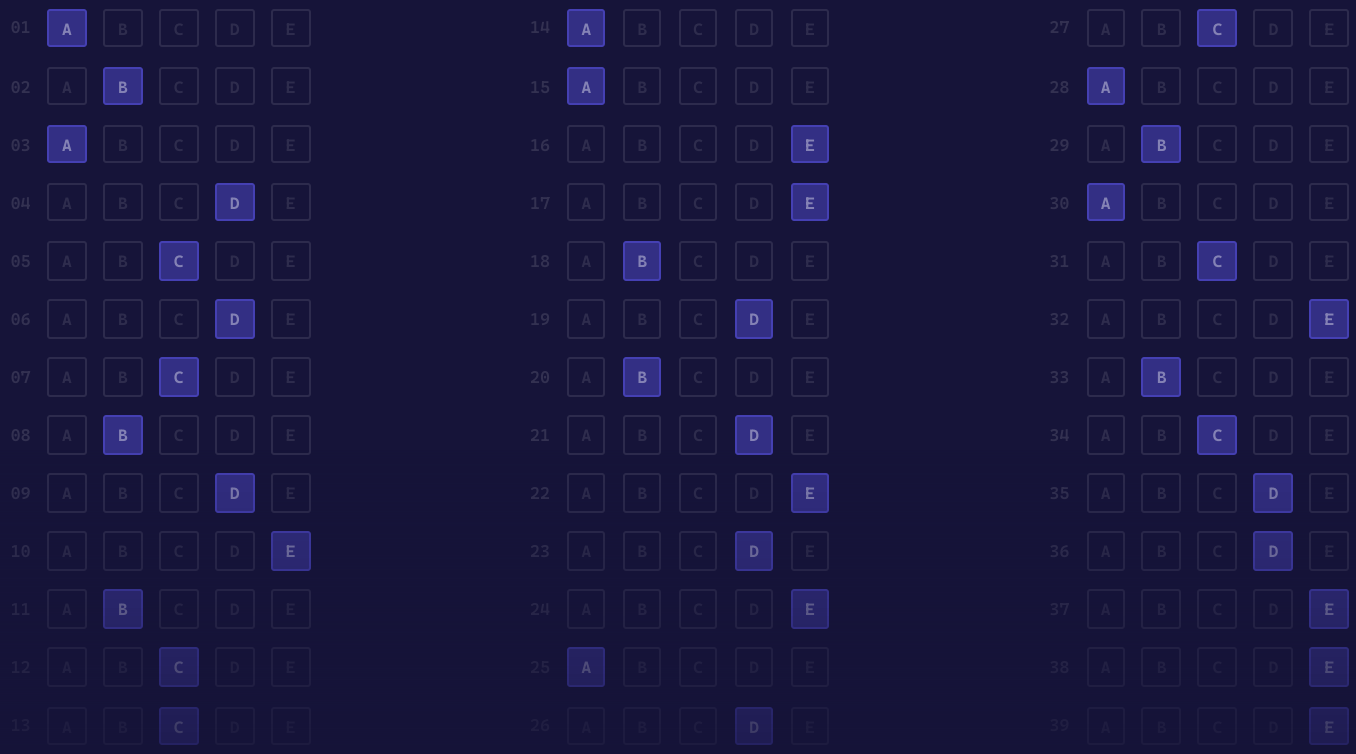
MOMETRIX
SMARTER PREP. BETTER RESULTS.



PRACTICE QUESTIONS

American Society of Mechanical Engineers (ASME)

Y18T

GDTP 2018 Certification - Technologist Level

EDITION	QUESTIONS	FORMAT
Demo	9	Limited Edition PDF

WEB www.mometrix.net

SUPPORT support@mometrix.net

Before you begin

What this is

9 practice questions for American Society of Mechanical Engineers (ASME) `Y18T`, each with its answer key and an explanation. Answer a question before you read its key — the explanation is worth more once you have committed.

If something looks wrong

Send us three things and we will check it:

- the exam code, `Y18T`
- the question number
- the email address on your order

Write to **support@mometrix.net**. We reply within one business day.

QUESTION 1 SINGLE CHOICE

On an engineering drawing, a cylindrical feature is dimensioned with a size of Ø25.4 ±0.05. According to ASME Y14.5-2018, what is the interpretation of this dimension regarding its limits of size?

A The feature must measure exactly 25.4 mm with a tolerance of ±0.05 applied to the MMC boundary

B The feature must lie between two specific size limits (25.35 and 25.45) at the extremes of individual features of size

C The tolerance applies only at the calculated MMC virtual condition of the feature

D The tolerance is a unilateral application that applies only on one side of the nominal size

ANSWER B

WHY ASME Y14.5-2018 defines limits of size as the maximum and minimum size limits assigned to a feature of size. The dimension 25.4 ±0.05 establishes two specific limits (25.35 and 25.45) between which the feature must lie. This is a bilateral tolerance applied to the limits of size, not the virtual condition (which requires MMC/LMC modifiers), and not a unilateral tolerance.

QUESTION 2 SINGLE CHOICE

On a printed drawing, the datum feature symbol is shown attached to the top surface of a rectangular block. Per ASME Y14.5-2018, the datum feature symbol identifies which of the following?

A The theoretical center point of the feature used to establish the datum axis

B The actual surface or feature used to establish a datum for measurement reference

C The storage location of the inspection records for the feature

D The process control attribute applied during fabrication of the feature

ANSWER B

WHY The datum feature symbol placed on a drawing identifies the datum feature—the actual surface (or feature of size) that is used to establish a datum for measurement reference. It does not indicate a center point or virtual condition by itself, nor does it relate to records or process control.

QUESTION 3 SINGLE CHOICE

A part drawing specifies a position tolerance of Ø0.3 with a MMB (Maximum Material Boundary) modifier placed on the feature control frame relative to datums A and B. Which of the following best describes the meaning of the MMB modifier?

A The tolerance applies at the function gage and cannot be inspected with standard measuring equipment

B The placement of the symbol does not affect the interpretation because the modifier is redundant

C The datum reference frame shifts when the feature departs from its MMB condition

D The geometric tolerance is independent of feature size, applied only at LMC

ANSWER C

WHY The MMB (Material Moved Boundary) modifier is applied to a datum reference and indicates that the datum feature may shift (i.e., the datum reference frame may be translated) when the datum feature departs from its Maximum Material Boundary. This allows the part to be inspected in its floating state. It does not relate to function gaging, is not redundant, and does not apply at LMC.

QUESTION 4 SINGLE CHOICE

According to ASME Y14.5-2018, which dimensioning practice should be used when the dimensional limits of a feature are too close together to permit the use of a clear plus and minus tolerance format?

A Apply the tolerances in millimeter units and ignore the original inch notation

B Switch to specifying the limits directly using a limit dimensioning format

C Use basic dimensions only and remove the tolerance callouts from the drawing

D Keep the plus/minus format but increase the nominal size to allow more room on the page

ANSWER B

WHY ASME Y14.5-2018 states that limit dimensioning should be used when the limits of size are too close together to permit the use of plus or minus tolerancing with clarity. The other options either violate the standard (changing units, dropping tolerances) or do not address the clarity problem.

QUESTION 5 SINGLE CHOICE

A drawing calls out three datums (A, B, and C) labeled primary, secondary, and tertiary in a feature control frame. Per ASME Y14.5-2018, this sequencing establishes what?

A The order of machining operations used to create each datum feature on the part

B The required sequence for the inspection report and sign-off procedure

C The order in which the datum reference frame is locked, restricting degrees of freedom in sequence

D The ranking of inspectors by experience level authorized to verify each datum

ANSWER C

WHY The order of letters in the feature control frame (A, B, C) indicates the precedence of the datums in establishing the datum reference frame. Each successive datum restricts additional degrees of freedom when the part is constrained in measurement equipment. The sequence is a measurement-related requirement, not a manufacturing, inspection-reporting, or personnel hierarchy.

QUESTION 6 SINGLE CHOICE

A hole in a part is dimensioned as Ø10.0 with a position tolerance of Ø0.5 referenced to datum A-B-C. The tolerance zone implied by the position callout, per ASME Y14.5-2018, is defined as:

A Two parallel planes 0.5 mm apart and perpendicular to datum plane A

B A cylindrical zone Ø0.5 within which the axis of the hole must lie, oriented and located per the datums

C A square zone of 0.5 mm on each side aligned with the primary datum direction

D A spherical zone Ø0.5 centered on the theoretical exact position

ANSWER B

WHY ASME Y14.5-2018 specifies that the tolerance zone for a position tolerance on a hole is cylindrical (ØØ) when called out with a diameter symbol on a feature of size. The axis of the hole must lie entirely within that cylindrical tolerance zone, which is located and oriented relative to the stated datum reference frame (A-B-C). The other options either misinterpret the shape of the zone or the geometry being controlled.

QUESTION 7 SINGLE CHOICE

On a part drawing, circular run-out is specified on a cylindrical surface relative to datum A. Compared to total run-out, circular run-out:

A Controls the entire surface along the full length of the feature in a single revolution

B Controls only a single cross-section of the feature per measurement and is applied to individual circular elements

C Is the same tolerance under a different historical name and applies identically

D Applies only to flat surfaces and cannot be used on cylindrical features

ANSWER B

WHY Circular run-out controls a single circular cross-section (one measurement plane) at a time, measuring FIM (Full Indicator Movement) as the part is rotated. Total run-out, by contrast, controls the entire surface simultaneously along its length while the part is rotated. Circular run-out is specifically applicable to circular cross-sectional features, not just flat surfaces, and is functionally different from total run-out.

QUESTION 8 SINGLE CHOICE

A drawing shows an angular dimension of 30° with a tolerance of ±0.5°. Per ASME Y14.5-2018, the general angular tolerance applies to the feature being dimensioned. Which statement best describes how this angular tolerance interacts with other tolerances on the drawing?

A The angular tolerance cancels any geometric orientation tolerance referencing the same surface

B The angular tolerance applies only when orientation controls like perpendicularity or parallelism are absent

C The angular tolerance must always be tighter than any position tolerance on the same feature

D Angular tolerances only apply to features on the secondary datum plane

ANSWER B

WHY ASME Y14.5-2018 establishes that tolerancing principles generally apply, and where a specific geometric tolerance (such as perpendicularity, parallelism, or angularity) is specified, it takes precedence and the general plus/minus tolerance does not stack. The other statements either contradict the standard's hierarchy of controls or have no basis in ASME Y14.5.

QUESTION 9 MULTIPLE CHOICE

Which of the following statements about feature control frames in ASME Y14.5-2018 are correct? (Choose two.)

A **A feature control frame may contain multiple symbols and multiple tolerance values stacked in sequence**

B Each feature control frame applies to only one geometric characteristic on one feature

C **Datum references are placed inside the feature control frame to relate the tolerance to those datums**

D Feature control frames must always read horizontally from left to right across the drawing

ANSWER **A · C**

WHY ASME Y14.5-2018 permits a feature control frame to contain multiple symbols and tolerance values arranged sequentially (stacked or aligned), and to include datum references within the same frame that relate the geometric tolerance to the stated datums. The other statements are incorrect: a single feature control frame can include combined tolerances (e.g., position with profile and datums), and feature control frames may be placed parallel with the dimension line of the feature being controlled, not strictly horizontally across the drawing.