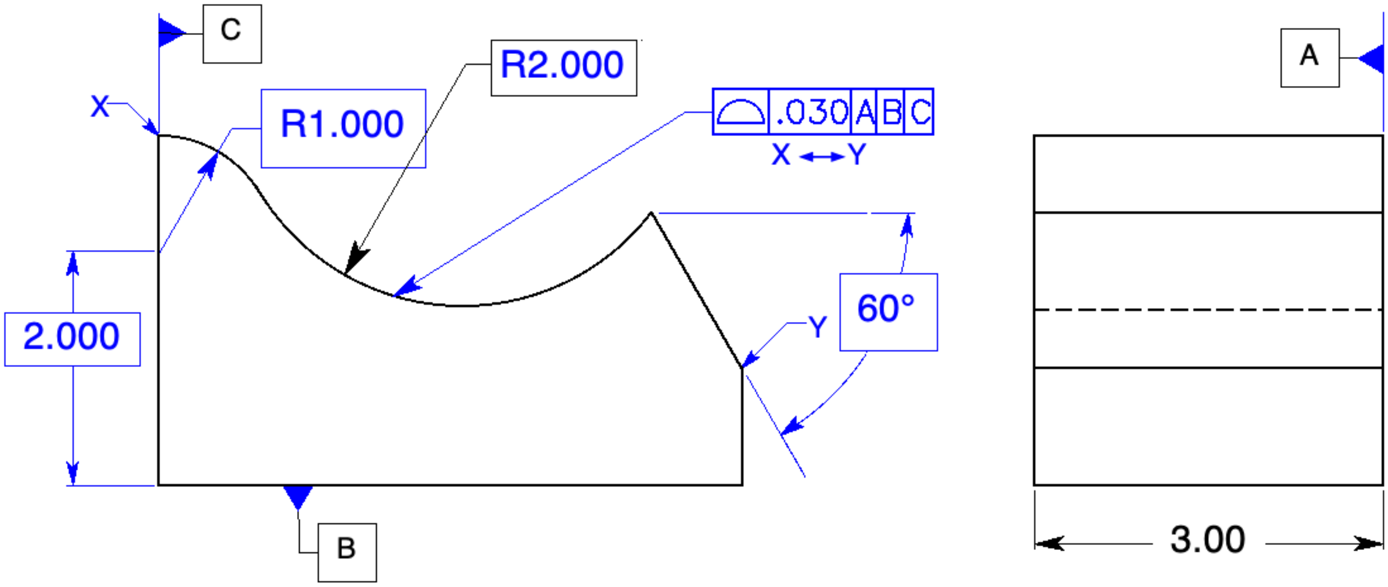


Figure 11-19 Controlling the profile of a surface between two points: Prob. 2.

2. Control the top surface between points X and Y in Fig. 11-19, specify a profile tolerance of .030, located to datum features A, B, and C.

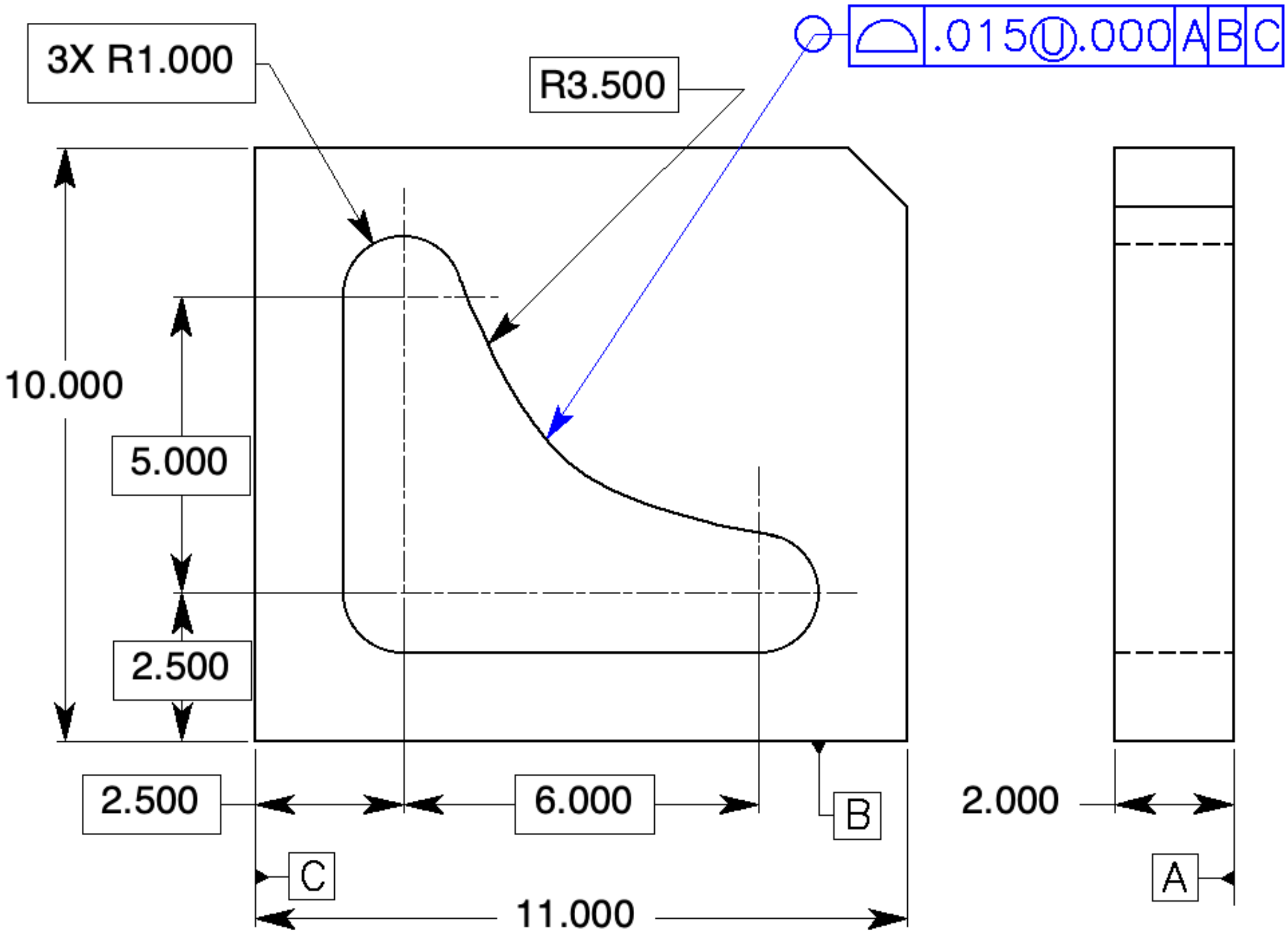


Figure 11-20 Control the location of a profile: Prob. 3.

3. Control the entire surface of the die cavity in Fig. 11-20 to the datum features indicated within a tolerance of .015 in the direction that would remove material from the true profile.

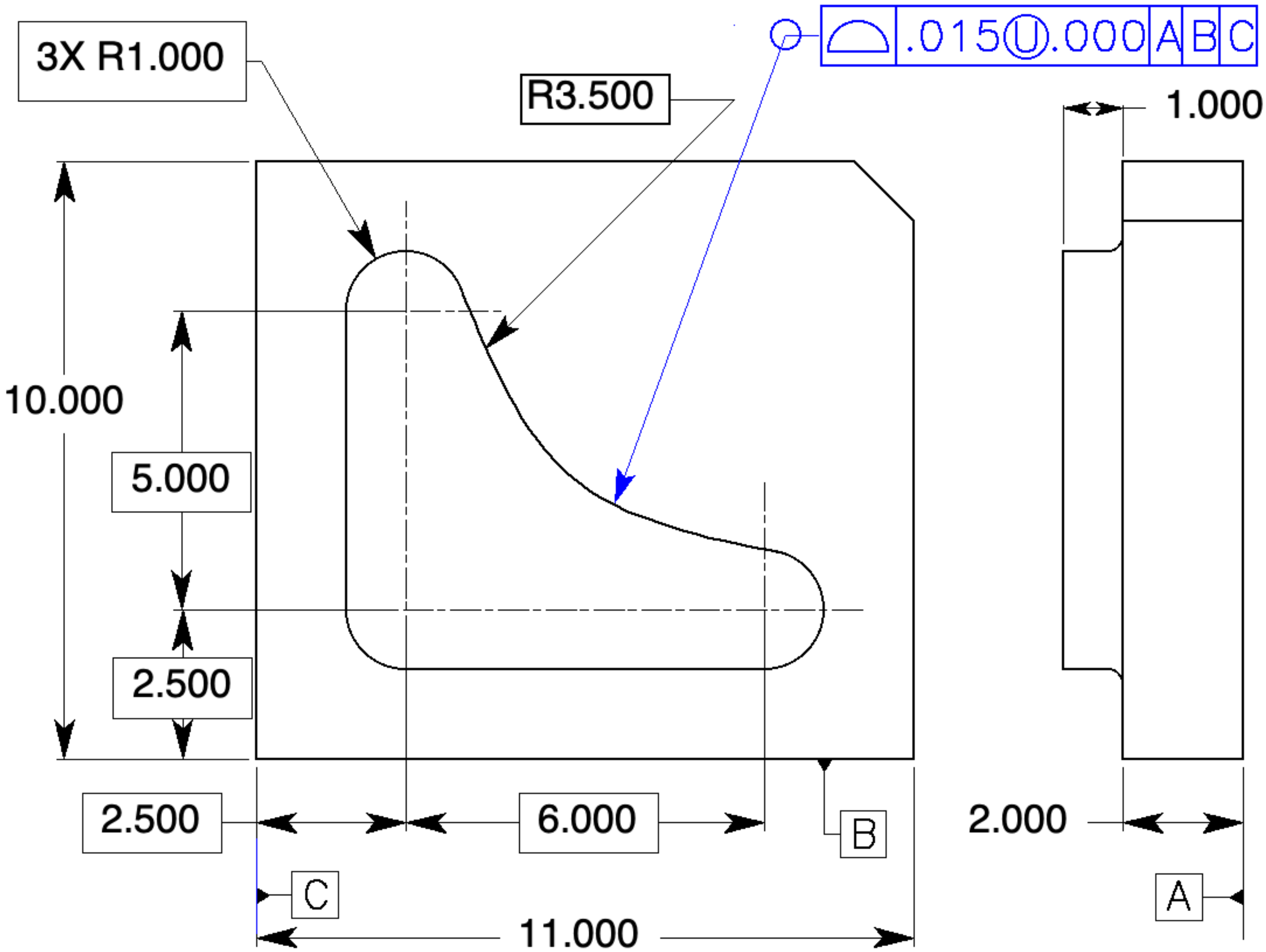


Figure 11-21 Locating a mating profile: Prob. 4.

3. Control the entire surface of the punch in Fig. 11-21 to the datum features indicated within a tolerance of .015 in the direction that would remove material from the true profile.

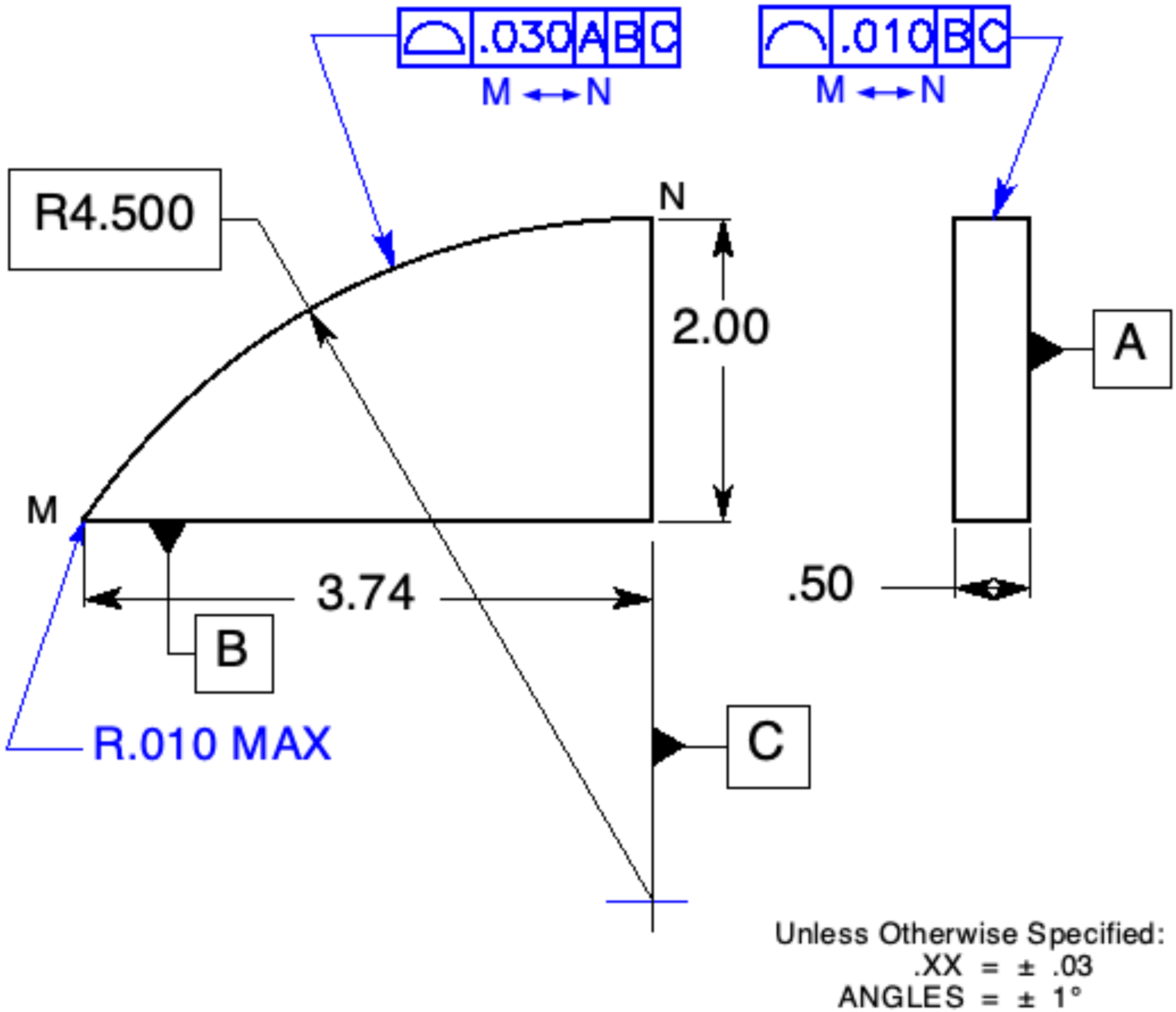


Figure 11-22 Control a profile with combined tolerances: Prob. 5.

5. Use the profile of a surface control to tolerance the curve in Fig. 11-22 within .030 between points M and N. Each line element in the profile must be parallel to datum features B and C within .010. The point at M may not exceed a radius of .010.

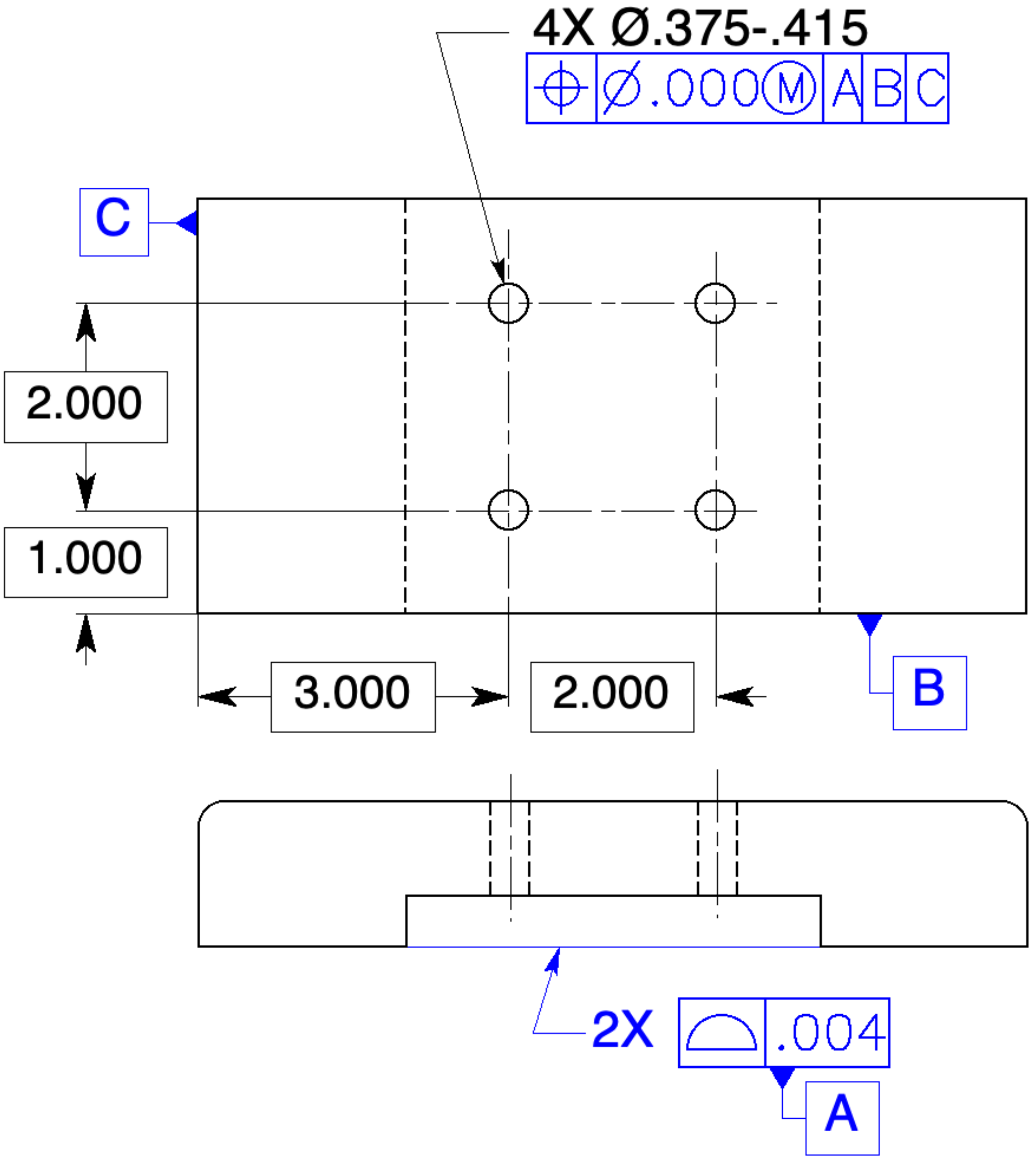


Figure 11-23 Controlling coplanarity with the profile of a surface: Prob. 6.

6. The primary datum feature is the two lower coplanar surfaces in Fig. 11-23. Specify the primary datum feature to be coplanar within .004.

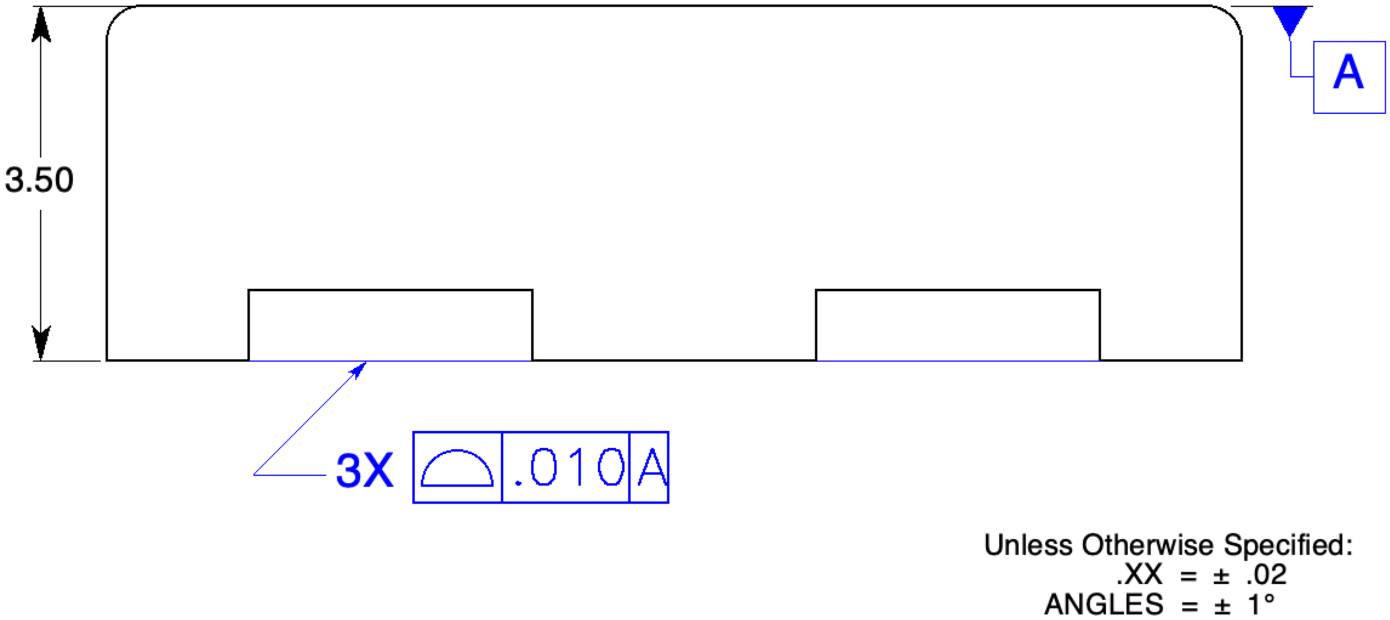


Figure 11-24 Controlling coplanarity to a datum feature: Prob. 7.

7 Control the bottom three surfaces in Fig. 11-24 coplanar and parallel to datum feature A within .010.

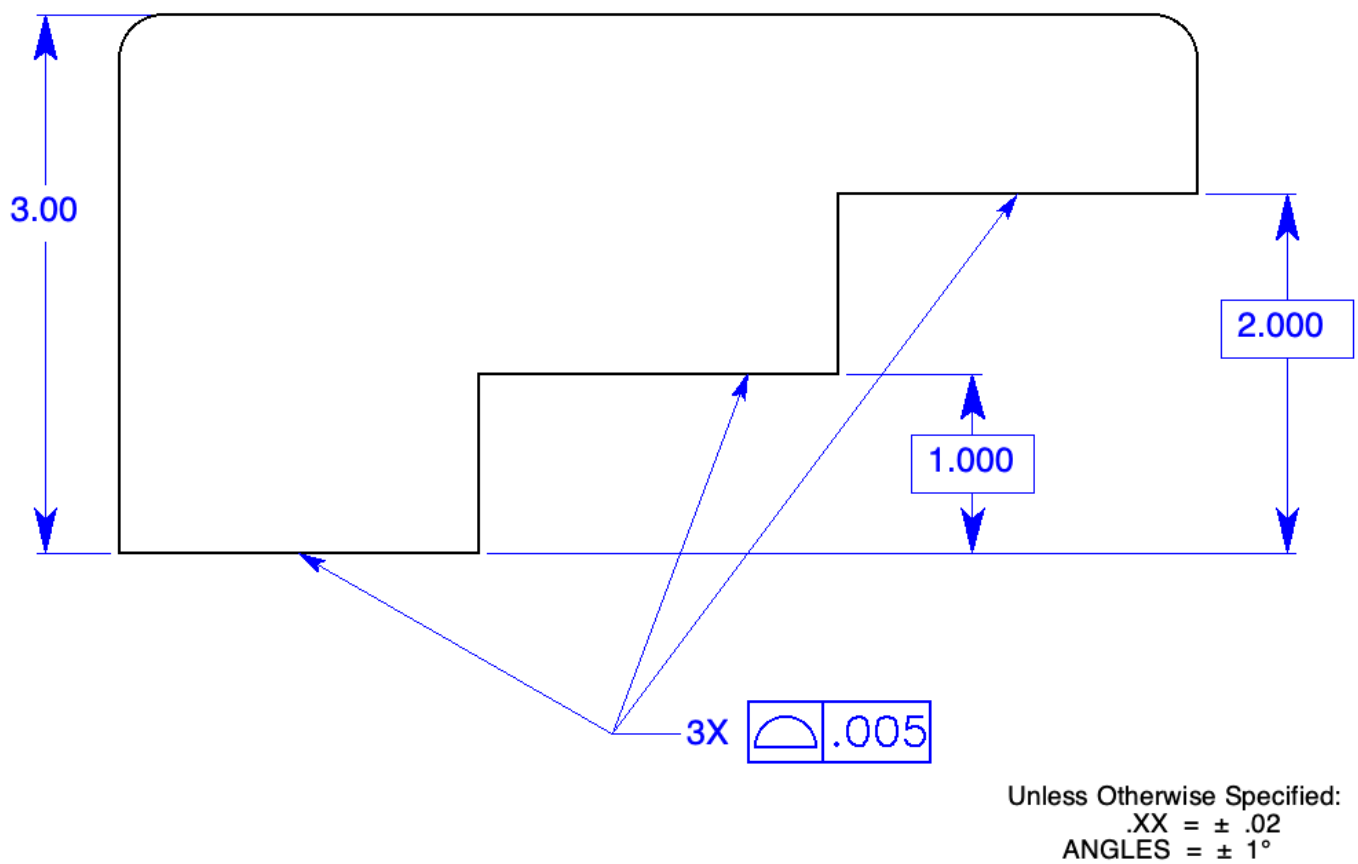


Figure 11-25 Controlling a stepped part with a profile of a surface: Prob. 8.

8. Control the three bottom surfaces in Fig. 11-25 flat, parallel, and stepped one basic inch apart within .005.

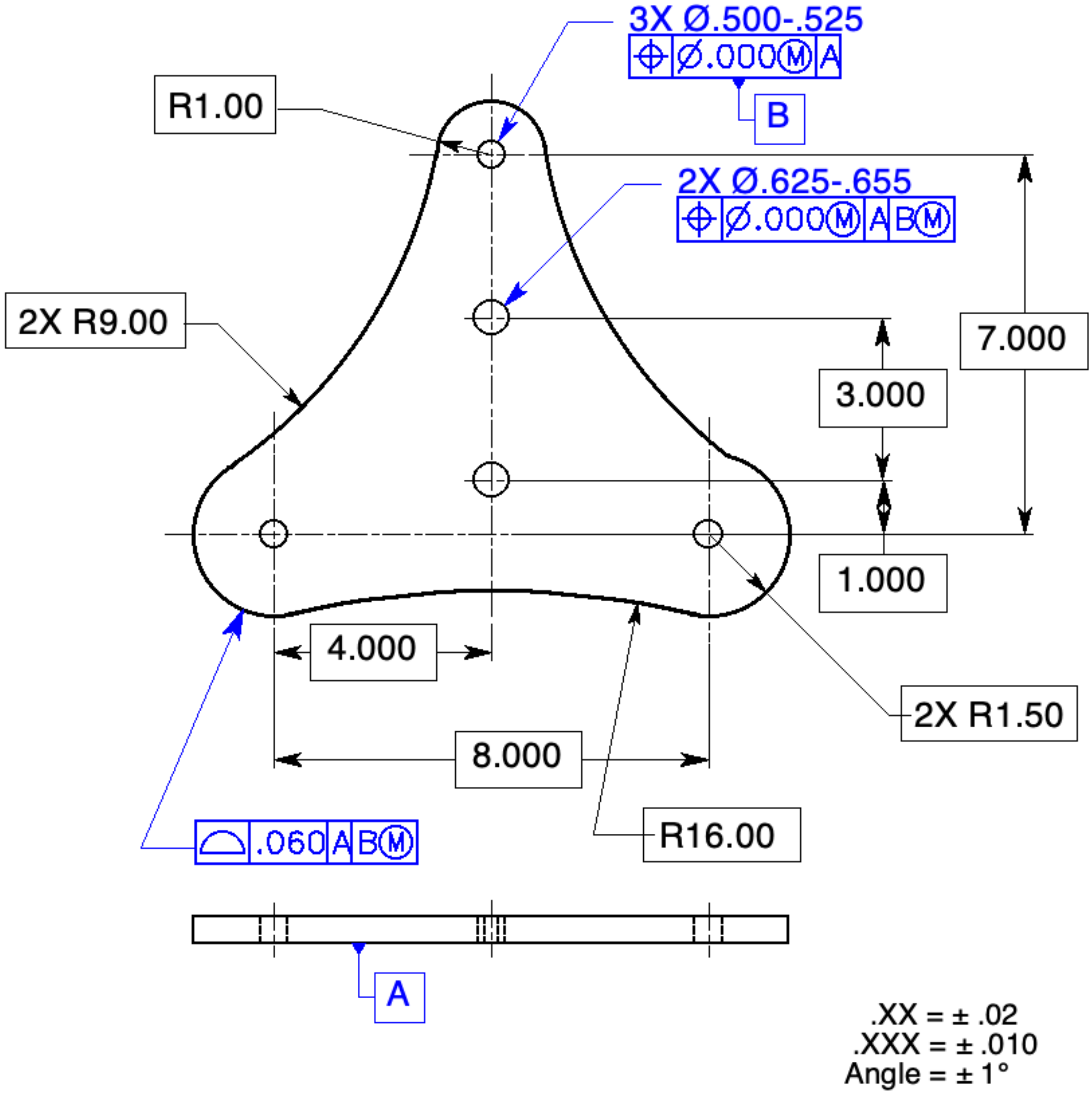


Figure 11-26 Controlling a profile to datum features of size: Prob. 9.

9. Tolerance the drawing in Fig. 11-26. Specify controls locating the hole-patterns to each other and perpendicular to the back of the part. The holes are for 1/2-inch and 5/8-inch bolts respectively. Specify a control locating the outside profile of the part to the hole-patterns and perpendicular to the back of the part within a tolerance of .060. Specify MMC and MMB wherever possible.

Are there other answers to this problem?

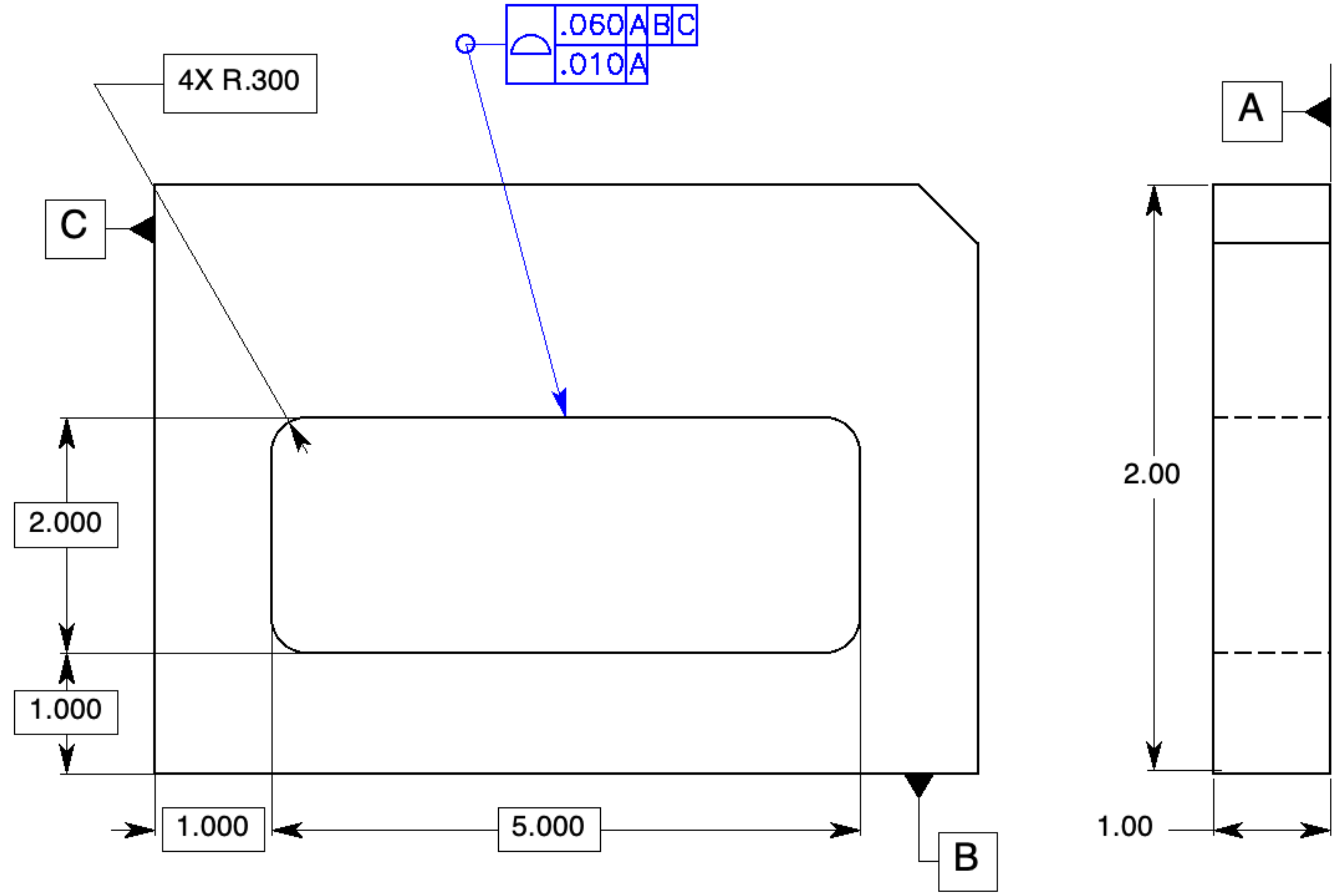
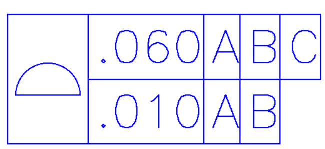


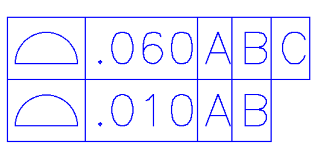
Figure 12-27 Composite profile tolerancing: Probs. 10 through 12.

10. In Fig. 11-27, specify a profile tolerance for the center cutout that will control the size, form, and orientation to datum feature A within .010, and locate the cutout within .060 to the datum features indicated. Complete the drawing.

11. Draw a profile tolerance below that will satisfy the requirements for problem 10 and in addition, *orient* the cutout parallel to datum feature B within .010.



12. Draw a profile tolerance below that will satisfy the requirements for problem 10 and in addition, *locate* the cutout to datum feature B within .010.



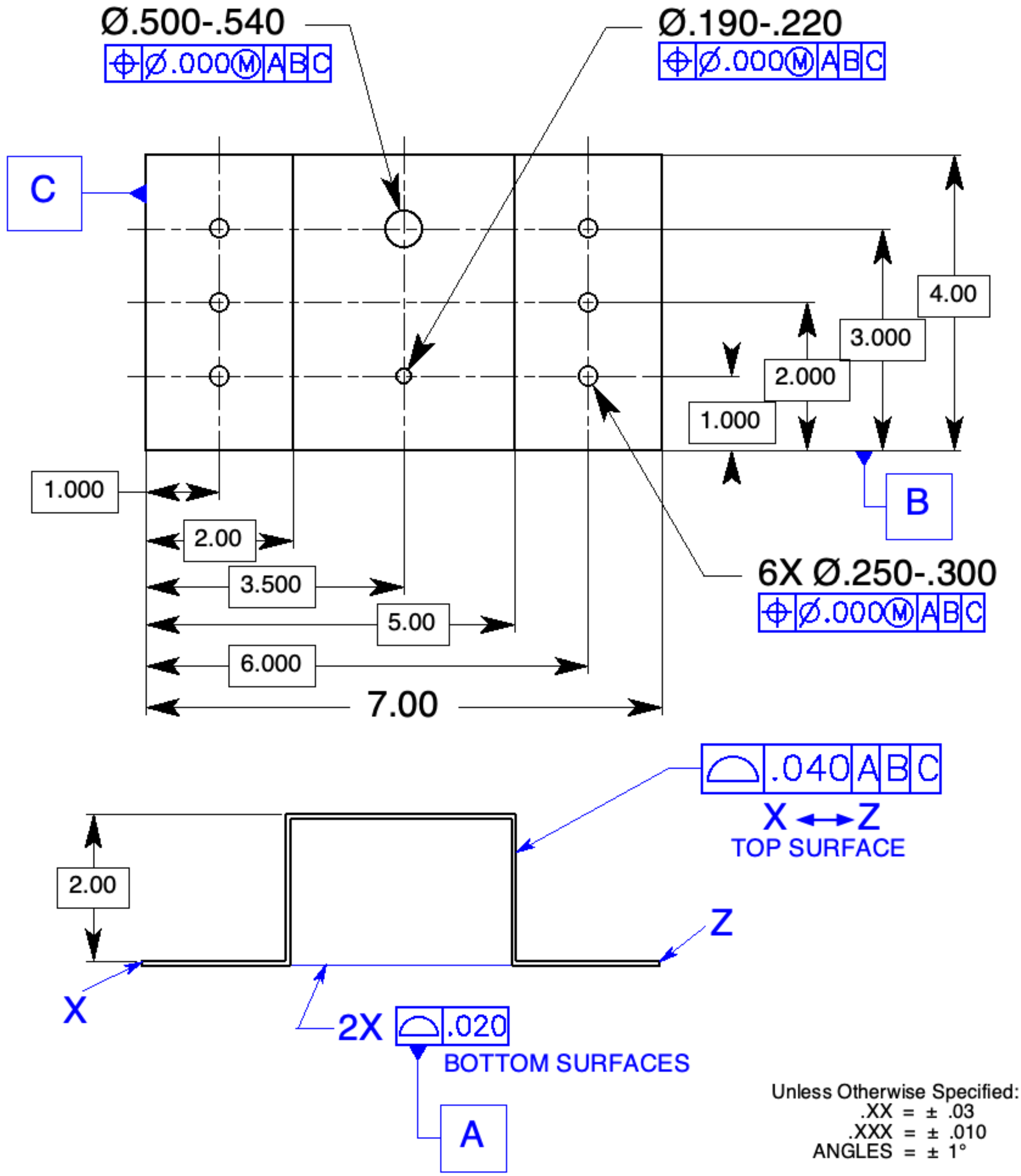


Figure 11-28 Controlling the surface of a sheet metal part with profile: Prob. 13.

13. Specify the lower two surfaces of the bottom of the sheet metal part in Fig. 11-28 coplanar within .020. Tolerance the holes with geometric tolerancing. The MMC for each hole is the virtual condition for the mating fastener. Specify the profile of the top surface of the sheet metal part within .040. Use MMC and MMB wherever possible.

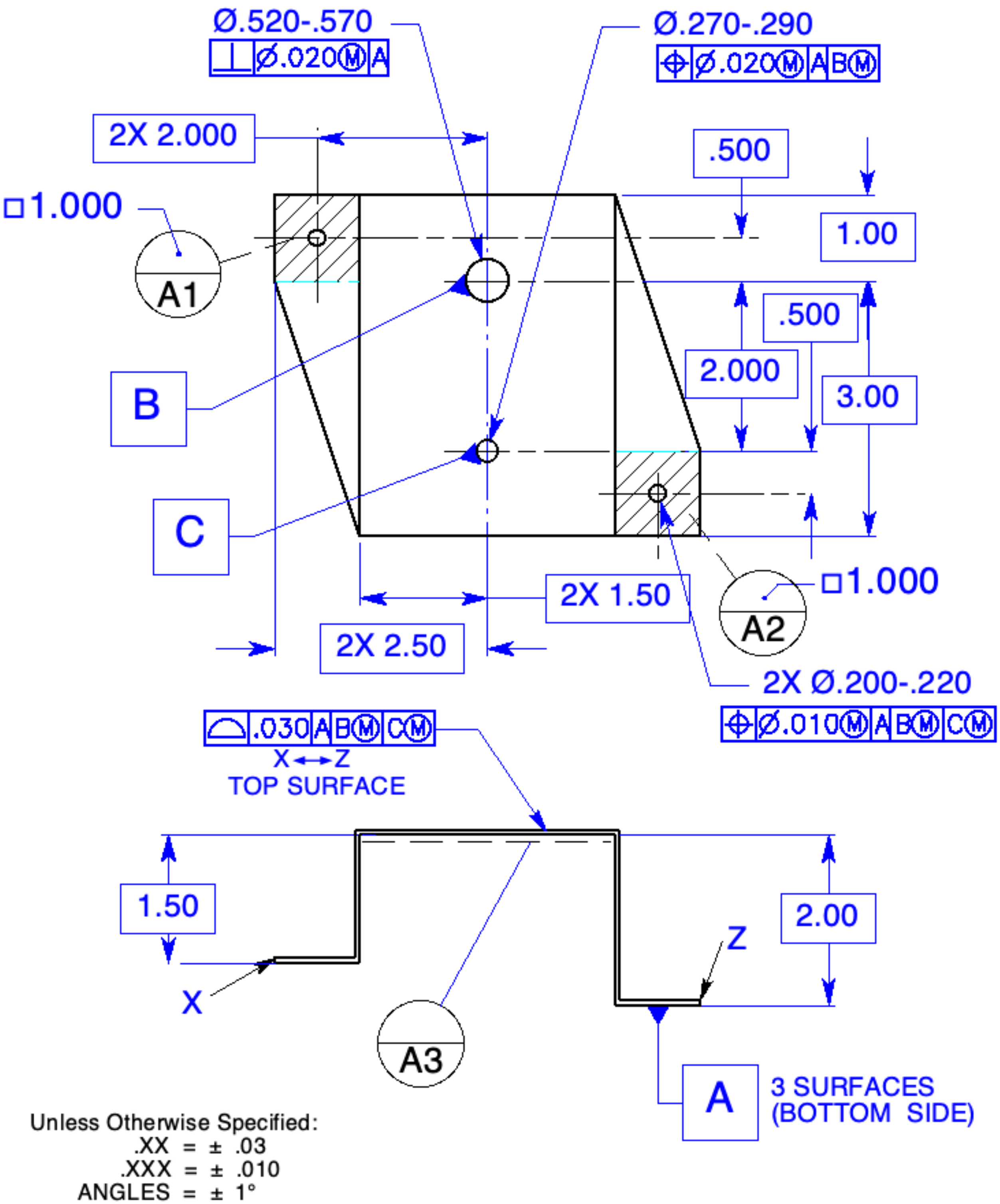


Figure 11-29 Controlling the surface of a sheet metal part with profile and datum targets: Prob. 14 9 (p. 246)

14. Tolerance the holes in Fig. 11-29 with geometric tolerancing to datum features A, B, and C. The sizes of the holes are for ½-inch, ¼-inch, and #10 fastener sizes. Specify the profile of the top surface of the part within a tolerance of .030 to datum features A, B, and C. Use MMC and MMB wherever possible.

Chapter 12

A Strategy for Tolerancing Parts

Chapter Review

Page 265

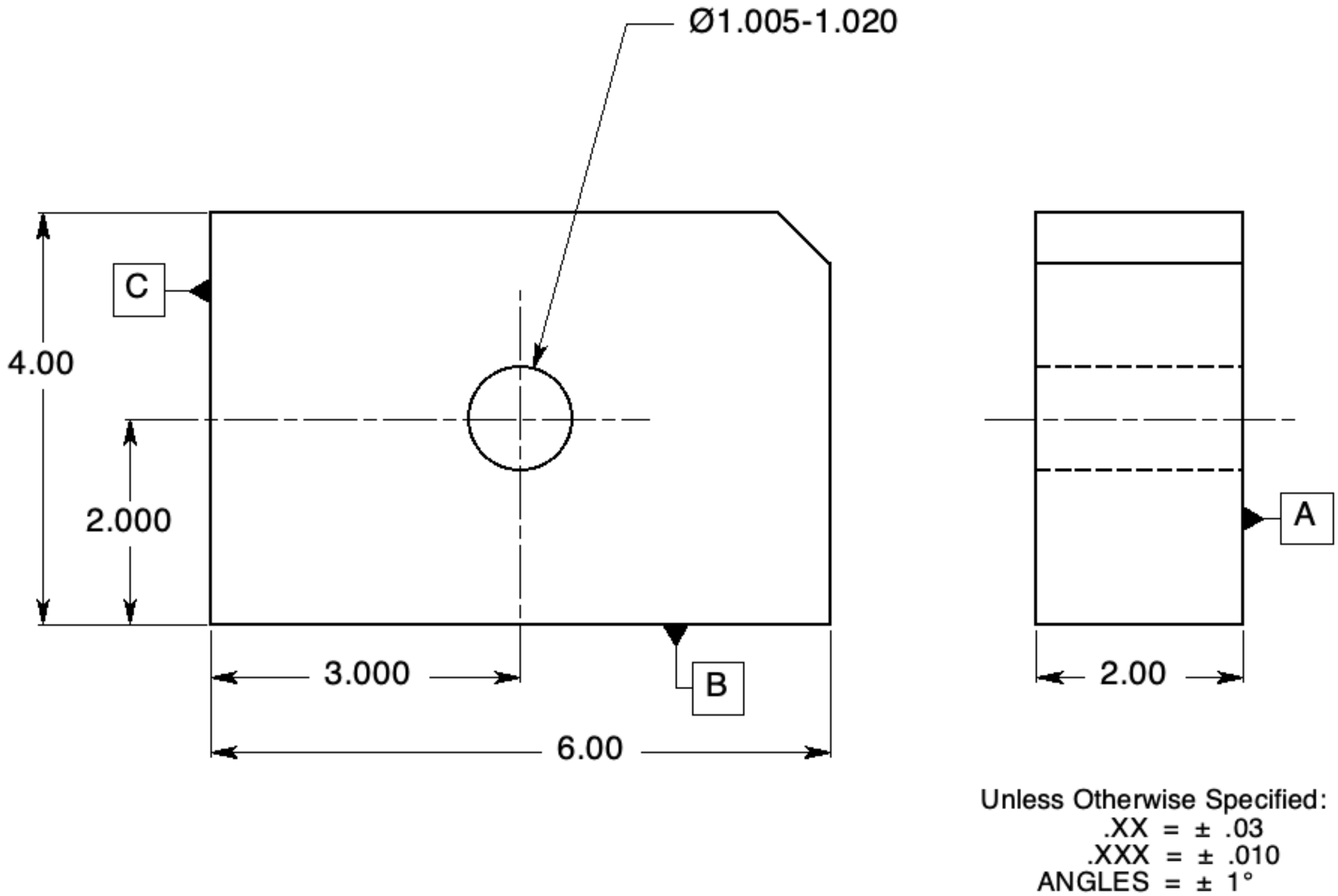


Figure 12-20 A hole located and oriented to datum features A, B, and C: Questions 1 through 6.

1. What type of geometric tolerances applies to the primary datum feature in a drawing like the drawing in Fig. 12-20? *Form tolerance*

2. What geometric tolerance applies to the primary datum feature in the drawing in Fig.12-20? *Flatness*

3. The primary datum feature controls *the orientation* of the part being controlled.

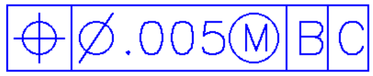


Figure 12-21 A feature control frame with two location datum features: Question 4.

4. If the feature control frame for the hole in Fig. 12-20 happened to be the one shown in Fig. 12-21, what relationship would the one-inch diameter hole have to datum features B and C?

The tolerance zone of the one-inch hole would be parallel to datum feature B and parallel to datum feature C and at the same time , located with basic dimensions up from datum feature B and over from datum feature C.

Figure 12-22 A feature control frame with three datum features: Question 5.

5. If the feature control frame for the hole in Fig. 12-20 happened to be the one shown in Fig. 12-22, what relationship would the one-inch diameter hole have to datum features A, B, and C?

The tolerance zone of the one-inch hole would be perpendicular to datum feature A, located with basic dimensions up from datum feature B and over from datum feature C.

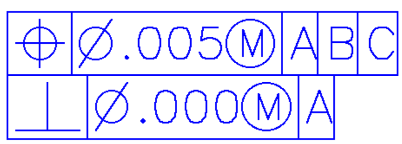
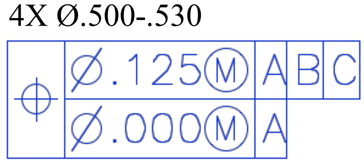


Figure 12-23 A position feature control frame with a refinement: Question 6.

6. Complete the feature control frame in Fig. 12-23 so that it will refine the orientation of the hole in Fig. 12-20 within a cylindrical tolerance of .000 at MMC.

7. Draw a feature control frame to locate a pattern of holes within cylindrical tolerance zones .125 in diameter at MMC to their datum features, A, B, and C. Refine the feature-to-feature relationship perpendicular to datum feature A and located to each other within a .000 positional tolerance at MMC.

4X Ø.500-.530



8. What is the orientation tolerance for the pattern of holes specified in the answer for Question 7?
Perpendicular to datum feature A within a cylindrical tolerance zone of .000 in diameter at MMC.

9. Keeping in mind that the primary datum controls orientation, explain how you would select a primary datum on a part. *Key points in selecting a primary datum feature are:*

   Select a functional surface

   Select a mating surface

   Select a sufficiently large, accessible surface that will provide repeatable positioning in a datum reference frame while processing and ultimately in assembly

10. How would you determine which datum feature should be secondary and which should be tertiary?
The secondary datum may be more important because it is larger than the tertiary datum or because it is a mating surface.

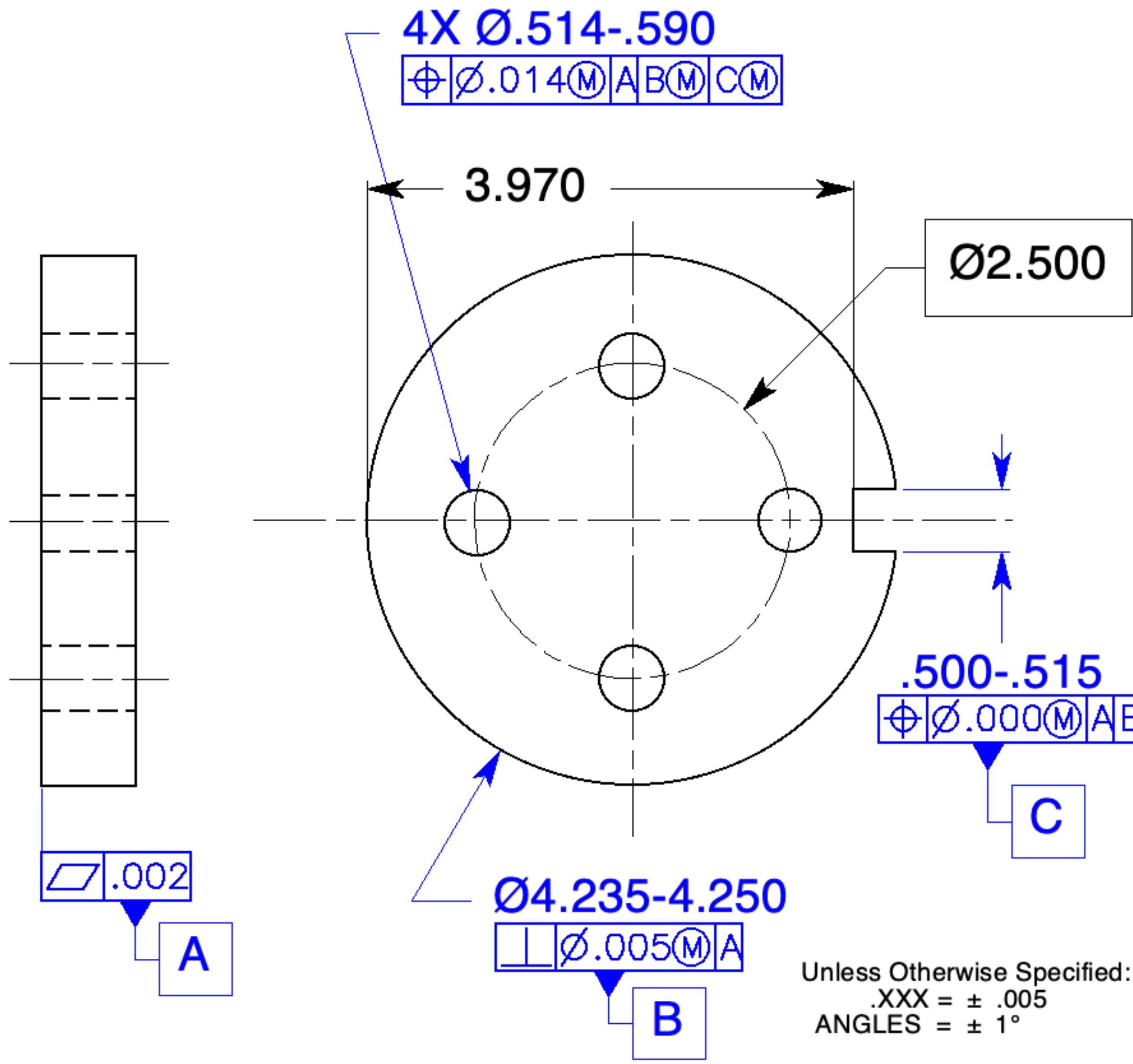


Figure 12-24 Four-hole pattern of features: Questions 11 through 17.

11. Select a primary datum feature and specify a form control for it.
12. Select a secondary datum feature and specify an orientation control for it. The virtual condition of the mating outside diameter is Ø 4.255.
13. Tolerance the keyseat for a 1/2-inch key.
14. Tolerance the 1/2-inch clearance holes for 1/2-inch floating fasteners.
15. Are there other ways this part can be toleranced? How

Yes, the hole pattern could be datum feature C, or datum feature C could be left off entirely.

16. If the outside diameter is actually produced at 4.240, how much shift tolerance is available?

If datum feature B were perfectly perpendicular to datum feature A, there would be a Ø.015 cylindrical shift tolerance zone

17. If the outside diameter is actually produced at 4.240 and the keyseat is actually produced at .505, how much can this part actually shift? Sketch a gage about the part.

If datum feature B were perfectly perpendicular to datum feature A, the outside diameter could shift back and forth .015 and up and down .005. The part could rotate some.

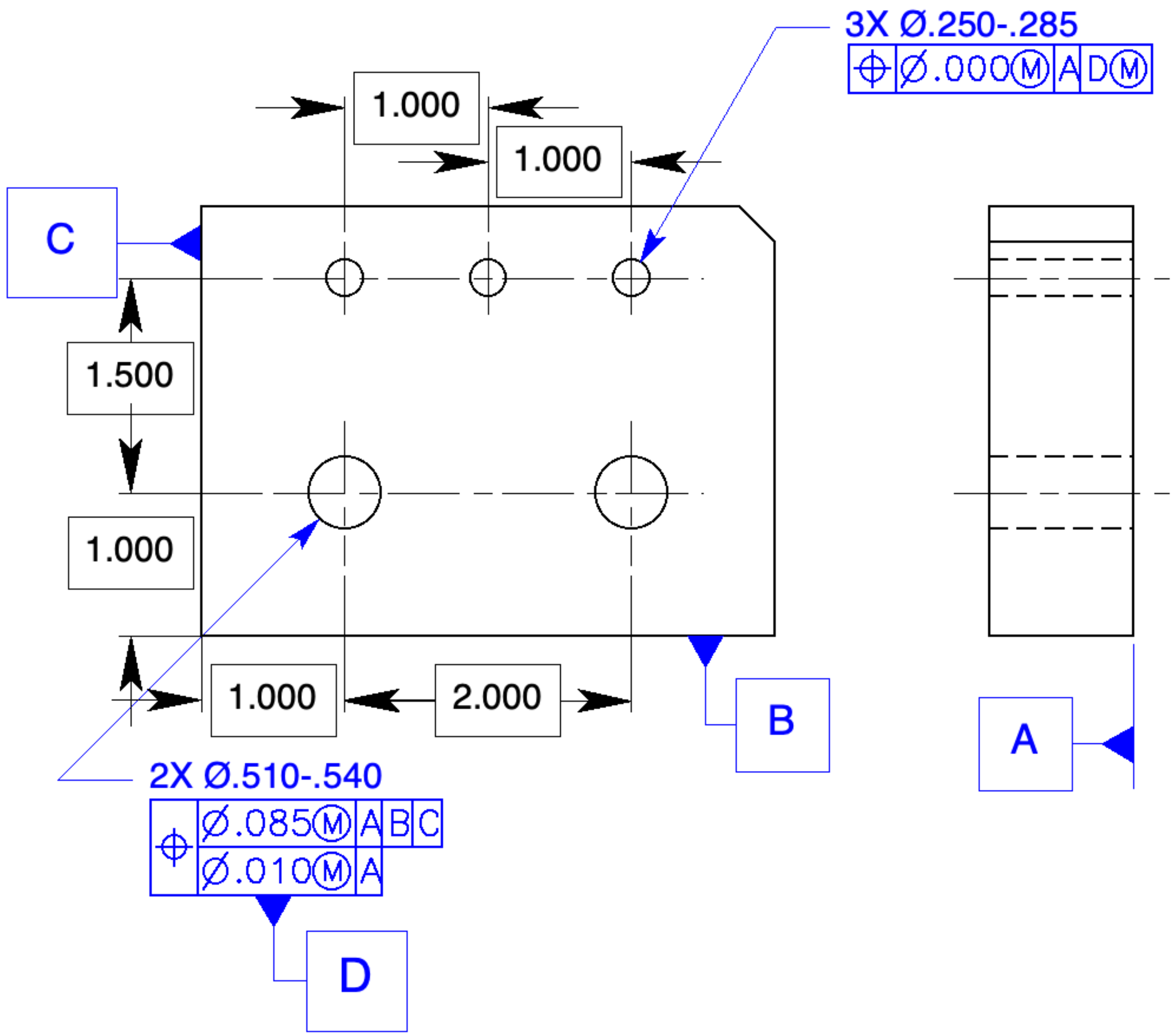


Figure 12-25 A pattern of features located to a second pattern of features: Questions 18 through 21.

18. Locate the two-hole pattern to the surface datum features with a positional tolerance of .085 at MMC. Locate the same two holes to each other and orient them to datum feature A within a cylindrical tolerance of .010 at MMC.
19. Locate the three-hole pattern to the two-hole pattern within a .000 positional tolerance.
20. The two-hole pattern is specified as a datum feature at MMB, at what size does each of the two holes apply?

    Virtual condition with respect to datum feature A – Ø.500

Problems

Page 270

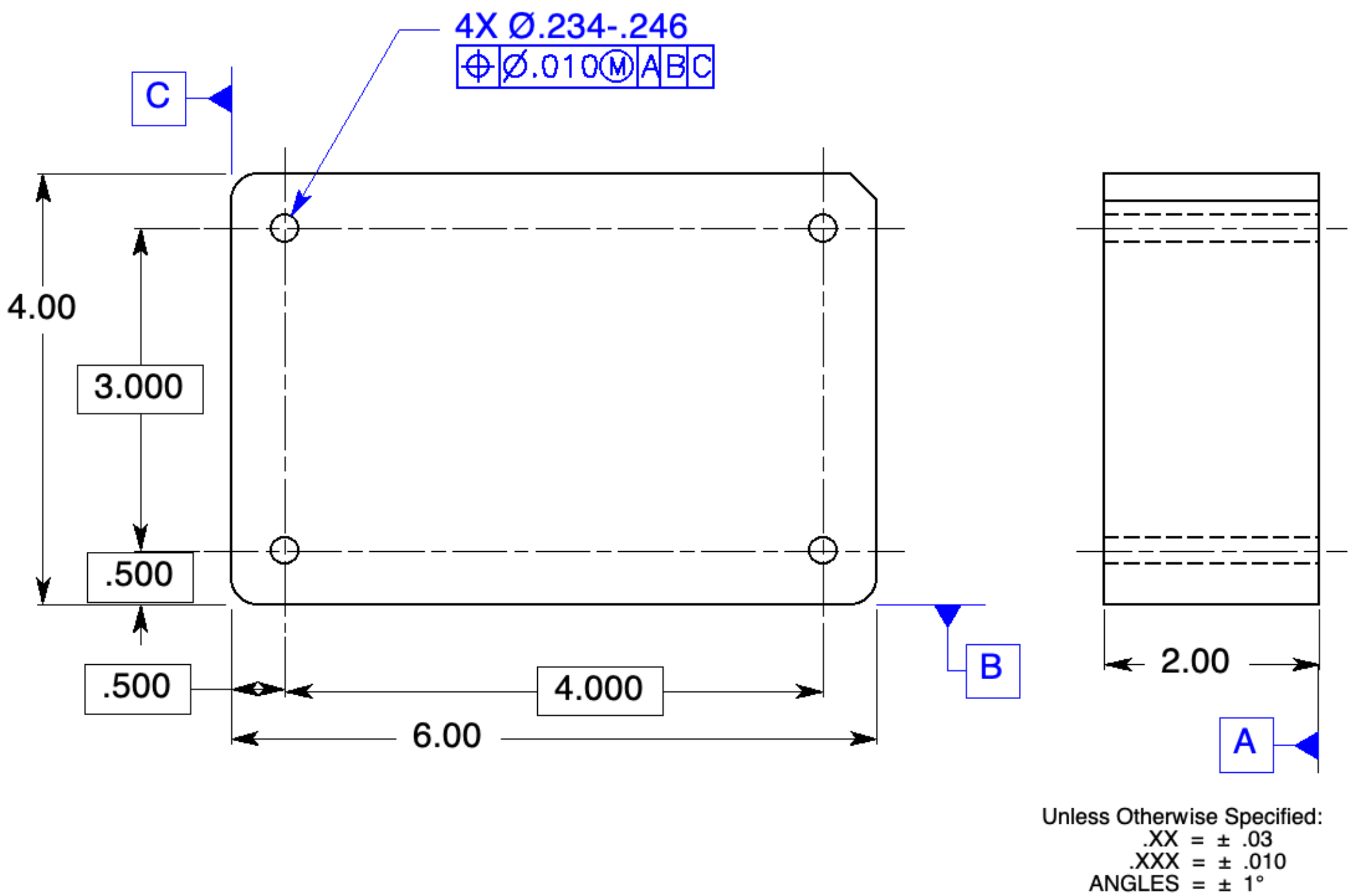


Figure 12-26 Tolerancing fixed fasteners: Prob. 1.

1. Tolerance the four-hole pattern in Fig. 12-26 for #10 (Ø.190) cap screws as fixed fasteners. Specify a tolerance of .010 at MMC for the clearance holes and 60% of the total tolerance for the threaded holes in the mating part.

- How flat is datum feature A? *Within .060 – in reality probably within .015*.
- How perpendicular are datum features B and C to datum feature A and to each other? *±1°*

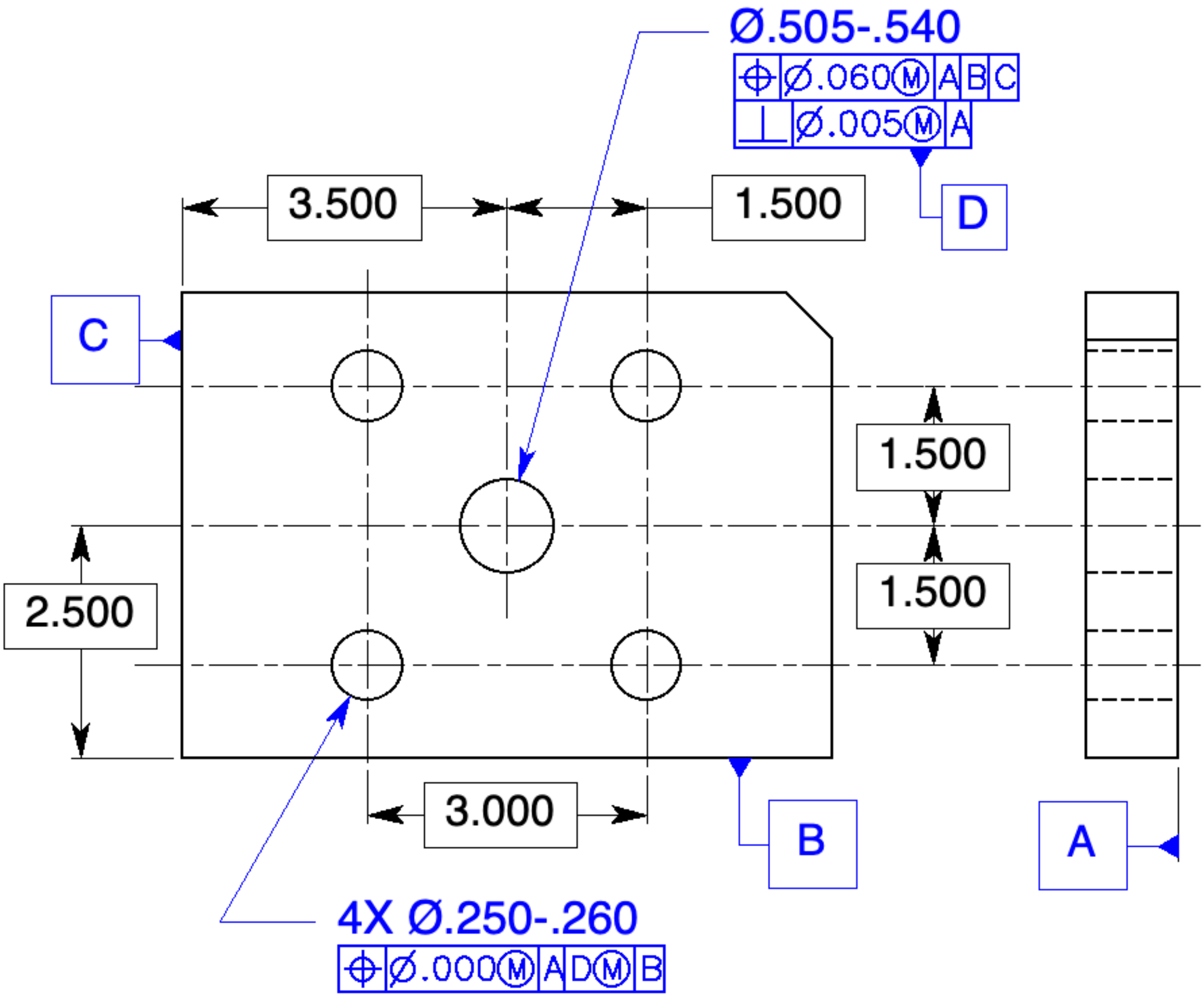


Figure 12-27 Tolerancing a pattern of features to a datum feature of size: Prob. 2.

2. In Fig. 12-27, tolerance the center hole to the outside edges with a tolerance of .060. Refine the orientation of the half-inch hole to the back of the part within .005. Locate the four-hole pattern to the center hole. Clock the pattern to a surface. The four-hole pattern mates with a part having four pins with a virtual condition of .250 in diameter. Give each feature all of the tolerance possible.

- At what size does the center hole apply for the purposes of positioning the four-hole pattern? _______
- *Virtual condition of datum feature D with respect to datum feature A–Ø.500*
- If the center hole is produced at a diameter of .535, how much shift of the four-hole pattern is possible? *Ø.035 if datum feature D is perfectly perpendicular to datum A*

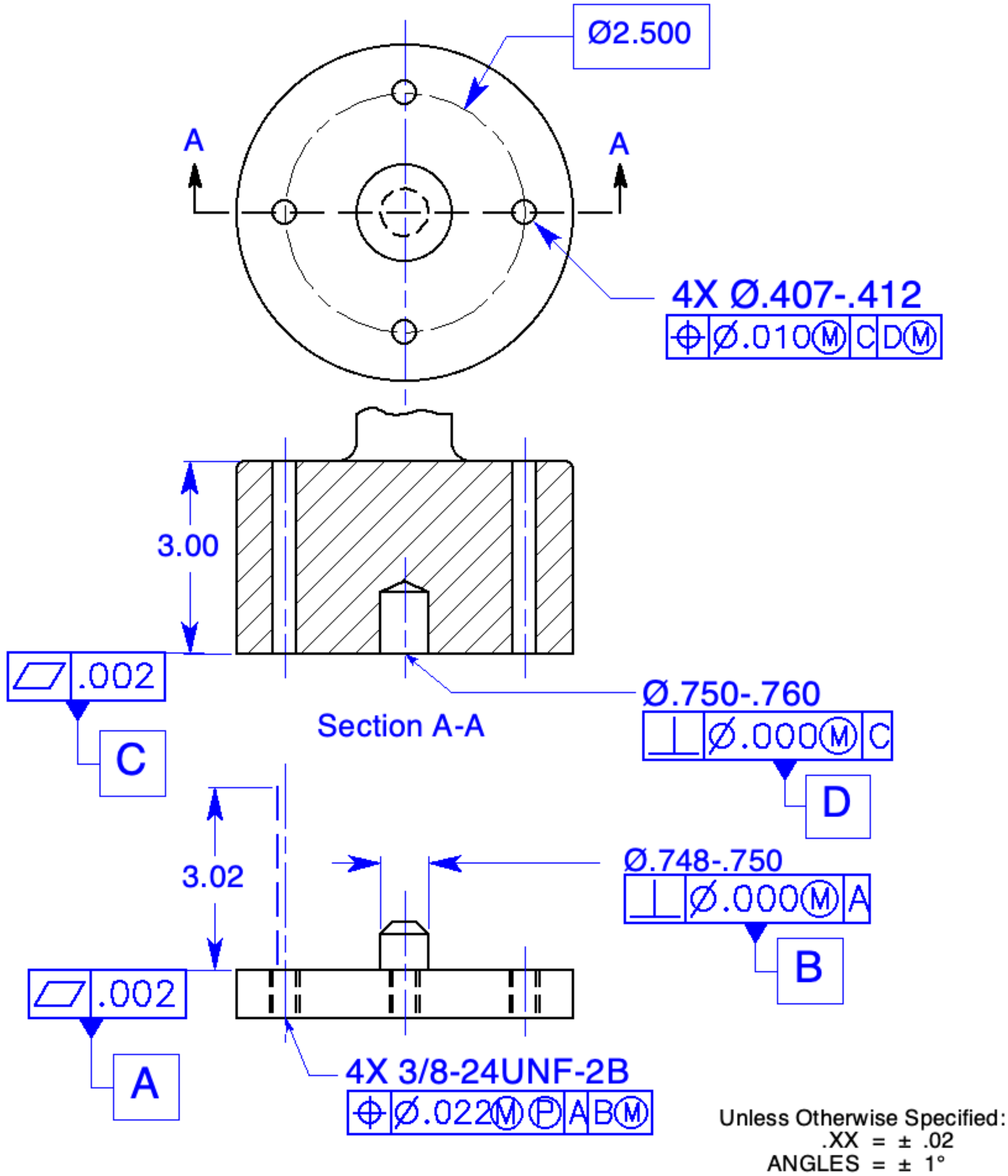


Figure 12-28 Tolerance two features for assembly: Prob. 3.

3. Tolerance the two parts on the drawing in Fig. 12-28. Specify a flatness control of .002 on each of the primary datum features. Specify the appropriate orientation control to control the relationships between the primary and secondary datum features. Finally, complete the location tolerances for the hole patterns using a positional tolerance of .010 for the clearance holes. Use MMC/MMB wherever possible.

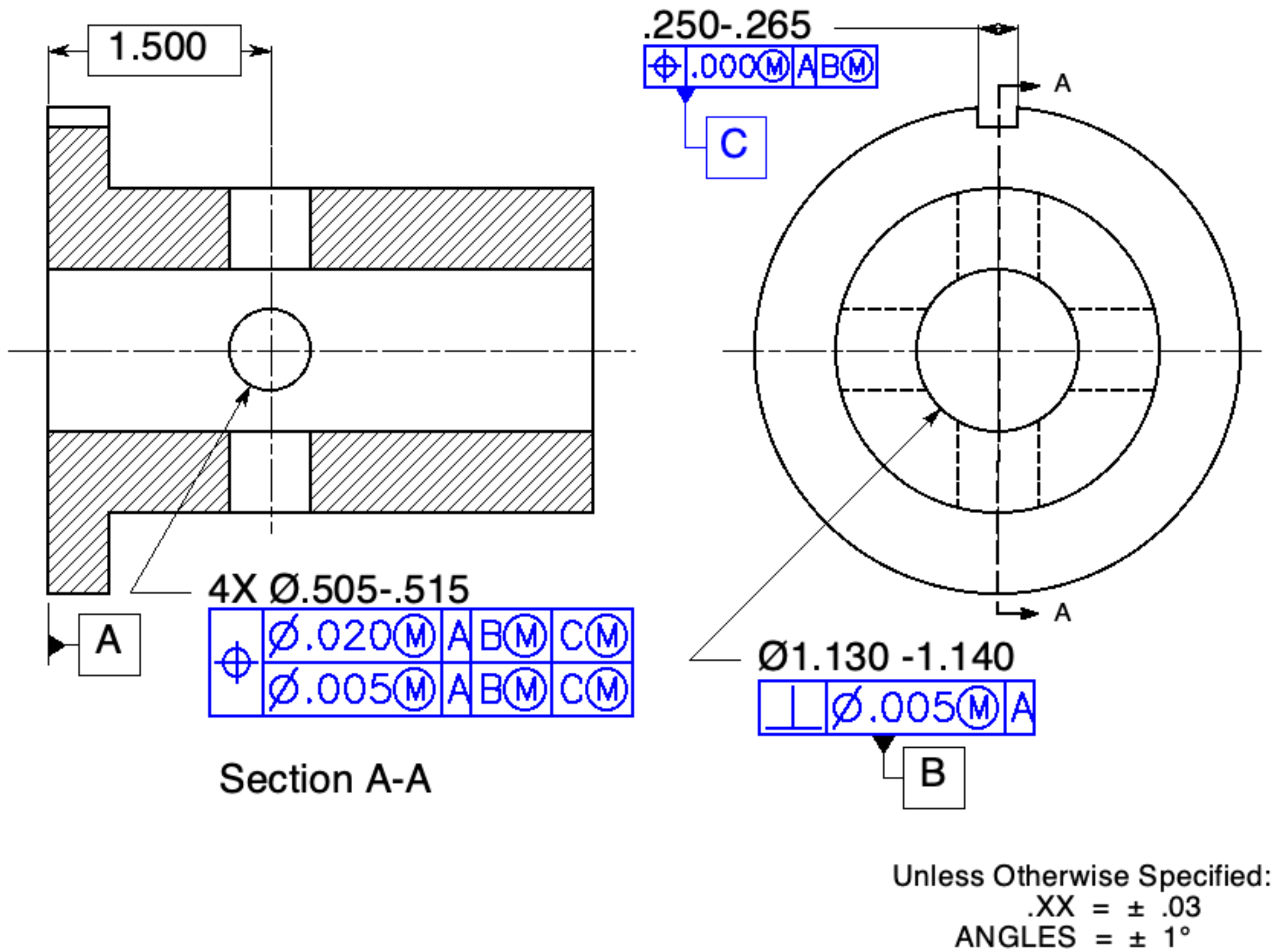


Figure 12-29 Tolerancing a four-hole pattern with a composite positional tolerance: Prob. 4.

4. In Fig. 12-29, orient the center bore to datum feature A with a geometric tolerance. The mating shaft is 1.125 in diameter. Orient the ¼-inch keyseat to datum feature A, and locate it to the center bore. Position the four-hole pattern within a tolerance of .020 parallel to datum feature A, centered on the 1.130 bore, and clocked to the keyseat. Refine the location of the 4 holes in the 4-hole pattern to each other, and refine the orientation of the four-hole pattern parallel to datum feature A, perpendicular to the center bore, and parallel and perpendicular to the center plane of the keyseat within .005. Use MMC and MMB wherever possible.

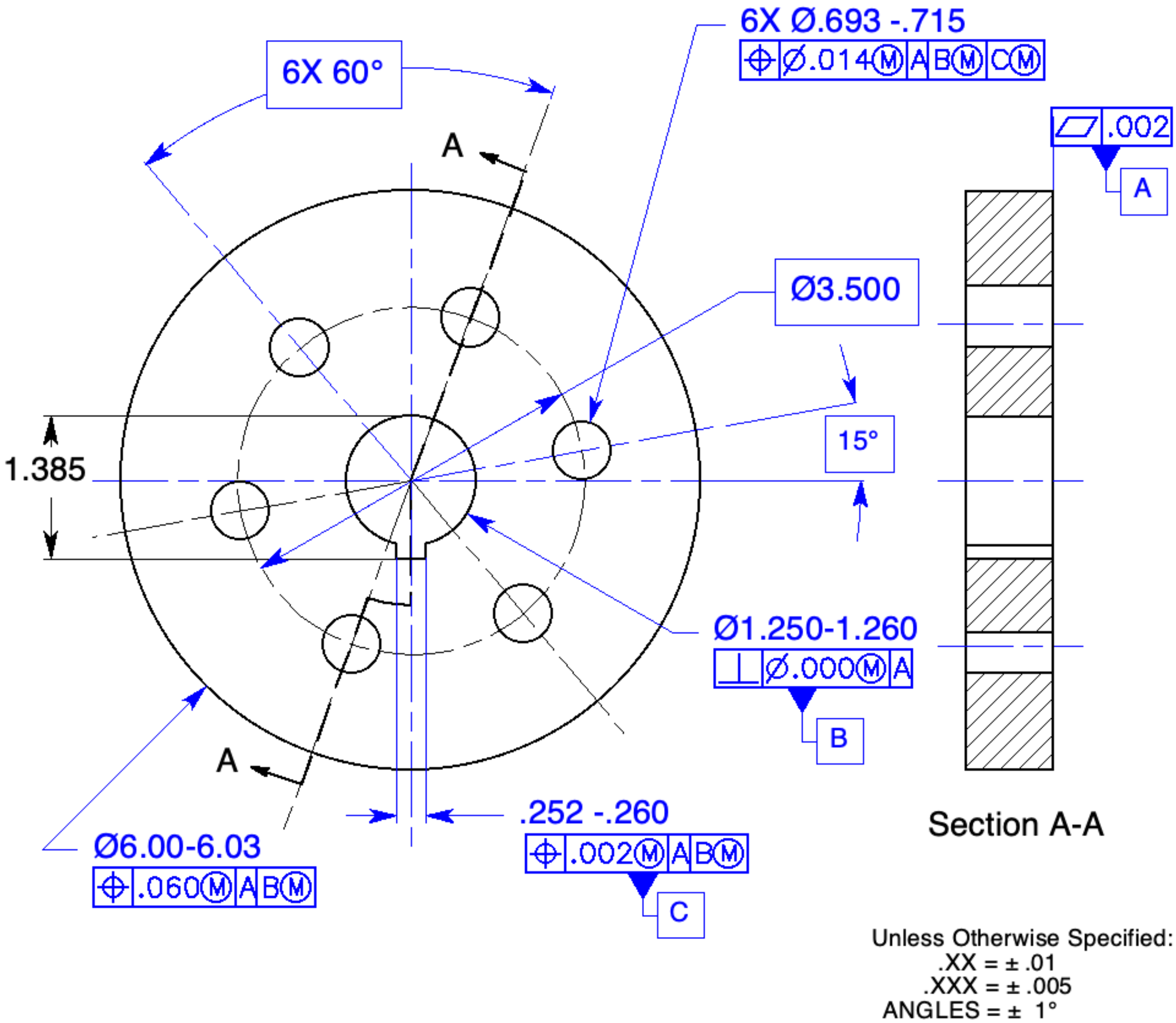


Figure 12-30 Tolerancing a pattern of features to a datum feature of size and a keyway: Prob. 5.

5. In Fig. 12-30, the inside diameter and the back are mating features. Select the primary datum feature. (Consider a form control.) The virtual condition of the mating shaft is 1.250 in diameter. Control the relationship between the primary and secondary datum features. Locate the keyway for a ¼-inch key. Locate the six-hole pattern for 5/8-inch (Ø.625) cap screws with a positional tolerance of .014 as fixed fasteners with the mating part. Use MMC and MMB wherever possible.

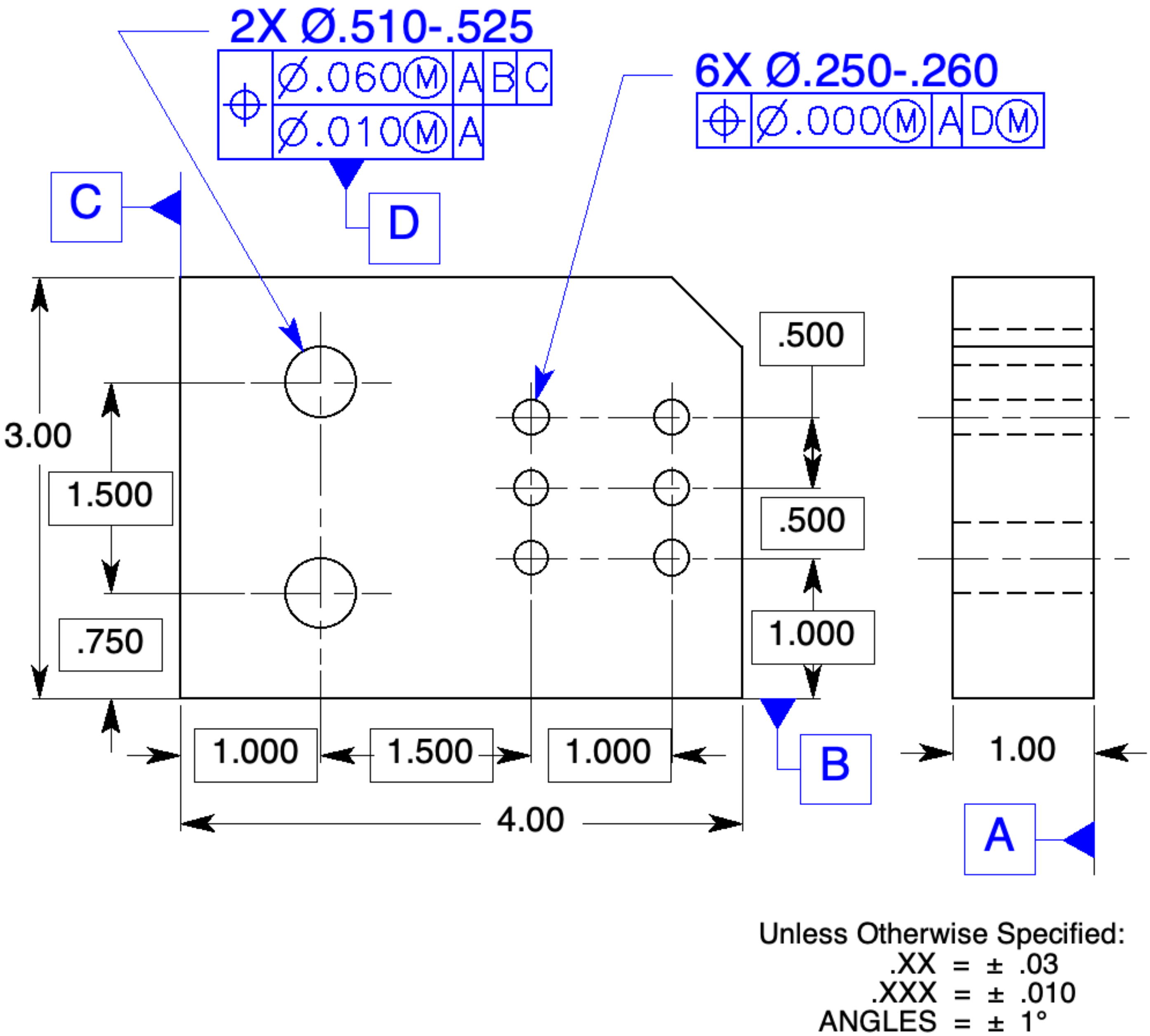


Figure 12-31 Tolerance a pattern of features to a second pattern of features: Prob. 6.

6. In Fig. 12-31, the location of the hole patterns to the outside edges is not critical; a tolerance of .060 at MMC is adequate. The location between the two half-inch holes and their orientation to datum feature A must be within .010 at MMC. Control the six-hole pattern to the two-hole pattern within .000 at MMC. The mating part has virtual condition pins of .500 and .250 in diameter. Complete the tolerance.

- At what size does the two-hole pattern apply for the purposes of positioning the six-hole pattern? *Virtual condition with respect to orientation – Ø .500*
- If the two large holes are produced at a diameter of .520, how much shift of the six-hole pattern is possible? *Ø.020*

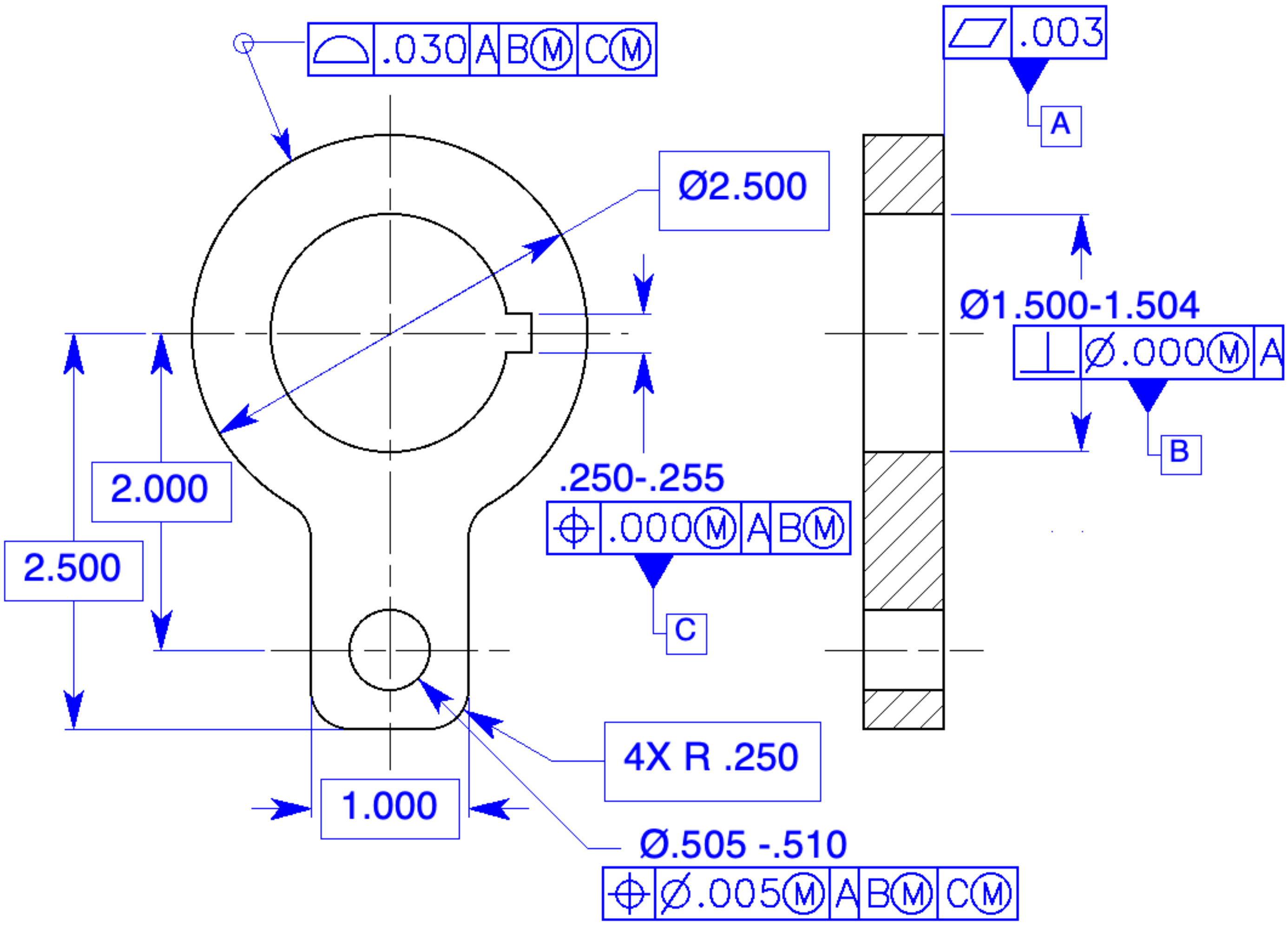


Figure 12-32 Tolerance a feature of size to a second feature of size: Prob. 7.

7. In Fig. 12-32, the inside diameter and the back are mating features. Select the primary datum feature. (Consider a form control.) The virtual condition of the mating shaft is 1.500 in diameter. Locate the keyway for a ¼-inch key. Locate the ½-inch diameter hole for a mating pin .498-.500 in diameter. Control the outside edge of the part with a tolerance of .030.

Chapter 13

Graphic Analysis

Chapter Review

Page 288

1. List the advantages of graphic analysis.

   Provides functional acceptance

   Reduces cost and time

   Eliminates gage tolerance and wear allowance

   Allows function verification of MMC/MMB, RFS/RMB, and LMC/LMB

   Allows verification of any shape tolerance zone

   Provides a visual record for the material review board

   Minimizes storage required

2. List the factors that affect the accuracy of graphic analysis.

   The accuracy of the graphs and overlay gage

   The accuracy of the inspection data

   The completeness of the inspection process

   The drawing's ability to provide common drawing interpretations

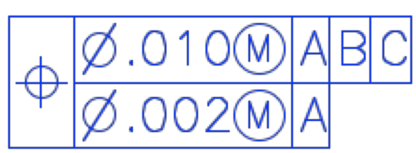


Figure 13-12 A composite feature control frame: Questions 3 through 7

3. A piece of graph paper with datums, true positions, tolerance zones, and actual feature locations drawn on it is called a *data graph*.

4. The upper segment of the composite feature control frame, the drawing, and the inspection data dictates the configuration of the ____*data graph*____.

5. A piece of tracing paper with true positions and tolerance zones traced or drawn is called a ____ *tolerance zone overlay gage*____

6. The lower segment of the feature control frame, the drawing, and the inspection data dictate the configuration of the____*overlay gage*____.

7. If the tracing paper can be adjusted to include all feature axes within the ____*tolerance zones*____ on the tracing paper, the feature-to-feature relationships are in tolerance.

Figure 13-13 A datum feature of size specified at MMB: Questions 8 through 11.

8. To inspect a pattern of features controlled to a datum feature of size, the feature control frame, the drawing, and the inspection data dictate the configuration of the ____*data graph*____.

9. Draw the actual location of each feature on the data graph. If each feature axis falls inside its respective tolerance zone, the part is ____*in tolerance*____.

10. If any of the feature axes fall outside its respective tolerance zone, ____*the part still may be acceptable if there is enough shift tolerance to shift all axes into their respective tolerance zones*____.

11. If the tracing paper can be adjusted to include all feature axes on the overlay gage within their respective tolerance zones on the data graph and datum axis D contained within its shift tolerance zone while orienting datum feature B on the overlay gage parallel to datum feature B on the data graph, the pattern of features is____*in tolerance*____.

Problems

Page 290

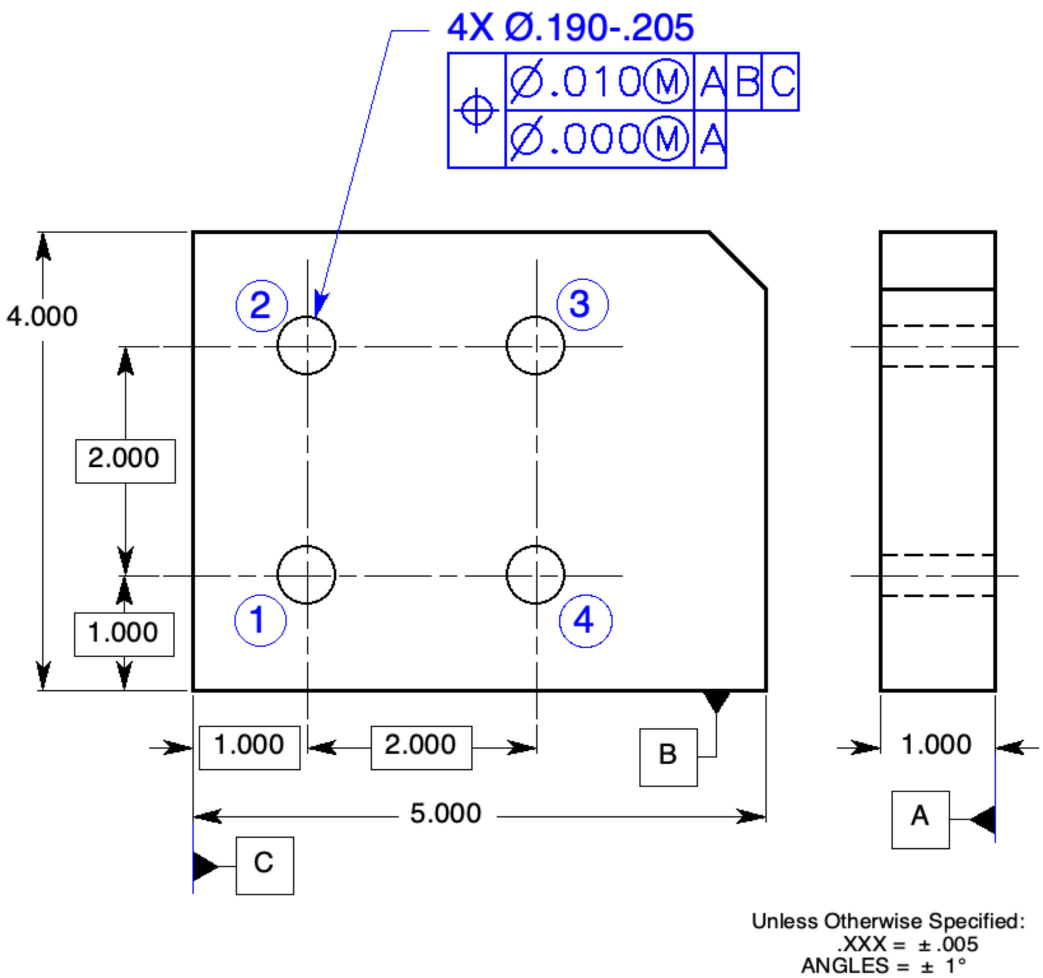


Figure 13-14 A pattern of features controlled with a composite tolerance: Prob. 1.

Feature Number	Feature Location from Datum C X-Axis	Feature Location from Datum B Y-Axis	Feature Size	Departure from MMC (Bonus)	Datum-to-Pattern Tolerance Zone Size	Feature-to-Feature Tolerance Zone Size
1	1.002	1.003	Ø .200	***.010***	***Ø .020***	***Ø .010***
2	1.005	3.006	Ø .198	***.008***	***Ø .018***	***Ø .008***
3	3.005	3.002	Ø .198	***.008***	***Ø .018***	***Ø .008***
4	3.003	.998	Ø .196	***.006***	***Ø .016***	***Ø .006***

Table 13-3 Inspection Data for Graphic Analysis of Prob. 1

1. A part was made from the drawing in Fig. 13-14; the inspection data was tabulated in Table 13-3. Perform a graphic analysis of the part. Is the pattern within tolerance?_____*Yes*_________

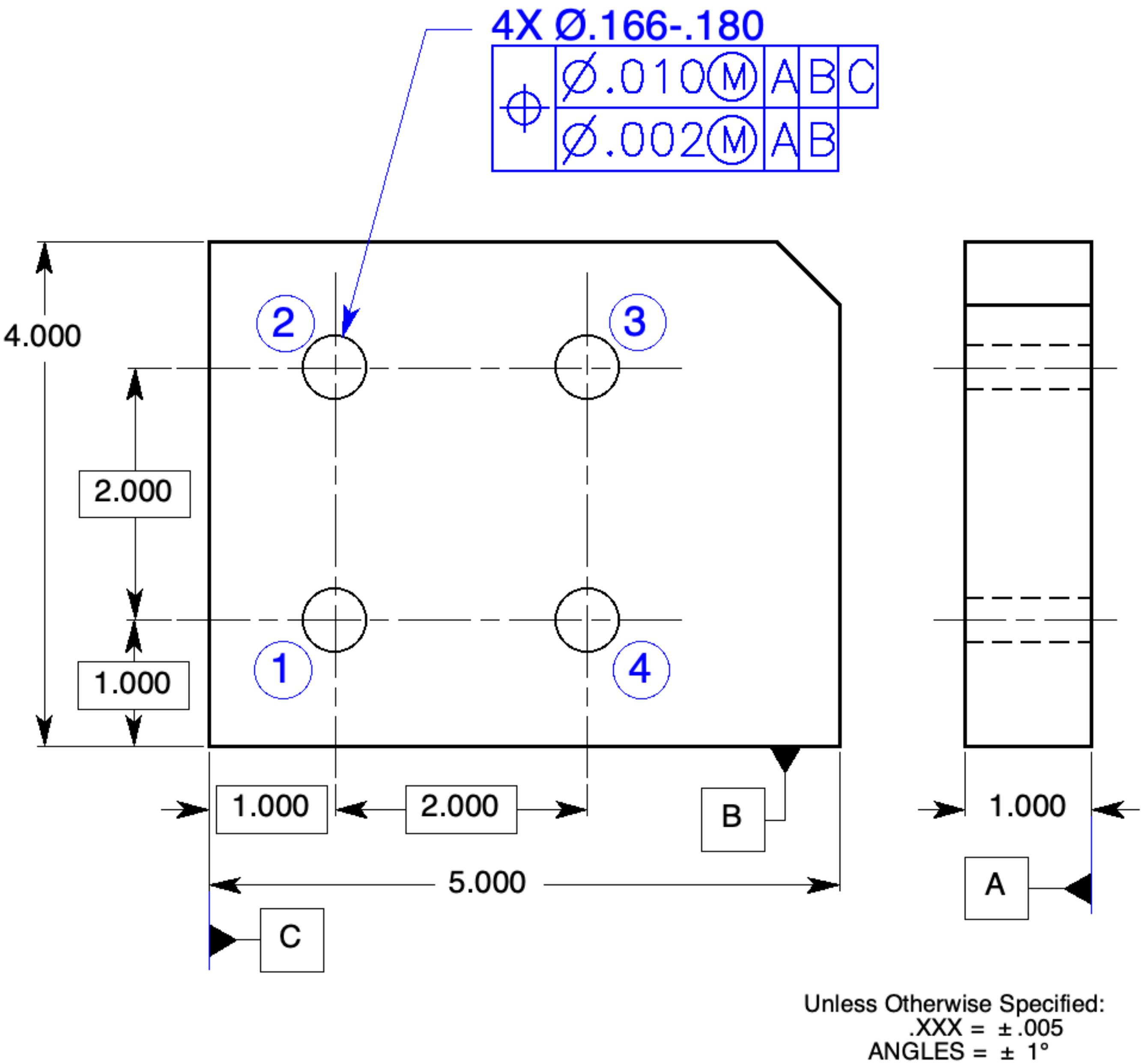


Figure 13-15 A pattern of features controlled with a composite tolerance: Prob. 2.

Feature Number	Feature Location from Datum C X-Axis	Feature Location from Datum B Y-Axis	Feature Size	Departure from MMC (Bonus)	Datum-to-Pattern Tolerance Zone Size	Feature-to-Feature Tolerance Zone Size
1	1.004	.998	Ø.174	*.008*	*Ø.018*	*Ø.010*

2	.995	3.004	Ø .174	*.008*	*Ø .018*	*Ø .010.*
3	3.000	3.006	Ø .172	*.006*	*Ø .016*	*Ø .008*
4	3.006	1.002	Ø .176	*.010*	*Ø .020*	*Ø .012*

Table 13-4 Inspection Data for Graphic Analysis of Prob. 2

2. A part was made from the drawing in Fig. 13-15; the inspection data were tabulated in Table 13-4.

   Perform a graphic analysis of the part. Is the pattern within tolerance? *No*

   (The pattern must remain parallel to datum B because datum B has been repeated in the lower segment of the feature control frame.)

   If it is not in tolerance, can it be reworked? If so, how? *The pattern will be in tolerance if hole numbers 2 and 3 are enlarged by about .004.*

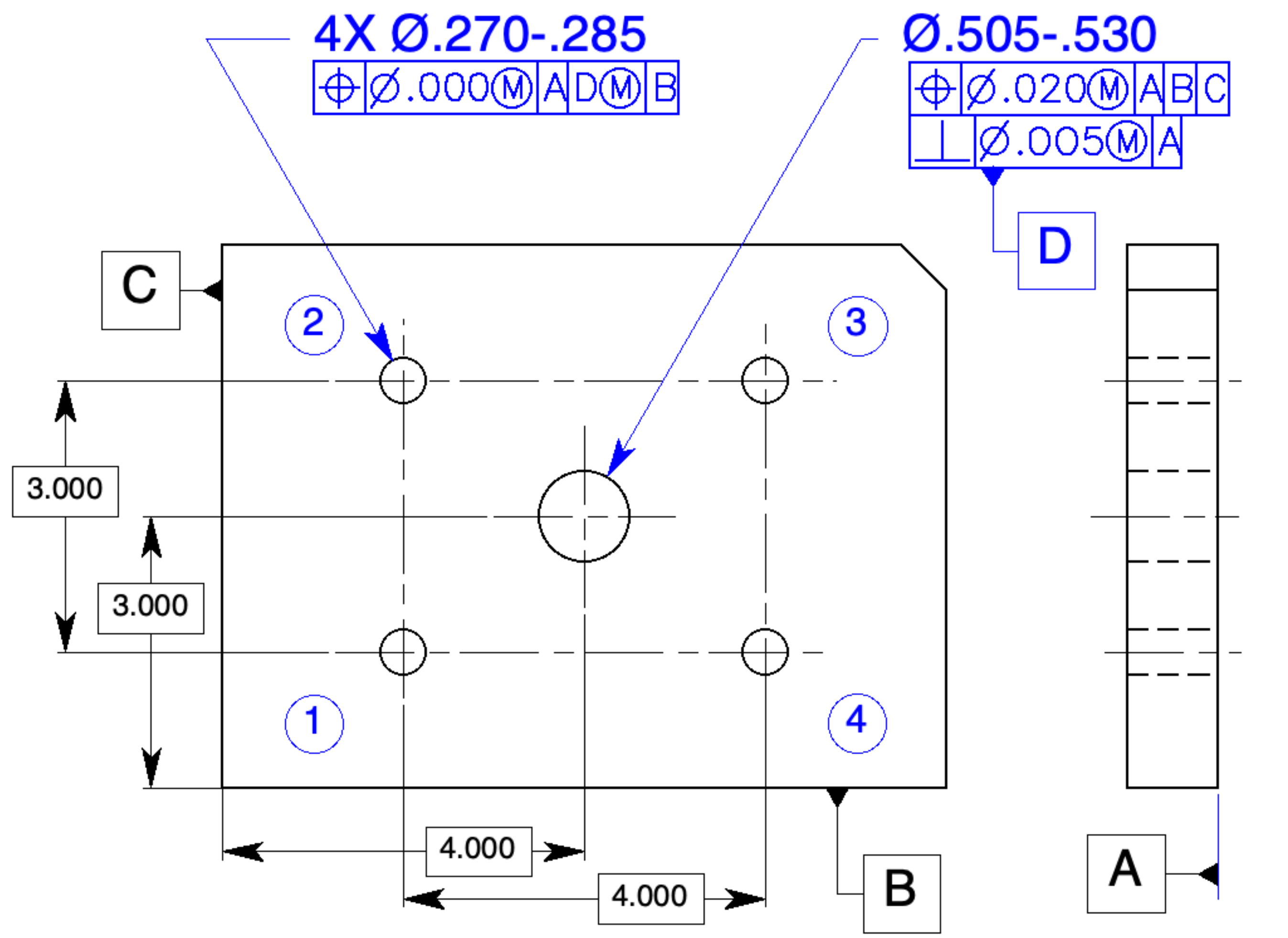


Figure 13-16 A pattern of features controlled to a size feature: Prob. 3.

Feature Number	Feature Location from Datum D X-Axis	Feature Location From Datum D Y-Axis	Actual Feature Size	Departure from MMC (Bonus)	Total Geometric Tolerance
1	-1.992	-1.493	Ø .278	*.008*	*Ø .008*
2	-1.993	1.509	Ø .280	*.010*	*Ø .010*
3	2.010	1.504	Ø .280	*.010*	*Ø .010*
4	2.010	-1.490	Ø .282	*.012*	*Ø .012*
Datum			Ø .520	Shift Tolerance = *Ø .020*	

Table 13-5 Inspection Data for Graphic Analysis of Prob. 3

3. A part was made from the drawing in Figure 13-16; the inspection data was tabulated in Table 13-5.

Perform a graphic analysis of the part. Is the pattern within tolerance? *Yes*

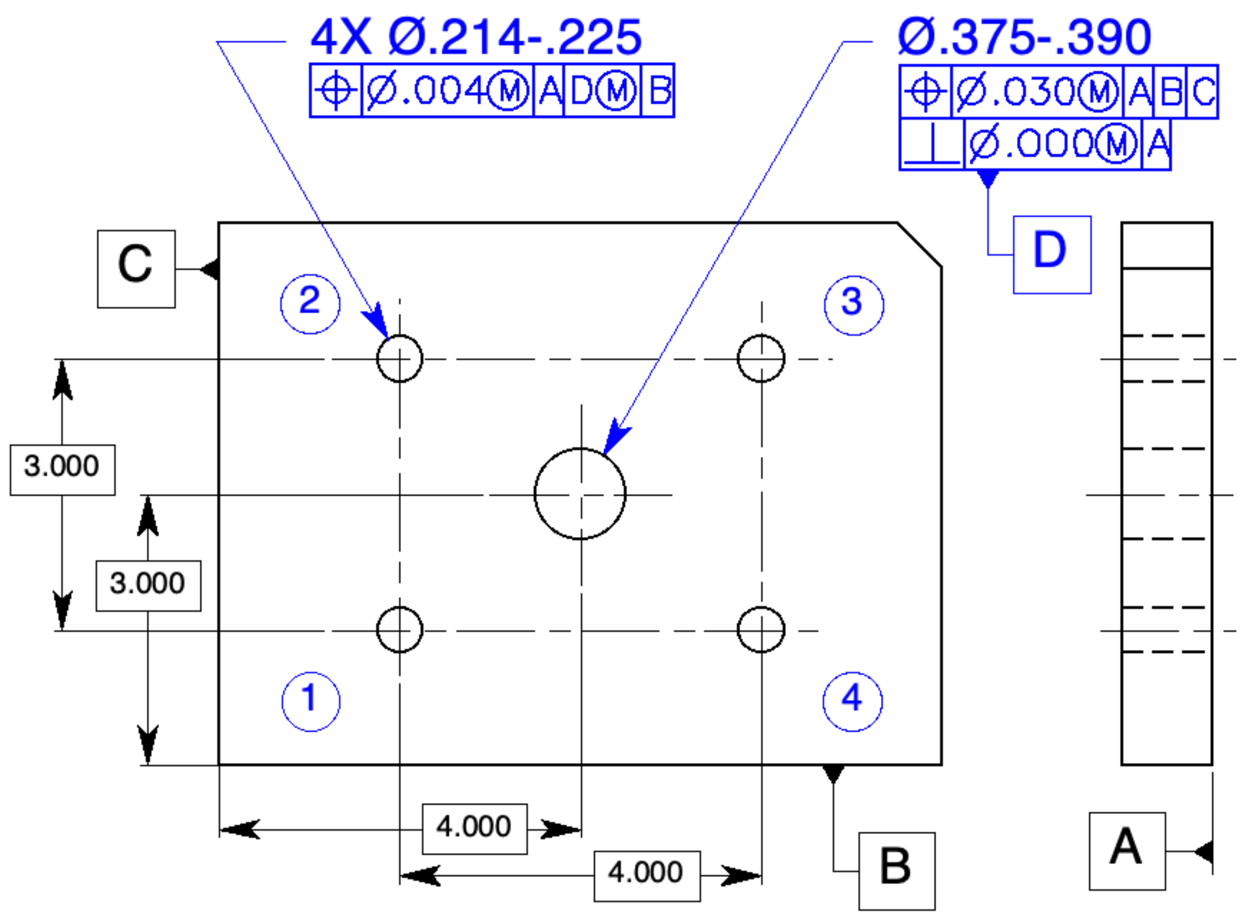


Fig. 13-17 A pattern of features controlled to a size feature: Prob. 4.

Feature Number	Feature Location from Datum D X-Axis	Feature Location From Datum D Y-Axis	Actual Feature Size	Departure from MMC (Bonus)	Total Geometric Tolerance
1	-1.995	-1.495	Ø .224	*.010*	*Ø .014*
2	-1.996	1.503	Ø .218	*.004*	*Ø .008*
3	2.005	1.497	Ø .220	*.006*	*Ø .010*
4	1.997	-1.506	Ø .222	*.008*	*Ø .012*
Datum			Ø .380	Shift Tolerance = *Ø .005*	

Table 13-6 Inspection Data for the Graphic Analysis of Prob. 4

4. A part was made from the drawing in Fig. 13-17; the inspection data was tabulated in Table 13-6.

   Perform a graphic analysis of the part. Is the pattern within tolerance? *No*

   If it is not in tolerance, can it be reworked, if so, how? *The pattern will be in tolerance if all holes are enlarged to their LMC size.*

Appendix A

Concentricity and Symmetry

Page 301

Note: *The concentricity and symmetry controls have been deleted from the ASME Y 14.5-2018 standard. However, many readers of this text continue to use or are required to read drawings toleranced to an earlier version of the standard. Consequently, the chapter in the previous (second) edition on concentricity and symmetry (Chap. 10) appears here in its entirety as App. A.*

1. Both concentricity and symmetry controls are reserved for a few *unique tolerancing applications*.

2. Both concentricity and symmetry employ the same tolerancing *concept*; they just apply to different *geometries*.

3. Concentricity is that condition where the median points of all diametrically opposed points of a surface of revolution are congruent with *the axis (or center point) of a datum feature*.

4. Concentricity is a *location* control. It has a *cylindrical* shaped tolerance zone that is coaxial with *the datum axis*.

5. Concentricity tolerance only applies on a *RFS* basis. It must have at least *one datum feature* that applies only at *RMB*.

6. For concentricity, the aggregate of all *median points* must lie within a *cylindrical* tolerance zone whose axis is coincident with the axis of the *datum feature*.

7. Concentricity can be inspected, for acceptance only, by placing a *dial indicator* on the toleranced surface of revolution and rotating the part about its *datum axis*.

8. To reject parts and to inspect features such as regular polygons and ellipses, the traditional method of *differential measurements* is employed.

9. The concentricity tolerance is often used to accurately control *balance* for high speed rotating parts.

10. Concentricity is time consuming and expensive to *inspect*, but less expensive to *manufacture* than the runout tolerance.

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

12. Symmetry is a *location* control.

13. Symmetry has a tolerance zone that consists of *two parallel planes* evenly disposed about the *center plane or axis* of the datum feature.

14. Symmetry tolerance applies only at *RFS*.

15. Symmetry must have at least one *datum feature* that only applies at *RMB*.

16. The aggregate of all *median points* must lie within a tolerance zone defined by *two parallel planes* equally disposed about the center plane or axis of the *datum feature*.

17. The symmetry tolerance is independent of both *size and form*.

18. Differential measurement excludes *size, shape, and form* while controlling the *median points* of the feature.

19. The symmetry tolerance is often used to accurately control *balance* for rotating parts or to insure equal *wall thickness*.

20. Specify symmetry only when it is necessary because it is *time consuming and expensive* to manufacture and inspect.

Problems

Page 303

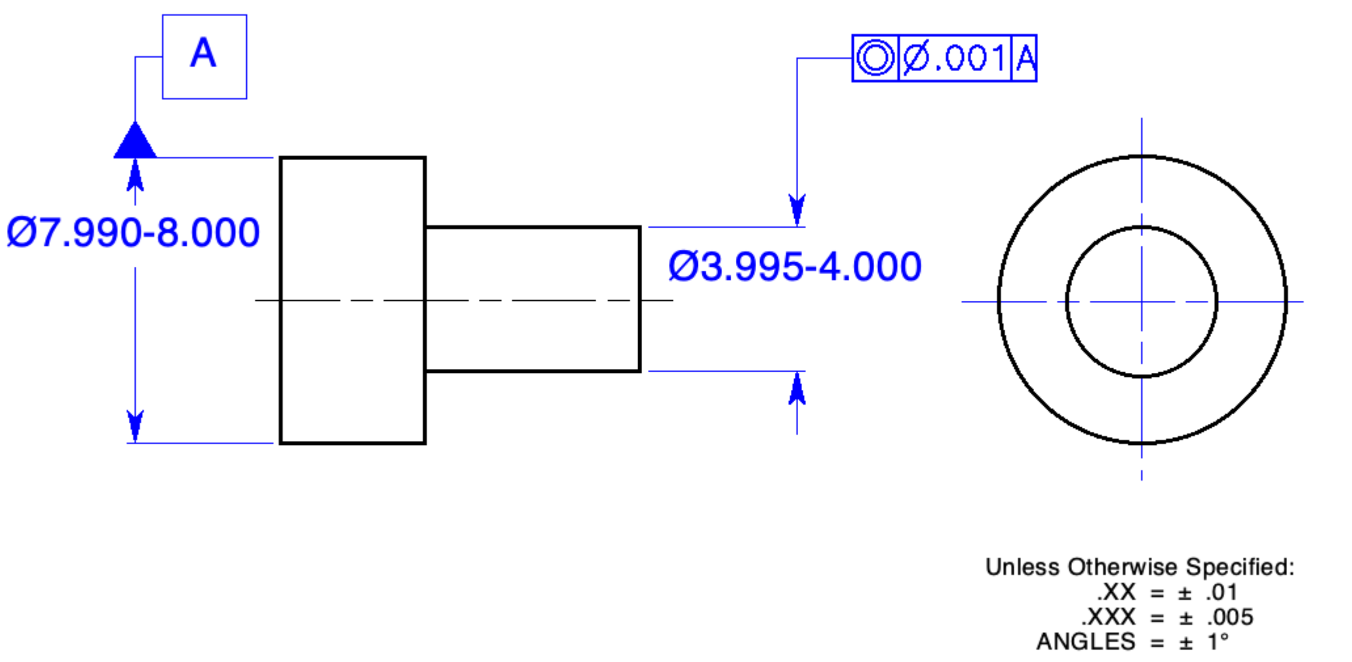


Figure A-7 Controlling the coaxiality of a cylinder: Prob. 1.

1. The mass of this high-speed rotating part in Fig. A-7 must be accurately balanced. The form of the surface is sufficiently controlled by the size tolerance. Specify a coaxiality control for the axis of the 4-inch diameter within a cylindrical tolerance of .001 at RFS to datum feature A at RMB.

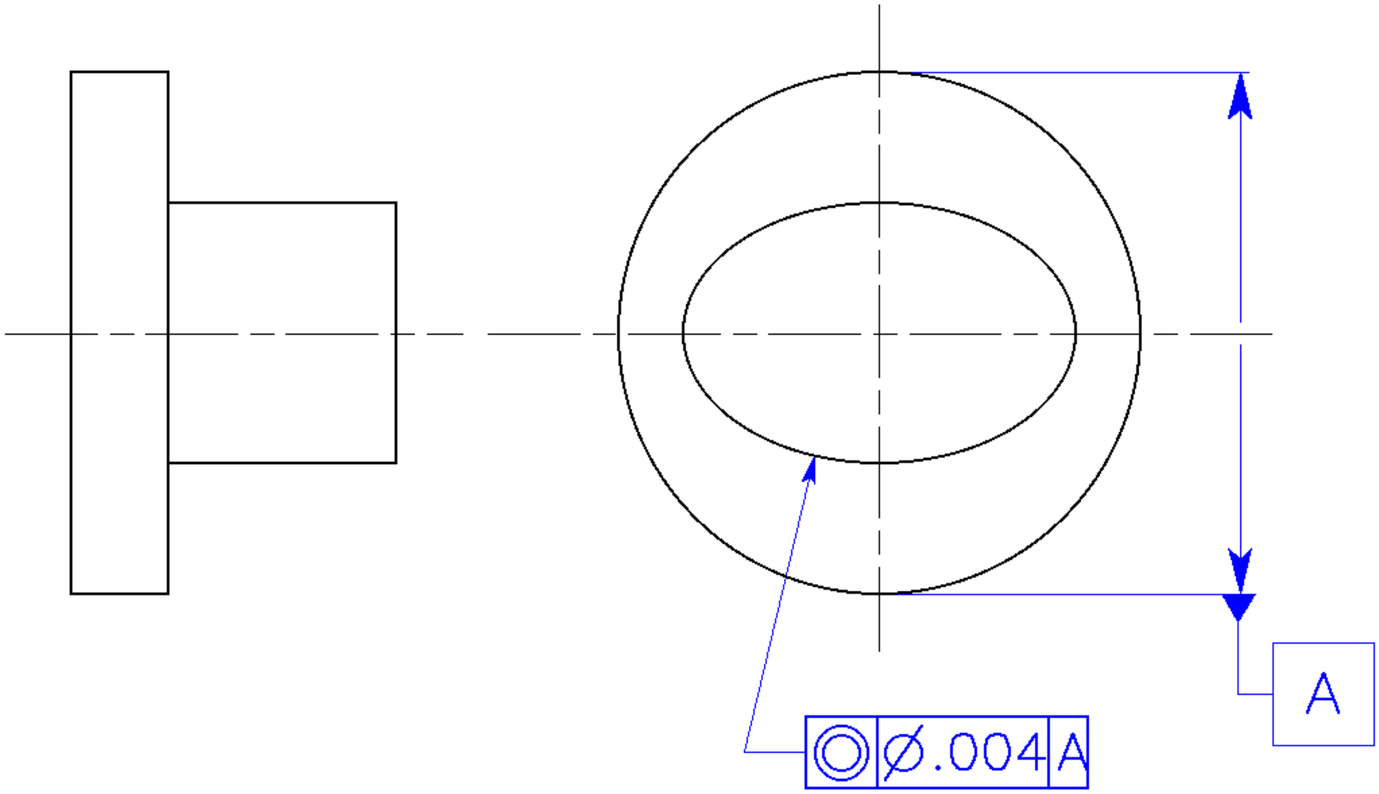


Figure A-8 Controlling the coaxiality of an ellipse: Prob. 2.

2. The mass of the ellipse in Fig. A-8 must be accurately balanced. Specify a coaxiality control that will locate the median points of the ellipse within a cylindrical tolerance of .004 at RFS to datum feature A at RMB.

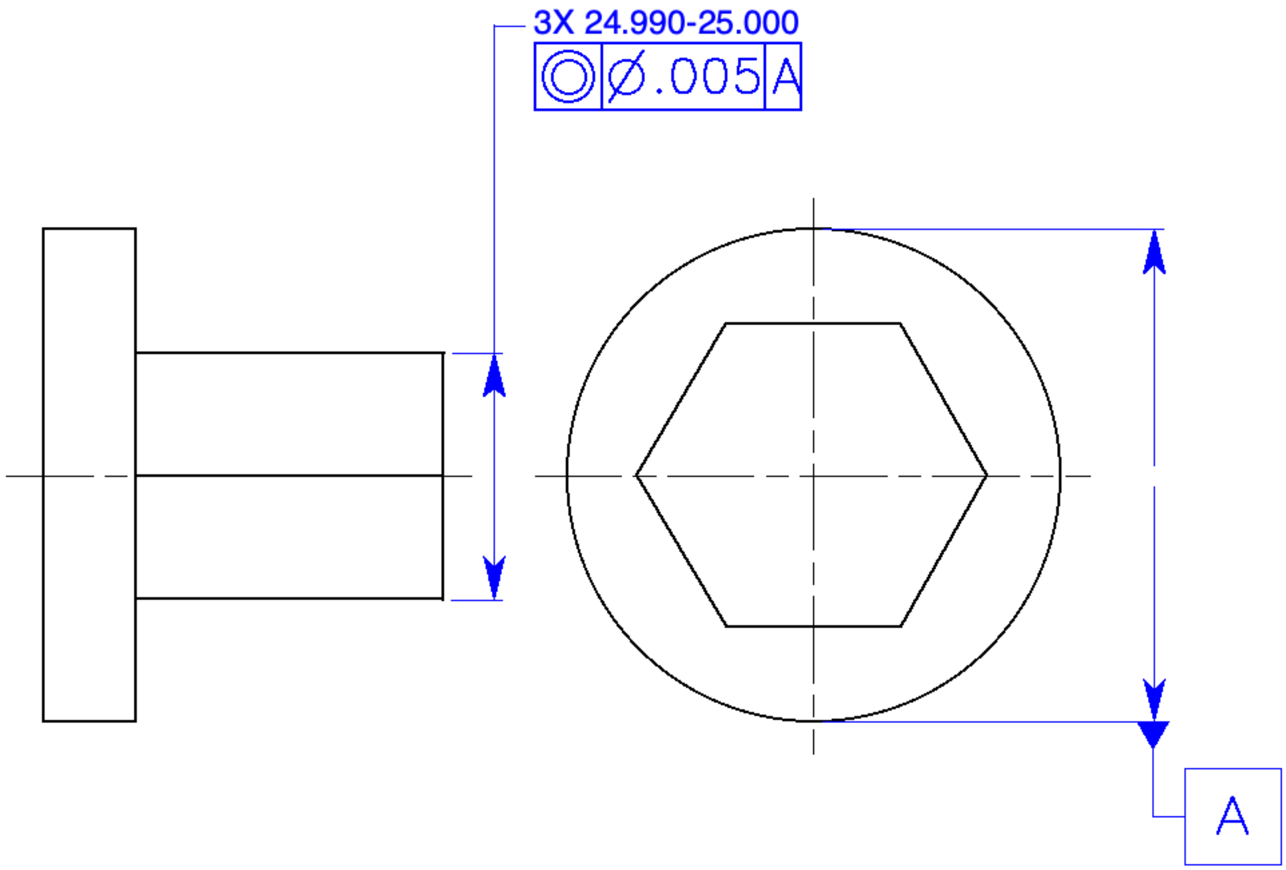


Figure A-9 Controlling the coaxiality of the hexagon: Prob. 3.

3. The mass of the hexagon in Fig. A-9 must be accurately balanced. Specify a coaxiality control for the median points of the hexagon within a cylindrical tolerance of .005 at RFS to datum feature A at RMB.

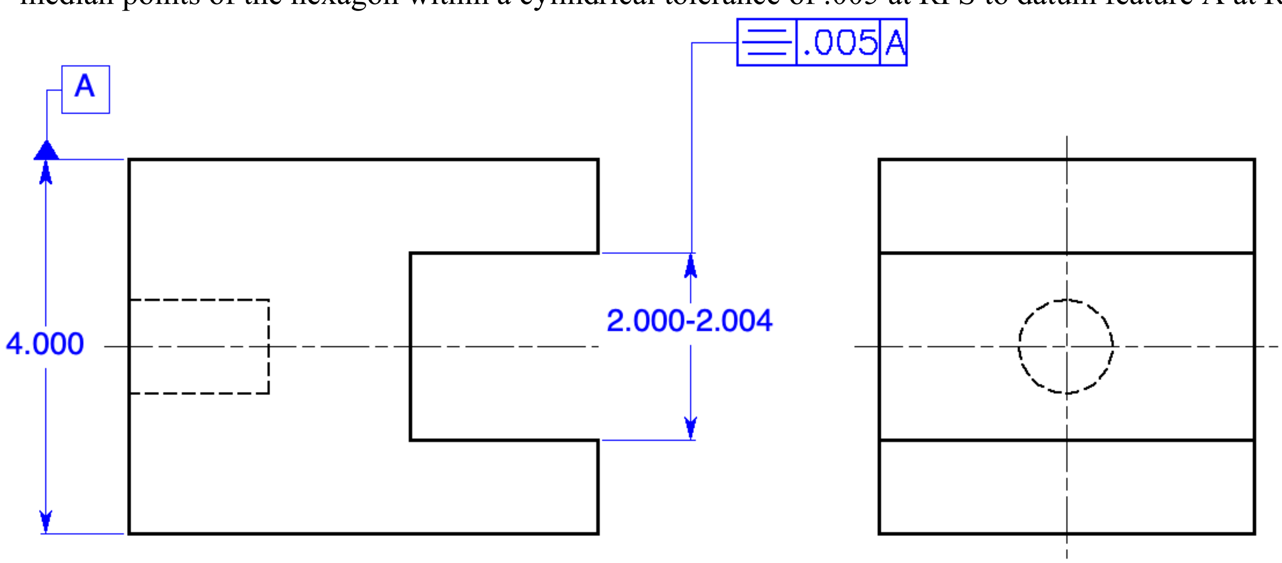


Figure A-10 Controlling the symmetry of a slot: Prob. 4.

4. The mass of the part in Fig. A-10 must be accurately balanced. Specify a geometric tolerance that will centrally locate the slot in this part within a tolerance of .005 at RFS to datum feature A at RMB.