


ISO Geometrical Tolerancing Exercise Workbook Answer Guide

Based on ISO 1101:2004
and Companion Standards

Companion to
ISO Geometrical Tolerancing Exercise Workbook v1.04
by Alex Krulikowski



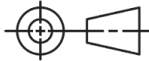
SAE International
400 Commonwealth Drive, Warrendale, PA 15096
www.sae.org/gdt • 1-800-866-0909 • 1-734-744-5940

		ISO 1101:2004	ISO 2692:2006			
		ISO 8015:1985	ISO 5459:1981			
		ISO 2768-H-E	ISO 128-1:2003			
		ISO 1302:2002				
	DRAWN BY	DATE	ALL DIMENSIONS ARE IN MILLIMETERS	ETI EFFECTIVE TRAINING INC.		
	CHECK BY	DATE		TITLE:	REV	A
	APPROVED	DATE		PINION, DRIVE		
APPLICATION				SIZE	DRAWING NO.:	SHEET: 1 OF 1
THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF EFFECTIVE TRAINING INC. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF EFFECTIVE TRAINING INC. IS PROHIBITED.				DO NOT SCALE DRAWING	SCALE	

Use the figure above to answer questions 1 - 4.

- What does the specification of ISO 1101:2004 indicate on the drawing?
 - The interpretation of geometrical tolerance symbols
 - The independency principle
 - Drawing general tolerances
 - None of the above
- Why does the specification of ISO 1101:2004 include the date?
 - To indicate the year the drawing was created
 - To indicate the year the drawing was released
 - To indicate which version of the standard applies
 - None of the above



		ISO 1101:2004	ISO 2692:2006			
		ISO 8015:1985	ISO 5459:1981			
		ISO 2768-H-E	ISO 128-1:2003			
		ISO 1302:2002				
	DRAWN BY	DATE	ALL DIMENSIONS ARE IN MILLIMETERS	ETI EFFECTIVE TRAINING INC.		
	CHECK BY	DATE		TITLE:	REV	A
	APPROVED	DATE		PINION, DRIVE		
APPLICATION				SIZE	DRAWING NO.:	SHEET: 1 OF 1
THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF EFFECTIVE TRAINING INC. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF EFFECTIVE TRAINING INC. IS PROHIBITED.				DO NOT SCALE DRAWING	SCALE	

Use the figure above to answer questions 1 - 4.

3. Why is ISO 1302:2002 specified on the drawing?
 - a. This standard is required whenever ISO 1101 is specified
 - b. This standard establishes requirements for general tolerances
 - c. Many drawings require more than the basic three standard references
 - d. None of the above

4. Which ISO standards are recommended to be specified on every engineering drawing?
 - a. 1101, 8015, 2768
 - b. 1101, 128, 129
 - c. 1101, 8015, 1302
 - d. 1101, 8015, 129



5. An important reason to use ISO standards on engineering drawings is to...
- Invoke 3rd angle projection
 - Invoke the metric system
 - Aid in global sourcing of parts
 - None of the above
6. One of the principles of ISO 128-1:2003 is that...
- Limit dimensions may be expressed in different units than basic dimensions
 - Dimensions shall be placed to be read from the bottom or left hand side of the drawing
 - Drawings should be language independent
 - Dimensions shall be in SI units
7. When a drawing specifies ISO 1101:2004, what standard does it invoke?
- ISO 5459:2004
 - ISO 129-1:2004
 - ISO 1660:2004
 - None of the above
8. One of the principles of ISO 129-1:2004 states that all dimensional information shall be _____ .
- Expressed as geometric tolerances
 - Expressed in millimeters
 - Complete and shown on the drawing
 - None of the above

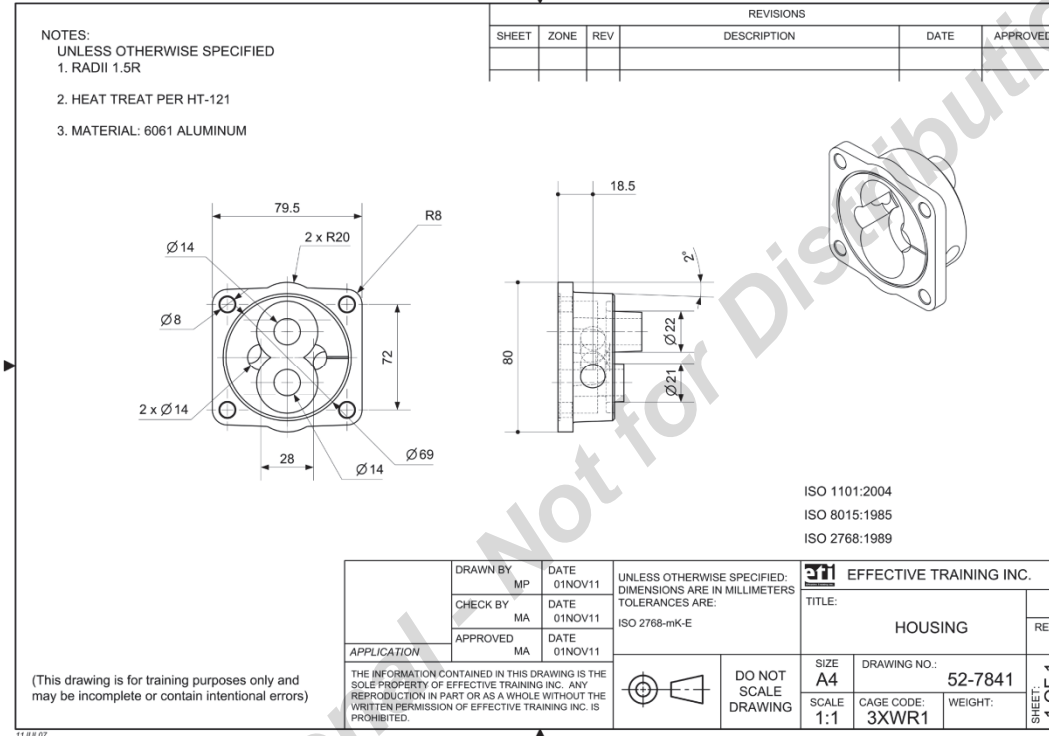


9. When ISO 1101:2004 is specified on a drawing, approximately how many additional ISO standards are invoked?
- a. Less than 5
 - b. More than 5, but less than 10
 - c. More than 30
 - d. None of the above

True / False - Indicate if each statement is true or false.

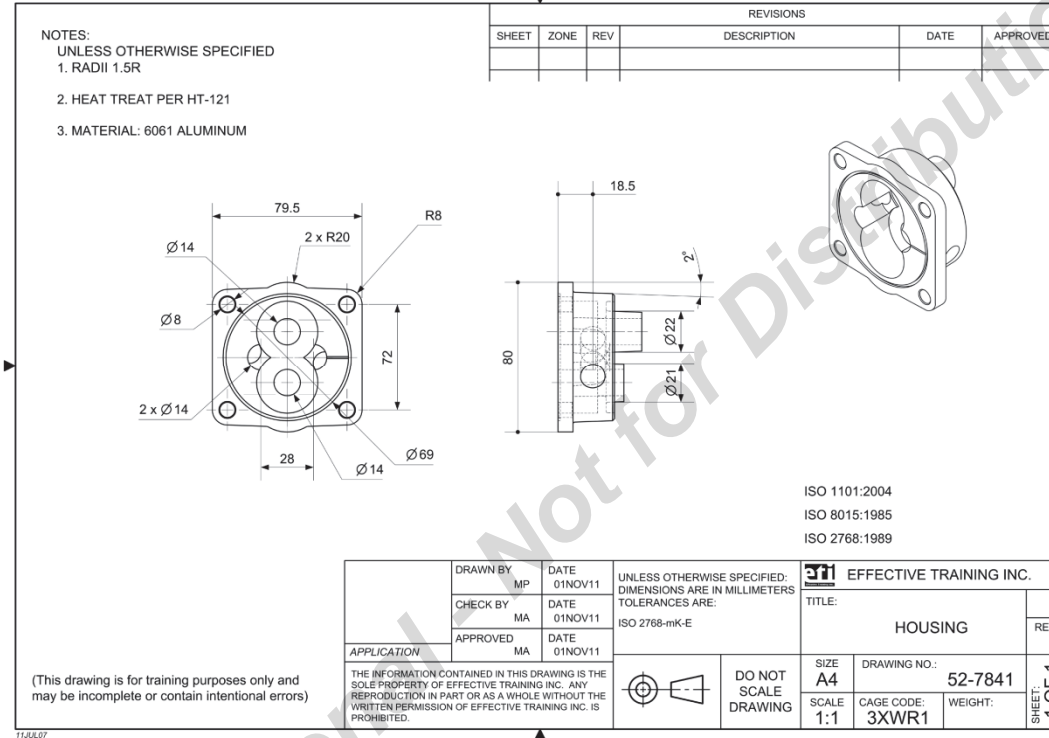
10. T F In the ISO standards system, it is preferred that drawings should be language dependent.
11. T F The International Organization for Standards uses “ISO” as a name because it means equal.
12. T F Indicating the ISO 8015 standard on a drawing invokes the “SI” units.
13. T F ISO-compliant drawings may use any specified dimensional units.
14. T F Applicable ISO standards should be specified in or near the title block.
15. T F Geometrical tolerances should be specified in accordance with functional requirements.





- A dimension line may terminate with an arrowhead, oblique line, or _____.
 - Leader line
 - Triangle
 - Double circle
 - None of the above
- What is the short horizontal line associated with a leader line called?
 - Leader line terminator
 - Reference line
 - Note line
 - None of the above





3. Which method of orthographic projection is used on this drawing?

- a. 1st angle
- b. 2nd angle
- c. 3rd angle
- d. 4th angle



Multiple Choice - Select the best answer.

4. What type of dimension is derived from other dimensions and given for informational purposes only?
- A basic dimension
 - A theoretically exact dimension
 - An auxiliary dimension
 - None of the above
5. A _____ is a straight or curved line that indicates a dimension graphically on a drawing.
- Leader line
 - Visible line
 - Dimension line
 - Angular dimension
6. Where a linear tolerance is indicated as “20f7,” it is referred to as a _____.
- Limit dimension
 - Limit deviations
 - Tolerance class symbol
 - None of the above



Multiple Choice - Select the best answer.

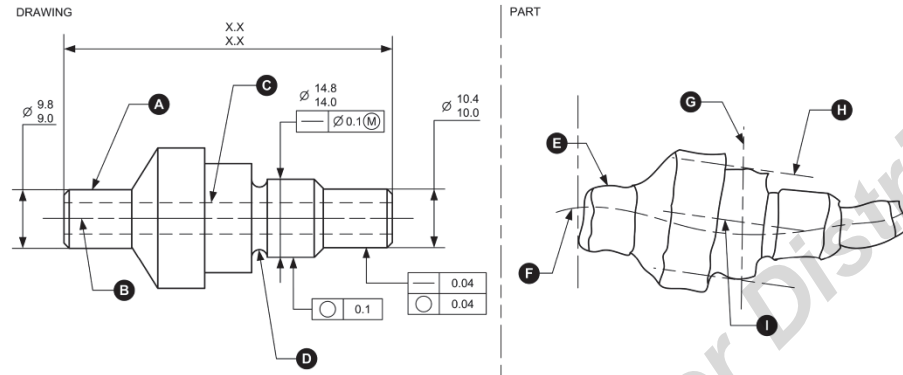
7. Which statement best describes the tolerance zone of a $40^{\circ} \pm 2^{\circ}$ angular dimension between two planar surfaces?
- a. The space between two planes, one at 38° and one at 42°
 - b. The allowable range of angles between two contacting lines
 - c. The allowable range of angles between two contacting planes
 - d. None of the above
8. What is one problem with coordinate dimensions?
- a. Square tolerance zones
 - b. No tolerance accumulation
 - c. Round tolerance zones
 - d. None of the above



True / False - Indicate if each statement is true or false.

9. T F Drawings made to ISO standards may use first-angle projection.
10. T F On drawings made to ISO standards, decimal separators may use either a comma or a point symbol.
11. T F Dimension lines of diameters must always have two terminators.
12. T F An auxiliary dimension is for informational purposes only.
13. T F Coordinate tolerancing should be used to define simple parts.
14. T F An open circle dimension line terminator indicates the origin of running dimensions.





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

1. Use the figure above to fill in the table.

Feature Domain Terminology			
Feature Labeled	Domain Appropriate Name	Feature Labeled	Domain Appropriate Name
A	1 – Nominal integral feature	F	6 – Extracted derived feature
B	2 – Nominal derived feature	G	4 – Associated derived feature
C	1 – Nominal integral feature	H	3 – Associated integral feature
D	1 – Nominal integral feature	I	4 – Associated derived feature
E	5 – Real integral feature	J	3 – Associated integral feature
1 - Nominal integral feature 2 - Nominal derived feature 3 - Associated integral feature		4 - Associated derived feature 5 - Real integral feature 6 - Extracted derived feature	



Multiple Choice - Select the best answer.

2. The ISO standards define a feature in terms of...
 - a. 4 states
 - b. 4 phases
 - c. 3 states
 - d. 3 domains

3. Geometrical tolerances such as flatness and circular run-out are categorized as which type of standards in the GPS master plan?
 - a. Fundamental
 - b. Global
 - c. General
 - d. Complementary

4. In the ISO standards system, what does the abbreviation GPS stand for?
 - a. Global production standards
 - b. Geometrical product specifications
 - c. Globally practiced standards
 - d. Global positioning system



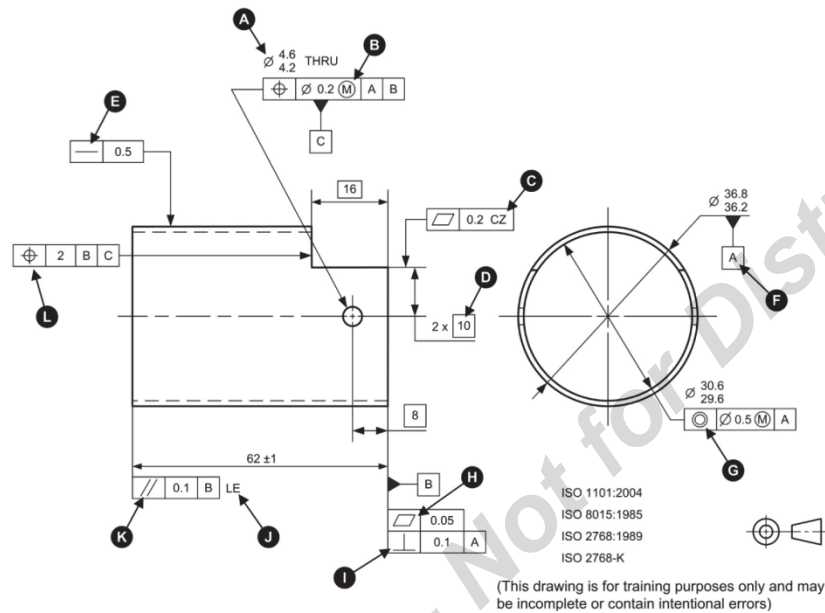
Multiple Choice - Select the best answer.

5. What is a nominal feature?
- a. A set of points extracted from a real feature
 - b. An integral feature of perfect form
 - c. An actual workpiece surface
 - d. A surface shown on the drawing

True False - Select the best answer.

6. T F The GPS master plan is explained in ISO/TR 14638:1995.
7. T F There are only four areas or “chain links” in the master plan.
8. T F Geometrical tolerance specification and verification are part of the same chain link.
9. T F ISO standards define a feature to be a portion of a part, such as a hole or tab.
10. T F ISO standards differentiate between a drawing feature and a workpiece feature.
11. T F An associated feature is a surface as shown on a drawing.

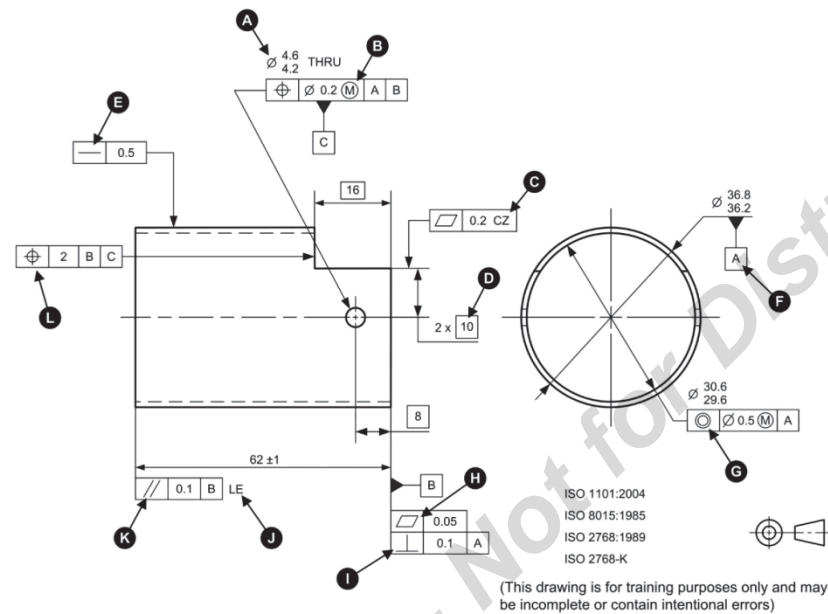




1. Match the symbol name with the label from the drawing above.

Geometrical and Common Symbol Identification	
Symbol Name	Drawing Label
Diameter	A
Flatness	H
Common Zone	C
Coaxiality	G
Maximum Material Requirement	B





1. Match the symbol name with the label from the drawing above.

Geometrical and Common Symbol Identification	
Symbol Name	Drawing Label
Parallelism	K
Perpendicularity	I
Line Element	J
Position	L
Datum Feature Identification	F
Theoretical Exact Dimension	D
Straightness	E



Multiple Choice - Select the best answer.

2. The GPS system divides geometrical tolerances into how many types?
- a. 2
 - b. 4
 - c. 14
 - d. 19
3. The geometrical tolerances that are classified as location tolerances are coaxiality, position, _____, and _____.
- a. Concentricity, symmetry
 - b. Perpendicularity, angularity
 - c. Circular run-out, total run-out
 - d. None of the above
4. Which type of geometrical tolerance can limit the most types of deviations?
- a. Form
 - b. Orientation
 - c. Location
 - d. None of the above



Multiple Choice - Select the best answer.

5. What does the abbreviation TED stand for?
- Technically exact dimension
 - Theoretical extraction data
 - Test and extraction data
 - Theoretically exact dimension
6. Which TED values may be implied on a drawing?
- 0mm, 45°, and 90°
 - 0mm, 100mm, and 1000mm
 - 0°, 90°, and 180°
 - 0°, 45°, and 90°
7. Where a dimension is enclosed in a frame (or box), what method should be used to tolerance the dimension?
- General linear or angular tolerances
 - Tolerance class (IT) symbols
 - Direct plus-minus limits
 - None of the above
8. What does the LD symbol mean?
- Least diameter
 - Least squares diameter
 - Aligned diameter
 - Minor diameter



Multiple Choice - Select the best answer.

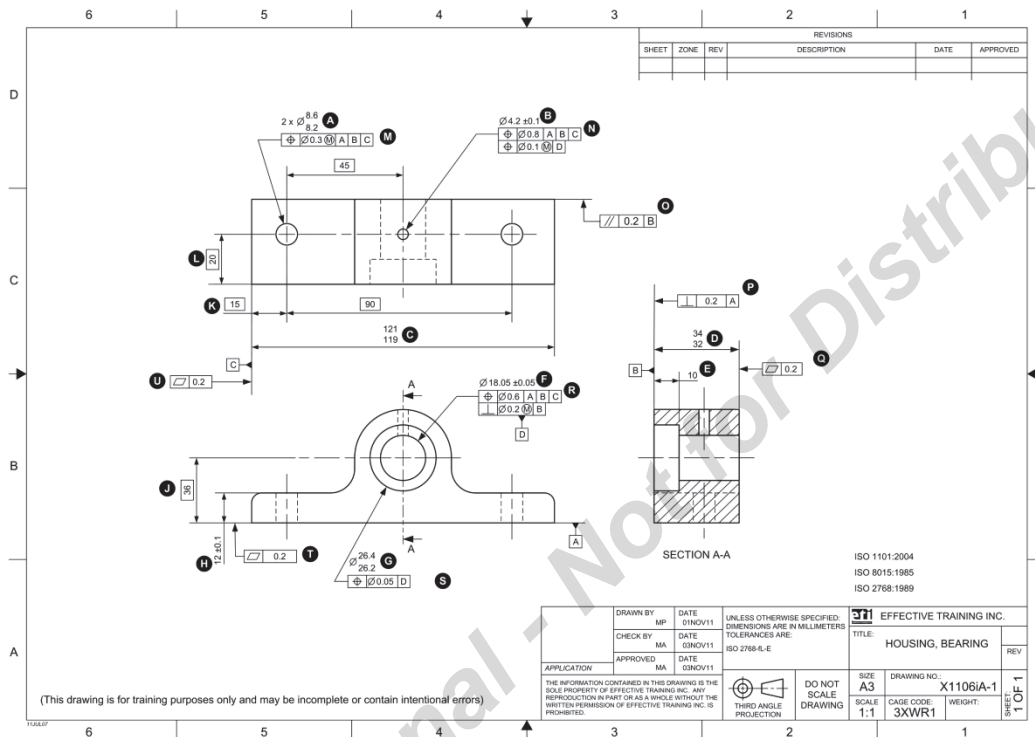
9. What does the NC symbol mean?
- a. Nominal corner
 - b. Non-concave
 - c. No common zone
 - d. None of the above
10. How can you determine if a tolerance frame applies to the surface of an integral feature?
- a. The tolerance frame is placed below the size dimension.
 - b. The tolerance frame is not connected to a center line.
 - c. The tolerance frame is directed to the outline of the feature.
 - d. The tolerance frame is placed in line with the size dimension.
11. What is the minimum number of compartments that a tolerance frame may consist of?
- a. 1
 - b. 2
 - c. 3
 - d. 4



True / False - Indicate if each statement is true or false.

12. T F An implied TED does not require a tolerance.
13. T F The LE symbol is interpreted as “line element.”
14. T F The space between two concentric circles is one of the nine common tolerance zone shapes.
15. T F The SR symbol is interpreted as “spherical relief.”
16. T F Concentricity and coaxiality tolerances are indicated with the same geometrical symbol.
17. T F One of the nine types of ISO tolerance zones is the space between two arcs.
18. T F The ISO GPS system has a symbol for square.

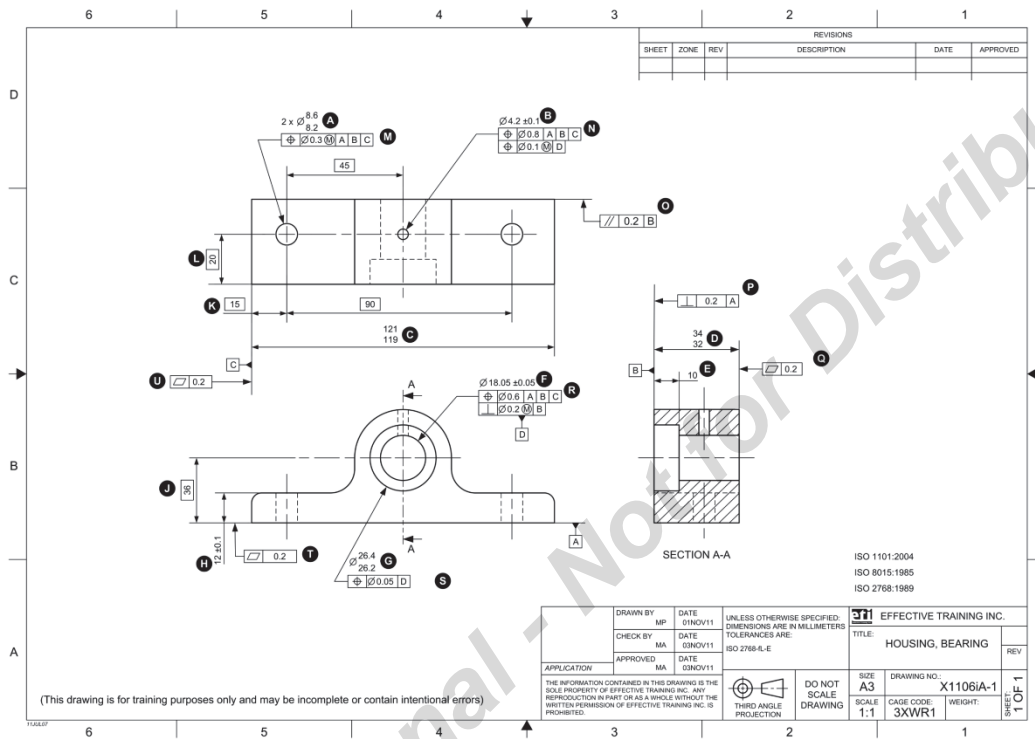




2. In the chart below, enter the value of the MMS and LMS for each dimension label, or indicate if a material condition does not apply.

Dimension	MMS	LMS	Does Not Apply
A	8.2	8.6	
B	4.1	4.3	
C	121	119	
D	34	32	
E			X





2. In the chart below, enter the value of the MMS and LMS for each dimension label, or indicate if a material condition does not apply.

Dimension	MMS	LMS	Does Not Apply
F	18	18.1	
G	26.2	26.4	
H	12.1	11.9	
J			X
K			X
L			X



Multiple Choice - Select the best answer.

3. The term “maximum material condition” is most relevant when working with _____ features of size.
- Nominal
 - Derived
 - Extracted
 - None of the above
4. Where the independency principle applies, a linear tolerance controls only the _____ of a feature of size.
- The straightness requirements of line elements
 - Actual local sizes
 - Actual mating envelopes
 - Spacing between features
5. Where no relationship is specified (MMR or LMR), a geometrical tolerance applies _____.
- To the envelope principle
 - At RFS
 - At MMC by default
 - None of the above



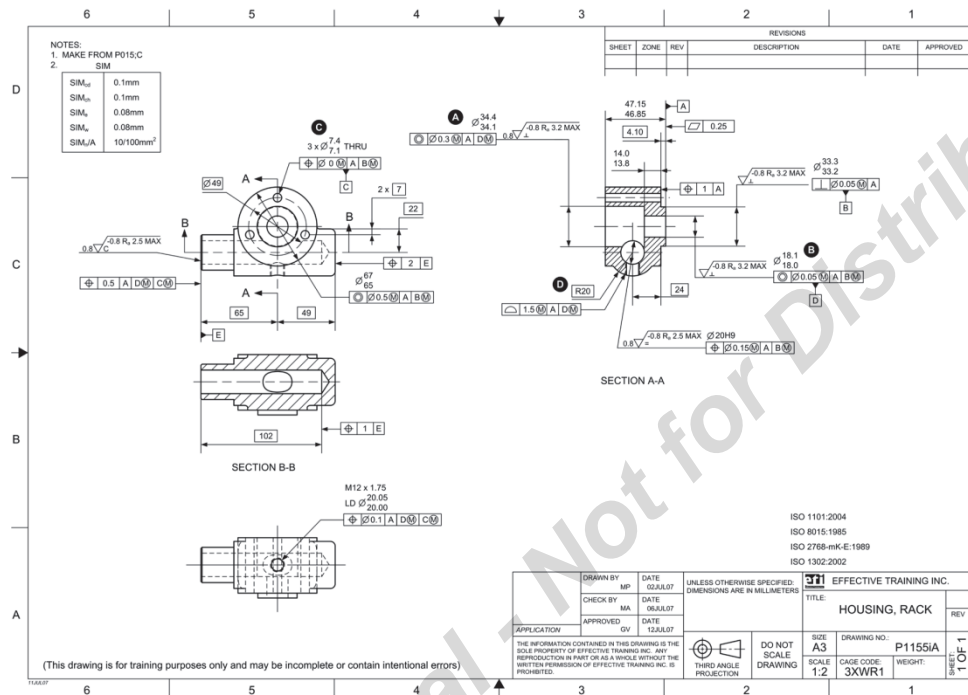
Multiple Choice - Select the best answer.

6. On an engineering drawing, _____ is the value representing the smallest size of an internal feature of size.
- Actual local size
 - Least material condition
 - Maximum material size
 - None of the above

True / False - Indicate if each statement is true or false.

7. T F Maximum material size is a drawing domain term equal to MMC.
8. T F A feature of size may have many actual local sizes.
9. T F A feature of size has either an MMS or an LMS, but not both.
10. T F In the ISO GPS system, RFS is the default condition.



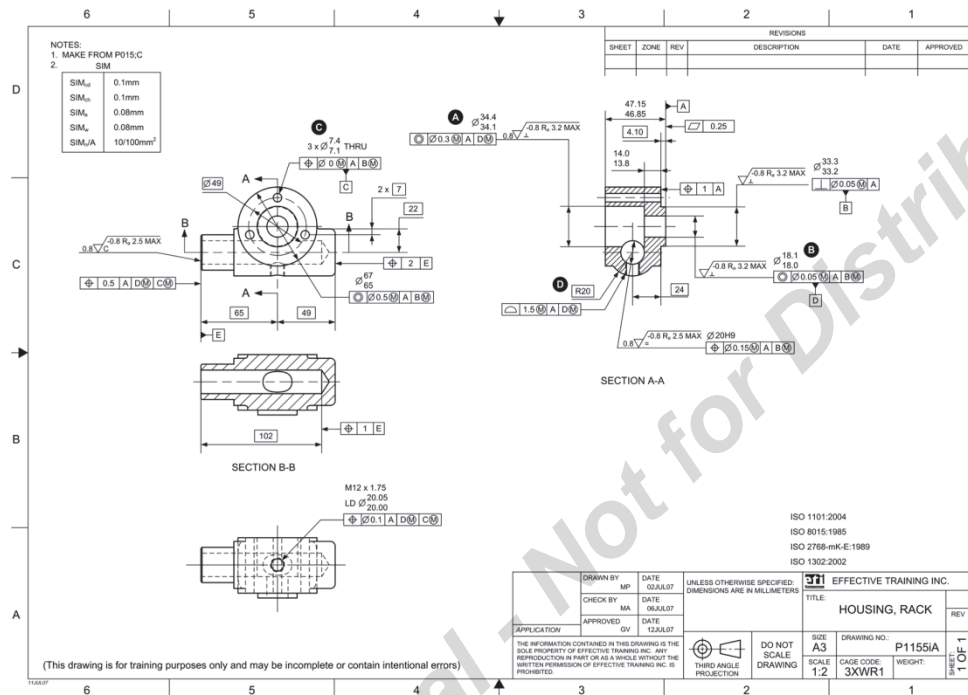


1. Is the envelope requirement or the independency the default for this drawing?

The envelope principle

How is this indicated? **By the specification of ISO 2768-mK-E: 1989 on the drawing**

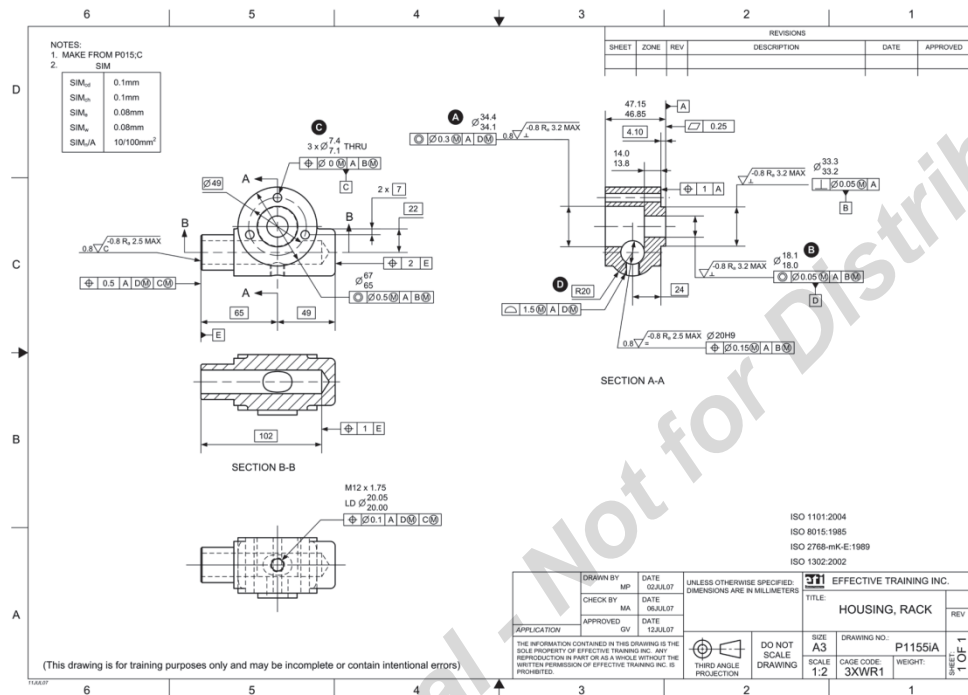




2. In the chart below, fill in the MMS and LMS for each dimension label. If a dimension does not have an MMS or LMS, place an “X” in the does not apply column.

Label	MMS	LMS	Does Not Apply
A	34.1	34.4	
B	18	18.1	
C	7.1	7.4	
D			X





3. Which ISO standard invokes the independency principle as a default for engineering drawings?

ISO 8015:1985

Where this standard is invoked, which additional specification is required?

Near the title block; ISO 2768:1989

4. Does the independency principle apply to any dimensions on this drawing? **NO**

Explain why or why not. **ISO-mK-E invokes a general envelope requirement**



Multiple Choice - Select the best answer.

5. Drawings that specify ISO 8015:1985 must also specify which additional standard?
- a. ISO 1101:2004
 - b. ISO 13715:2000
 - c. ISO 2768:1989
 - d. ISO 128:1999
6. The independency principle states that:
- a. Each specified dimensional or geometrical requirement shall be met independently.
 - b. Orientation tolerances shall not limit form variations.
 - c. Location tolerances shall not limit orientation variation.
 - d. Angles are features of size.
7. How can a designer invoke the envelope requirement for the entire drawing?
- a. Specify ISO 2768 with a -E suffix
 - b. Specify ISO -E near the title block
 - c. Specify E near the title block
 - d. None of the above



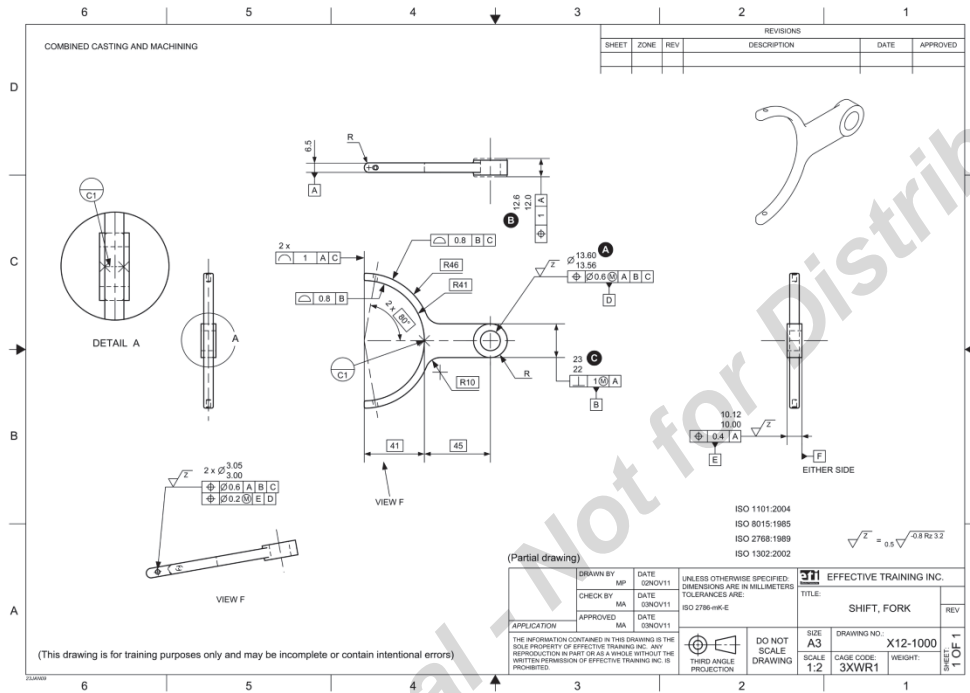
Multiple Choice - Select the best answer.

8. The envelope requirement states that a feature of size cannot violate the envelope of perfect form at:
- a. Maximum material size
 - b. Least material size
 - c. Maximum and least material size
 - d. None of the above

True / False - Indicate if each statement is true or false.

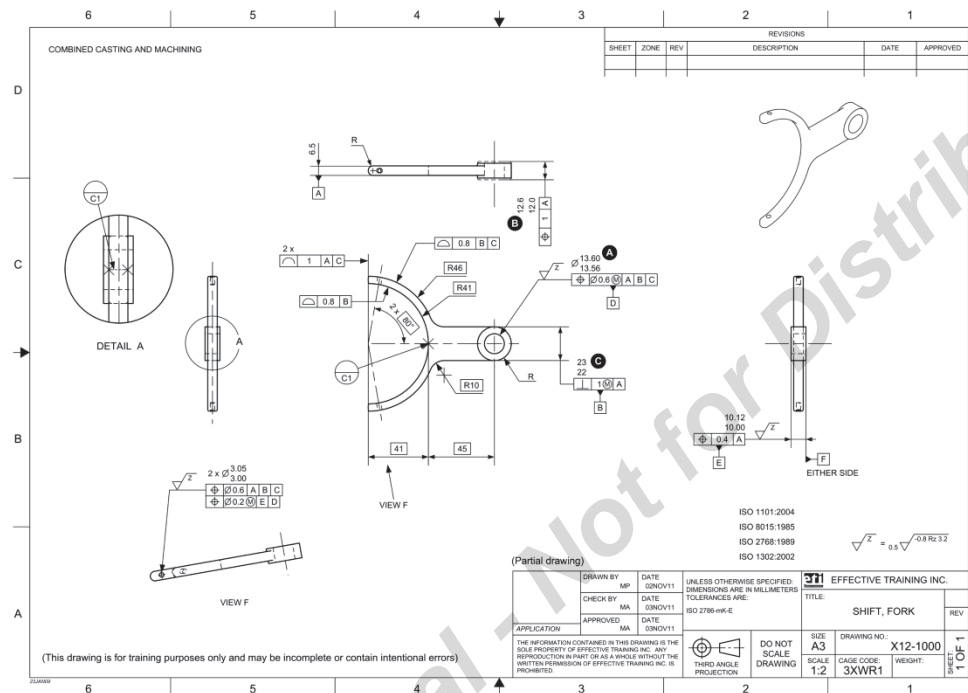
9. T F ISO 2768 defines independency and the envelope requirement.
10. T F ISO 8015 invokes the RFS default condition.
11. T F Where the independency principle applies, the drawing requires an indication for each characteristic (size, form, orientation, and location).
12. T F The envelope requirement is typically used when a feature of size assembles with another feature of size.
13. T F Using the \textcircled{M} symbol always invokes the envelope requirement.





1. What is the MMVS of diameter labeled **A**? 12.96
2. How many places are collective requirements invoked? 3
3. How many times is reciprocity applied on this drawing? 0





4. How do collective requirements affect the width labeled **C** ?
- The width may increase beyond the size limits
 - The perpendicularity deviation is required to be less than 1
 - C** The perpendicularity deviation may exceed 1
 - None of the above
5. What type of boundary is associated with the feature of size labeled **B** ?
- a** Extreme boundary
 - MMVC
 - LMVC
 - None of the above



Multiple Choice - Select the best answer.

6. When should the RPR be considered for use in a tolerance frame?
- a. To control a minimum distance
 - b. To prohibit the use of a collective requirement
 - c. To enlarge the size tolerance
 - d. None of the above
7. When should the LMR be considered for use in a tolerance frame?
- a. To enlarge the size tolerance
 - b. To prohibit the use of a collective requirement
 - c. To control a minimum distance
 - d. None of the above
8. What is a collective requirement?
- a. An additional requirement to MMR that changes the size tolerance that the tolerance feature of size cannot violate
 - b. An LMVC or MMVC boundary that the toleranced feature of size cannot violate
 - c. A requirement that a geometrical tolerance applies to any size of toleranced features of size
 - d. None of the above



Multiple Choice - Select the best answer.

9. What is the reciprocity requirement?

- a. An additional requirement that changes the size tolerance of a feature of size by adding the collective requirements of MMR and LMR
- b. A requirement for a feature of size that defines a geometrical feature of the same type and of perfect form, with a value equal to LMVS
- c. A requirement that a geometrical tolerance applies at any size of toleranced feature of size
- d. None of the above

10. What is the maximum material requirement?

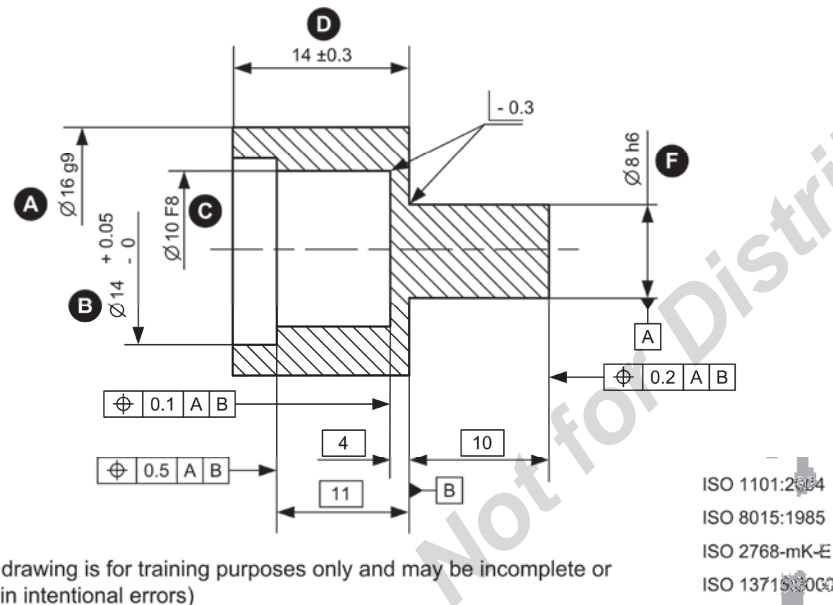
- a. An additional requirement to MMS that changes the size tolerance that the toleranced feature of size cannot violate
- b. A requirement for a feature of size that defines a geometrical feature of the same type and of perfect form, with a value equal to LMVS
- c. A requirement that a geometrical tolerance applies at any size of toleranced feature of size
- d. None of the above



True / False - Indicate if each statement is true or false.

11. T F An MMVC boundary may limit geometrical deviations.
12. T F An LMVC can only apply when the RPR symbol is specified.
13. T F To calculate an MMVS for an internal feature of size, a geometrical tolerance is added to the MMS.
14. T F Where RPR is indicated, a collective requirement also applies.
15. T F Reciprocity permits a geometrical tolerance to vary more than the specified value.

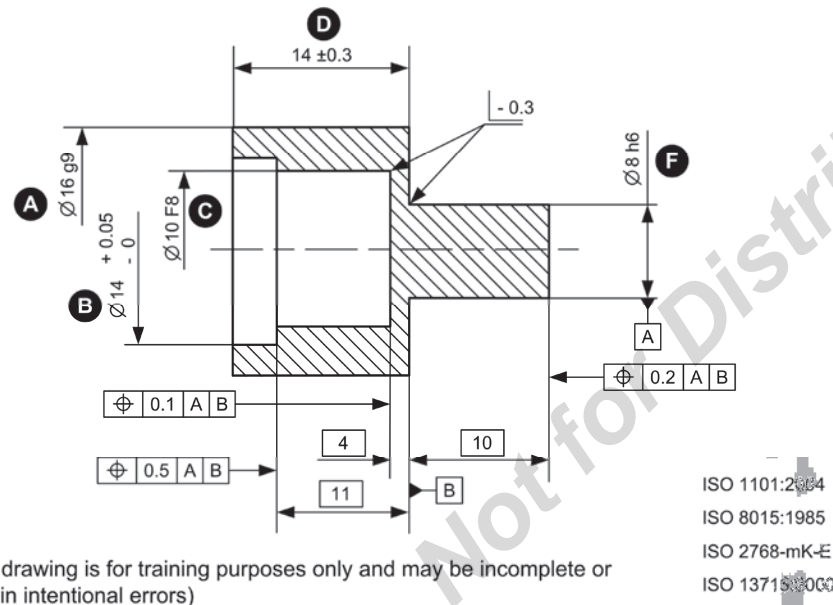




Use the figure above to answer questions 1 - 5.

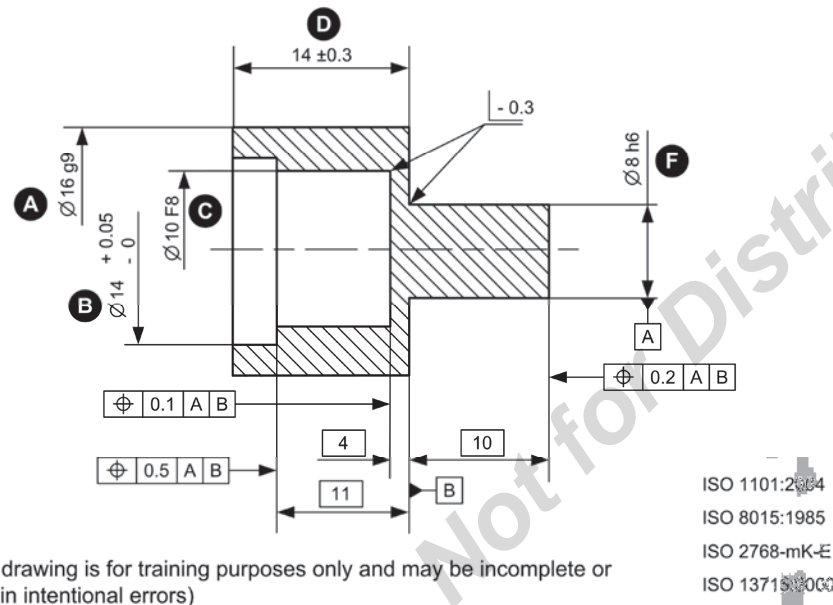
1. What part of the dimension **A** indicates the fundamental deviation? Letter "g"
2. What term is used to describe the "ø10" on the dimension labeled **C**?
 - a. The actual size
 - b. The basic size**
 - c. The minimum size
 - d. The maximum size





3. What difficulty might a designer have in specifying the dimension labeled **B** as a limit and fit code?
- $\varnothing 14$ is not a preferred standardized basic size
 - The -0 limit does not match any fundamental deviation
 - The $+0.05$ limit is too coarse a tolerance for the limits and fits system
 - None of the above
4. What does the letter “h” indicate about the limit and fit system code for the dimension labeled **F**?
- The specification is a hole designation
 - The MMS is $\varnothing 8.000$
 - The envelope principles applies
 - None of the above





5. Which manufacturing process would definitely be capable for producing the dimension labeled **C**?
 Hint: See Figure 4.6 on page 80.
- Milling
 - Drilling
 - Reaming
 - None of the above



Multiple Choice - Select the best answer.

6. A straight line that represents the basic size from which deviations and tolerances are referred is called:
- a. A fundamental deviation origin
 - b. A basic shaft
 - c. A basic hole
 - d. None of the above
7. What are IT grades 5 – 11 are usually applied to?
- a. Measuring tools
 - b. Large manufacturing tolerances
 - c. Production parts
 - d. Coarser (less controlled) manufacturing operations
8. What benefits are provided by the standardized system of limits and fits?
- a. It always results in a clearance fit
 - b. It invokes the envelope principle for the feature of size
 - c. It reduces the number of stock sizes and tools for manufacturing
 - d. It derives the type of fit based on manufacturing processes



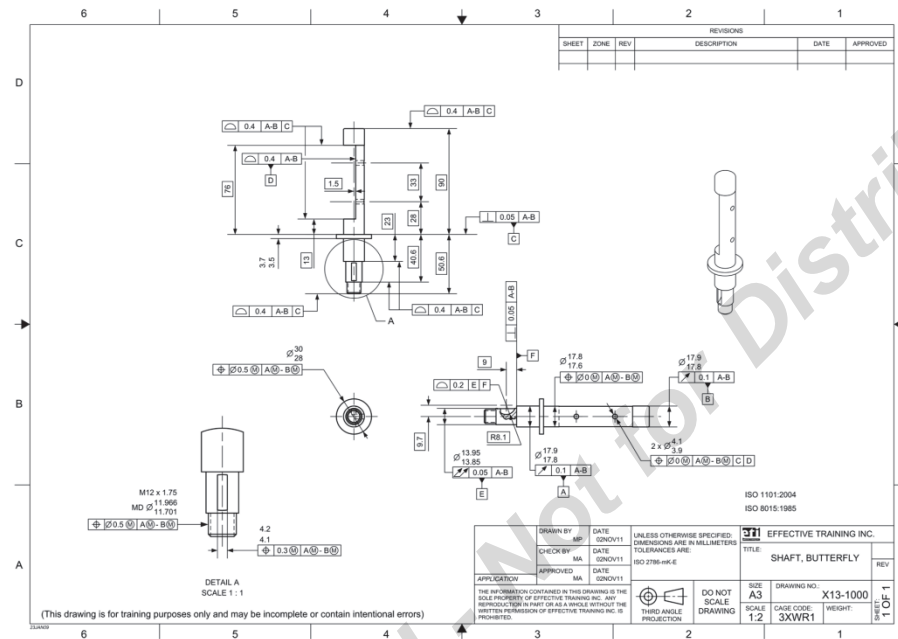
Multiple Choice - Select the best answer.

9. What size limits for the hole and the shaft are defined in the specification 20 D9/ h9?
- a. Hole 19.972 - 19.993 / Shaft 19.987 - 20.000
 - b. Hole 20.000 - 20.117 / 19.948 - 20.065
 - c. Hole 20.065 - 20.117 / 19.948 - 20.000
 - d. None of the above
10. What type of a fit is defined with the specification 8H11/k6?
- a. Locational clearance
 - b. Free running
 - c. Locational transition
 - d. None of the above

True / False - Indicate if each statement is true or false.

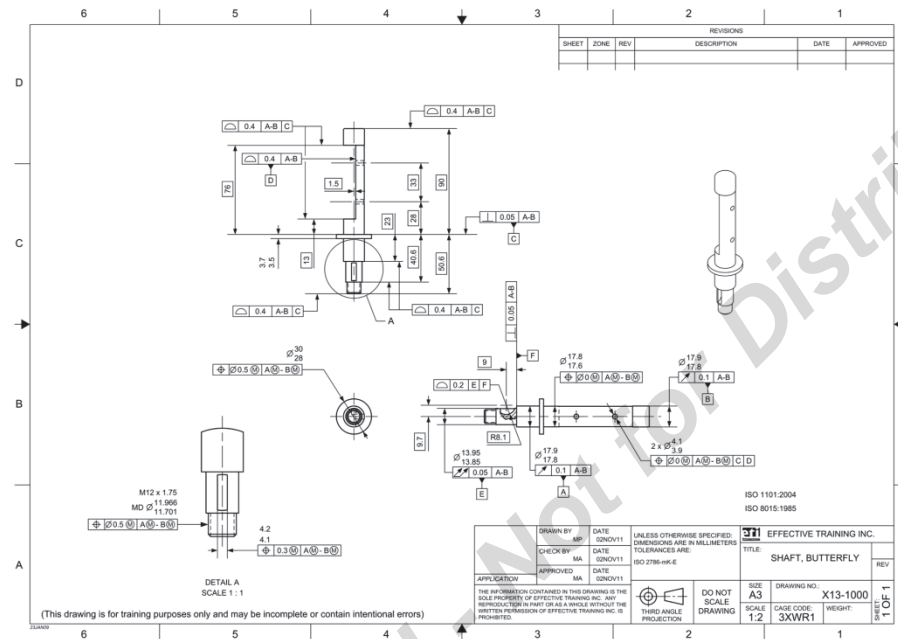
11. T F It may be necessary to apply the $\text{\textcircled{E}}$ symbol to limit and fit specifications.
12. T F Limit and fit codes may be applied to any shaped mating geometry.
13. T F All international tolerance (IT) grades begin at the zero line.
14. T F A transition fit may provide either a clearance or interference-type fit.





- Which datum letter frame symbols specify planar datums?
 - C, D, and F
 - B, C, and E
 - B, C, and D
 - None of the above
- How is datum feature E referenced in a geometrical tolerance?
 - As a primary datum feature
 - As a secondary datum feature
 - As a tertiary datum feature
 - None of the above





3. How is datum feature F referenced in a geometrical tolerance?
 - a. As a primary datum feature
 - b.** As a secondary datum feature
 - c. As a tertiary datum feature
 - d. None of the above

4. How is datum feature C referenced in geometrical tolerances?
 - a. As a primary datum feature
 - b.** As a secondary datum feature
 - c. As a tertiary datum feature
 - d. None of the above



Multiple Choice - Select the best answer.

5. What is a datum system?
- a. Three mutually perpendicular workpiece surfaces indicated by datum symbols
 - b. A group of two or more separate datums used as a combined datum indication for a tolerated feature
 - c. A theoretical exact geometrical reference to which a tolerated feature is related
 - d. None of the above
6. What is a benefit of the datum system?
- a. Workpiece surfaces have better quality
 - b. It relate dimensional measurements to workpiece function
 - c. It identifies manufacturing relationships between workpiece features
 - d. None of the above
7. What is the primary consideration for selecting datum features on a workpiece?
- a. The importance of the features to design function
 - b. Manufacturing preferences and tool capabilities
 - c. Identifying the largest surface on the workpiece
 - d. None of the above



Multiple Choice - Select the best answer.

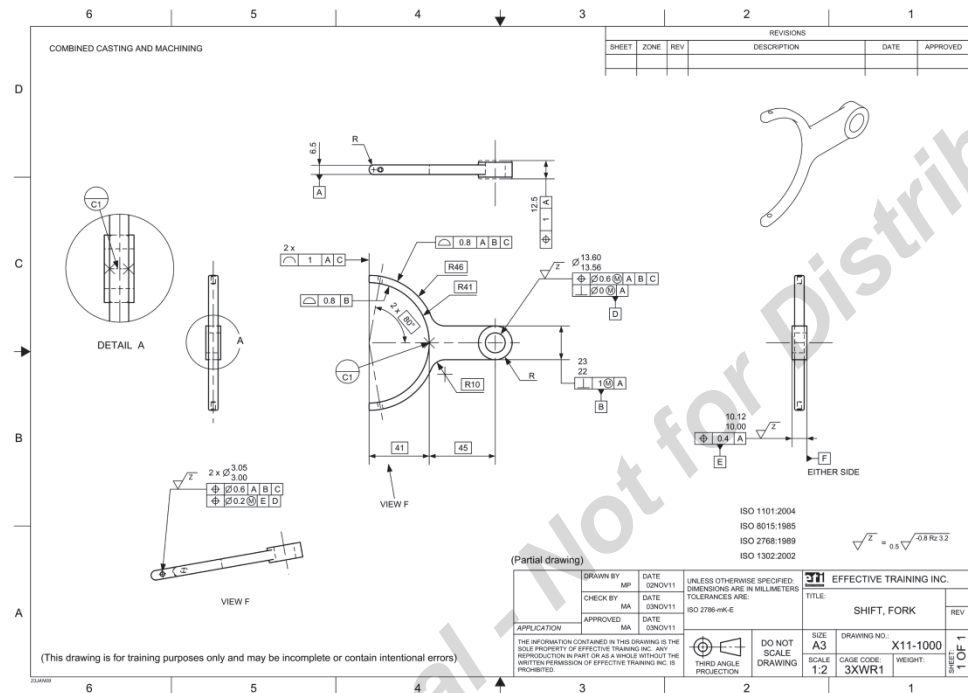
8. What effect does changing the order in which datum features are indicated in a tolerance frame have on the workpiece?
- a. It changes the manufacturing process
 - b. No effect as long as the same datum feature letters are used
 - c. It has significant effect on part measurement
 - d. None of the above
9. How many degrees of freedom are restricted when a planar surface is referenced as a primary datum feature?
- a. 1
 - b. 2
 - c. 3
 - d. None of the above
10. A datum is a theoretically exact geometrical reference such as...
- a. A plane, point, or axis
 - b. Planes, points, and formulas
 - c. Planes, lines, and curves
 - d. None of the above



True / False - Indicate if each statement is true or false.

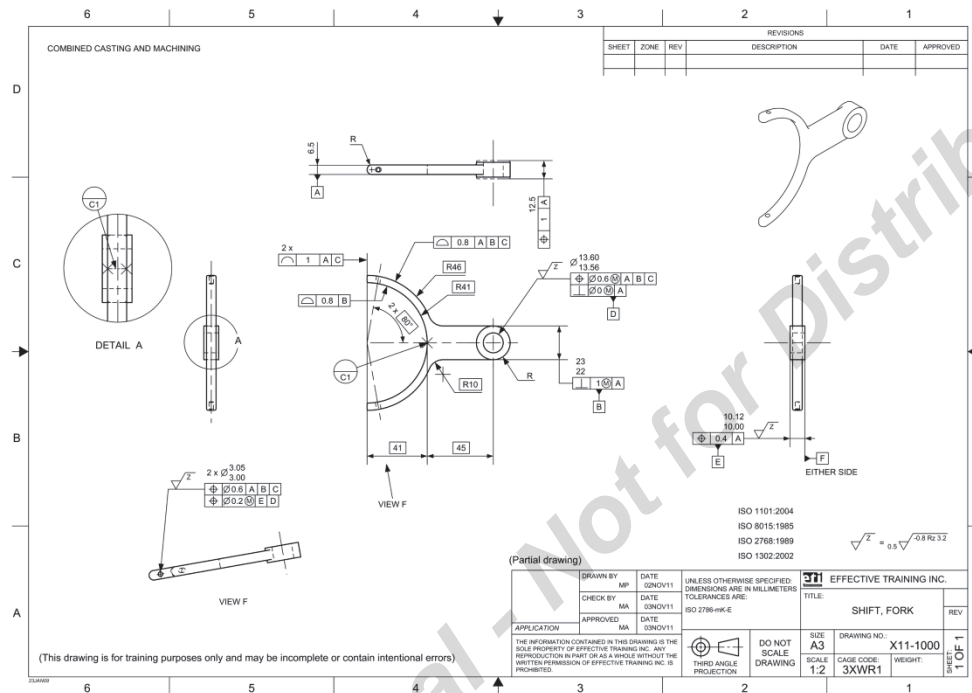
11. T F A datum feature is a real feature of a workpiece.
12. T F A datum feature is theoretical.
13. T F Every workpiece has eight degrees of freedom.
14. T F Datum reference sequence has an impact on part acceptance.
15. T F When rocking occurs between a datum feature and its simulated datum, the orientation of the datum feature should be optimized to produce the maximum distance between the datum feature and the simulated datum.





- Which datum is established by a datum target(s)?
 - A
 - B
 - C**
 - D
- Why is a datum target used on the drawing?
 - The workpiece would rock or wobble when a radius is used as a datum feature
 - Because datum feature C is not a full diameter
 - C** It is not practical to use the entire feature as a datum feature
 - None of the above





3. What type of datum target is shown on this drawing?

- a. Datum target point
- b. Datum target line**
- c. Datum target area
- d. Datum target curve

4. Which gage element could be used to simulate datum C?

- a. A spherical-tipped gage pin
- b. A flat-tipped gage pin
- c. The side of a cylindrical gage pin**
- d. None of the above



Multiple Choice - Select the best answer.

5. Which two symbols should always be included in the specification of a datum target?
- a. A datum target frame symbol and a datum letter frame symbol
 - b. A datum target frame symbol and a model coordinate symbol
 - c. A datum target frame symbol and a target type symbol
 - d. None of the above
6. Datum target areas may be specified by a datum target frame symbol and...
- a. A hatched area surrounded by a thin double-dashed line and TED(s)
 - b. TED dimensions to describe the diameter of the area
 - c. The diameter of the area shown inside with a datum target point symbol
 - d. None of the above
7. What is a datum target?
- a. Datum feature(s) on the real workpiece that receive special treatments
 - b. A workpiece surface used to simulate a datum
 - c. Points, lines, or limited areas on the workpiece used for contact with the manufacturing and inspection equipment to define the required datums
 - d. None of the above



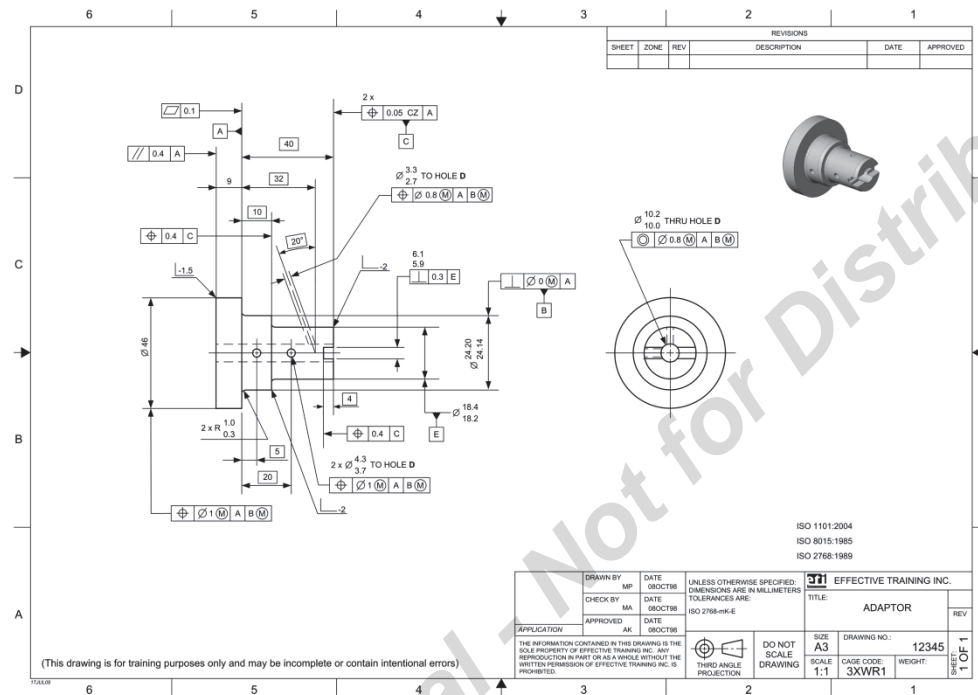
Multiple Choice - Select the best answer.

8. What is one precaution when using datum targets?
- a. Avoiding the use of datum targets on offset surfaces
 - b. Ensuring that the part has only one orientation and location in the inspection set-up
 - c. Avoiding the use of datum targets on irregular or complex geometry
 - d. None of the above

True / False - Indicate if each statement is true or false.

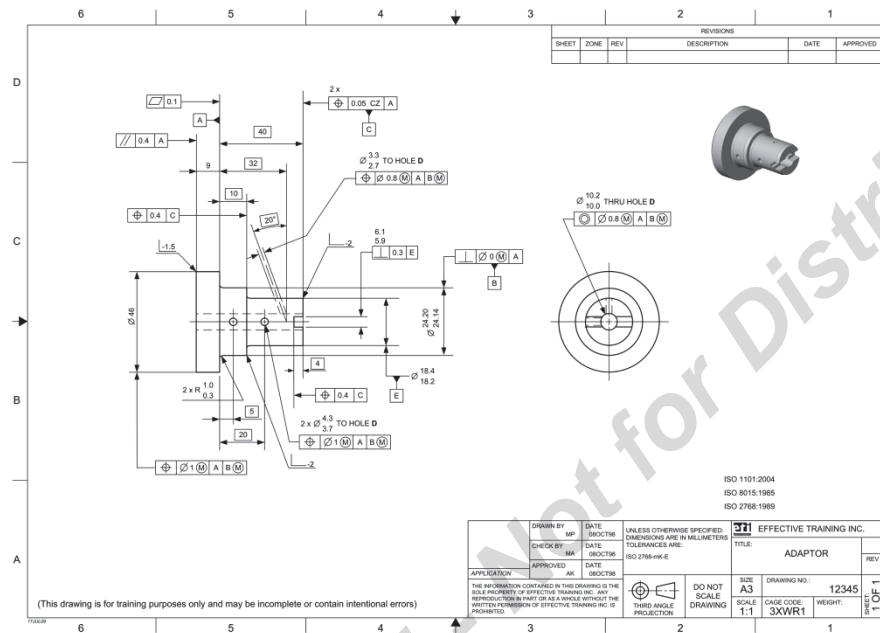
9. T F Datum target points and lines require a sharp (tip or edge) datum feature simulator.
10. T F A datum target area should be simulated with the same shape gage element as a datum target point.
11. T F Datum targets should be used where complex surfaces are datum features.
12. T F Datum targets are often specified where offset surface are datum features.





1. Which datum features are size datum features? B & E
2. When inspecting the position of the $\varnothing 46$ (left side of front view), how is datum B established?
 - a. The axis of the smallest circumscribed cylinder
 - b. The axis of the smallest circumscribed cylinder perpendicular to datum A
 - c. The axis of the MMVC cylinder perpendicular to datum A
 - d. The axis of the MMVC cylinder

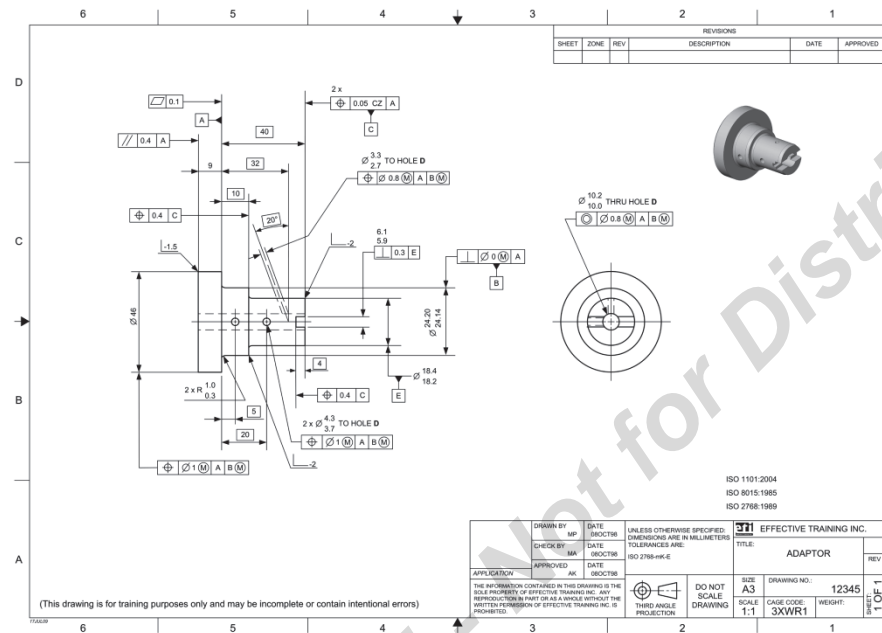




3. When inspecting the position tolerance applied to the $\varnothing 46$ (left side of front view), how much datum displacement is possible?
 - a. Zero
 - b. 0.06
 - c. 0.5
 - d. None of the above

4. Which datum feature requires an adjustable size datum feature simulator?
 - a. A
 - b. B
 - c. C
 - d. E





5. Which datum feature requires a fixed-size datum feature simulator?

- a. A
- b. B**
- c. C
- d. E

6. Which datum feature is subject to datum feature displacement?

- a. A
- b. B**
- c. C
- d. E



Multiple Choice - Select the best answer.

7. Where a pattern of two holes are referenced as a datum feature, how does ISO 2692:2006 require this to be shown in the tolerance frame?
- a. A
 - b. A-A
 - c. A $\text{\textcircled{M}}$
 - d. (A-A) $\text{\textcircled{M}}$
8. Where two coaxial diameters are referenced as common datum features, how is it shown in the tolerance frame?
- a. AB
 - b. A-B
 - c. A/B
 - d. None of the above
9. When the datum feature is a hole and referenced as primary at RFS, how should the datum be simulated?
- a. A cylindrical datum target pin
 - b. An MMVS pin
 - c. An adjustable pin
 - d. None of the above



Multiple Choice - Select the best answer.

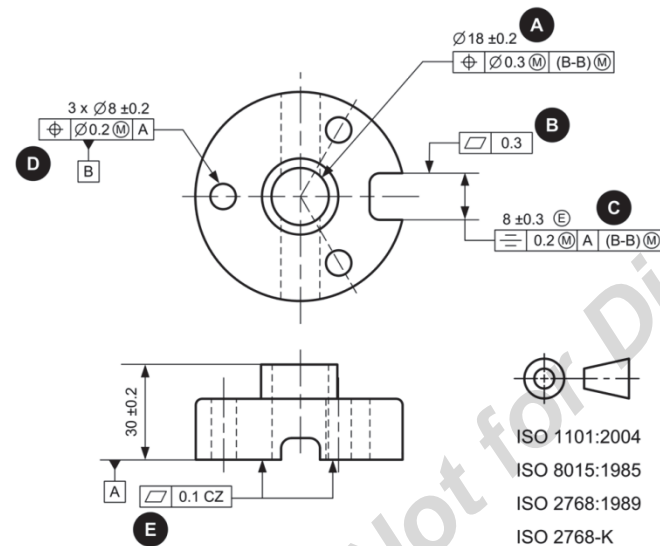
10. When the datum feature is a hole and referenced as primary at MMR, how should the datum be simulated?
- a. A cylindrical datum target pin
 - b. An MMVS pin
 - c. An adjustable pin
 - d. None of the above
11. When a geometrical tolerance references a threaded hole as the datum feature what is the default interpretation?
- a. The datum is simulated from the minor diameter
 - b. The datum is simulated from the major diameter
 - c. The datum is simulated from the pitch diameter
 - d. None of the above
12. Which statement is the best description of datum displacement?
- a. Accepting a minor violation of the engineering specifications to reduce scrap rates
 - b. Permissible movement of the datum feature within its MMVC boundary
 - c. The use of statistical data to replace the actual datum with an optimized datum
 - d. Moving the tolerance zone against the datum reference frame



True / False - Indicate if each statement is true or false.

13. T F Size datum features referenced with MMR are required to be equalized.
14. T F A size datum feature may produce a datum axis or median plane.
15. T F Where size datum features are not referenced with MMR, RFS is the default.
16. T F A single hole from a pattern of holes may be specified as a datum feature.



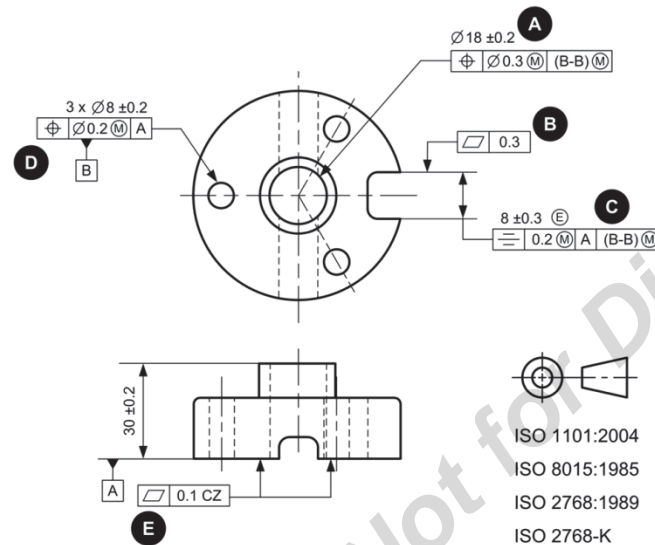


(This drawing is for training purposes only and may be incomplete or contain intentional errors)

Use the figure above to answer questions 1 - 5.

- What is the tolerance zone of the flatness tolerance labeled **B**?
 - The space between two parallel lines 0.3 apart
 - The space between two parallel planes 0.3 apart
 - The space within an MMVS boundary
 - None of the above
- The largest tolerance value that could be applied to the flatness tolerance labeled **B**, must be less than...
 - 0.2
 - 0.3
 - 0.4
 - 0.6





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

5. What happens if the CZ modifier is removed from the flatness tolerance labeled **E** ?
- The two surfaces must be flat within 0.1, but not coplanar
 - The two surfaces must be flat and coplanar within 0.1
 - The envelope requirement would limit flatness deviations
 - The extreme boundary cannot be calculated



Multiple Choice - Select the best answer.

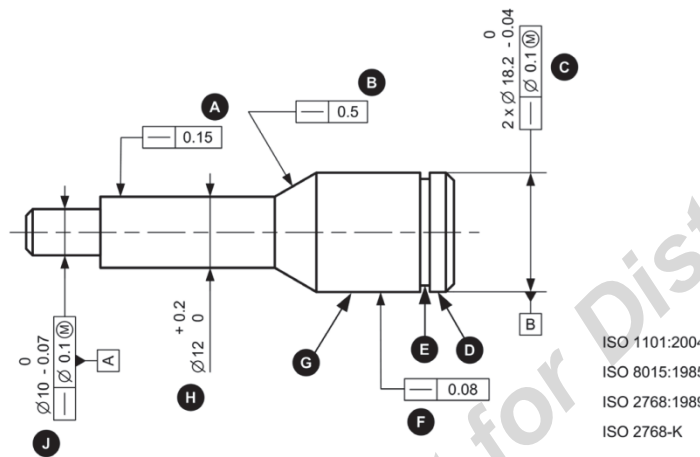
6. Which geometrical tolerance could provide an indirect flatness requirement?
- a. Concentricity
 - b. Symmetry
 - c. Position
 - d. None of the above
7. Which modifier can be applied to a flatness tolerance?
- a. NC
 - b. LE
 - c. $\text{\textcircled{R}}$
 - d. ACS
8. What is a common application for a flatness tolerance?
- a. Ensuring assembly
 - b. Gasket joint surfaces
 - c. Bearing journals
 - d. None of the above



True / False - Indicate if each statement is true or false.

9. T F A flatness tolerance can be applied with the NC modifier.
10. T F A flatness tolerance can be applied to a feature of size.
11. T F General geometrical tolerances can be used to indirectly control flatness deviations.
12. T F A flatness tolerance can override the envelope principle.
13. T F A flatness tolerance may affect the extreme boundary of a feature of size.



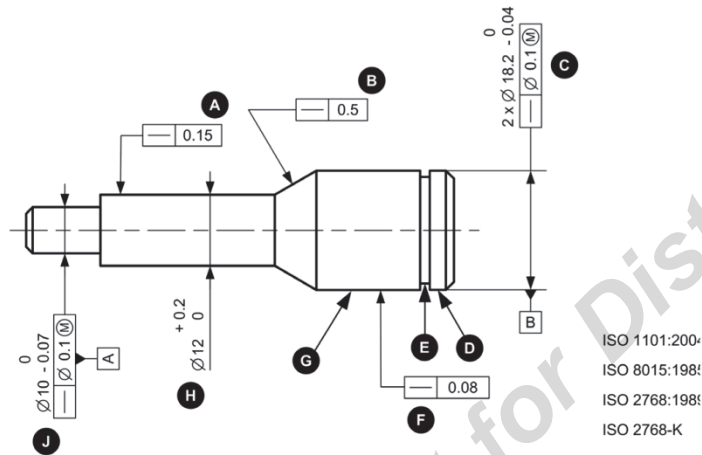


This drawing is for training purposes only and may be incomplete or contain intentional errors)

Use the figure above to answer questions 1 - 5.

- The straightness tolerance labeled **F** applies to which surface(s)?
 - D
 - E
 - G**
 - D and G
- Can the straightness tolerance labeled **B** be applied to a conical surface?
 - Yes – because a cone is a feature of size
 - Yes – because the integral line elements are nominally straight**
 - No – because straightness must be applied to linear or cylindrical surfaces
 - No – because this is not an integral feature



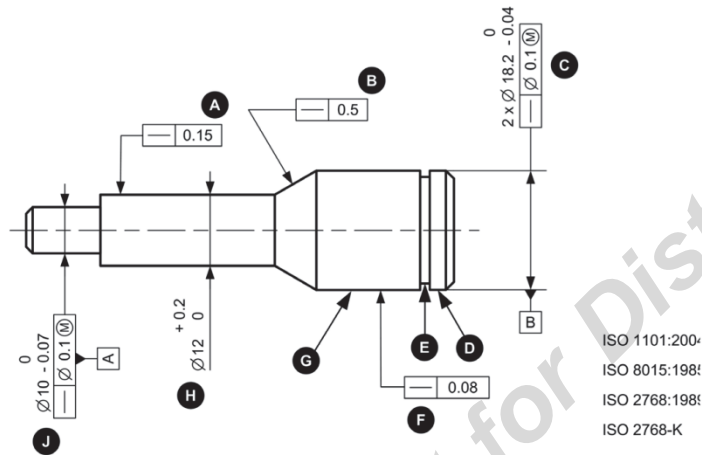


This drawing is for training purposes only and may be incomplete or contain intentional errors)

3. What does the straightness tolerance labeled **J** limit?
 - a. Straightness deviations of the axis
 - b. Straightness deviations of the derived nominal feature
 - c. Straightness deviations of the real feature
 - d.** Straightness deviations of the derived median line

4. What is the extreme boundary of the diameter labeled **H**?
 - a. $\varnothing 11.85$
 - b. $\varnothing 12.05$
 - c. $\varnothing 12.2$
 - d.** $\varnothing 12.35$





This drawing is for training purposes only and may be incomplete or contain intentional errors)

5. What is the maximum straightness deviation permitted by the tolerance frame labeled **C**?
- a. 0.04
 - b. 0.06
 - c. 0.1
 - d. 0.14**



Multiple Choice - Select the best answer.

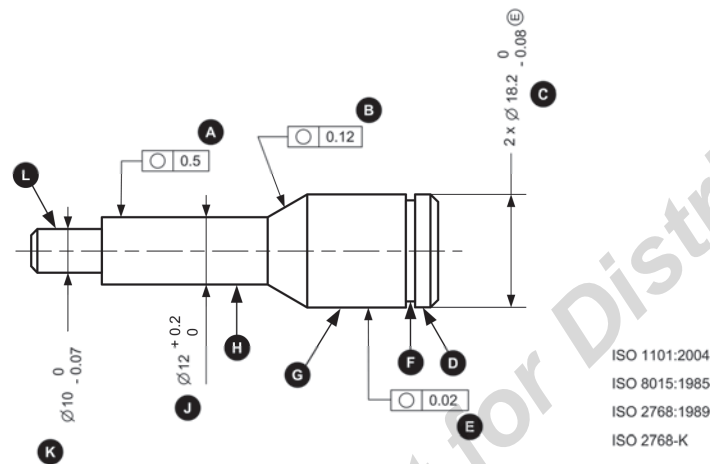
6. Which modifier can be used in a straightness tolerance applied to surface elements?
- a. CZ
 - b. LE
 - c. $\text{\textcircled{R}}$
 - d. \varnothing
7. Which geometrical tolerance can indirectly limit straightness deviations of a derived median line?
- a. Flatness
 - b. Position
 - c. Circular run-out
 - d. Profile any line



True / False - Indicate if each statement is true or false.

8. T F A straightness tolerance frame may have a primary datum reference.
9. T F A straightness tolerance should be indicated to integral features using a leader line.
10. T F A straightness tolerance can limit deviation of flatness.
11. T F A straightness tolerance with \textcircled{M} may have additional tolerance through collective requirements.



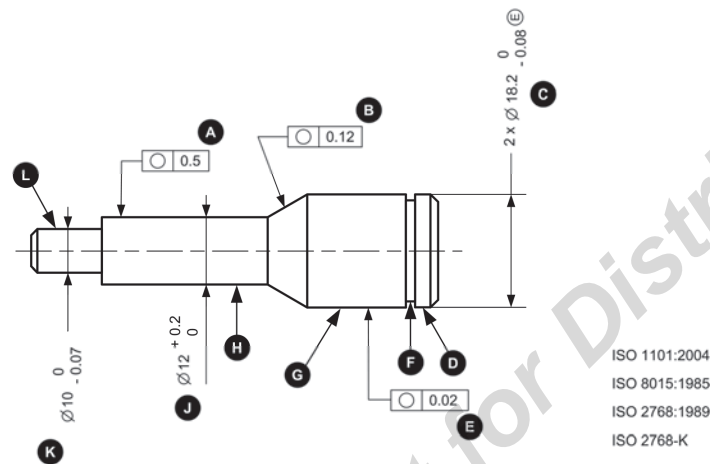


(This drawing is for training purposes only and may be incomplete or contain intentional errors)

Use the figure above to answer questions 1 - 4.

- What is the maximum permissible roundness deviation of the diameter labeled **G**?
 - 0
 - 0.02
 - 0.08
 - None of the above
- What defines the maximum permissible roundness deviation for the diameter labeled **H**?
 - The independency principle
 - The envelope requirement
 - The roundness specification
 - None of the above





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

3. What limits the roundness deviations for the diameter labeled **L** .
- The independency principle
 - The envelope requirement
 - The roundness specification
 - d.** None of the above
4. Which statement describes the tolerance zone for the roundness tolerance labeled **B** ?
- The space between two coaxial cones
 - b.** The space between two concentric circles
 - The space between two coaxial cylinders
 - None of the above



Multiple Choice - Select the best answer.

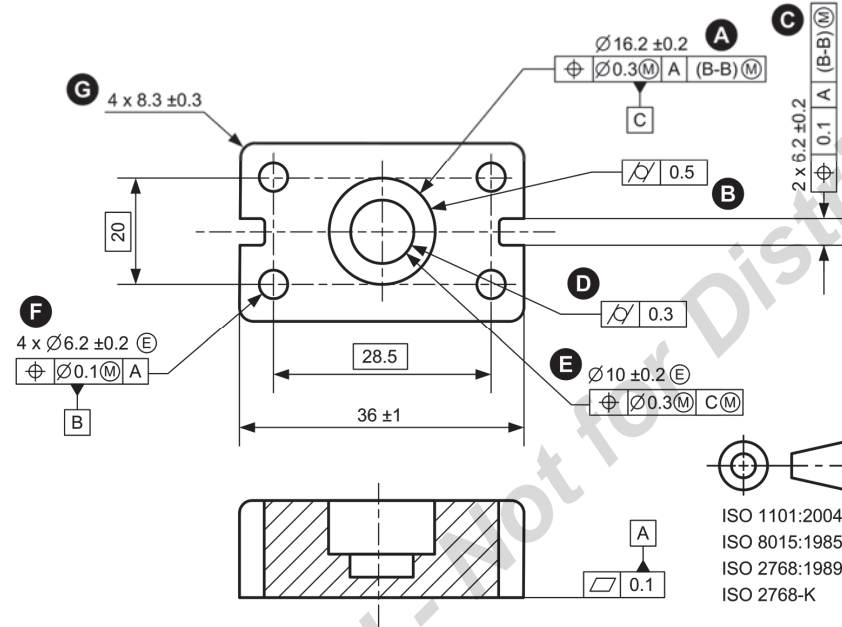
5. Which statement best describes an application for a roundness tolerance?
- a. Assembly
 - b. Limit lobing of a shaft
 - c. Centering or alignment
 - d. Minimum wall thickness
6. Where the envelope requirement applies to a cylindrical feature of size, a roundness tolerance...
- a. May have a tolerance value larger or smaller than the limits of size
 - b. Must have a tolerance value larger than the limits of size
 - c. Must have a tolerance value smaller than the limits of size
 - d. None of the above



True / False - Indicate if each statement is true or false.

7. T F Roundness is a property of a cylinder.
8. T F A roundness tolerance cannot affect the extreme boundary of a diameter.
9. T F A roundness tolerance may be used to improve rolling characteristics.
10. T F Roundness deviations may be indirectly controlled with general geometrical tolerances.
11. T F A roundness tolerance may override the envelope requirement.





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

Use the figure to answer questions 1-3 below.

1. What is the maximum permissible cylindricity deviation of diameter labeled **F** ? **0.4**
2. What is the maximum permissible cylindricity deviation of diameter labeled **A** ? **0.5**
3. What limits the cylindricity deviation of diameter labeled **E** ?
 - a. The envelope requirement
 - b. The MMVC
 - c. The cylindricity tolerance**
 - d. Nothing; it is undefined



Multiple Choice - Select the best answer.

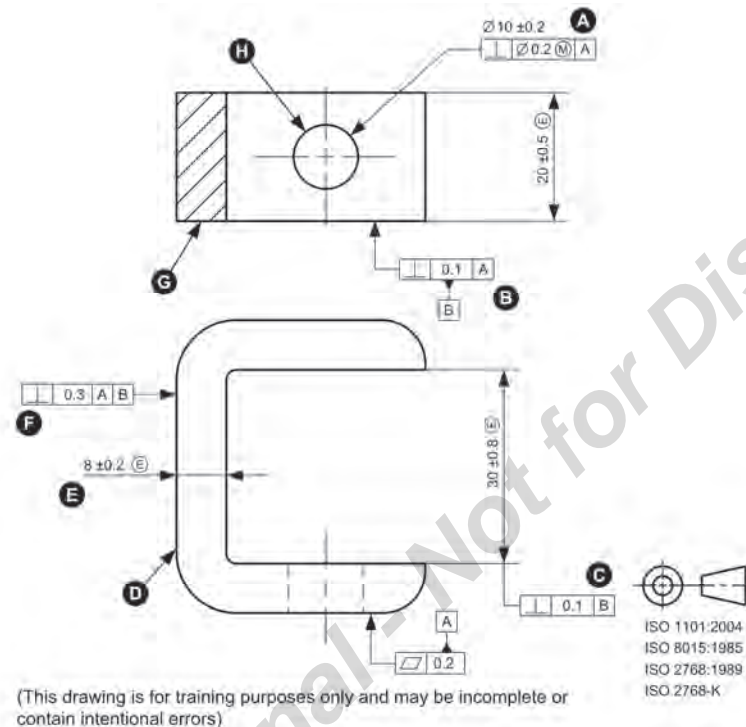
4. What is the tolerance zone for a cylindricity tolerance?
- a. The space inside a cylinder
 - b. The space between two coaxial cylinders
 - c. The space between two coaxial circles
 - d. The space between two circles
5. Which modifier can be used with a cylindricity tolerance?
- a. NC
 - b. CZ
 - c. \textcircled{M}
 - d. \textcircled{L}
6. Where the principle of independency is used, the cylindricity tolerance...
- a. Value must be less than the limits of size
 - b. Value must be greater than the limits of size
 - c. Applies to a derived feature
 - d. And the size tolerance are used to define the extreme boundary



True / False - Indicate if each statement is true or false.

7. T F The envelope requirement is an indirect cylindricity tolerance.
8. T F A cylindricity tolerance may be used to control form deviations of bearings and bushings.
9. T F A cylindricity tolerance may be specified with the \varnothing modifier.
10. T F A cylindricity tolerance may be specified with the $\text{\textcircled{F}}$ modifier.
11. T F A cylindricity tolerance limits roundness and straightness deviations.
12. T F A cylindricity tolerance may use the collective requirement concept.





Use the figure above to answer questions 1 - 5.

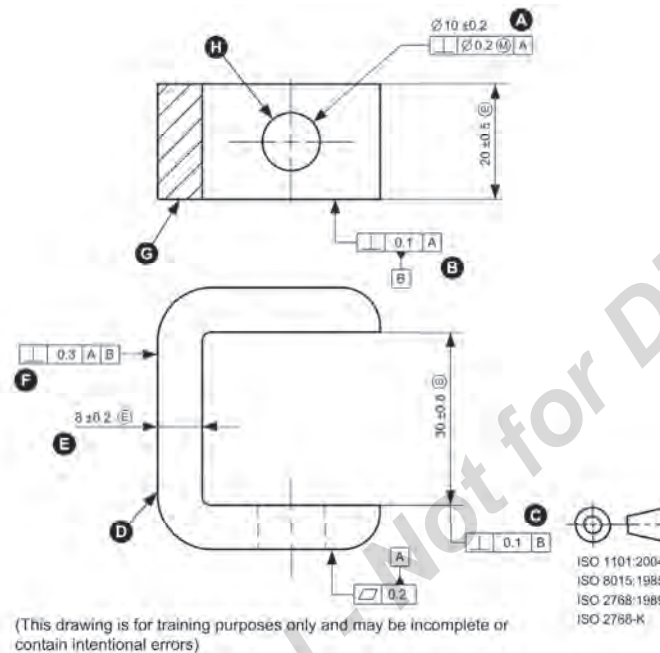
1. Describe the tolerance zone for the perpendicularity tolerance labeled **A**.

A $\varnothing 9.6$ MMVC perpendicular to datum plane A

2. Describe the tolerance zone for the perpendicularity tolerance labeled **F**.

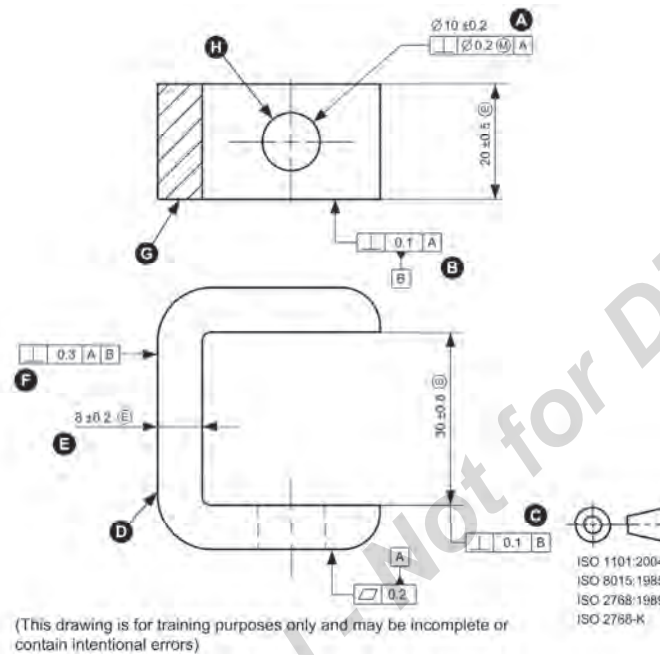
The space between two parallel planes 0.3 apart perpendicular to datum A and B





3. Which deviations are limited by the perpendicularity tolerance labeled **B** ?
- Form only
 - Orientation only
 - Form and orientation**
 - Form, orientation, and location
4. What is the perpendicularity tolerance labeled **C** is applied to?
- A median plane
 - An axis
 - A derived median surface**
 - An integral feature





5. Which statement applies to the geometrical tolerance labeled **A** ?
- The MMVS is 9.7.
 - The envelope requirement applies.
 - The collective requirement applies.**
 - The deviation of the derived median line is being limited by the tolerance zone.



Multiple Choice - Select the best answer.

6. Where a perpendicularity tolerance applies at RFS to a cylindrical feature of size, orientation and _____ deviations are limited.
- Circularity
 - Cylindricity
 - Straightness
 - None of the above
7. Where a perpendicularity tolerance is applied, which statement is true?
- A dimension of 90° must be specified.
 - A dimension of 90° is implied.
 - A TED of 90° automatically applies.
 - A TED of 90° must be specified.
8. Which modifier can be used with a perpendicularity tolerance applied to a surface?
- M
 - PD
 - CZ
 - None of the above



Multiple Choice - Select the best answer.

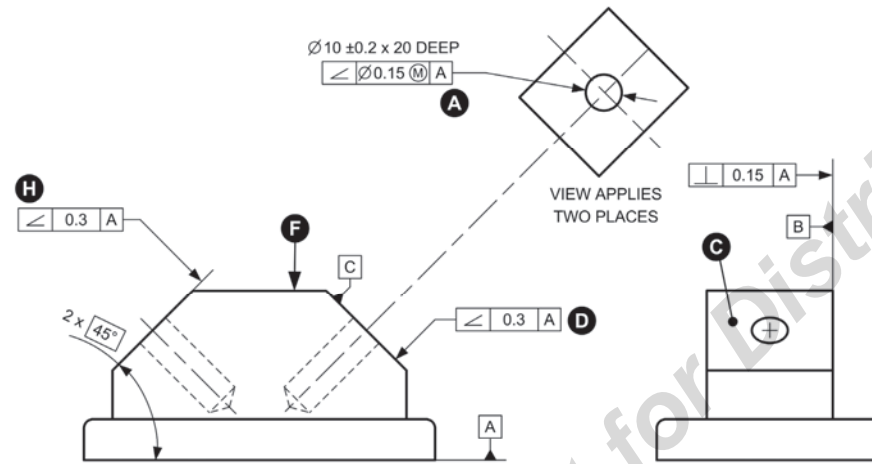
9. Which geometrical tolerance may indirectly limit perpendicularity deviations?

- a. Position
- b. Flatness
- c. Straightness
- d. None of the above

True / False - Indicate if each statement is true or false.

10. T F In a perpendicularity tolerance, a diameter modifier invokes the space within a cylinder as a tolerance zone.
11. T F A perpendicularity tolerance must include a datum indication.
12. T F A perpendicularity tolerance may control spacing between holes.





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

3. The flatness deviations of surface **C** are limited to ...
- a. 0.1
 - b. 0.2
 - c. 0.3**
 - d. Cannot determine from this drawing



Multiple Choice - Select the best answer.

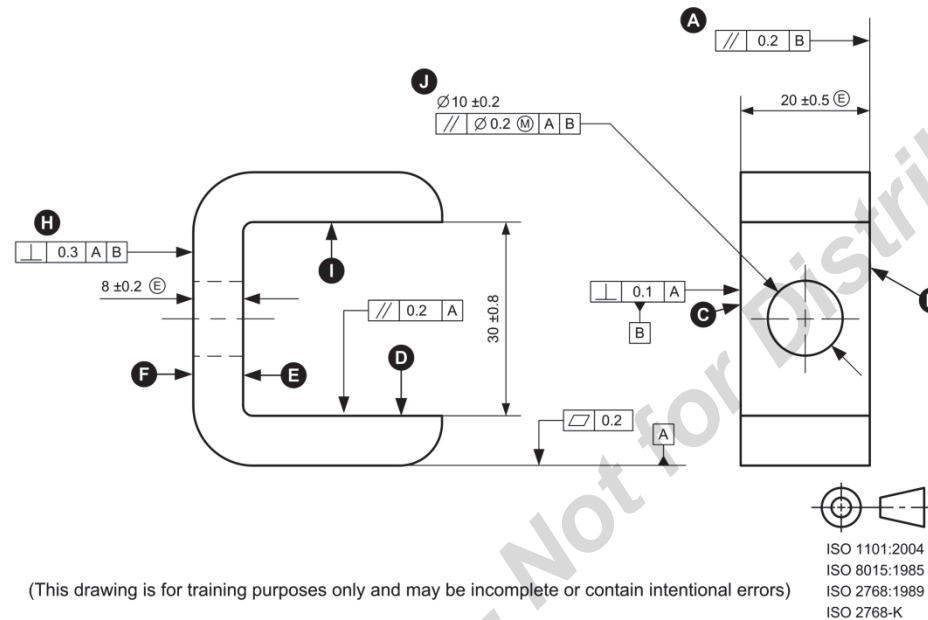
4. Which modifier can be used in an angularity tolerance applied to planar surfaces?
- a. LD
 - b. CZ
 - c. MMR
 - d. None of the above
5. When does an angularity tolerance apply to a derived median line?
- a. When an angularity tolerance is applied to a feature of size
 - b. When an angularity tolerance contains the ML modifier
 - c. When an angularity tolerance is applied to a diameter with MMR
 - d. When an angularity tolerance is applied to a diameter at RFS
6. Which geometrical tolerance could indirectly limit angularity deviations?
- a. Position
 - b. Straightness
 - c. Perpendicularity
 - d. None of the above
7. An angularity tolerance must include...
- a. An MMR modifier
 - b. A datum reference
 - c. The envelope requirement
 - d. None of the above



True / False - Indicate if each statement is true or false.

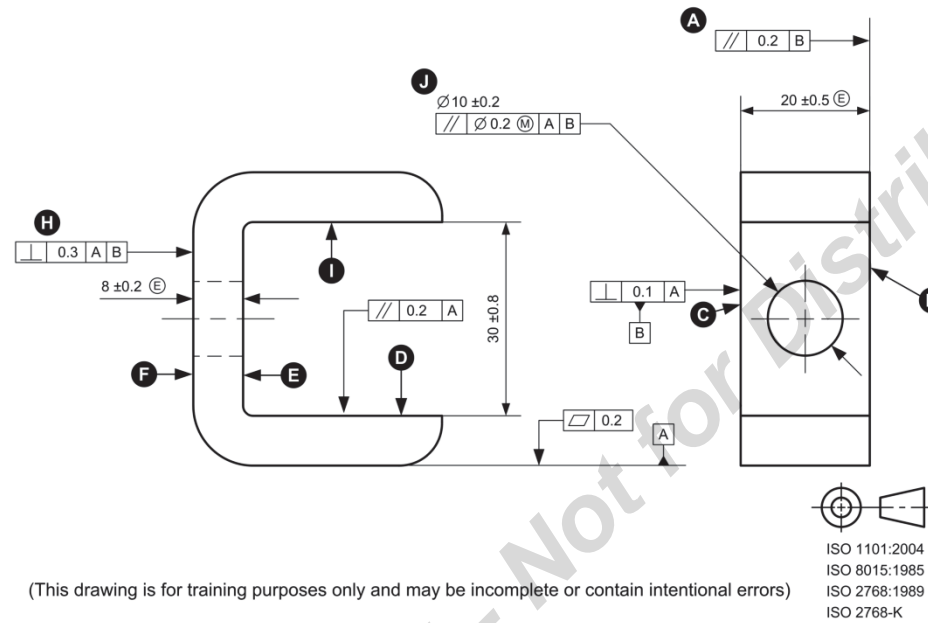
8. T F The envelope requirement may indirectly limit angularity deviations.
9. T F A TED is always required with an angularity tolerance.
10. T F Total run-out may indirectly limit angularity deviations.
11. T F Angularity may be specified with the CZ modifier.





3. Describe the tolerance zone for the parallelism tolerance applied to the hole labeled **J**.
- The space within a 9.6 diameter MMVC cylinder
 - The space within a 9.8 diameter MMVC cylinder
 - The space within a 10.4 diameter MMVC cylinder
 - None of the above
4. What is the maximum possible parallelism deviation of hole labeled **J**?
- 0.2mm
 - 0.4mm
 - 0.6mm
 - None of the above





5. Which modifier can be used with a parallelism tolerance applied to the size dimension of the hole labeled **J** ?

- a. MD
- b. PD
- c. LD
- d. None of the above



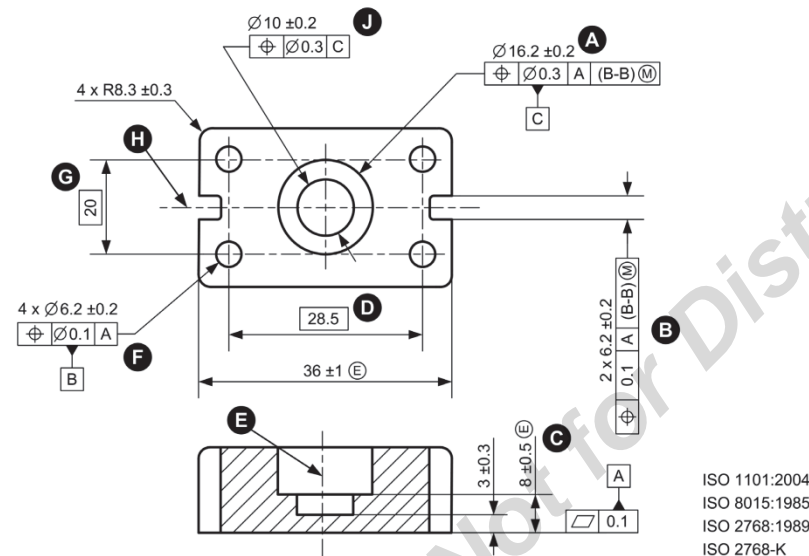
Multiple Choice - Select the best answer.

6. The datum reference of a parallelism tolerance specification must be:
- a. A planar datum
 - b. A feature of size (RFS)
 - c. A feature of size (MMR)
 - d. None of the above
7. When a parallelism tolerance at MMR is applied to the size dimension of a pattern of holes, what characteristics are being controlled?
- a. Only the orientation of the pattern
 - b. Only the spacing of the pattern
 - c. Both the spacing and orientation of the pattern
 - d. None of the above

True / False - Indicate if each statement is true or false.

8. T F A parallelism tolerance may be specified with the CZ modifier.
9. T F The specification of a parallelism tolerance, always affects the extreme boundary calculation.
10. T F A parallelism tolerance may be specified with the PD modifier.





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

Use the figure above to answer questions 1 - 5.

1. What is the maximum positional deviation permitted for the diameter labeled **J**? $\varnothing 0.3$
2. Which relationships of the hole pattern are tolerated by the position tolerance labeled **F**?
Spacing and orientation
3. Describe the tolerance zone for the position tolerance labeled **B**.
The space between two parallel planes 0.1 apart perpendicular to datum A and centered about datum axis B



Multiple Choice - Select the best answer.

6. What is one use for a tolerance of position?
- a. To locate a nominally flat surface
 - b. To control form deviations of a cylindrical feature of size
 - c. To invoke the envelope principle
 - d. None of the above
7. Why does a position tolerance with TEDs have an advantage over coordinate tolerancing?
- a. TEDs provide larger tolerances
 - b. TEDs prevent unwanted tolerance accumulation
 - c. TEDs protect part function
 - d. None of the above
8. Does a position tolerance always need to be specified with a datum reference?
- a. Yes, when position must define a location to the datum reference frame
 - b. Yes, when position must express an orientation to the datum reference frame
 - c. No, not when a position tolerance is only defining the spacing between feature of size
 - d. No, not when position is applied to a hole pattern



Multiple Choice - Select the best answer.

9. Where two or more groups of holes toleranced with position tolerances without datum references are shown on the same axis, they shall be considered...
- a. As a single pattern
 - b. As independent requirements
 - c. As only controlling the spacing between each hole pattern separately
 - d. None of the above
10. For which function would the designer use the MMR modifier in a position tolerance?
- a. Assembly
 - b. Centering
 - c. Wall thickness
 - d. None of the above

True / False - Indicate if each statement is true or false.

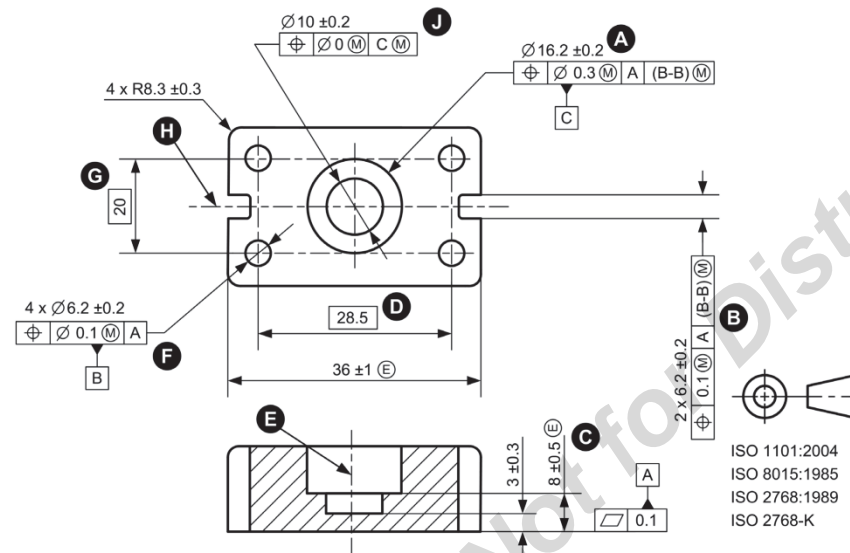
11. T F A position tolerance may be used without a datum reference.
12. T F A position tolerance may be used to define the location, orientation, and form of a planar surface.



True / False - Indicate if each statement is true or false.

13. T F The default condition for a position tolerance is MMR.
14. T F Where a position tolerance is applied to a nominally flat surface, the tolerance zone is the space between two parallel planes.
15. T F A position tolerance may be used to define a symmetrical relationship.



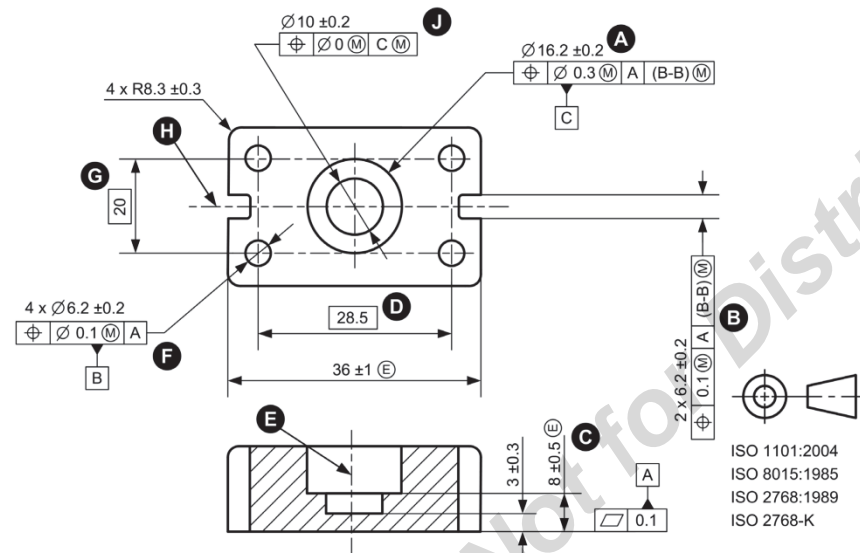


(This drawing is for training purposes only and may be incomplete or contain intentional errors)

Use the figure above to answer questions 1 - 4.

- Which position tolerances use collective requirements? As labeled: A, B, F, and J
- Identify the benefit of the zero tolerance at MMR used in the position tolerance labeled **J**.
 - It allows the workpiece to be inspected with a variable gage
 - It prevents the rejection of usable parts**
 - It allows the size tolerance to increase under certain conditions
 - None of the above





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

3. What is the MMVC boundary for the position tolerance labeled **A** ?
- 15.7 diameter
 - 15.8 diameter
 - 16.0 diameter
 - 16.7 diameter
4. What is the MMVC boundary for the position tolerance labeled **F** ?
- 5.9 diameter
 - 6.2 diameter
 - 6.3 diameter
 - None of the above



Multiple Choice - Select the best answer.

5. Where a position tolerance is applied with MMR:
- The datum reference must also specify MMR
 - The toleranced feature of size may violate its size tolerance
 - The tolerance zone is an MMVC
 - None of the above
6. What is a projected tolerance zone?
- A tolerance zone that is projected for the length of the feature of size throughout the workpiece
 - A tolerance zone that is projected through the workpiece
 - A tolerance zone that is projected above the surface of the workpiece
 - None of the above
7. Where a position tolerance indicated with the \textcircled{M} and \textcircled{R} modifiers is applied to a hole, the size tolerance of the hole must be:
- Between MMVS and LMS
 - Between MMS and LMS
 - Less than the position tolerance
 - None of the above



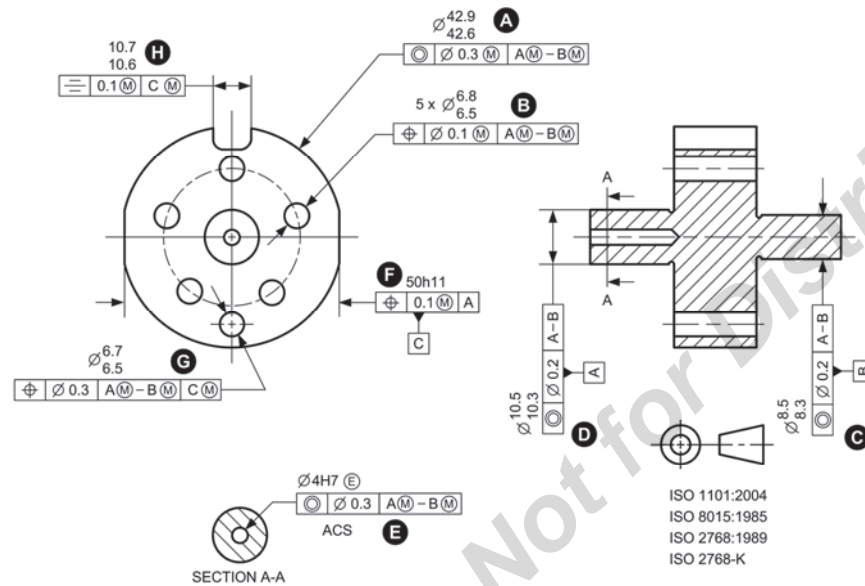
Multiple Choice - Select the best answer.

8. Which geometrical tolerance can be an indirect position tolerance?
- a. Profile of any surface
 - b. Coaxiality
 - c. Flatness
 - d. None of the above

True / False - Indicate if each statement is true or false.

9. T F When a position tolerance is used to limit the location deviation of a feature of size, it must contain the MMR or LMR modifier.
10. T F When a position tolerance with no datum references is applied to coaxial diameters, it controls the spacing and location of the diameters.
11. T F When a zero tolerance at MMR is indicated in a position tolerance, the MMVC boundary is equal to the MMC envelope requirement boundary.
12. T F A position tolerance with LMR is often used to limit a minimum distance.



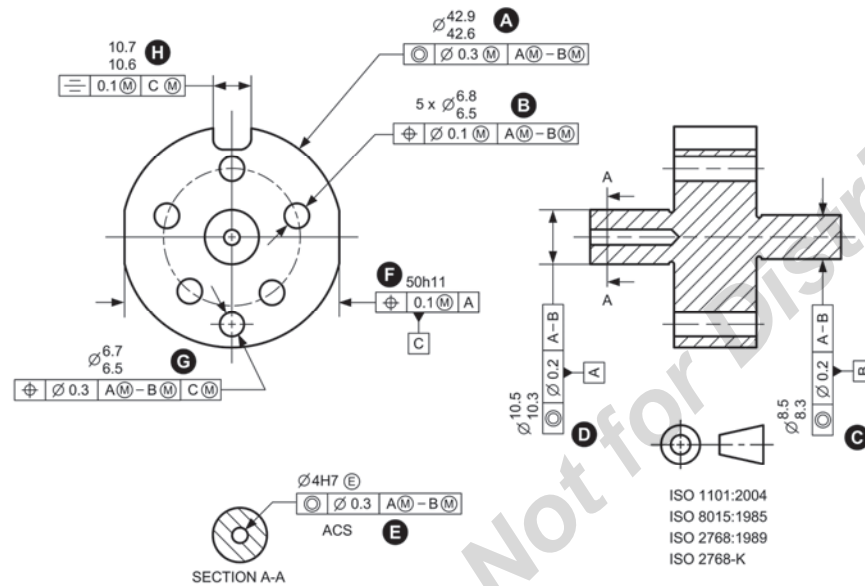


(This drawing is for training purposes only and may be incomplete or contain intentional errors)

Use the figure above to answer questions 1 through 5.

1. Which tolerance frame(s) specify a coaxiality tolerance? **As labeled: A, C, and D**
2. Which tolerance frame(s) specify a concentricity tolerance? **As labeled: E**
3. What is the shape of the tolerance zone for the geometrical tolerance labeled **E**?
The space within a 0.3 diameter circle





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

The MMR modifier may be used in a coaxiality tolerance

4. Is the MMR modifier allowed as shown in the geometrical tolerance labeled **A**? Yes
5. Which geometrical tolerances use collective requirements? As labeled: A, B, F and H



Multiple Choice - Select the best answer.

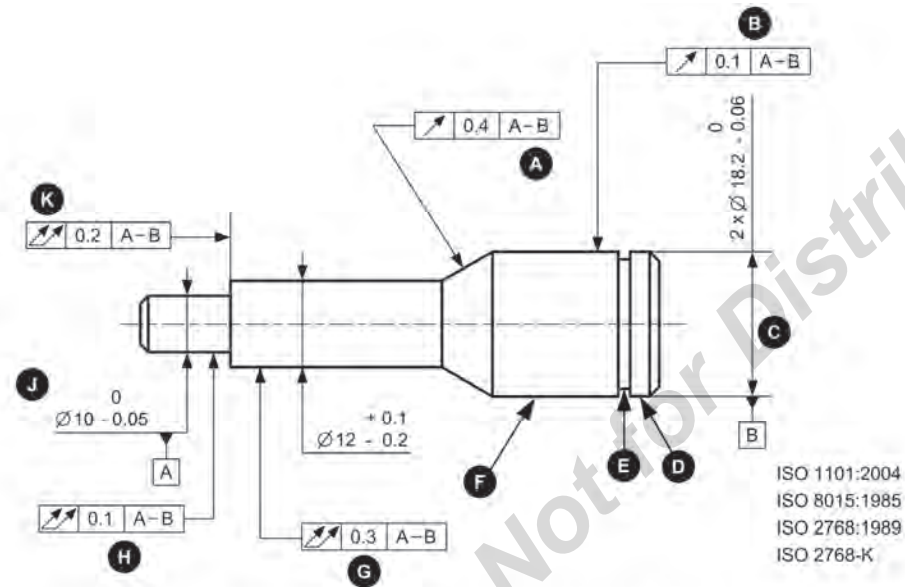
6. What can a coaxiality tolerance be applied to?
- a. A nominally planar surface that is aligned with the datum axis
 - b. A median line of a feature of size
 - c. Any integral feature
 - d. None of the above
7. What can a concentricity tolerance be applied to?
- a. A cylindrical surface
 - b. A derived median line
 - c. A center point
 - d. None of the above
8. What is the tolerance zone for a symmetry tolerance?
- a. The space between two parallel planes
 - b. The space between two parallel lines
 - c. The space between two parallel lines or planes
 - d. None of the above
9. Where a coaxiality tolerance at RFS is applied to a cylindrical feature of size:
- a. Deviations of the extracted median line are limited by the tolerance zone
 - b. Deviations of the extracted median surface are limited by the tolerance zone
 - c. Deviations of the surface are limited by the tolerance zone
 - d. None of the above



True / False - Indicate if each statement is true or false.

10. T F A symmetry tolerance is a geometric tolerance that defines a requirement for the permissible location deviation of a median surface.
11. T F A coaxiality tolerance is a geometric tolerance that defines a requirement for the permissible location deviation of a median line.
12. T F The M modifier may be applied in a symmetry tolerance frame.
13. T F A total run-out tolerance may provide an indirect symmetry tolerance.
14. T F A coaxiality tolerance must be specified with the \emptyset symbol.



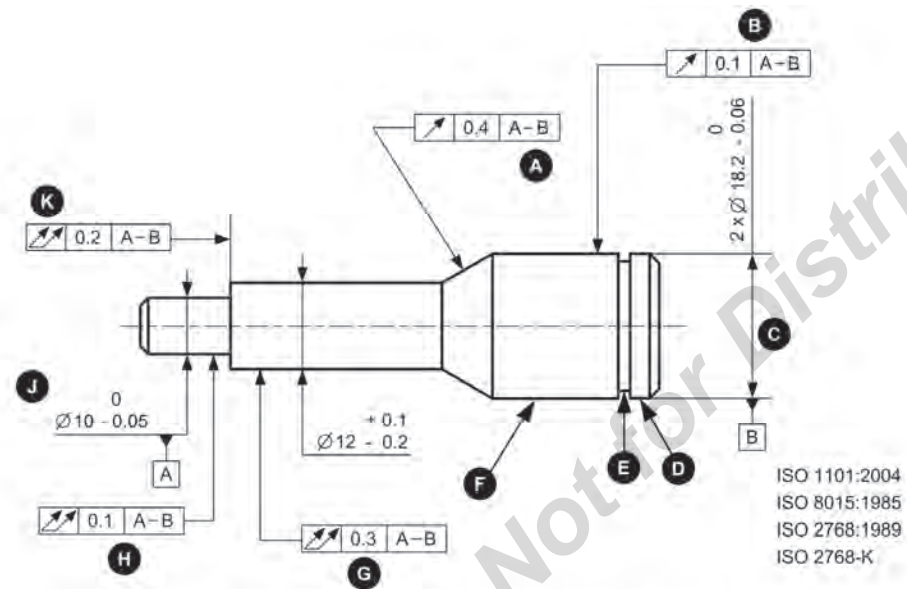


(This drawing is for training purposes only and may be incomplete or contain intentional errors)

Use the figure above to answer questions 1-5.

- Which geometrical tolerances use collective requirements? **None**
- Which type of datum axis is used with the run-out tolerance labeled **H**?
Two nominally coaxial diameters a sufficient distance apart
- Describe the tolerance zone for the run-out tolerance labeled **B**.
The space between two concentric circles 0.1 apart, centered about datum axis A-B





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

4. Describe the tolerance zone for the run-out tolerance labeled **G**.

The space between two coaxial cylinders 0.3 apart, centered about datum axis A-B

5. Describe the tolerance zone for the run-out tolerance labeled **K**.

The space between two parallel planes 0.2 apart, 90° to datum axis A-B



Multiple Choice - Select the best answer.

6. What types of deviations are limited by a circular run-out tolerance?
- a. Concentricity
 - b. Size
 - c. Form
 - d. None of the above
7. Common uses for run-out tolerances include...
- a. Assembly
 - b. Bearing journals
 - c. Weight reduction
 - d. None of the above
8. How does the envelope requirement affect the specification of circular run-out?
- a. The run-out tolerance value must be less than the limits of size
 - b. The run-out tolerance must be greater than the limits of size
 - c. The envelope requirement may define the limits of form deviations
 - d. None of the above



Multiple Choice - Select the best answer.

9. Which deviation can be limited by a total run-out tolerance zone?

- a. Cylindricity
- b. Size
- c. Concentricity
- d. None of the above

10. Which modifier may be specified with a run-out tolerance?

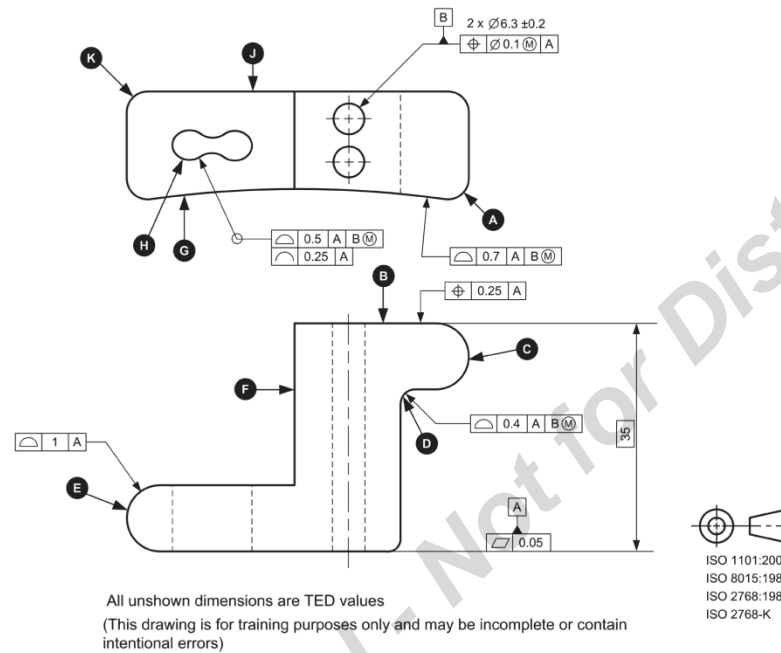
- a. \varnothing
- b. \textcircled{M}
- c. \textcircled{R}
- d. None of the above



True / False - Indicate if each statement is true or false.

11. T F Run-out tolerances may be specified with a datum displacement.
12. T F A total run-out tolerance zone applies to the entire surface simultaneously.
13. T F Total run-out applied to a planar surface limits location and perpendicularity deviations.
14. T F A circular run-out tolerance zone applies at each cross section plane perpendicular to the datum axis.
15. T F Circular run-out and total runout limit identical types of deviations.





Use the figure above to answer questions 1 - 4.

- What is the extent of the $\left[\begin{array}{|c|c|c|c|} \hline \text{⌀} & 0.4 & A & B \text{ (M)} \\ \hline \end{array} \right]$ tolerance zone? List the labels of the surface(s) that the profile tolerance applies to. **Surface labeled "D"**
- What is the extent of the $\left[\begin{array}{|c|c|c|c|} \hline \text{⌀} & 0.7 & A & B \text{ (M)} \\ \hline \end{array} \right]$ tolerance zone? List the labels of the surface(s) that the profile tolerance applies to. **Surface labeled "G"**
- Can the position tolerance on surface labeled **B** be replaced with a profile tolerance? **No**
Why? **Profile tolerances cannot be used to tolerance planar surfaces**



Multiple Choice - Select the best answer.

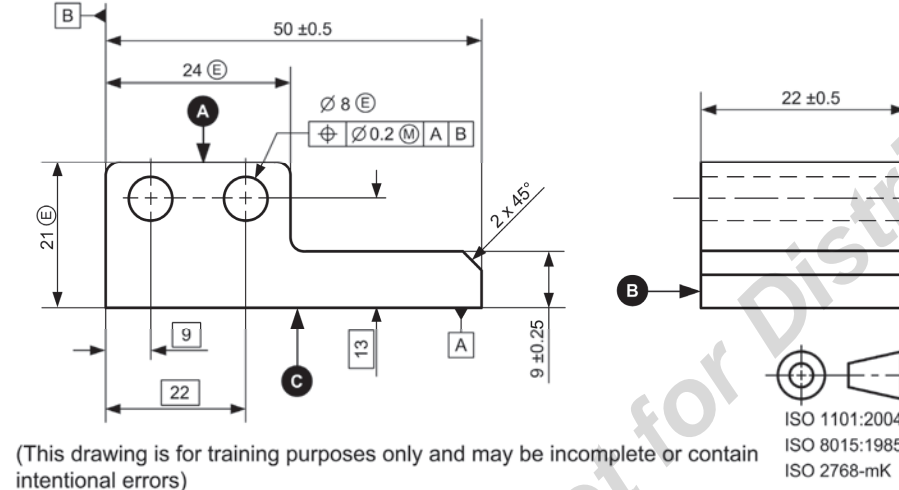
6. Profile any surface is typically used to define which type of design applications?
- Shafts
 - Precision fits
 - Threaded holes
 - Plastic components (complex shapes)
7. Which geometrical tolerance is an indirect profile any surface tolerance?
- Position
 - Circular run-out
 - Cylindricity
 - None of the above
8. Which modifier can be used in a profile any surface tolerance?
- PD
 - LE
 - CZ
 - None of the above



True / False - Indicate if each statement is true or false.

9. T F A profile any line tolerance is often specified as a refinement of profile any surface.
10. T F A profile any line tolerance may be specified without a datum reference.
11. T F A profile any surface tolerance requires a datum reference to limit size deviations.
12. T F A profile any surface tolerance zone is not required to be equally distributed about the true profile.





Use the figure above to answer questions 1 - 6.

1. What is the linear tolerance for the 24 dimension? ±0.2
2. What is the angular tolerance for the basic 2 x 45° dimension? ±1°
3. If the tolerance (±0.5) had been omitted, what would the tolerance on 50mm be? ±0.3
 Why do you think the designer specified a tolerance? The designer probably felt that ±0.5 better represents the functional limits of the feature, and larger tolerances are more economical



Multiple Choice - Select the best answer.

7. Which class of general tolerance is required for all drawings referencing ISO 8015?
- a. General linear tolerances
 - b. General angular tolerances
 - c. General geometrical tolerances
 - d. General form tolerances
8. What does the “ISO 2768-L” specification indicate?
- a. A class of fine general linear tolerances
 - b. A class of fine general geometrical tolerances
 - c. A class of coarse general linear tolerances
 - d. A class of coarse general geometrical tolerances
9. How can a general cylindricity tolerance be established for a drawing?
- a. Specify ISO 2768-cK
 - b. Specify ISO 2786-mK
 - c. Specify ISO 2768-L
 - d. None of the above



Multiple Choice - Select the best answer.

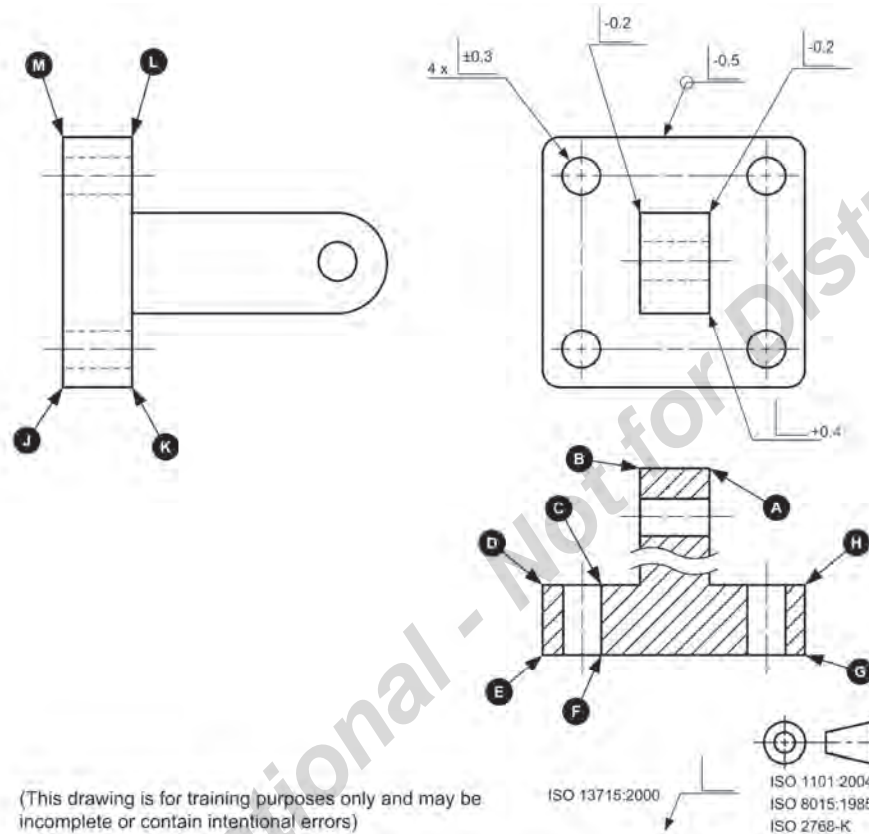
10. Which statement describes an advantage of using ISO 2768 general tolerances?
- a. The drawing focuses on evaluating the best general tolerances
 - b. Reduced inspection levels save money
 - c. Manufacturing quality is determined by customary process capability
 - d. None of the above
11. Which statement describes a disadvantage of using ISO 2786 general tolerances?
- a. Quality is determined by “customary process capability”
 - b. Drawing users should be familiar with ISO 2768
 - c. The accuracy of the workshop should be known
 - d. None of the above



True / False - Indicate if each statement is true or false.

12. T F ISO 2768 can be used to invoke the envelope requirement as a general requirement for all features of size on a drawing.
13. T F A general parallelism tolerance may depend upon the general flatness tolerance.
14. T F Under ISO 2786-H, all symmetrical relationships have the same tolerance.
15. T F ISO 2768-K permits four different coaxiality tolerance values.
16. T F The size of a diameter may be tolerated by a general linear tolerance.





Use the figure above to answer questions 1 through 4.

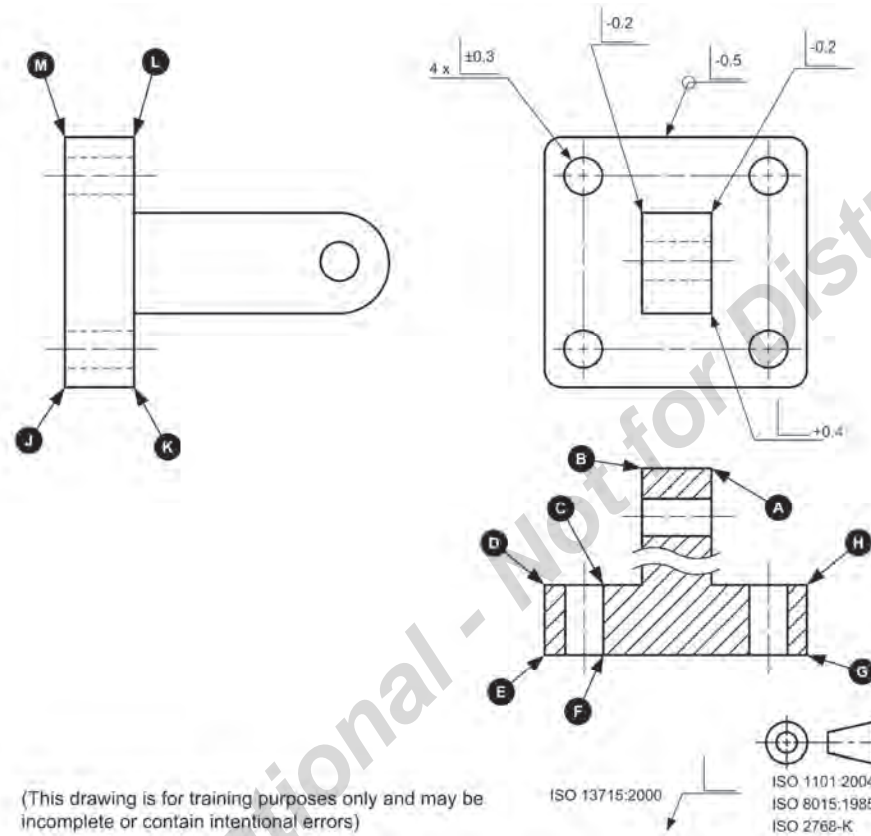
1. Which edges are defined by the -0.5 edge specification?

As labeled: D, E, G, H, J, K, L, and M

2. Which edges are defined by the ± 0.3 edge specification?

As labeled: C and F





3. What type(s) of deviations is permitted for edge **C**?

Burrs and undercuts up to 0.3 in any direction

4. Which edge specifications are not direction sensitive?

-0.2, -0.2, -0.5, ±0.3



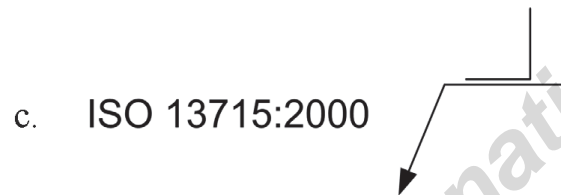
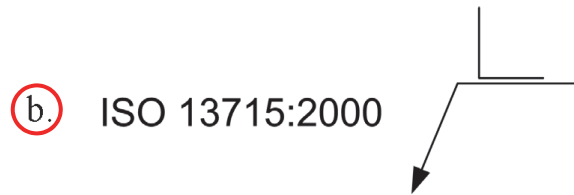
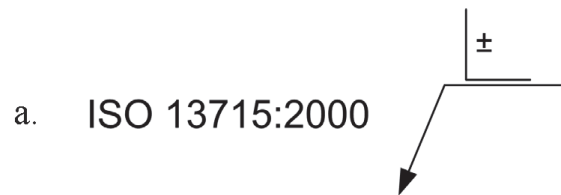
Multiple Choice - Select the best answer.

5. What does the negative sign on the tolerance (-0.3) specify for an external edge?
- a. Passing permitted
 - b. Undercut permitted
 - c. Burr permitted
 - d. None of the above
6. What does the positive sign on the tolerance (+0.3) specify for an internal edge?
- a. Passing permitted
 - b. Undercut permitted
 - c. Burr permitted
 - d. None of the above



Multiple Choice - Select the best answer.

7. Which method simply invokes the corners standard on an engineering drawing?



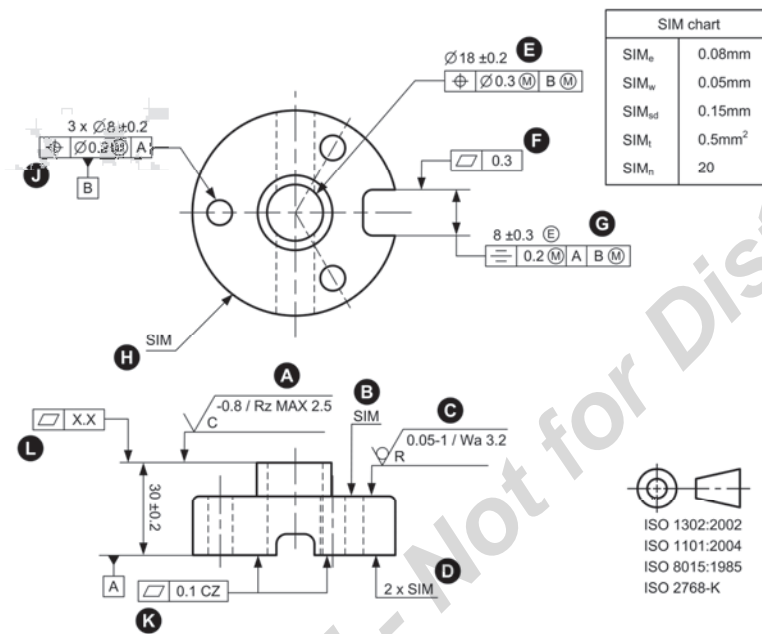
d. None of the above



True / False - Indicate if each statement is true or false.

8. T F Internal features may have a “sharp” edge.
9. T F The standard for edge specifications is often invoked through ISO 1101.
10. T F A sharp edge is an edge with less than $\pm 0.08\text{mm}$ of deviation.
11. T F The edge specification may be used to provide a general edge tolerance.
12. T F Negative tolerance values (-0.2) can only be specified to internal edges.





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

Use the figure above to answer questions 1 - 5.

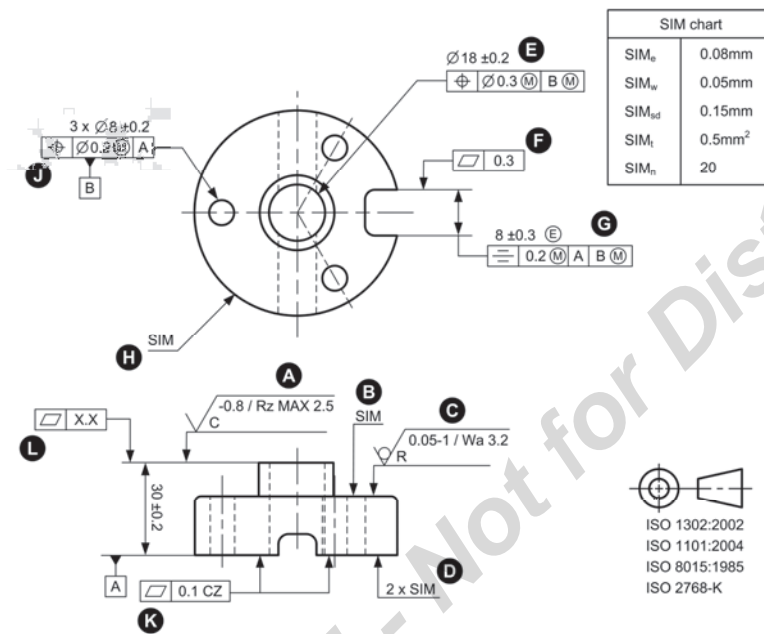
1. Interpret / describe the sampling length, roughness parameter, and direction of lay for the surface texture specification labeled **A**.

The sampling length is 0.8mm; using the Rz roughness parameter, all elements of the surface must be within 2.5 micrometers max, and the direction of the lay is circular

2. What is the evaluation length of the specification labeled **A**?
(Hint: the evaluation length is a multiple of the sample length.)

4mm (0.8 x 5)





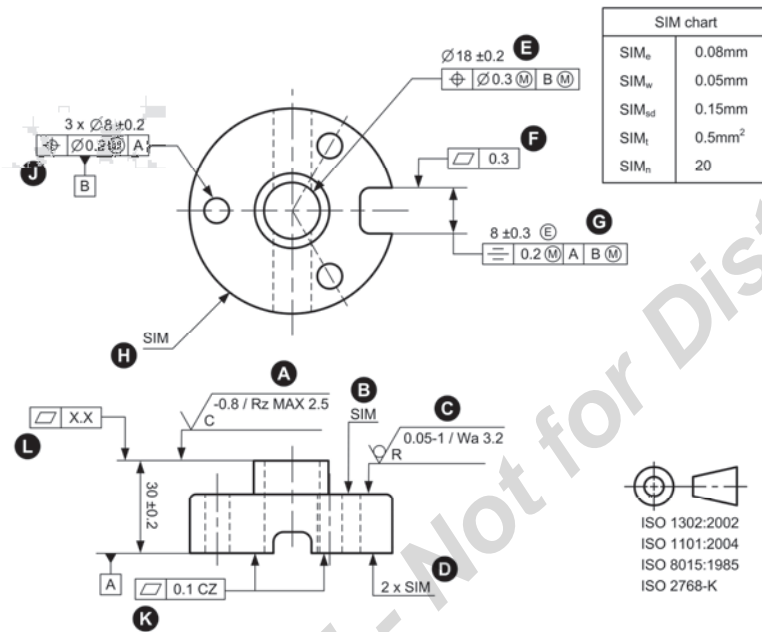
(This drawing is for training purposes only and may be incomplete or contain intentional errors)

3. Interpret / describe the transmission band (filters), roughness parameter, 16% rule, and direction of lay for the surface texture specification labeled **C**.

Using a transmission band of 0.05 for any length of 1mm, using the Wa waviness parameter, elements of the surface must be within 3.2 micrometers. The 16% rule applies. The direction of lay is radial.

4. What type of surface imperfection(s) are limited by the SIM parameter of this drawing?
- Crack
 - Burr
 - Flash
 - None of the above





(This drawing is for training purposes only and may be incomplete or contain intentional errors)

5. The SIM specification labeled **B** and surface texture specification labeled **C** apply to the same feature. Should one specification be removed? **No** Why or why not? _____

The surface texture and the surface imperfection specifications control different parameters. Surface imperfections are filtered out of a surface texture verification.



Multiple Choice - Select the best answer.

6. ISO 1302 establishes what condition as a default for surface texture?
- a. Machining is required
 - b. Sampling lengths of 0.5mm
 - c. The 16% rule
 - d. None of the above
7. _____ is the distance used for identifying the irregularities that characterize the profile under evaluation.
- a. Sampling length
 - b. Evaluation length
 - c. Transmission band
 - d. None of the above
8. The _____ is either the sum of the sampling lengths or the entire length of the surface.
- a. Transmission band
 - b. Real surface
 - c. Evaluation length
 - d. None of the above



Multiple Choice - Select the best answer.

9. Why is surface texture important?
- a. Surface texture affects product performance and durability
 - b. Secondary processes can improve appearance and texture
 - c. Cast and extruded forms have imperfect textures
 - d. None of the above
10. The measure of surface texture is generally determined in terms of its roughness profile, waviness profile and _____.
- a. Surface imperfections
 - b. Form
 - c. Sampling length
 - d. None of the above



True / False - Indicate if each statement is true or false.

11. T F ISO 1302 defines the SIM parameters and their drawing specification.
12. T F Dents and pores are examples of combined recession and raising imperfections.
13. T F Skidding and pitting are classified as appearance imperfections.
14. T F SIM_n is the symbol defining the number of imperfections per unit area.
15. T F Roughness height is the height of irregularities compared to a reference line.

