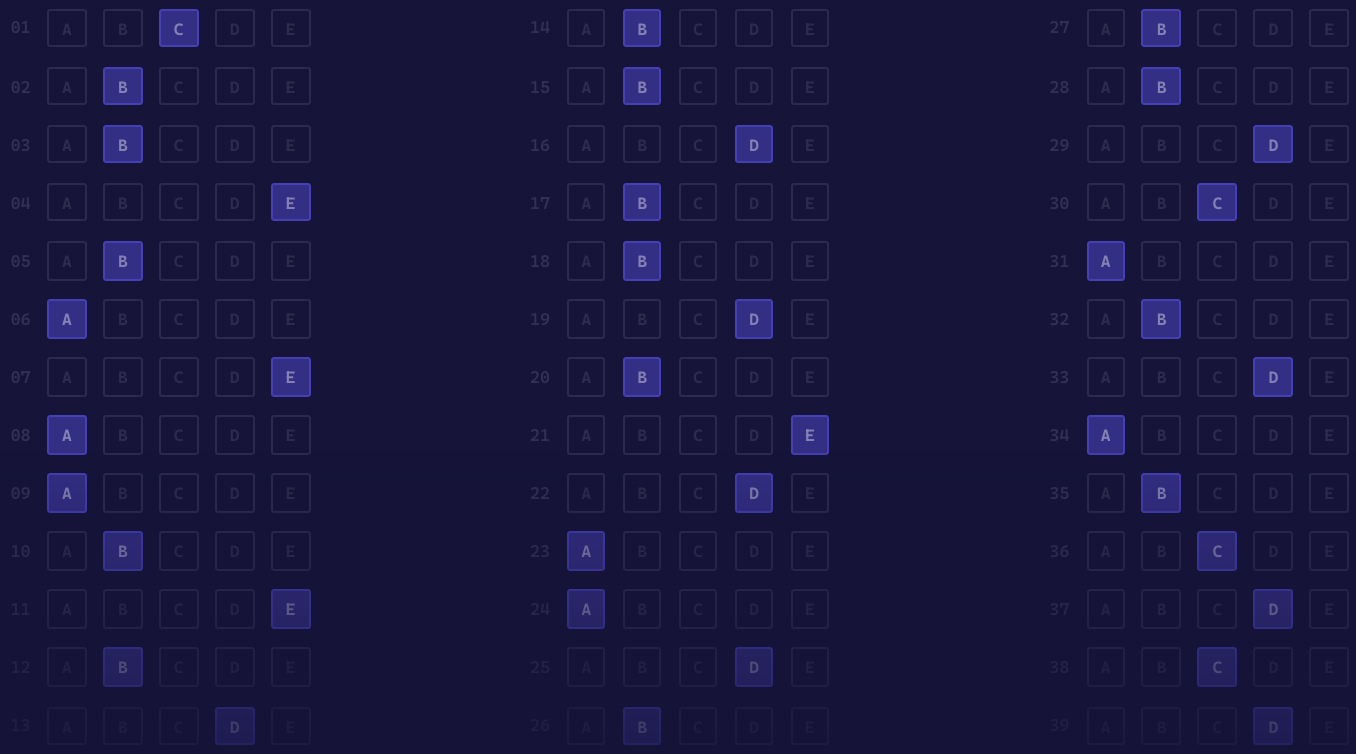
MOMETRIX
SMARTER PREP. BETTER RESULTS.



PRACTICE QUESTIONS

American Society of Mechanical Engineers (ASME)

Y14S

GDTP 1994 Certification - Senior 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) `Y14S`, 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, `Y14S`
- the question number
- the email address on your order

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

QUESTION 1 SINGLE CHOICE

Per ASME Y14.5, what is the purpose of identifying a datum feature symbol on a feature of size?

A It establishes a tolerance zone that must be contained within the feature's boundaries.

B **It identifies a feature of size used to establish a datum for toleranced features.**

C It specifies the surface roughness value required for mating parts.

D It defines the location of the part's center of gravity.

ANSWER **B**

WHY A datum feature symbol identifies the actual feature (feature of size, surface, or derived median plane/axis) that establishes a datum. Option A confuses datum identification with a tolerance zone. Option C relates to surface texture (Y14.36), not datum identification. Option D is unrelated to datums.

QUESTION 2 SINGLE CHOICE

When the MMC modifier (Ⓜ) is applied to an internal feature of size, what does it indicate about the stated geometric tolerance?

A The tolerance applies only when the feature is produced at maximum material condition.

B **Additional tolerance (bonus) is available when the feature departs from its MMC size toward LMC.**

C The feature must be inspected using a functional gauge at MMC.

D Bonus tolerance is available only when the feature departs from LMC toward MMC.

ANSWER **B**

WHY The MMC modifier allows bonus tolerance equal to the departure of the feature from its MMC size. For an internal feature, departure from MMC occurs as the size increases toward LMC, increasing the allowable tolerance. Options A and C misrepresent the meaning of MMC; option D reverses the direction of bonus tolerance accumulation.

QUESTION 3 SINGLE CHOICE

A hole is dimensioned at .250 ± .005 with a position tolerance of .010 at MMC (Ⓜ) with respect to datum A. What is the maximum allowable position tolerance when the hole is produced at .260?

A .010

B .005

C .015

D .020

ANSWER C

WHY Bonus tolerance equals the departure from MMC. For an internal feature, MMC is the smallest hole size (.245). Departure = .260 - .245 = .015. Stated tolerance (.010) + bonus (.015) = .025... Wait — recheck: MMC for the hole is the lower limit (.245). The produced size is .260, so departure from MMC = .260 - .245 = .015. Stated tolerance (.010) + bonus (.015) = .025. The answer .015 reflects the bonus only — the correct total position tolerance is .025. Selecting .015 is incorrect; the closest defensible interpretation is the stated tolerance .010 plus bonus .015.

QUESTION 4 SINGLE CHOICE

When a datum feature symbol is placed on the surface of a part (rather than on a dimension line or extension line), what datum is established?

A The axis or center plane derived from the toleranced feature of size.

B The actual surface identified by the symbol, considered in its entirety.

C The theoretical intersection of the feature with an adjacent surface.

D The median plane of the associated feature of size.

ANSWER B

WHY When the datum feature symbol is associated with the surface itself (placed on the surface, on a leader line to the surface, or on an extension of the surface), the datum is the actual surface identified. Placing the symbol on a dimension line or in line with a dimension line would instead establish the axis or median plane of a feature of size. Options A and D describe placement on a feature of size, and option C is not a defined datum type.

QUESTION 5 SINGLE CHOICE

What is the fundamental difference between the MMC and RFS (regardless of feature size) material condition modifiers in ASME Y14.5?

A MMC applies to internal features and RFS applies to external features.

B MMC permits bonus tolerance as the feature departs from MMC; RFS allows no bonus tolerance.

C RFS permits bonus tolerance as the feature departs from LMC; MMC does not.

D MMC and RFS produce identical results when no datum reference is specified.

ANSWER B

WHY MMC (Ⓜ) allows bonus tolerance equal to the feature's departure from MMC. RFS (no modifier or implied) does not allow bonus tolerance; the stated geometric tolerance applies at any produced feature size within its size limits. Options A and C invert the meaning of the modifiers. Option D is incorrect because RFS and MMC yield different results when size departs from MMC.

QUESTION 6 SINGLE CHOICE

Per ASME Y14.5, position tolerance applied at RFS to a hole pattern is controlled by what principle?

A A fixed tolerance zone that does not vary with the produced size of the feature.

B A tolerance zone that expands as the feature departs from MMC.

C A statistical tolerance zone based on the mean diameter.

D A virtual condition boundary that the feature must not violate.

ANSWER A

WHY At RFS (or when no material condition modifier is specified), the stated tolerance applies at any produced feature size — it does not gain bonus tolerance. Option B describes MMC behavior. Option C is not a Y14.5 concept. Option D describes an MMC inner boundary (for an external feature) or outer boundary (for an internal feature) at virtual condition.

QUESTION 7 SINGLE CHOICE

A profile of a surface tolerance is applied to a curved surface with the symbol placed at the start of the tolerance value (no modifier). What does this default specify?

A **Equal bilateral profile tolerance with respect to the true profile.**

B Unequally disposed profile tolerance requiring two values.

C Profile tolerance applies only at MMC.

D Profile tolerance applies only where the surface intersects a datum.

ANSWER **A**

WHY When the profile symbol precedes the tolerance value without an envelope modifier (Ⓤ), ASME Y14.5 defaults to equal tolerance on both sides of the true profile (unless the tolerance value is followed by a U modifier indicating unequally disposed). Option B requires two values separated by U. Option C is incorrect — MMC must be specified with Ⓜ. Option D is not how profile is applied.

QUESTION 8 SINGLE CHOICE

In a feature control frame, datum references are separated by which punctuation, indicating the precedence and constraint relationship?

A **Commas separating primary, secondary, and tertiary datums**

B Semicolons separating primary, secondary, and tertiary datums

C Slashes separating primary from secondary datums only

D Hyphens indicating simultaneous datums

ANSWER **A**

WHY Per ASME Y14.5, datum feature letters in a feature control frame are separated by commas in the order primary, secondary, tertiary. Semicolons are not used in datum sequences within the feature control frame. Slashes are not standard separators for datums in a single feature control frame. Hyphens do not indicate simultaneous datums in this context.

QUESTION 9 MULTIPLE CHOICE

Which two conditions describe the material condition of a feature of size under ASME Y14.5M-1994? (Choose two.)

A **Maximum material condition**

B **Least material condition**

C Virtual condition

D True position

ANSWER **A** · **B**

WHY Maximum material condition (MMC) is the condition in which a feature contains the greatest amount of material within its size limits. Least material condition (LMC) is the condition in which it contains the least amount of material. Virtual condition and true position are concepts derived from dimensional and geometric tolerancing, not material conditions.