Test Answers and Exam Booklet

Geometric Tolerancing

Contents

Geometric Tolerancing

ANSWERS TO THE GEOMETRIC TOLERANCING TEST

PART 1. QUESTIONS

1. *Give a brief definition of geometric tolerancing (GT).* Geometric tolerancing is the dimensioning and tolerancing of a part's individual features where the permissible variations relate to characteristics of form, profile, orientation, runout, or the relationship between features.

2. *Give an example of how general tolerancing has some control on form.* General tolerancing has some control on form by establishing inexact limits of a feature's size without revealing the feature's actual dimensions.

3. *Define* ***datum.*** A datum is the theoretically perfect surface, plane, point, or axis that's established from the true geometric *counterpart* of the datum feature.

4. *Describe or show an example of how a datum feature symbol is shown on a plane surface and a centerline axis.* A datum feature symbol is shown on a plane surface either by being placed directly on the edge view of the surface or by being placed on an extension line from the edge view of the surface. If the symbol is placed on the extension line, it must be placed offset from the dimension line arrowheads. A datum feature symbol is shown on a centerline axis either by being aligned with the dimension line arrowhead or by being placed on the feature, leader shoulder, or feature control frame.

5. *What is another name for the three-plane dimensioning system?* Another name for the three-plane dimensioning system is a datum reference frame.

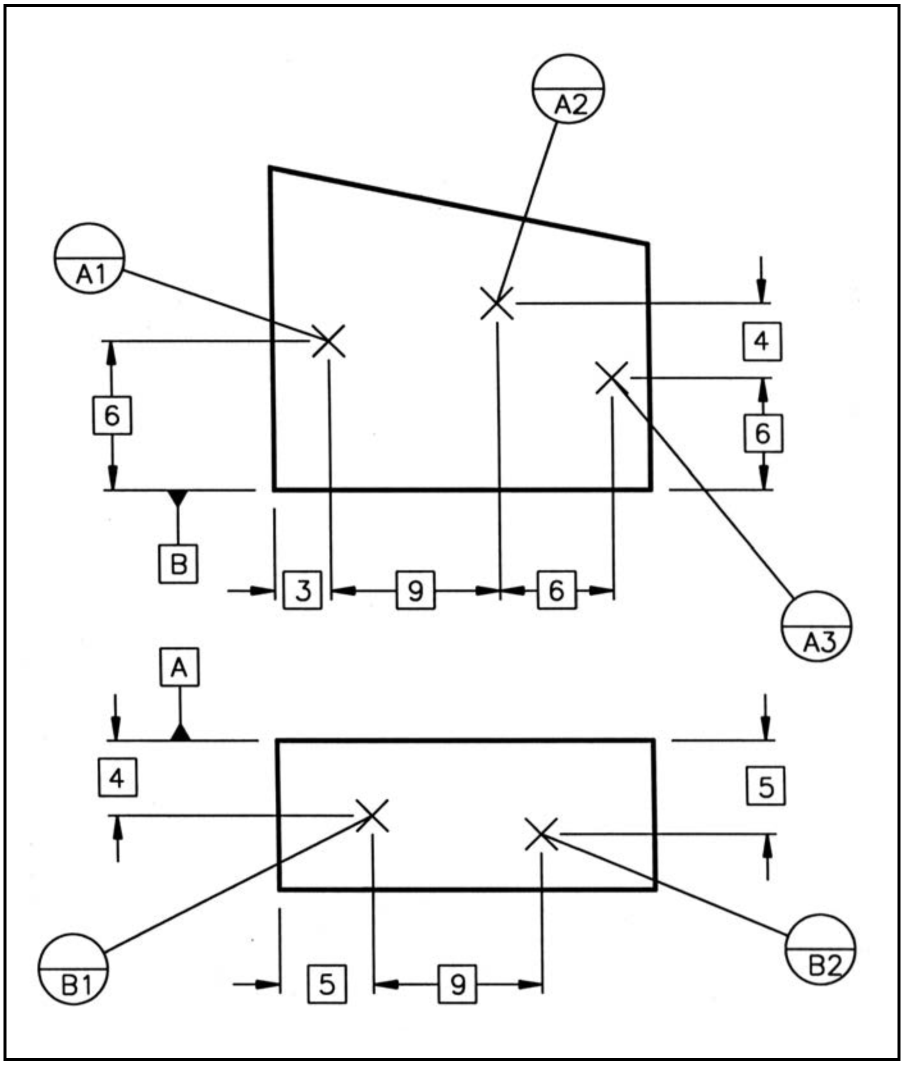
6. *Name the elements of the three-plane dimensioning system.* The elements of the three-plane dimensioning system are the primary datum plane, the secondary datum plane, and the tertiary datum plane.

7. *Discuss the order of precedence of datum features.* The primary datum feature is the one that has the most functional importance. The secondary feature is next in importance, and the third is last. Labeling datum features can allow you to relate the part to the data reference frame. By listing the datum reference letters from left to right in the feature control frame, you can reveal your desired order of precedence.

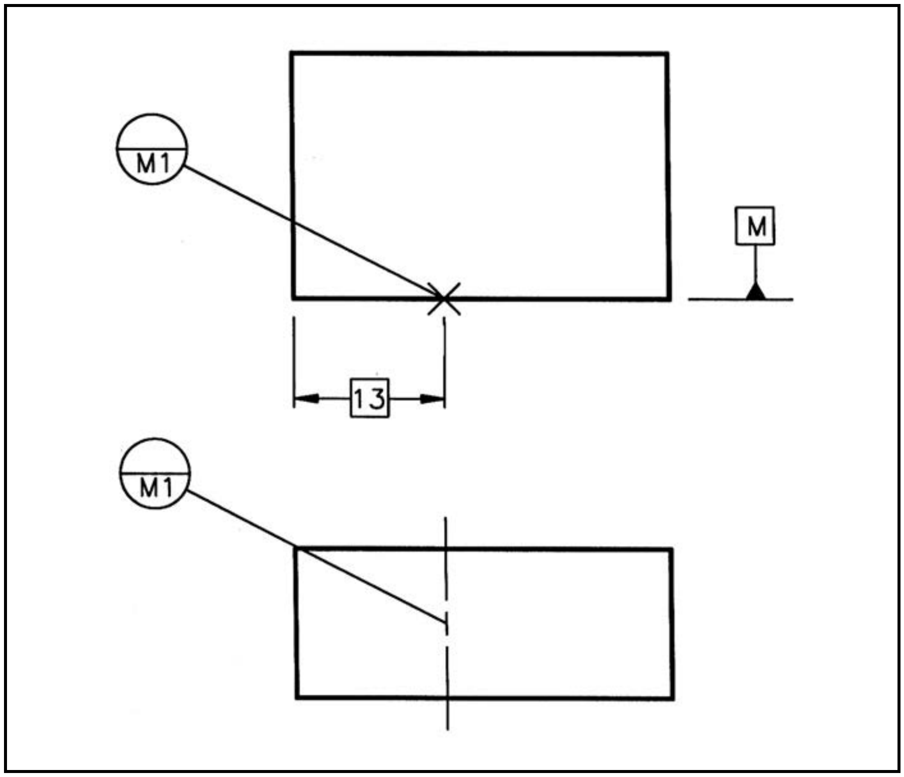
8. *When is it practical to use a partial surface datum?* Using a partial surface datum is practical when an entire surface has related features located in one or more concentrated places.

9. *Describe how coplanar surface datums are drawn.* Coplanar surface datums are drawn so that the two datum features are shown on two different surfaces, but on the same plane.

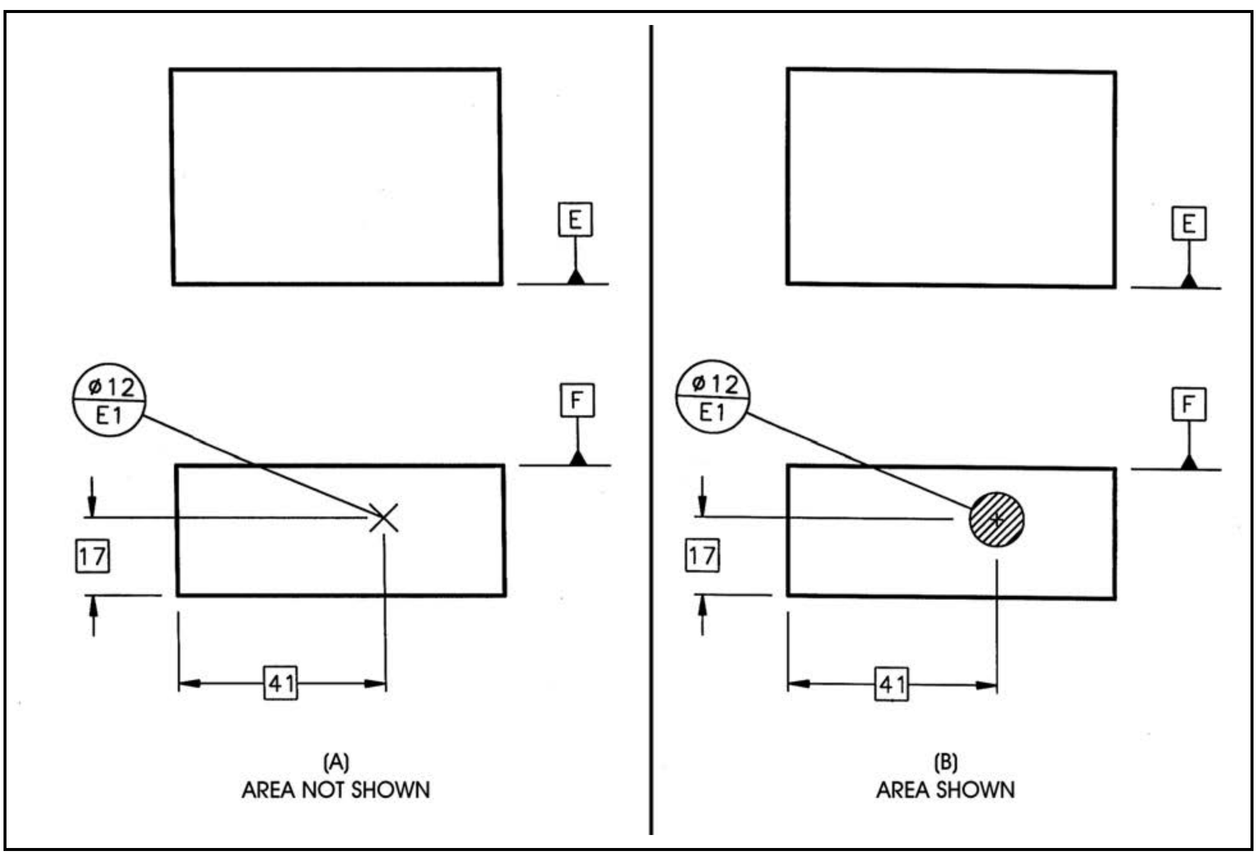
10. *Show an example of how datum target points are located and identified on a primary datum surface.* An example of datum target points on a primary datum surface is shown here.



11. *Show an example of how a datum target line may be shown on a drawing.* An example of a datum target line follows.

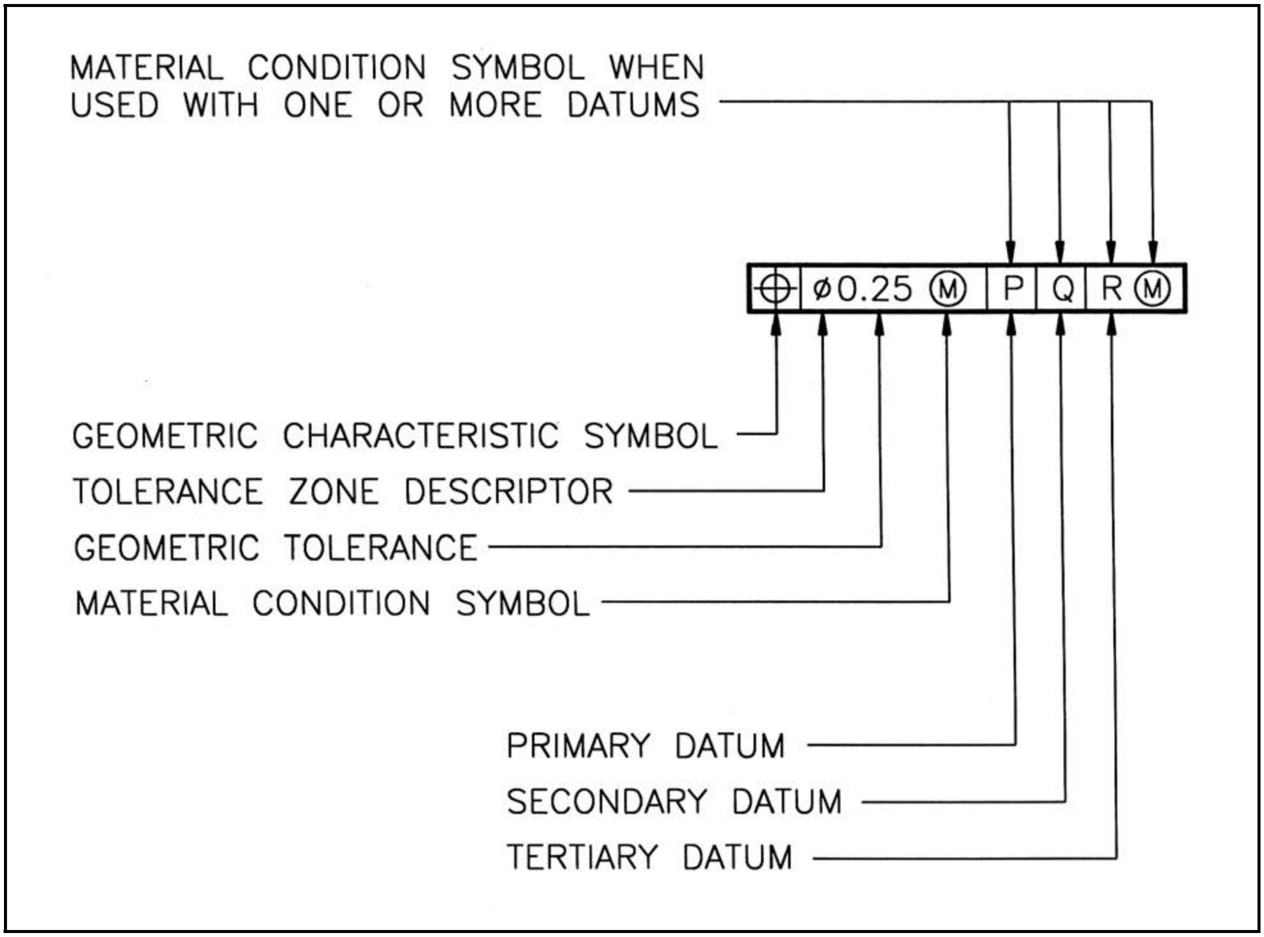


12. *Show an example of how datum target areas may be shown on a drawing.* In this illustration is an example of how a datum target area may be shown on a drawing.

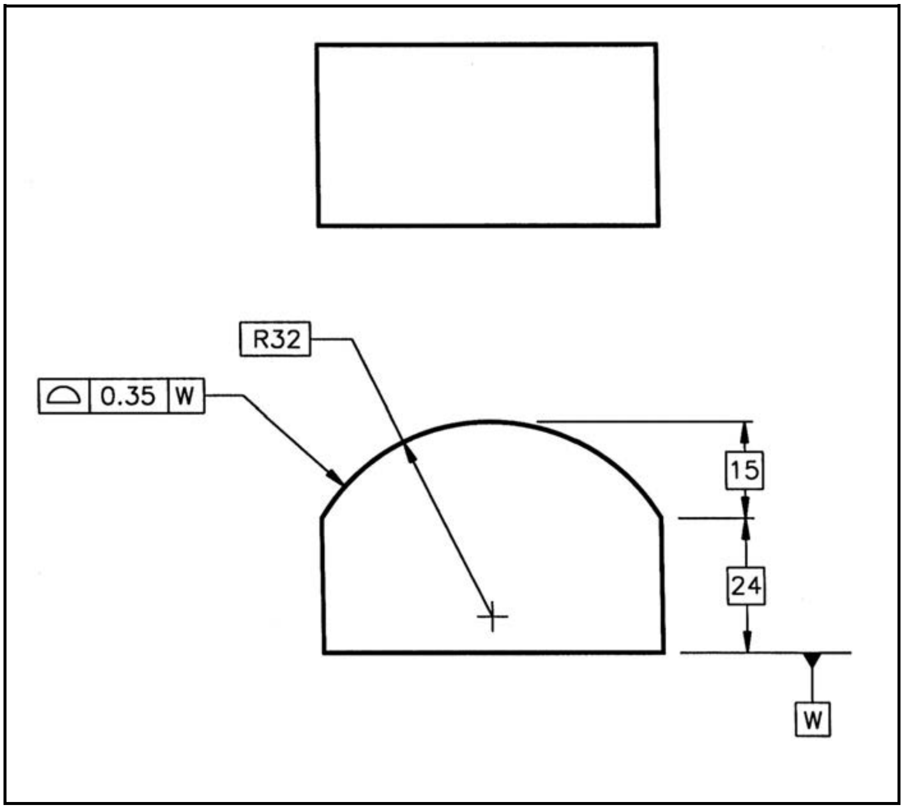


13. *What is a feature control frame used for?* A feature control frame is used to relate a geometric tolerance to a part feature.

14. *Show and label the complete order of elements in a feature control frame.* The complete order of elements in a feature control frame is illustrated here.



15. *Define basic and show an example of basic dimension.* In the phrase "basic dimension," the word "basic" refers to the basis from which permissible variations are established. A basic dimension is thus any size or location dimension used to identify the theoretically exact size, profile, orientation, or location of a feature or datum target. An example of how basic dimensions would appear on a drawing is shown here.



16. *List the geometric characteristics for form, profile, orientation, location, and runout.*

Form

- straightness
- flatness
- circularity
- cylindricity

Profile

- profile of a line
- profile of a surface

Orientation

- parallelism
- perpendicularity
- angularity

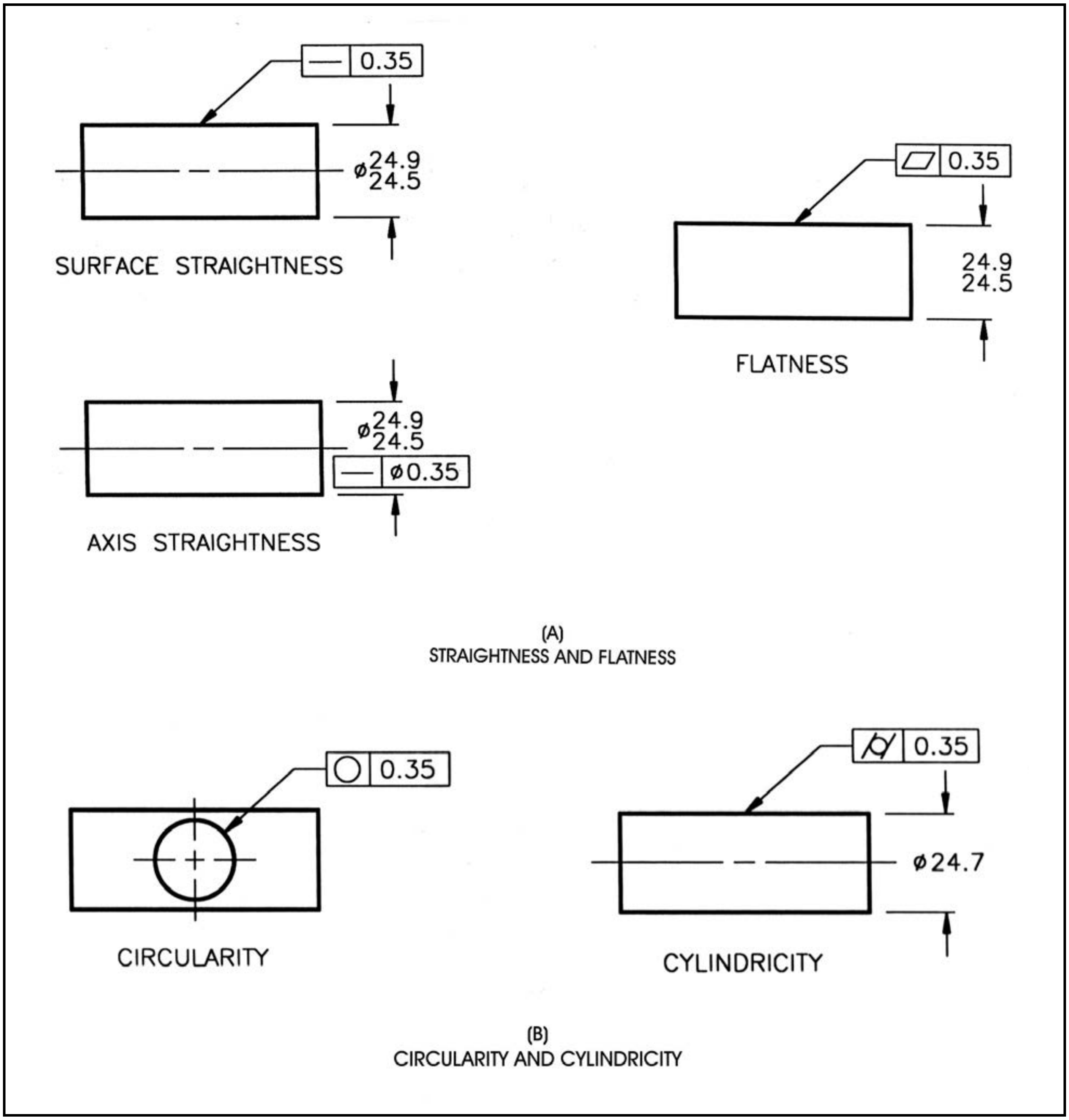
Location

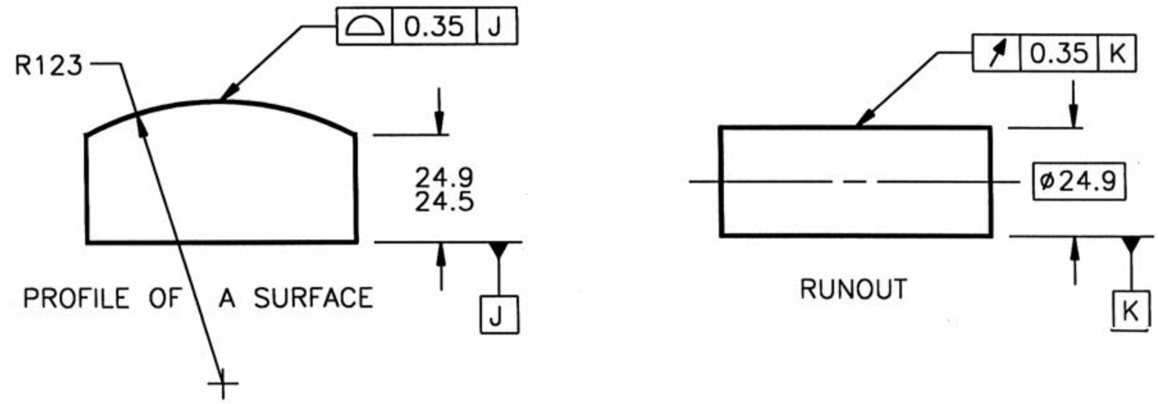
- concentricity
- symmetry
- position

Runout

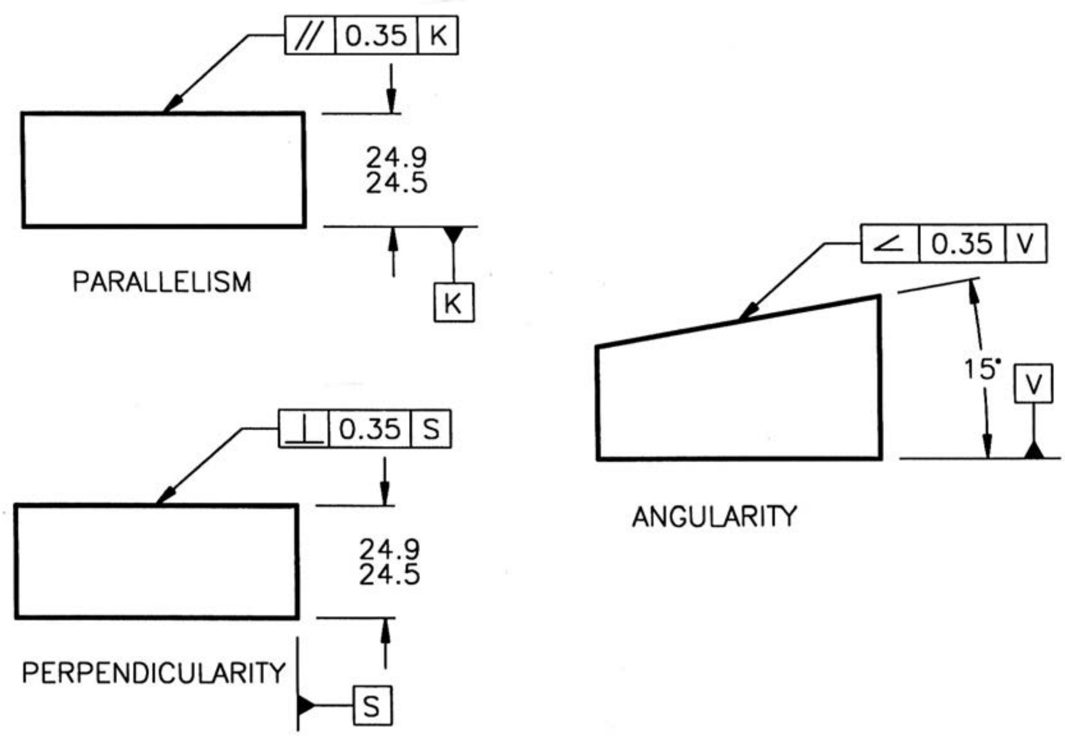
- circular runout
- total runout

17. *Show examples of how the following geometric characteristics are represented on a drawing: straightness, flatness, circularity, cylindricity, profile of a surface, parallelism, perpendicularity, angularity, and runout.* Examples of these geometric characteristics are shown here.





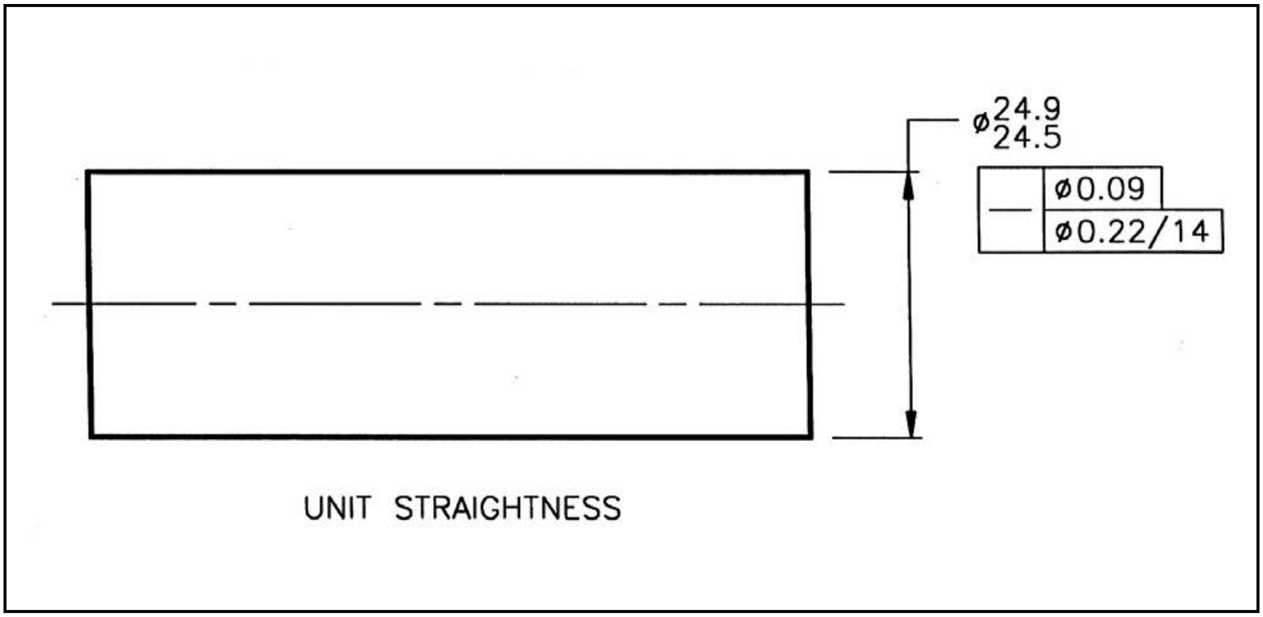
(C)
PROFILE OF A SURFACE AND RUNOUT



(D)
PARALLELISM, PERPENDICULARITY, AND ANGULARITY

18. *Which geometric tolerance is more confining, circularity or cylindricity?* Cylindricity is a more confining geometric tolerance than circularity.

19. *Show an example of how unit straightness may be shown on a drawing.* How unit straightness can be indicated on a drawing is shown here.



20. *Which profile tolerance is most confining?* The profile tolerance that's most confining is the profile of a surface because it affects all the surface elements of a feature equally.

21. *Must the parallelism geometric tolerance zone be within the related size tolerance?* Yes, the geometric tolerance must fall within the size tolerance of the part.

22. *What does the note, EACH ELEMENT, denote when placed below a feature control frame?* When placed below a feature control frame, the note EACH ELEMENT indicates that a single line element is being specified (rather than the surface).

23. *Is a basic angle required to establish an angular surface for an angularity geometric tolerance?* Yes, a basic angle from a datum must be specified.

24. *Name the runout tolerance that is most confining.* The most confining runout tolerance is total runout.

25. *How are the specified limits of total runout determined?* The specified limits of total runout are determined by placing a chain line in the linear view adjacent to the desired location. The chain line is located with basic dimensions.

26. *Describe or show an example of how a surface portion can have a specified runout tolerance.* A portion of a surface can have a specified runout tolerance if you don't want to control the whole surface by that tolerance. This type of runout tolerance is indicated by placing a chain line near the particular portion of the surface. The feature control frame points to the chain line.

27. *Define* ***perfect form envelope.*** A perfect form envelope is the true geometric form of a feature established at maximum material condition (MMC).

28. *Clearly define regardless of feature size (RFS).* Regardless of feature size means that the geometric tolerance remains as the specified value regardless of the actual size of the feature.

29. *When is RFS implied?* RFS is always implied for all geometric characteristics and related datums unless otherwise specified.

30. *Clearly define maximum material condition (MMC) as related to the effect of MMC on the geometric tolerance.* As the actual produced size of a feature departs from the maximum material condition toward the least material condition (LMC), the geometric tolerance is allowed to increase equal to the change from MMC. The maximum amount of change is at the LMC produced size.

31. *Clearly define least material condition (LMC) as related to the effect of LMC on the geometric tolerance.* As the actual produced size deviates from LMC toward MMC, the tolerance is allowed an increase equal to the amount of change from LMC. An LMC value is never permitted to be less than LMC.

32. *True or False: position tolerances must have a correlated material condition symbol (MMC or LMC) applied to the tolerance and related datums.* False. MMC or LMC is applied to the position tolerance and datum reference only as needed for design.

33. *Define the concentricity geometric characteristic.* The concentricity geometric characteristic is the relationship of the axes of cylindrical shapes. Perfect concentricity exists when the axes of two or more cylindrical features are in perfect alignment.

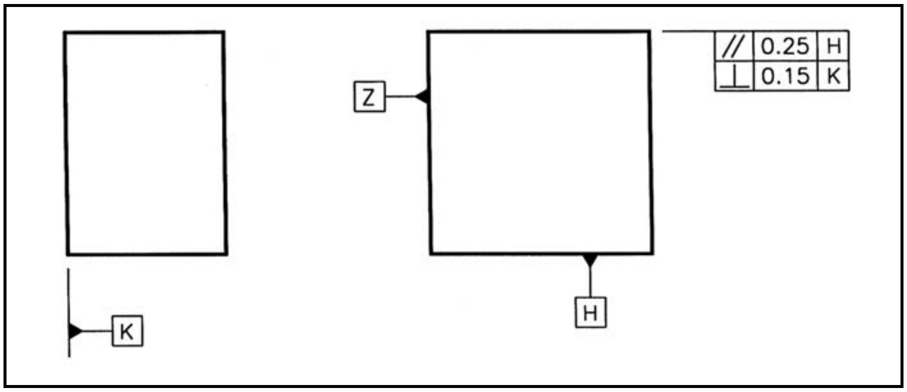
34. *Define the symmetry geometric characteristic.* The symmetry geometric characteristic is the center plane relationship between two or more features. Perfect symmetry exists when the center plane of two or more features is in alignment.

35. *Describe the position tolerance zone and how it is located.* The position tolerance zone is a zone within which the center, axis, or center plane of a feature or size is permitted to vary from the true position. The position tolerance zone is located by basic dimensions.

36. *Define* ***true position.*** True position is the theoretically exact position of a center, axis, or center plane of a feature.

37. *Describe a projected tolerance zone.* A projected tolerance zone is used in situations where there's the possibility of interference with mating parts. In cases such as this, the tolerance zone could be extended or projected away from the primary datum controlling the axis of the related feature. (The amount of projection is made equal to the thickness of the mating part.)

38. *Show an example of a combination control.* An example of a combination control follows.



Part 2. Calculations

1. *Given:*
   a. Shaft ∅24.00 / 23.92.
   b. Straightness geometric tolerance 0.02.

   What is the geometric tolerance at the actual sizes specified below for the type of straightness and material condition shown? Compare your answers with those in the table shown here.

ACTUAL SIZE	SURFACE STRAIGHTNESS	AXIS STRAIGHTNESS	
	RFS	RFS	MMC
24.00	0	.02	.02
23.99	.01	.02	.03
23.98	.02	.02	.04
23.96	.02	.02	.06
23.94	.02	.02	.08
23.92	.02	.02	.10

2. *Given:*
   a. Positional tolerance ∅0.02 at true position in reference to datums L, M, N.
   b. Hole size ∅8.50 / 8.40.

   What is the positional tolerance usingdifferent material condition symbols at the actual sizes specified below? Compare your answers with those in the table shown here.

ACTUAL SIZES	MATERIAL CONDITION AS APPLIED TO TOLERANCE		
	MMC	RFS	LMC
8.50	.12	.02	.02
8.49	.11	.02	.03
8.48	.10	.02	.04
8.46	.08	.02	.06
8.44	.06	.02	.08
8.42	.04	.02	.10
8.40	.02	.02	.12

3. *If the positional tolerance of the hole in problem 2 (above) is zero at MMC, then what would the positional tolerance be at the actual produced sizes given below?* Compare your answers with those in the table shown here.

ACTUAL SIZES	MMC
8.50	.10
8.48	.08
8.46	.06
8.44	.04
8.42	.02
8.40	0

4. *What is the virtual condition of a ∅12.2/12.0 hole that is located with a positional tolerance of ∅0.05 at MMC?* The virtual condition of the hole is equal to MMC of the hole minus the geometric tolerance: 12.0 – 0.05 = a diameter of 11.95.

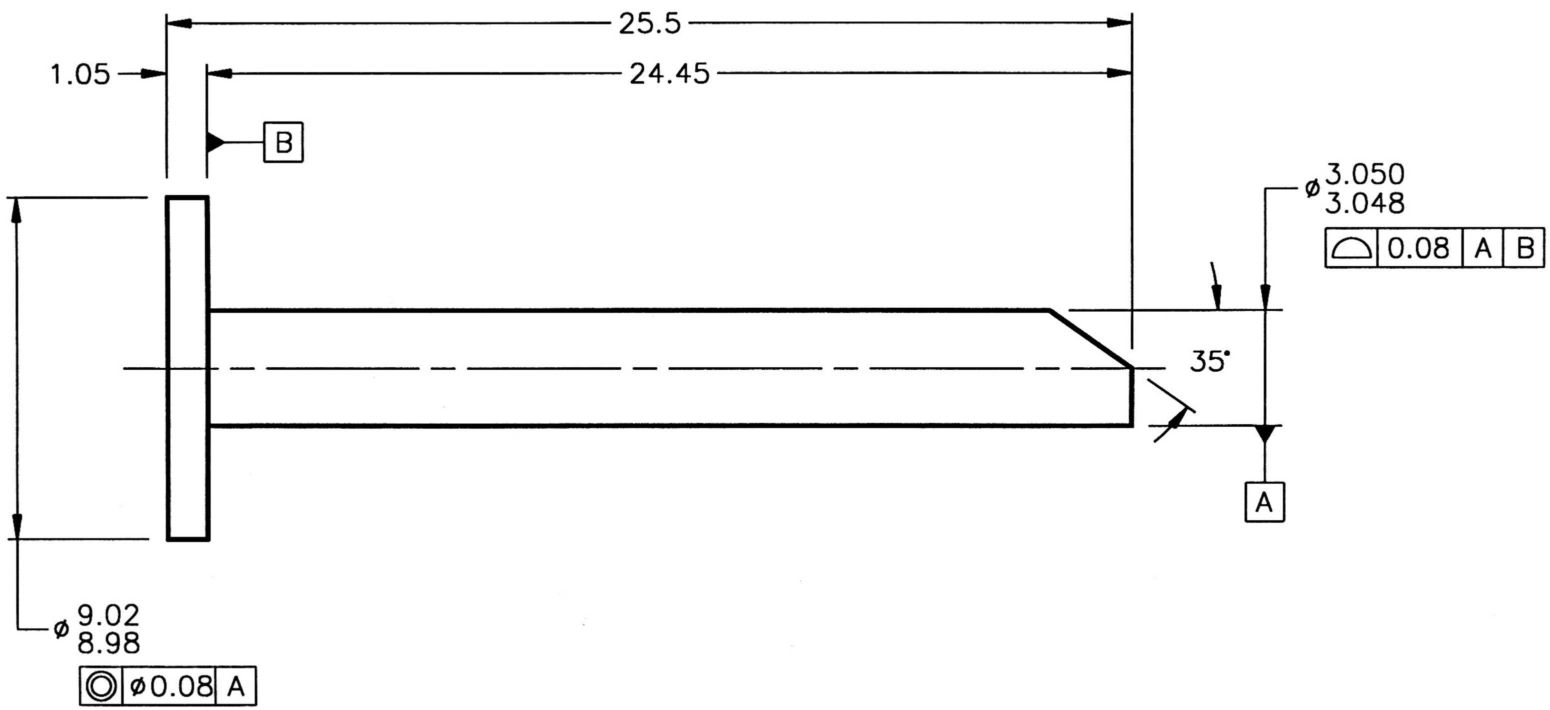
5. *What is the virtual condition of a ∅6.0/5.9 pin that is established with perpendicularity to a datum A by ∅0.02 at MMC?* The virtual condition of the pin is equal to MMC of the pin plus the geometric tolerance: 6.0 + 0.02 = a diameter of 6.02.

6. *Calculate the positional tolerance for the location of holes with the following specifications:*
   a. Floating fastener.
   b. Fastener: M20 × 2.5.
   c. Hole through two parts: ∅21.2 / 20.8.
   Positional tolerance for holes in part 1 equals 0.8, part 2 equals 0.8. (MMC of the hole minus MMC of fastener: 20.8 – 20 = 0.8)

7. *Calculate the positional tolerance for the location of holes with the following specifications:*
   a. Fixed fastener.
   b. Part 1 hole: ∅9.0 / 8.6
   c. Part 2 hole: M8 × 1.25.
   d. Equal positional tolerance for each part.
   Positional tolerance for holes in part 1 equals 0.3, part 2 equals 0.3. (Half the quantity MMC of the hole minus MMC of the fastener's thread: $\frac{8.6 - 8}{2} = 0.3$)

SAMPLE ANSWERS TO THE GEOMETRIC TOLERANCING PROBLEMS

Sample answers to the six problems are provided on the following drafting plates. Compare your finished drawings with those shown on the plates.



2. REMOVE ALL BURRS AND SHARP EDGES.
1. INTERPRET DIMENSIONS AND TOLERANCES PER ASME Y14.5M–1994.

NOTES:

UNSPECIFIED TOLERANCES:

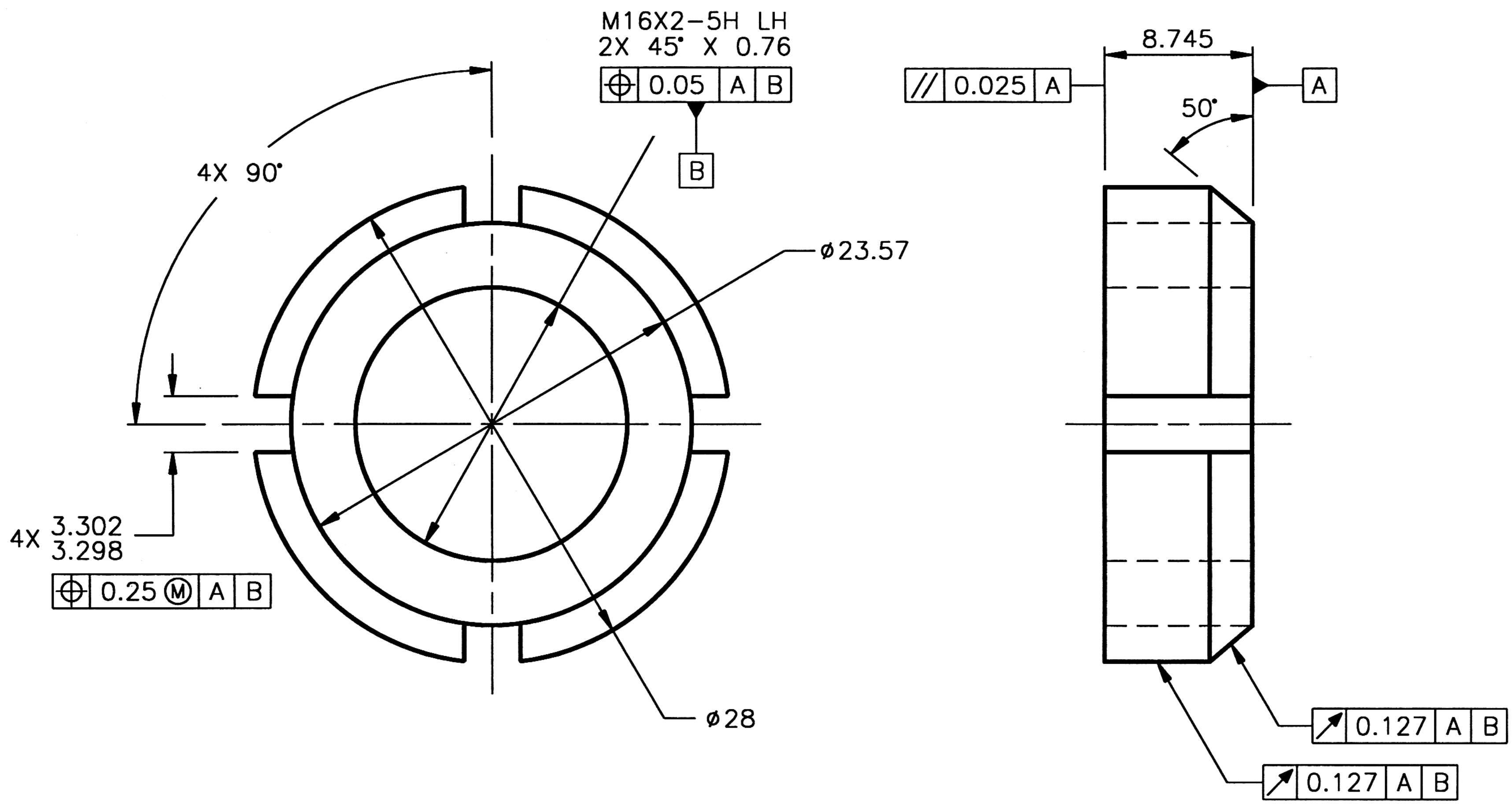
DECIMALS	mm	IN.
X	±2.5	±.1
XX	±0.25	±.01
XXX	±0.127	±.005
ANGULAR ±30'		
FINISH	3.2 μm	125 μIN.

FLOW PIN

DRAWING NO: PROBLEM 13–1

SCALE: 1:5

MATERIAL: BRONZE   FINISH ALL OVER 0.20 μm.



4. FINISH ALL OVER 1.6 μm.
3. △3: MARK PER AS478 CLASS D WITH 1193125 AND APPLICABLE DASH NUMBER.
2. REMOVE ALL BURRS AND SHARP EDGES.
1. INTERPRET DIMENSIONS AND TOLERANCES PER ASME Y14.5M-1994.

NOTES:

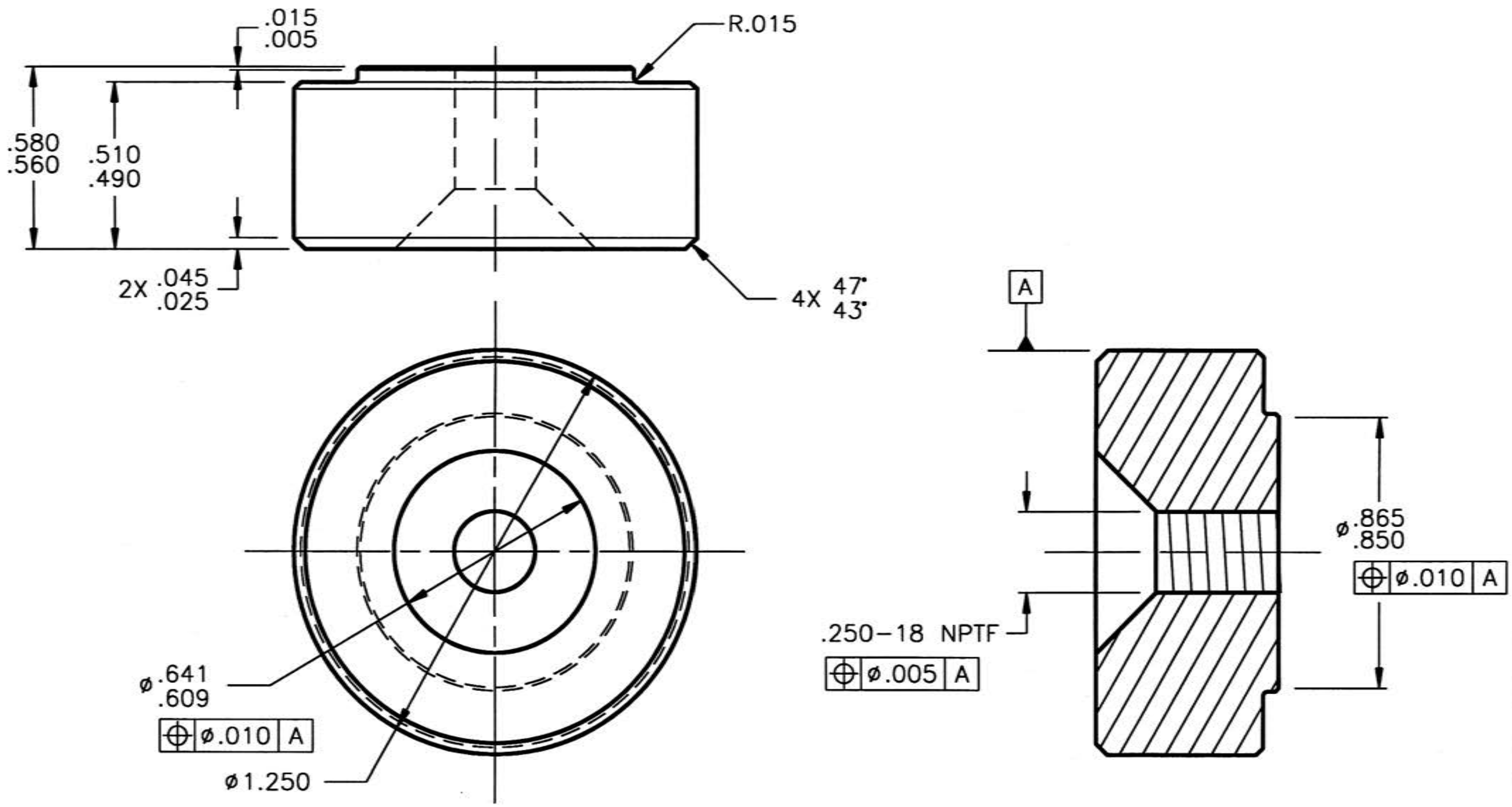
UNSPECIFIED TOLERANCES:

DECIMALS	mm	IN.
X	±2.5	±.1
XX	±0.25	±.01
XXX	±0.127	±.005
ANGULAR ±30'		
FINISH	3.2 μm	125 μIN.

LN2 TEST PUMP LOCK NUT

DRAWING NO:	SCALE:
PROBLEM 13-2	1:10

MATERIAL:
AMS 5732



2. REMOVE ALL BURRS AND SHARP EDGES.
1. INTERPRET DIMENSIONS AND TOLERANCES PER ASME Y14.5M–1994.

NOTES:

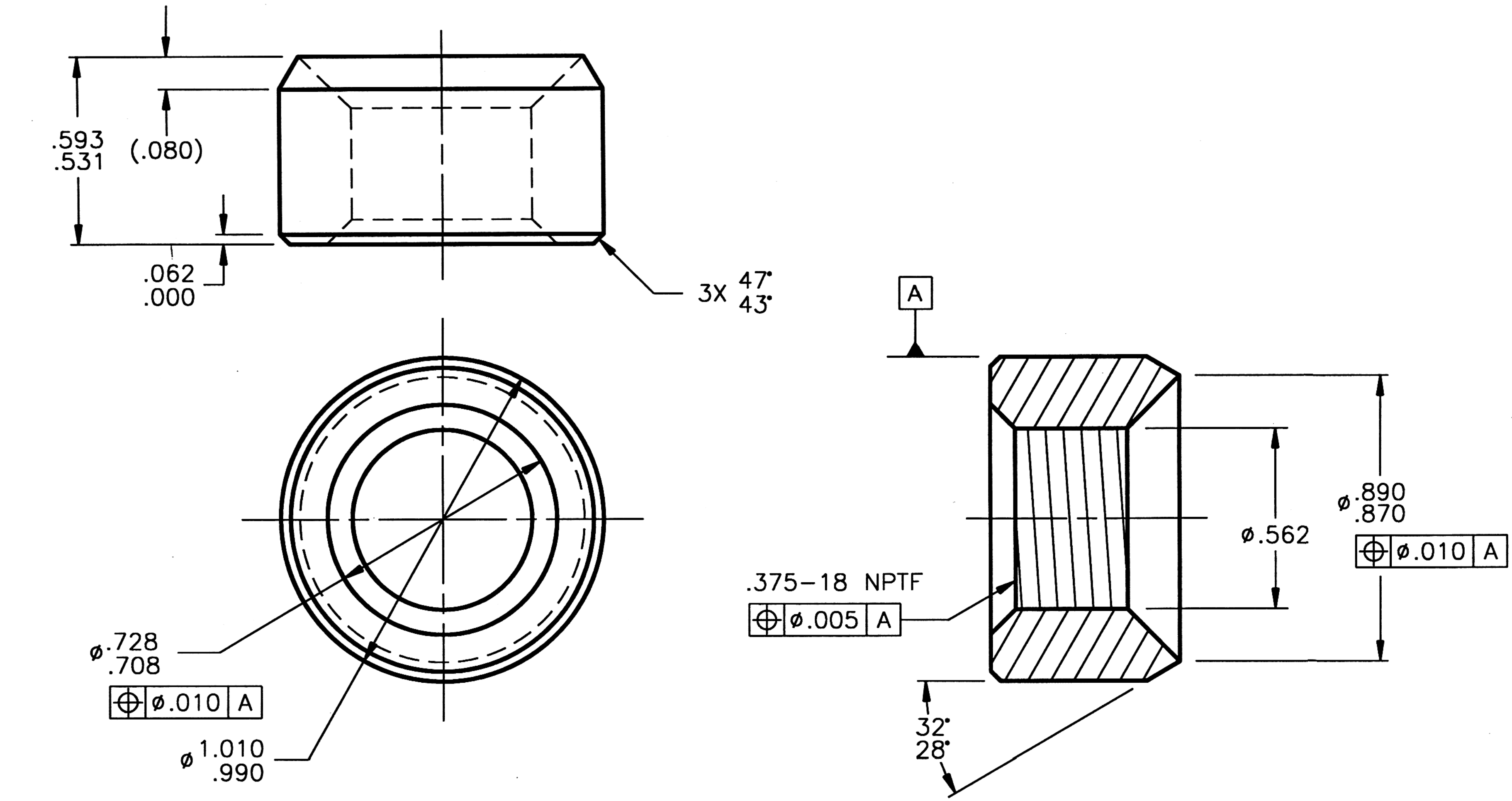
UNSPECIFIED TOLERANCES:

DECIMALS	mm	IN.
X	±2.5	±.1
XX	±0.25	±.01
XXX	±0.127	±.005
ANGULAR ±30'		
FINISH	3.2 μm	125 μIN.

HALF COUPLING

DRAWING NO:	SCALE:
PROBLEM 13–3	2:1

MATERIAL:
ø1.250 6061–T6 ALUMINUM



2. REMOVE ALL BURRS AND SHARP EDGES.
1. INTERPRET DIMENSIONS AND TOLERANCES PER ASME Y14.5M–1994.

NOTES:

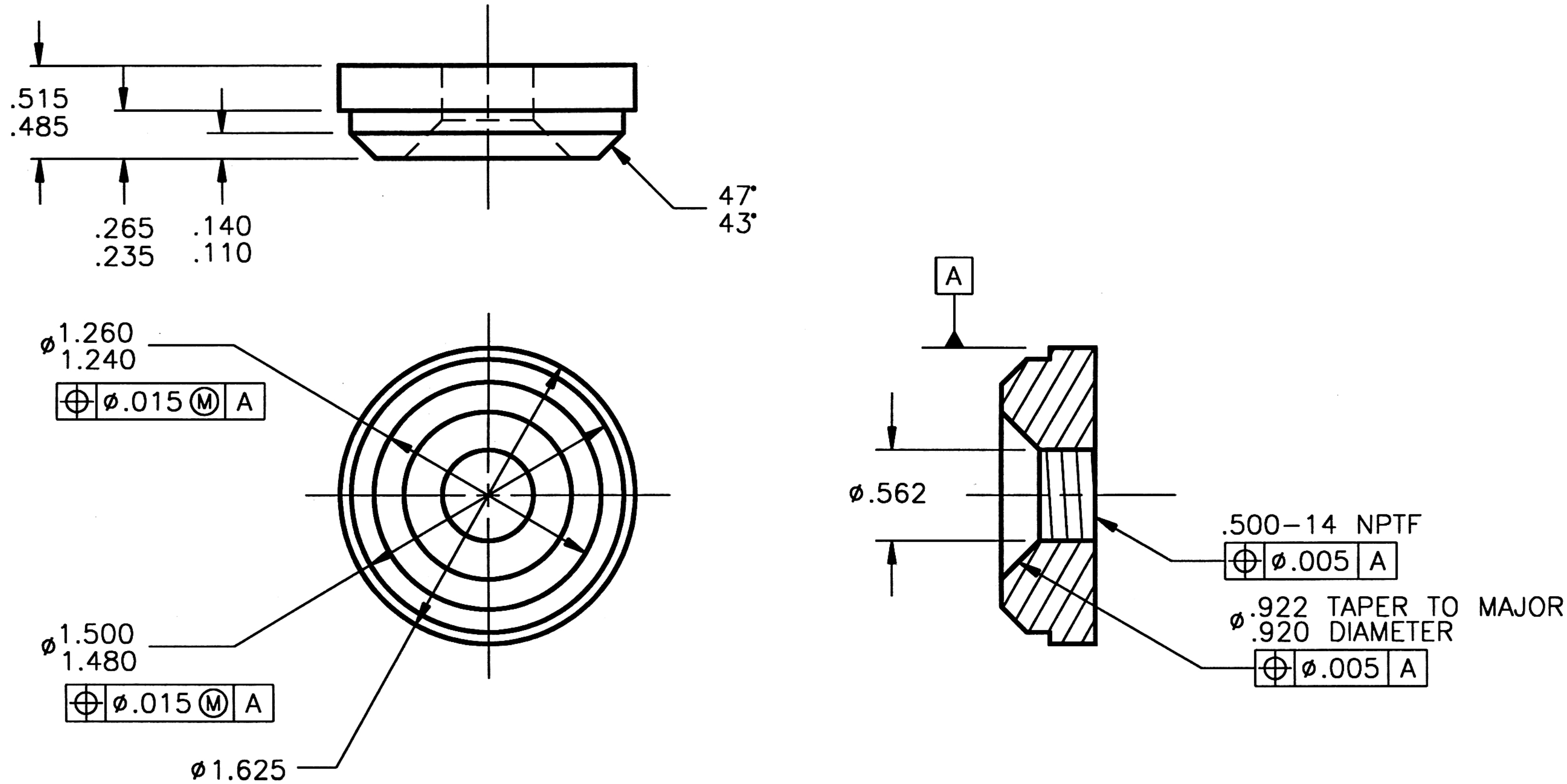
UNSPECIFIED TOLERANCES:

DECIMALS	mm	IN.
X	±2.5	±.1
XX	±0.25	±.01
XXX	±0.127	±.005
ANGULAR ±30'		
FINISH	3.2 μm	125 μIN.

COUPLING

DRAWING NO:	SCALE:
PROBLEM 13–4	2:1

MATERIAL:
AISI 1010, KILLED



2. REMOVE ALL BURRS AND SHARP EDGES.
1. INTERPRET DIMENSIONS AND TOLERANCES PER ASME Y14.5M–1994.

NOTES:

UNSPECIFIED TOLERANCES:

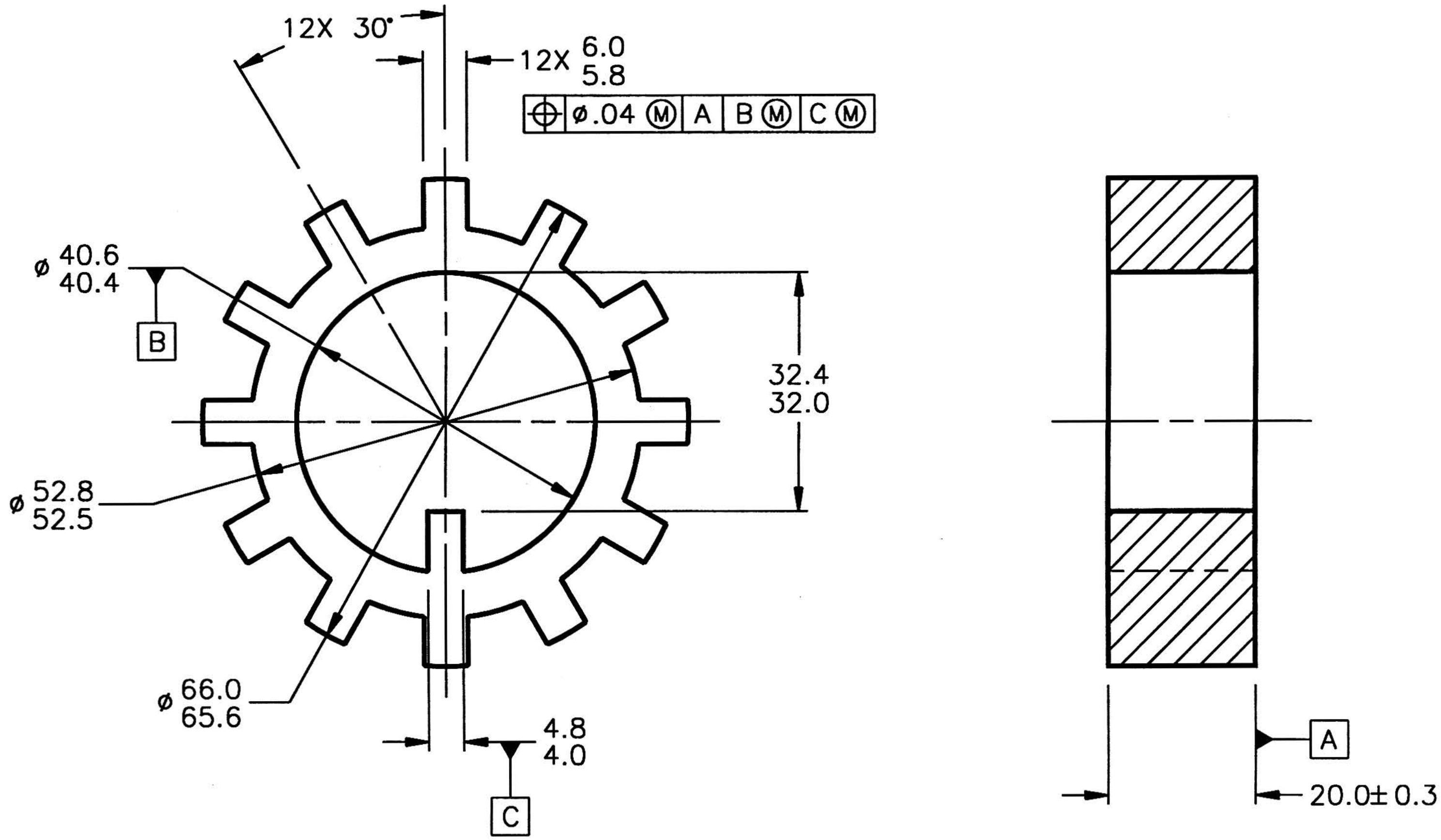
DECIMALS	mm	IN.
X	±2.5	±.1
XX	±0.25	±.01
XXX	±0.127	±.005
ANGULAR ±30'		
FINISH	3.2 µm	125 µIN.

HALF COUPLING

DRAWING NO: PROBLEM 13–5

SCALE: FULL

MATERIAL: ø1.625 6061–T6511



2. REMOVE ALL BURRS AND SHARP EDGES.
1. INTERPRET DIMENSIONS AND TOLERANCES PER ASME Y14.5M–1994.

NOTES:

UNSPECIFIED TOLERANCES:

DECIMALS	mm	IN.
X	±2.5	±.1
XX	±0.25	±.01
XXX	±0.127	±.005
ANGULAR ±30'		
FINISH	3.2 μm	125 μIN.

SPLINE PLATE

DRAWING NO:	SCALE:
PROBLEM 13–6	FULL

MATERIAL:
SAE 3135

NOTES

Geometric Tolerancing

EXAMINATION NUMBER:

05402601

Whichever method you use in submitting your exam answers to the school, you must use the number above.

For the quickest test results, go to http//www.takeexamsonline.com

When you feel confident that you have mastered the material in this study unit, complete the following examination. Then *submit only your answers to the school for grading,* using one of the examination answer options described in your "Test Materials" envelope. Send your answers for this examination as soon as you complete it. *Do not wait until another examination is ready.*

Questions 1–20: Select the one best answer to each question.

1. Where would you specify a larger positional tolerance for the pattern of features as a group?

   A. In the feature relating control
   B. In the pattern locating control
   C. Outside the feature control frame
   D. At the bottom of a two-part feature control frame

2. Which one of the following defines the correct left-to-right order of a feature control frame?

   A. Tolerance zone descriptor, geometric characteristic, geometric characteristic tolerance, and primary datum reference
   B. Geometric characteristic tolerance, geometric characteristic, tolerance zone descriptor, and datum reference
   C. Geometric characteristic, geometric characteristic tolerance, tolerance zone descriptor, and datum reference
   D. Geometric characteristic symbol, tolerance zone descriptor, geometric tolerance, material condition symbol, and multiple datum reference.

3. If an angularity symbol is being used appropriately, the basic angle from the datum must be

   A. 90°.
   B. any oblique angle.
   C. any size angle.
   D. less than 45°.

4. Which of the following are considered fixed fasteners?

A. Screws
B. Nuts
C. Bolts
D. Rivets

5. Which of the following is *not* an example of a location tolerance?

A. Position
B. Symmetry
C. Perpendicularity
D. Concentricity

6. A datum reference frame consists of how many mutually perpendicular planes?

A. Two
B. Three
C. Four
D. Five

7. Which of the following is *not* one of the five types of geometric tolerance?

A. Form
B. Runout
C. Profile
D. Straightness

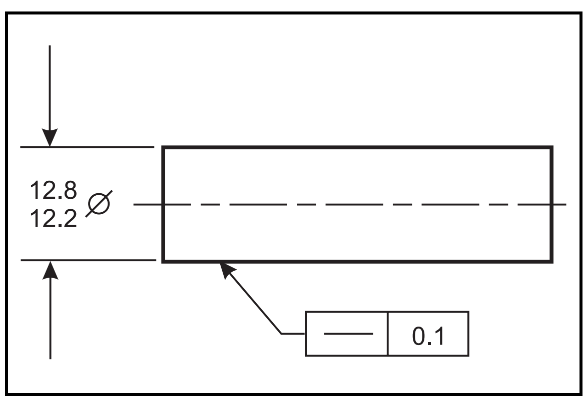
8. The location tolerance specifies the amount that the ________ of the feature is allowed to deviate from true position.

A. axis
B. edge
C. surface
D. radius

9. What does the acronym FIM stand for?

A. Feature in maximum
B. Form illustrates movement
C. Full indicator movement
D. Form indicates material

10. What geometric characteristic is implied in using the feature control frame shown here?



A. Surface straightness
B. Axis straightness
C. Unit straightness
D. Zone straightness

11. What are required to establish the relationship between the size or location of the feature and the geometric tolerance?

A. Polar coordinates
B. Material condition symbols
C. Tolerance notes on symmetrical features
D. Coplanar surface datums

12. Cylindricity is similar to circularity in that both

A. lie between two concentric cylinders.
B. have a cross-sectional tolerance.
C. lie between two concentric circles.
D. have a radius tolerance zone.

13. Which of the following is *not* a geometric characteristic that you would consider when contemplating form tolerance for a cylindrical shape?

   A. Straightness
   B. Circularity
   C. Concentricity
   D. Cylindricity

14. Which of the following indicates that the tolerance zone is split equally on each side of the specified perfect form?

   A. True profile tolerance
   B. Bilateral profile tolerance
   C. Coplanar profile tolerance
   D. Unilateral tolerance zone

15. Which of the following would be used to control the relationship of the feature surface and the true position at largest hole size?

   A. Positional tolerance at RFS
   B. Zero positional tolerance at MMC
   C. Positional tolerance at LMC
   D. Positional tolerance at MMC

16. When a datum feature symbol and a feature control frame are combined, the symbol is shown at the

   A. right end of the frame.
   B. center of the frame's base.
   C. left end of the frame.
   D. center of the frame's top.

17. A feature's true position may be defined as the theoretically exact position of the feature's

   A. datum.
   B. axis.
   C. feature control box.
   D. runout.

18. Which of the following does *not* belong in a feature control frame?

   A. A geometric characteristic symbol
   B. A material conditions symbol
   C. A decimal point
   D. A basic dimension

19. Which of the following are the two types of profile?

   A. The profile of a line and the profile of an arc
   B. The profile of a circle and the profile of an arc
   C. The profile of a line and the profile of a surface
   D. The profile of an arc and the profile of a surface

20. How are surface datums different from axis and center plane datums?

   A. When surface datums are shown on extension lines, they're offset from the dimension line arrows. Axis datums and center plane datums, on the other hand, can align with the dimension line's arrowhead.
   B. In surface datum symbols the equilateral triangle is filled, while in axis datums and center plane datums the equilateral triangle is left unfilled.
   C. Axis datums and center plane datums are verified from one of the existing surfaces.
   D. The surface datum is the only one to position the part in relation to a datum reference frame.