Fundamentals of Geometric Dimensioning & Tolerancing

(Based on ASME Y14.5M-1994)

2nd Edition

Answer Guide

Chapters listed below:

- Chapter 1
- Chapter 2
- Chapter 3
- Chapter 4
- Chapter 5
- Chapter 6
- Chapter 7
- Chapter 8
- Chapter 9
- Chapter 10
- Chapter 11
- Chapter 12

400 Commonwealth Drive, Warrendale, PA 15096
www.sae.org/gdt • 1-800-866-0909 • 1-734-744-5940

CHAPTER ONE: QUESTIONS AND PROBLEMS

1. Limit
2. Plus-minus
3. Plus-minus
4. Limit
5. Equal
6. Unilateral
7. Unequal
8. The decimal points and zero are omitted.
9. A zero precedes the decimal point
10.

Dimension	Max/Min limits	And its measured value was	This dimension would be Accepted	Rejected	Why
A	13.52 13.5	13.52001		X	over size
B	98 94	93.9999		X	under size
C	6.2 6	6.27001		X	over size
D	40.2 39.2	40.1999	X		within limits
E	16.8 16.2	16.80	X		within limits

11. ASME American Society of Mechanical Engineers
    Y14.5 The number of the standard
    M Metric
    1994 The year the standard was approved
12. Coordinate tolerancing is a dimensioning system where a part feature is located (or defined) by means of rectangular dimensions with given tolerances.
13. a. Square tolerance zones
    b. Fixed-size tolerance zones
    c. Ambiguous instructions for inspection
14. Geometric tolerancing raises product costs.

CHAPTER TWO: QUESTIONS AND PROBLEMS

1.

Letter	Feature of size dimension	Non-feature of size dimension
A		X
B	X	
C		X
D		X
E		X
F		X
G		X
H	X	
I		X
J		X
K	X	

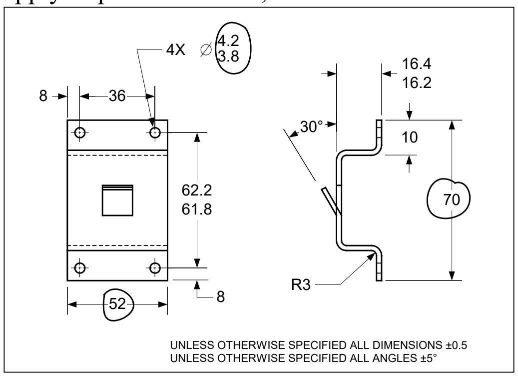
2.

Letter	MMC	LMC	Does not apply
A			X
B	70.5	69.5	
C			X
D			X
E			X
F			X
G			X
H	3.8	4.2	
I			X
J			X
K	52.5	51.5	

3. Actual local size is the value of any **individual** distance at any **cross section** of a feature.
4. In a feature of size, the surfaces or elements must be **opposed**.
5. There are two types of features of size: **internal** and **external**.
6. Actual mating envelope is a **variable** value.
7. The **largest** perfect feature counterpart that can be inscribed about the feature
8. The smallest diameter of a hole is its **maximum** material condition.
9. When a radius is specified, flats or reversals are **allowed**.
10. Non FOS
11. A **planar** FOS is a FOS that contains two parallel plane surfaces.
12. A radius without flats and reversals is referred to as a **controlled radius**.
13. The five types of geometric characteristic symbols are: Form, **orientation, profile, runout, and location**
14. a. **Geometric characteristic portion** c. **Datum reference portion**
    b. **Tolerance portion**

CHAPTER THREE: QUESTIONS AND PROBLEMS

1. Perfect form at MMC
2. a. Apply a straightness tolerance to a FOS.
   b. Apply a special note like, "Perfect form at MMC not required" to a FOS.
3.



4. 20.4
5. 9.8
6. The titleblock tolerance
7.

If dimension A was	The allowable form error on surface B is
12.8	0
12.7	0.1
12.6	0.2
12.5	0.3
12.4	0.4
12.3	0.5
12.2	0.6

8. (1) Pass the part through a gage with an opening equal to the MMC of the FOS
   (2) Two-point check with an instrument like a caliper
9. RFS applies with respect to the individual tolerance, datum reference, or both where no modifying symbol is specified.
10. A numerical value used to describe the theoretically exact size, true profile, orientation, or location of a feature of size or datum target
11. a. To define theoretically exact part features
    b. To define datum targets
12. Virtual condition: a condition (worst-case) boundary generated by the collective effects of a FOS specified at MMC or LMC and the geometric tolerance for that material condition.
13. Bonus tolerance: an additional tolerance for a geometric control
14. Inner boundary: a worst-case boundary generated by the smallest feature minus the stated geometric tolerance (and any additional tolerance, if applicable)
15. Outer boundary: a worst-case boundary generated by the largest feature plus the stated geometric tolerance (and any additional tolerance, if applicable)
16. A general term to refer to the extreme boundary for a FOS that is the worst case for assembly
17.

Use N/A for not applicable				If a FOS dimension is identified,			If a feature control frame is identified,		
Letter	Letter identifies a. . . FOS Dimension	Non-FOS Dimension	Feature Control Frame	Rule #1 Applies YES	NO	VC, OB, or IB is. . .	It applies to a. . . Feature	FOS	The amount of bonus tolerance permissible is. . .
A			✓				✓		0
B	✓			✓		63			
C			✓				✓		0
D			✓				✓		0
E	✓			✓		4.0			
F			✓					✓	0.4
G		✓							
H			✓				✓		0
I	✓				✓	37			
J			✓					✓	0.6
K	✓			✓		29.1			
L			✓					✓	1.0

CHAPTER FOUR: QUESTIONS AND PROBLEMS

1. Flatness is the condition where a surface has all of its elements in one plane.
2. Two parallel planes; the distance between the planes is equal to the flatness tolerance value.
3. The high points of the toleranced surface locate the first plane of a flatness tolerance zone.
4. 0.4
5. 0.4
6. a. Legal
   b. Illegal; MMC modifier not allowed
   c. Illegal; datum reference not allowed
   d. Illegal; ∅ modifier not allowed
7. Surface A= 0.1 Surface B= 0.4
8. 22.2
9. No; It must be a refinement of the size tolerance.
10. No; the flatness on surface "A" is controlled by the flatness symbol.
11.

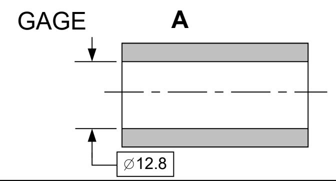
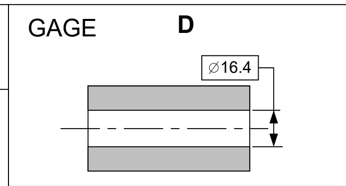
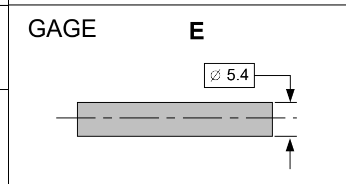
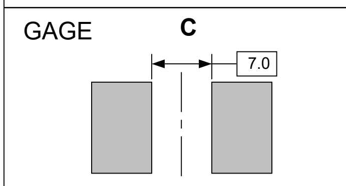
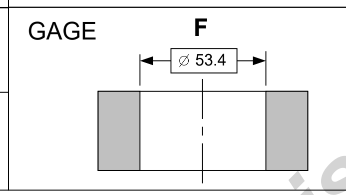
If the part was . . .	The flatness error of surface *B* would be limited to . . .	The flatness error of surface *A* would be limited to . . .
At MMC	0	0
At LMC	0.4	0.1
At 22.0	0.2	0.1

12. No
13. By contacting the toleranced surface against a surface plate and measuring the gap between the surface, the plate and the part surface
14. Straightness is the condition where each line element (axis or centerplane) is a straight line.
15. Two parallel lines 0.05 apart
16. 12.4
17. Rule #1 and the size dimension
18. Lay the pin on a surface plate and measure the gap between the pin surface and the surface plate.
19. a. Legal
    b. Illegal; MMC modifier not allowed on a surface
    c. Illegal; datum reference not allowed
    d. Illegal; ∅ modifier not allowed
20.

Dimension at letter	Is the straightness control applied to a Surface? or	FOS?	The VC, OB, or IB of the FOS is	Does Rule #1 apply to the FOS?
A		X	17.5	NO
B	X		N/A	N/A
C		X	3.35	NO
D		X	28.4	NO
E		X	3.9	NO
F	X		N/A	N/A
G		X	22	NO

21. Rule #1 and the size dimension
22. 12.4
23. B; E; G; I

24.

Gage			Gage		
	MAX BONUS 1.0	TOTAL ALLOWABLE TOLERANCE 1.2		MAX BONUS 0.4	TOTAL ALLOWABLE TOLERANCE 0.4
	MAX BONUS 0.2	TOTAL ALLOWABLE TOLERANCE 0.3		MAX BONUS 0.4	TOTAL ALLOWABLE TOLERANCE 0.8
	MAX BONUS 0.8	TOTAL ALLOWABLE TOLERANCE 1.0		MAX BONUS 2.0	TOTAL ALLOWABLE TOLERANCE 2.4

25. Circularity is a condition where all the points of a surface of revolution are equidistant from the axis.
26. Two coaxial circles with a radial distance between them equal to the circularity tolerance value
27. Rule #1 and the size dimension
28. 0.04
29. A; B; F
30. a. Illegal; ∅ modifier not allowed
    b. Illegal; Ⓢ modifier not allowed
    c. Legal
    d. Illegal; datum reference not allowed
31.

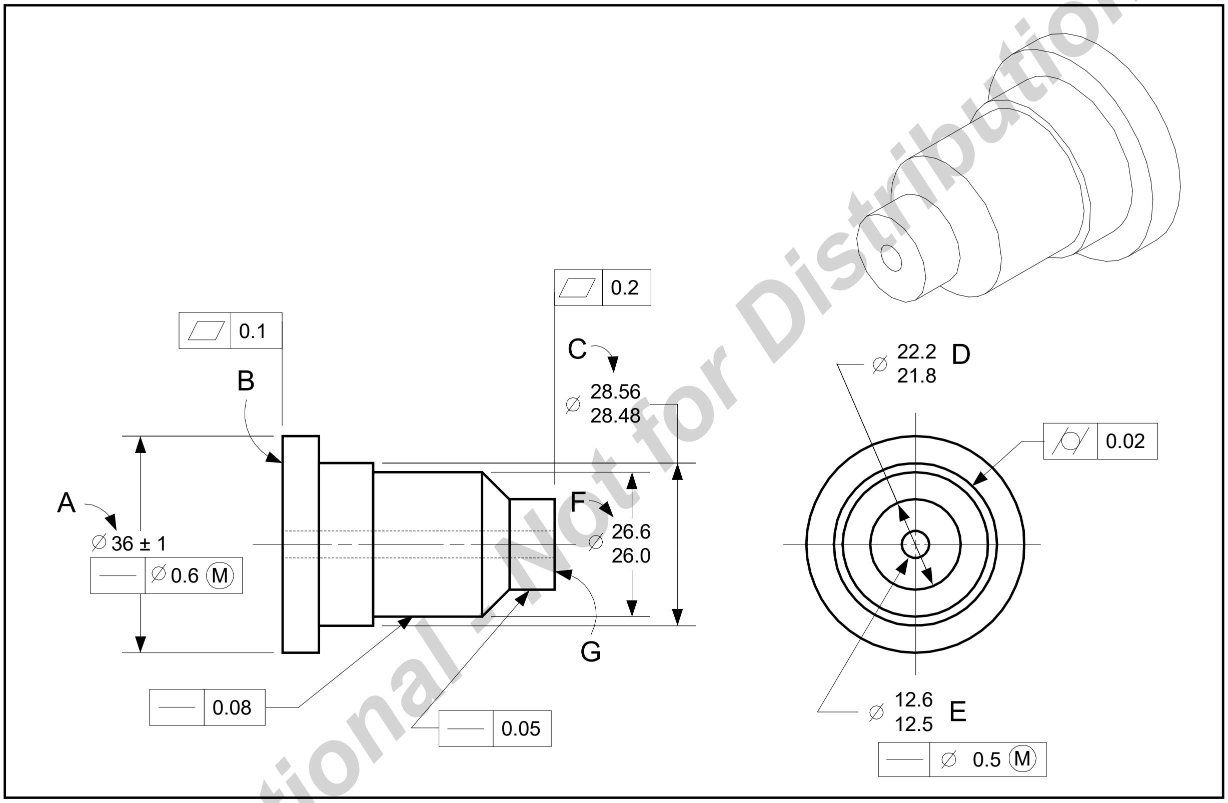
Diameter	WCB	Max circularity error possible	Max straightness of axis error possible	Max straightness of line element error possible	Rule #1 applies (YES/NO)
A	**9.8**	**0.8**	**0.8**	**0.8**	**YES**
B	**14.9**	**0.1**	**0.9**	**0.9**	**NO**
C	**10.4**	**0.04**	**0.04**	**0.04**	**YES**
D	**20.7**	**0.05**	**0.3**	**0.3**	**NO**
E	**5.8**	**0.2**	**0.5**	**0.5**	**NO**
F	**18.6**	**0.02**	**0.4**	**0.4**	**YES**

32. By comparing an enlarged outline of the circular cross-section to a set of concentric circles
33. Cylindricity is the condition of a surface of revolution in which all points of the surface are equidistant from a common axis.
34. Two coaxial cylinders with a radial distance between them, equal to the cylindricity tolerance value
35. Rule #1 and the size dimension
36. 0.04
37. A; C; F
38. A sampling of points of the part's cylindrical surface are compared to two coaxial cylinders (similar to circularity, but in 3-D)
39. A. Illegal, no datum reference allowed
    B. Legal
    C. Illegal, cannot use the LMC modifier
    D. Illegal, cannot use the diameter modifier

40.

Diameter	Rule #1 applies (YES/NO)	WCB	Max straightness of axis error possible	Max circularity error possible	Max cylindricity error possible
A	**NO**	**10**	**0.4**	**0.2**	**0.2**
B	**YES**	**16.2**	**0.2**	**0.03**	**0.2**
C	**YES**	**24.2**	**0.05**	**0.05**	**0.05**
D	**YES**	**12.82**	**0.02**	**0.02**	**0.02**

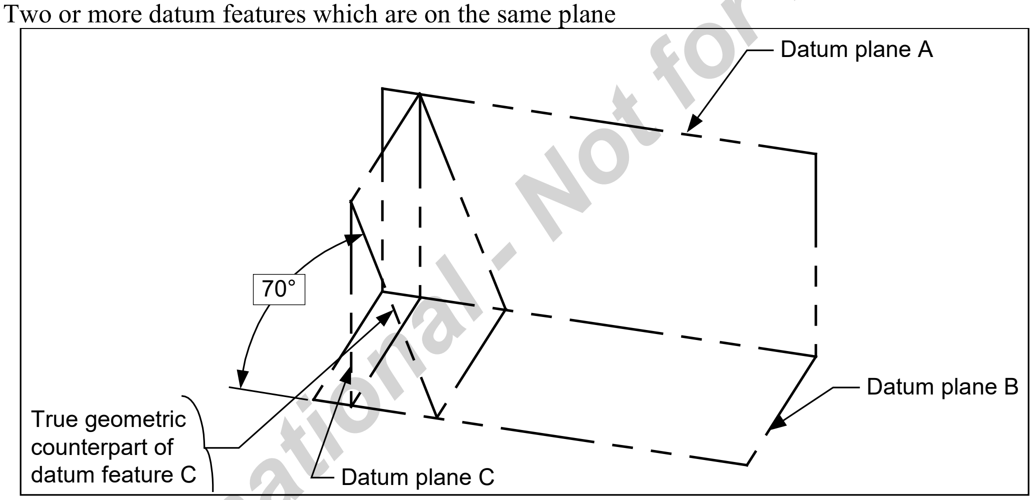
41.



CHAPTER FIVE: QUESTIONS AND PROBLEMS

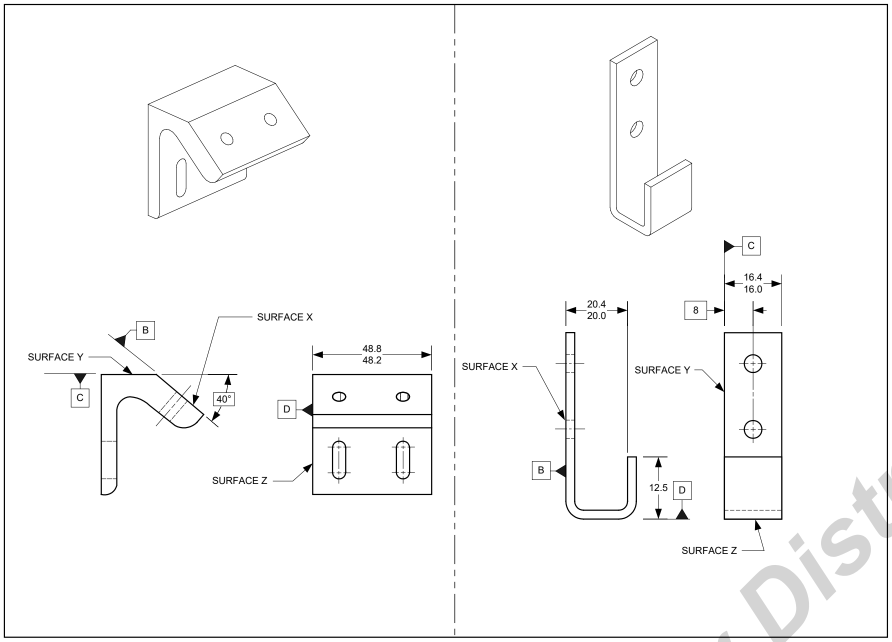
1. A system of symbols & rules that communicate to the drawing user how measurements are to be made
2. a. Aids in making dimensional measurements as intended by the designer
   b. Aids in making repeatable dimensional measurements
   c. Aids in communicating part functional relationships
3. An assumed plane, axis, or point from which a measurement is made.
4. a. Does not clearly communicate which part surfaces should touch the inspection equipment
   b. Does not communicate the sequence to bring the part into contact with the inspection equipment.
5. a. Good parts rejected
   b. Bad parts accepted
6. A theoretical plane, point, or axis from which dimensional measurements are made
7. A part feature that contacts a datum
8. A perfect plane
9. A

10. a. Connect the base of the datum symbol on the edge view of a surface or on an extension line of a surface
    b. Connect the base of the datum symbol to an extension line of a dimension. The base must be offset from the dimension lines.
11. How the part is mounted and located in its assembly
12. A set of three datum planes - mutually perpendicular
13. It is not shown; the general tolerance for angles either from the titleblock tolerances or a general note
14. a. Movement along the X axis
    b. Movement along the Y axis
    c. Movement along the Z axis
    d. Rotation around the X axis
    e. Rotation around the Y axis
    f. Rotation around the Z axis
15. a. Datum Plane E - 3
    b. Datum Plane B - 2
    c. Datum Plane A - 1
16. a. Movement along the Z axis
    b. Rotation around the X axis
    c. Rotation around the Y axis
17. a. Rotation around the Z axis
    b. Movement along the Y axis
18. Two or more datum features which are on the same plane
19.



20. A geometric tolerance associated with the dimension, will reference the datum reference frame
21. Symbols that describe the shape, size, and location of gage elements that are used to establish datum planes
22. a. Whenever it is not practical to use the whole surface as a datum feature
    b. Whenever the designer suspects the part may rock (or wobble) when the part contacts the datum plane
23. To ensure there will be minimum variation between gages

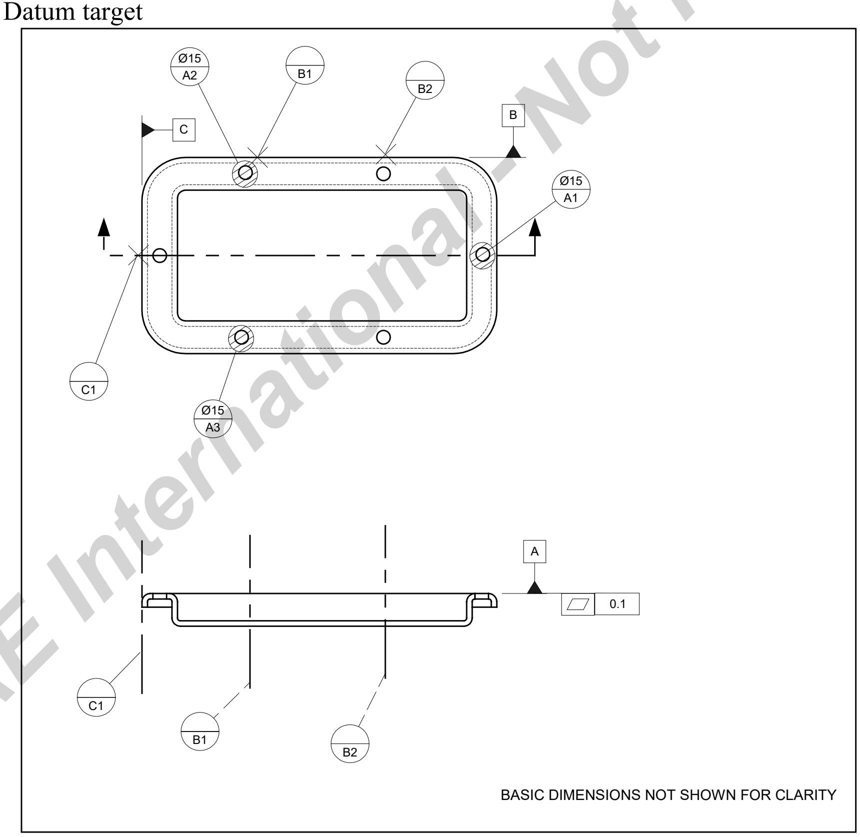
24.



25. 1. Basic dimensions should be used to define and locate the datum targets.
2. The datum reference frame must restrain the part in all six degrees of freedom.
3. The part dimensioning must ensure that the part will rest in the gage in only 1 orientation/location

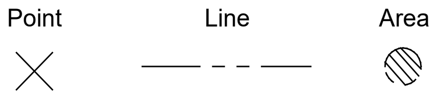
26. Datum target

27.

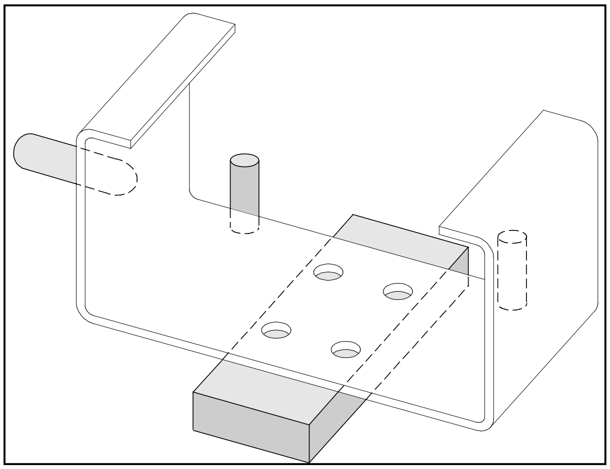


28. No; The flatness is checked from the high points of the surface.

29.



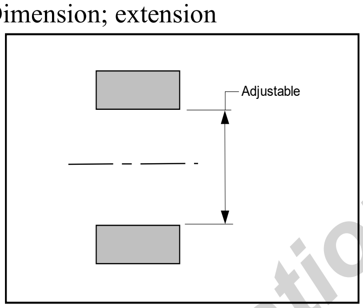
30.



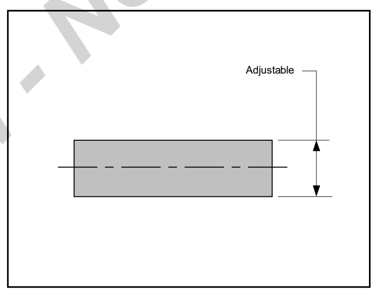
CHAPTER SIX: QUESTIONS AND PROBLEMS

1. Diameter; planar
2. Axis; centerplane
3. Rule #2; modifier
4. Surface; leader; feature control frame
5. Dimension; extension

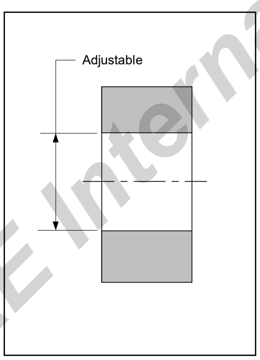
6.



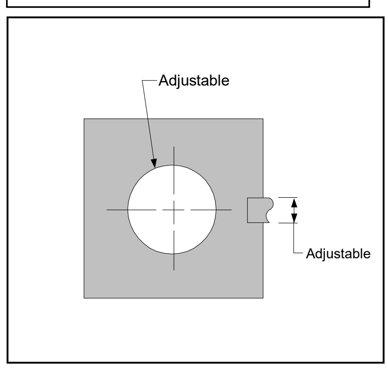
7.



8.

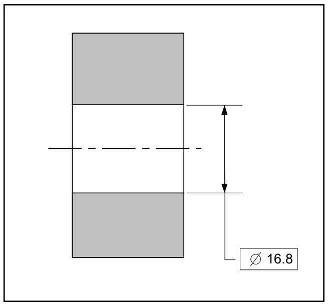


9.

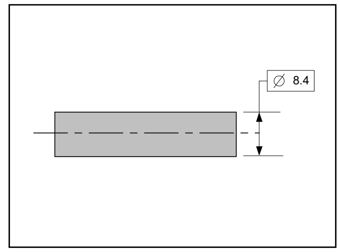


10. Two or more diameters that are shown on the same centerline as datum feature use.
11. The datum feature simulator (gage) is a fixed size.
12. The allowable movement between the part datum feature and the gage is called datum shift.
13. a. Where a straightness control is applied to a datum feature
    b. Where a secondary or tertiary datum feature of size in the same datum reference frame are controlled by a location or orientation control with respect to each other
14. When the Ⓜ modifier is shown in the datum portion of the feature control frame

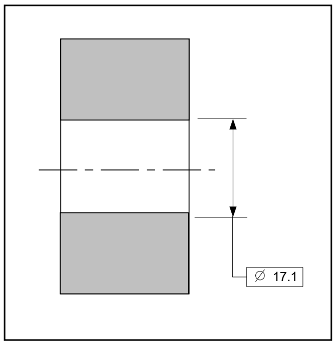
15.



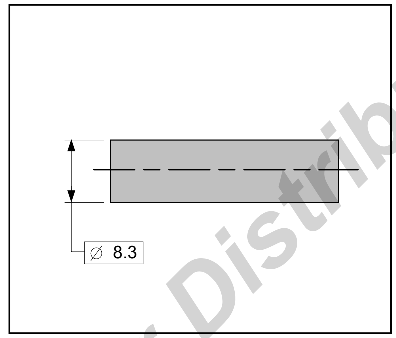
16.



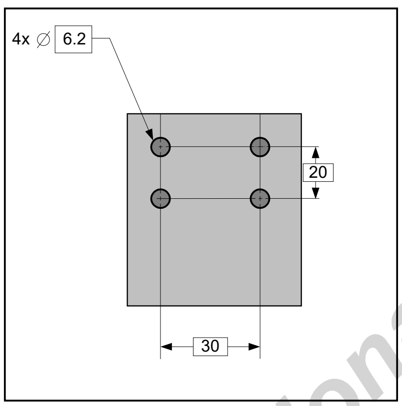
17.



18.



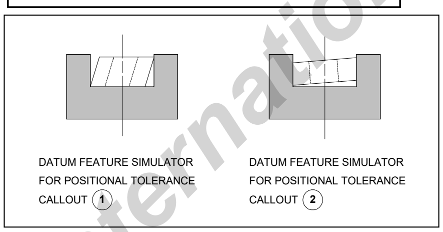
19.



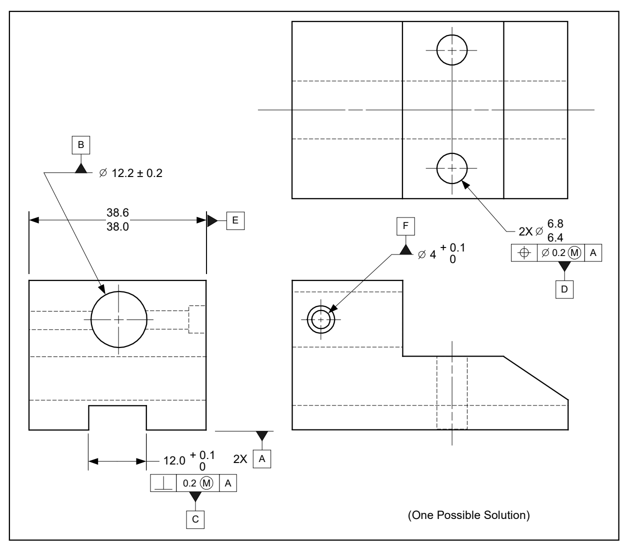
20.

Using the drawing from question. . .	The Maximum Permissable datum shift is. . .
15	0.2
16	0.4
17	0.5
18	0.5
19	0.4

21.



22.



CHAPTER SEVEN: QUESTIONS AND PROBLEMS

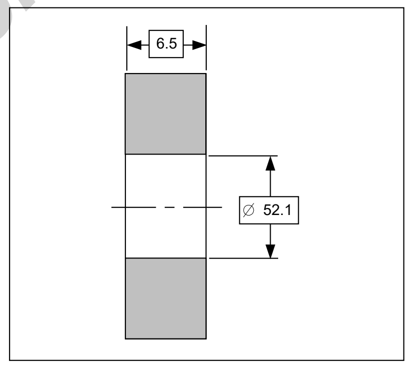
1. Two parallel planes 0.2 apart
2.

The flatness of surface. . .	Is controlled by...	Is limited to. . .
B	Rule #1 and Size dim.	0.6
C	Flatness Control	0.1
D	Perpendicularity Control	0.2
E	Rule #1 and Size dim.	0.6

3. A general note
4. 90° basic to datum plane A
5. The tolerance zone would be oriented to both datum plane A and datum plane B.
6. a. Two parallel planes
   b. A cylinder
6. Any three of the four statements below
   1. The tolerance zone is two parallel planes.
   2. The tolerance value is the distance between the planes.
   3. All elements of the surface must be within the tol. zone.
   4. The flatness of the surface is also controlled.
8. A cylinder 0.1 dia at MMC and 0.3 dia at LMC.
9.

If the actual size of dia. B is...	The bonus tolerance possible is...	The perpendicularity tolerance zone diameter would be...
52.0	0	0.1
51.9	0.1	0.2
51.8	0.2	0.3

10.



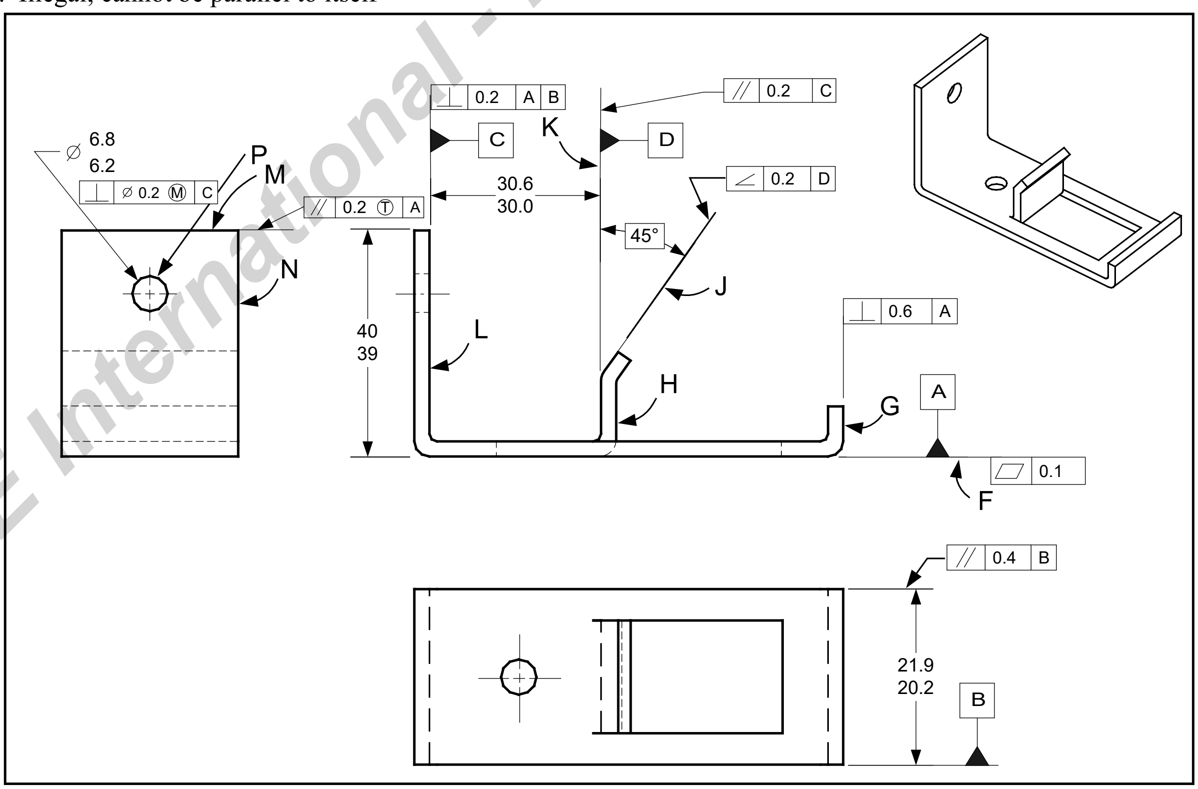
11. a. A bonus tolerance is permissible
    b. A fixed gage may be used.
    c. The axis or centerplane must be within the tolerance zone
12. a. Legal
    b. Legal
    c. Legal
    d. Illegal - no RFS modifier
    e. Legal
    f. Illegal - needs a datum reference
13. a. Two parallel planes
    b. A cylinder
14. Any three of the four statements below
    1. Tolerance zone is two parallel planes
    2. Tolerance zone is oriented to datum planes with a basic angle
    3. All elements of the surface must be within the tol. zone.
    4. Flatness also controlled within the tolerance value
15. Two parallel planes 0.1 apart.
16. The 30° basic angle.
17.

The flatness of surface. . .	Is controlled by...	Is limited to. . .
B	Rule #1 and size dim.	0.2
C	Angularity control	0.1

18. Yes
19. Yes
20. Yes
21. a. Tolerance zone is usually a cylinder
    b. Basic angle orients the tolerance zone in one direction
    c. Implied basic angle applies in the other direction
22. a. 2 parallel planes
    b. A cylinder
23. Two parallel planes 0.1 apart
24.

The flatness of surface. . .	Is Controlled by	is limited to. . .
B	Rule #1 and size dim.	0.6
C	Flatness Control	0.1
D	Perpendicularity Control	0.2
E	Parallelism Control	0.1

25. The dimension between the surfaces
26. Oriented parallel to datum plane A; located within the 36.0 - 36.6 dimension
27. 0.1
28. 0.6
29. 2 parallel planes 0.2 apart
30. It denotes that only the tangent plane of the toleranced surface needs to be within the tolerance zone.
31. a. Illegal; the tolerance value is too large.
    b. Illegal; the toleranced feature is perpendicular to datum referenced.
    c. Legal
    d. Legal
    e. Illegal; the toleranced feature is perpendicular to the datum referenced.
    f. Illegal; cannot be parallel to itself
32.



CHAPTER EIGHT: QUESTIONS AND PROBLEMS

1. A geometric tolerance that defines the location tolerance of a feature of size from its true position
2. The theoretically exact location of a FOS as defined by basic dimensions
3. a. Implied basic 90° angles
   b. Implied basic zero dimension
4. a. Cylindrical tolerance zones
   b. Additional tolerance
   c. Prevents tolerance accumulation
   d. Protects the part function
   e. Lower manufacturing costs
   f. Permits the use of functional gages
5. a. The distance between the features of size
   b. The location of the features of size
   c. The coaxiality of features of size
   d. The symmetry of features of size
6. A theoretical boundary limits the location of the surfaces of a feature of size.
7. The axis or centerplane of a FOS must be within the tolerance zone.
8. A 0.2 diameter cylindrical tolerance zone
9. Two parallel planes 0.1 apart
10. A 0.3 diameter cylinder
11. a. Boundary tolerance zone
    b. Bonus tolerance is permissible.
    c. A functional gage may be used.
12.

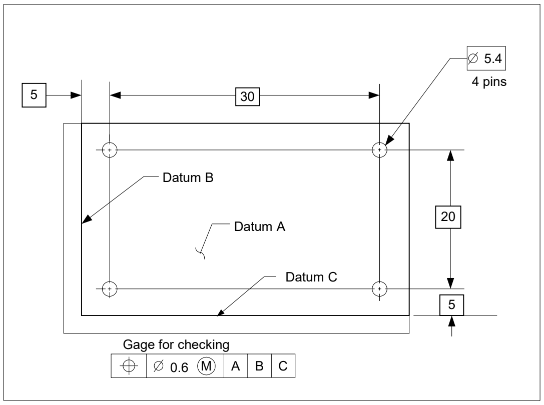
For the TOP callout labeled...	The shape of the tolerance zone is...	The max permissible bonus is...	The max permissible datum shift is...
A	**39.0 boundary**	**0.6**	**0.3**
B	**11.6 dia. cylinder**	**0.4**	**0.3**
C	**3.9 dia. cylinder**	**0.1**	**0.45**

13.

For the TOP callout labeled...	The shape of the tolerance zone is...	The max permissible bonus is...	The max permissible datum shift is...
A	**2.8 dia. boundary**	**0.6**	**0.6**
B	**42.3 dia. cylinder**	**1.0**	**0.6**
C	**5.9 boundary**	**0.8**	**0.6**

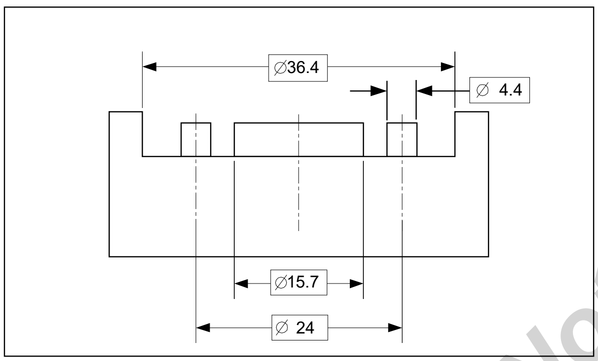
14. a. Illegal; no datum reference
    b. Legal
    c. Illegal; cannot be applied to a surface
    d. Illegal; cannot use a toleranced dimension to A (see "Design Tip" on page 162), and datum reference B must be referenced at MMC
15. A gage that verifies functional requirements of part features as defined by the geometric tolerances.
16. a. The gage represents the worst case mating part
    b. Parts can be verified quickly
    c. Gage is economical to produce
    d. No special skills
15. A sketch of a functional gage

18.



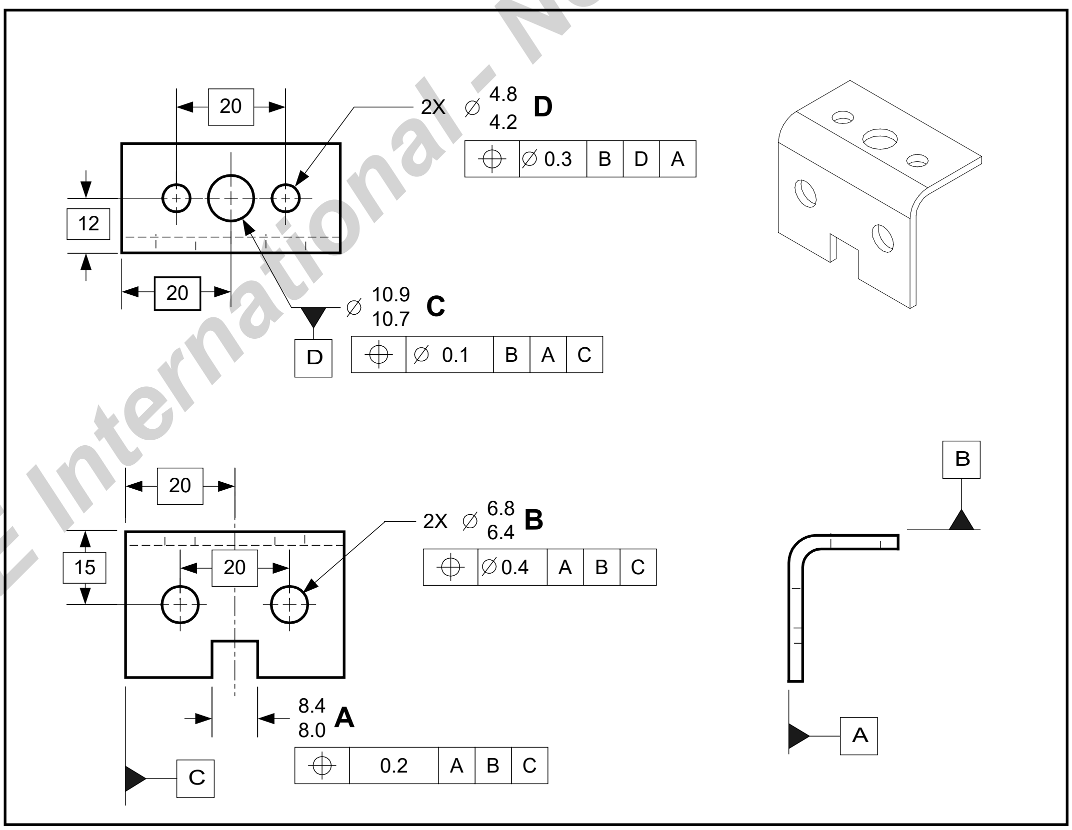
⌀ 5.4
4 pins
5
30
Datum B
Datum A
20
Datum C
5
Gage for checking
⌖ ⌀ 0.6 Ⓜ A B C

19.



⌀36.4
⌀ 4.4
⌀15.7
⌀ 24

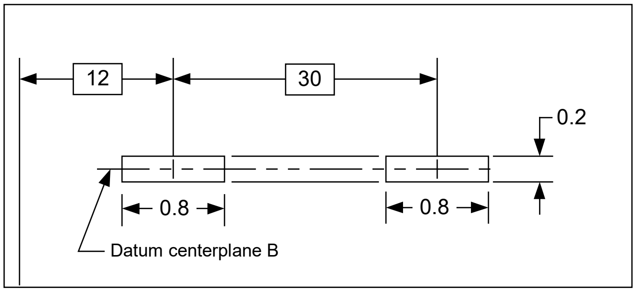
20.



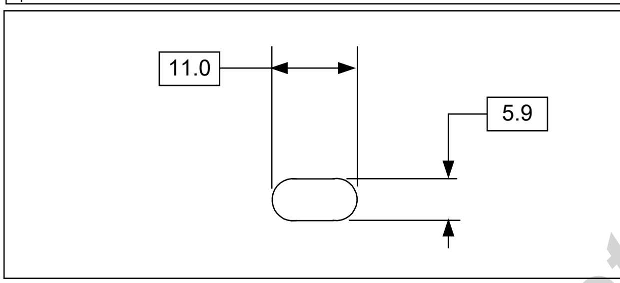
20
2X ⌀ 4.8 4.2 D
⌖ ⌀ 0.3 B D A
12
20
⌀ 10.9 10.7 C
D
⌖ ⌀ 0.1 B A C
B
20
2X ⌀ 6.8 6.4 B
15
20
⌖ ⌀0.4 A B C
8.4 8.0 A
⌖ 0.2 A B C
C
A

CHAPTER NINE: QUESTIONS AND PROBLEMS

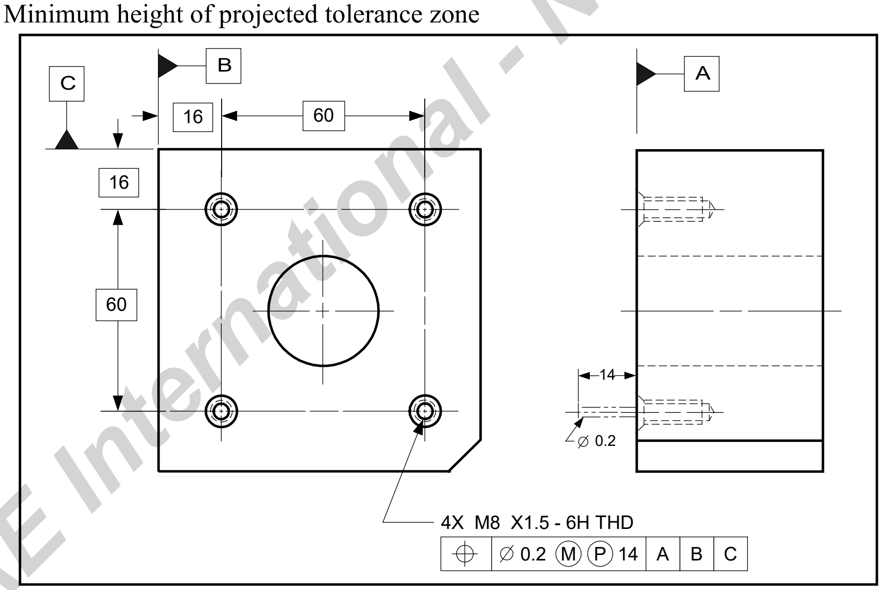
1. cylindrical
2. The TOP control
3. 0.2 0.3 0.4 0.5 0.6
4.



5.



6. There is no axis interpretation for the location of the elongated holes
7. Projected tolerance zone
8. Minimum height of projected tolerance zone
9.



10. 2 parallel planes
11.

Position Tolerance Zone Width at Centerline				
Toleranced Feature AME	Datum Feature AME			
	10.0	10.2	10.4	10.6
14.2	2.0	2.2	2.4	2.6
14.4	2.2	2.4	2.6	2.8
14.6	2.4	2.6	2.8	3.0

12. 0.6
13.

Toleranced Hole AME	⌖ Tolerance Zone Diameter
4.6	0.2
4.4	0.4
4.2	0.6
4.0	0.8

14. 4.0
15. 4.0
16.

Hole AME	⌖ Tol. Dia.	Bonus Tol.	Total Location Tol. Dia.
4.0	0	0	0
4.2	0	0.2	0.2
4.4	0	0.4	0.4
4.6	0	0.6	0.6
4.8	0	0.8	0.8

17. A tolerance stack is a calculation used to find the extreme max. or min. distance on a part.
18. Max. X = 2.3 Min. X = 1.3
19. Max. X = 16.6 Min. X = 15.4
20. $H = F + 2T$ and $T = \frac{H - F}{2}$
21. 0.7 (for housing) 0.7 (for cover)
22. 1.4
23. 0.3
    0.3
24. 0.3
    0.3

CHAPTER TEN: QUESTIONS AND PROBLEMS

1. A cylinder coaxial with the datum axis
2. Two parallel planes centered about the datum centerplane
3. The concentricity control
4. A 0.02 diameter cylinder coaxial with datum axis A
5. Yes
6. Median points
7. 0.01
8.

CONCEPT	CONCENTRICITY	TOTAL RUNOUT	TOP(RFS)
Describe the shape of the tolerance zone?	A cylinder	Two coaxial cylinders	A cylinder
What characteristic of the toleranced feature must be within the tolerance zone?	Median points of two point measurements	Surface elements of the dia.	Axis of the AME
Does rule #1 still apply to the toleranced feature?	Yes	Yes	Yes
What type of characteristics of the toleranced feature are being controlled?	Location, orientation	Location, orientation, form	Location, orientation

9. a. Illegal; dia. symbol missing
   b. Illegal; cannot use projected tolerance zone modifier
   c. Illegal; cannot use RFS modifier
   d. Legal
   e. Illegal; cannot use MMC modifier

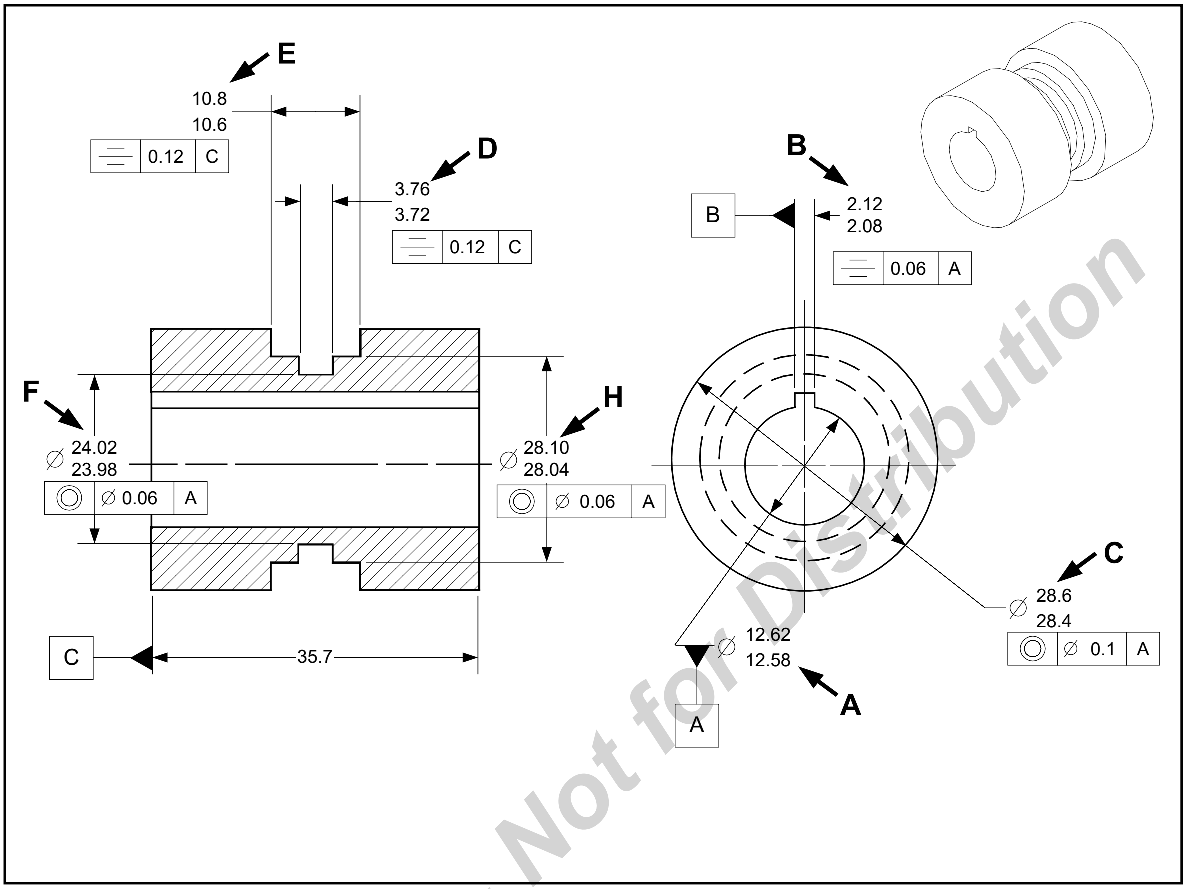
10. A median point is the mid-point of a two point measurement.
11. Two parallel planes 0.4 apart
12. Yes
13. Median points
14. 0.2
15.

CONCEPT	SYMMETRY	POSITION (RFS)
Tolerance zone shape	Two parallel planes	Two parallel planes
What characteristic of the toleranced feature must be within the tolerance zone?	Median points of two point measurement	Centerplane of AME
Does Rule #1 apply to the toleranced feature?	Yes	Yes
What type of characteristics of the toleranced feature are being controlled?	Location; orientation	Location; orientation

16.

A	⌯ | 0.5 | A	Legal
B	⌯ | 0.5 Ⓜ | A	Illegal; cannot use MMC modifier
C	⌯ | 0.5 Ⓢ | AⓈ	Illegal; cannot use RFS modifier
D	⌯ | 0.5 Ⓛ | AⓁ	Illegal; cannot use LMC modifier
E	⌯ | 0.5 Ⓟ10 | A	Illegal; cannot use projected tolerance zone modifier

17.



CHAPTER ELEVEN: QUESTIONS AND PROBLEMS

1. A composite tolerance that is to control the functional relationship (location, orientation, and form) of one or more features to a datum axis
2. a. A single diameter of sufficient length c. A surface and a diameter at right angles
   b. Two coaxial dias. a sufficient distance apart to create a single datum axis with centers located on the datum axis
   c. A surface and a diameter at right angles
3. Two coaxial circles
4. a. Axis location c. Form error (roundness)
   b. Axis orientation
5.

DIA	MAX POSSIBLE AXIS OFFSET FROM DATUM AXIS A
B	0.1
C	0.15
D	0.4

6.

QUESTION	APPLIES TO DIA **B**	DIA **C**	DIA **D**	DIA **E**
The size of the diameter is limited to?	0.4	0.1	0.2	0.2
The roundness of the diameter is limited to?	0.3	0.1	0.2	0.2
The maximum offset between the diameter axis and datum axis A is...	0.15	0.05	0.5	0.1
Describe the tolerance zone for the runout controls applied to the diameter.	2 coaxial circles with 0.3 radial seperation	2 coaxial circles with 0.1 radial seperation	2 coaxial circles with 1.0 radial seperation	2 coaxial circles with 0.2 radial seperation
How many places should the runout control be checked on this diameter?	Inspector's judgement			
What is the outer boundary (virtual condition) of this diameter?	6.7	20.3	13.6	18.4

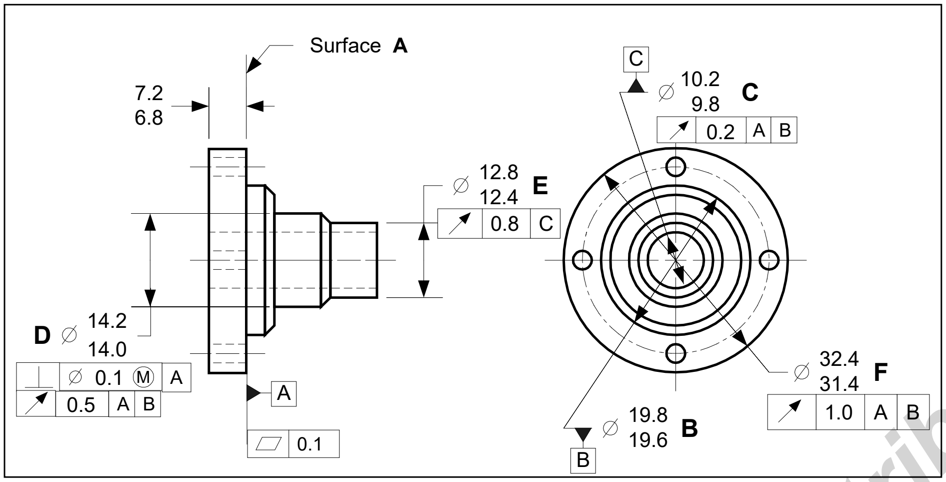
7. a. Legal c. Illegal; cannot use MMC modifier
   b. Legal d. Illegal; cannot use diameter modifier
8. A composite control affecting the form, orientation, and location of all surface elements simultaneously of a diameter (or surface) relative to a datum axis
9. Two coaxial cylinders whose centers are located on the datum axis
10. a. Axis offset d. Straightness
    b. Axis orientation e. Circularity
    c. Taper
11. a. Illegal; cannot use diameter modifier c. Illegal; cannot use RFS modifier
    b. Illegal; cannot use projected tolerance zone modifier d. Legal
12. B = 0.05
    C = 1.0
    D = 0.5
13.

QUESTION	APPLIES TO DIA **B**	DIA **C**	DIA **D**	DIA **E**
The size of the diameter is limited to?	0.2	0.4	0.2	0.2
The roundness of the diameter is limited to?	0.2	0.4	0.06	0.2
The maximum offset between the diameter axis and datum axis A is...	0.6	0.2	0.03	N/A
Describe the tolerance zone for the runout controls applied to the diameter.	2 coaxial cylinders with 1.2 radial seperation	2 coaxial cylinders with 0.4 radial seperation	2 coaxial cylinders with 0.06 radial seperation	N/A
What is the outer boundary (virtual condition) of this diameter?	25.4	13.2	14.66	10.2

14. A; B; D; E
15.

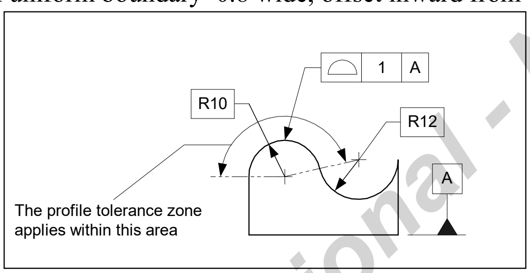
DISTANCE	MAX	MIN
A	**2.75**	**1.25**
B	**2.60**	**1.50**
C	**3.10**	**1.10**
D	**1.10**	**0.85**
E	**2.15**	**1.05**

16.

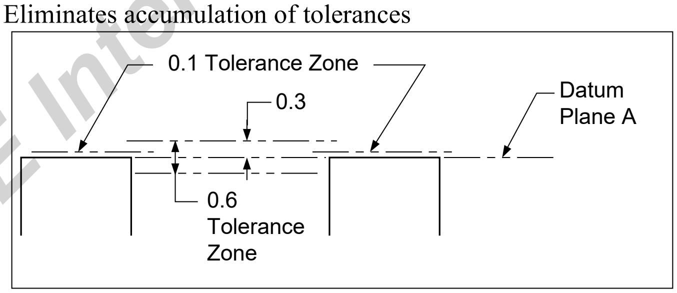


CHAPTER TWELVE: QUESTIONS AND PROBLEMS

1. Form
2. Datum related feature control
3. The exact profile of a part as described by basic dimensions
4. a. Size c. Orientation
   b. Location d. Form
5. A uniform boundary 0.8 wide, centered around the true profile
6. A uniform boundary 0.8 wide, offset inward from the true profile
7.



8. a. Use of a note
   b. Use of the between symbol
   c. Use of the all around symbol
9. a. Clear definition of the tolerance zone
   b. Communicates datums and datum sequence
   c. Eliminates accumulation of tolerances
10.



11.

This profile callout	Controls the (size, location, orientation, form)	Within	Relative to
⌓ | 1.5 | A | B | C	Location (or size)	1.5	A B C
⌓ | 1.0 | B	Orientation	1.0	B
⌓ | 0.2	Form	0.2	

12. a. Legal
    b. Illegal; needs basic dimension to define true profile
    c. Illegal; tolerance value too large
    d. Legal
    e. Legal
    f. Illegal; needs basic dimension to define true profile
    g. Illegal; cannot use an MMC modifier
13. Two uniform lines at any cross section of the surface.
14. Callout 1 - Two cones 0.6 apart and centered around the true profile
    Callout 2 - Two line elements 0.1 apart, located within the tolerance zone of the upper callout, oriented relative to datum axis A
15. a. Illegal - in conflict with the location dimension.
    b. Legal
    c. Illegal - tolerance value too large
    d. Legal
16.

DISTANCE	MAX	MIN
A	28.5	27.5
B	42	40
C	28.4	27.6
D	2.4	1.6
E	30.8	29.2

17.