

01	A	B	C	D	E
02	A	B	C	D	E
03	A	B	C	D	E
04	A	B	C	D	E
05	A	B	C	D	E
06	A	B	C	D	E
07	A	B	C	D	E
08	A	B	C	D	E
09	A	B	C	D	E
10	A	B	C	D	E
11	A	B	C	D	E
12	A	B	C	D	E
13	A	B	C	D	E

14	A	B	C	D	E
15	A	B	C	D	E
16	A	B	C	D	E
17	A	B	C	D	E
18	A	B	C	D	E
19	A	B	C	D	E
20	A	B	C	D	E
21	A	B	C	D	E
22	A	B	C	D	E
23	A	B	C	D	E
24	A	B	C	D	E
25	A	B	C	D	E
26	A	B	C	D	E

27	A	B	C	D	E
28	A	B	C	D	E
29	A	B	C	D	E
30	A	B	C	D	E
31	A	B	C	D	E
32	A	B	C	D	E
33	A	B	C	D	E
34	A	B	C	D	E
35	A	B	C	D	E
36	A	B	C	D	E
37	A	B	C	D	E
38	A	B	C	D	E
39	A	B	C	D	E

PRACTICE QUESTIONS

American Society of Mechanical Engineers (ASME)

18Tec

GDTP (Y14.5 - 2018 Edition) Exam - Technologist Level

EDITION

Demo

QUESTIONS

9

FORMAT

Limited Edition PDF

Before you begin

What this is

9 practice questions for American Society of Mechanical Engineers (ASME) 18Tec , 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, 18Tec
- the question number
- the email address on your order

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

QUESTION 1

MULTIPLE CHOICE

Per ASME Y14.5-2018, which of the following statements about the material condition modifiers MMC and LMC are correct? (Choose two.)

- ☒ **A** MMC represents the condition of a feature when it contains the maximum amount of material within its stated limits of size
- ☐ **B** LMC represents the condition of a feature when it contains the maximum amount of material within its stated limits of size
- ☒ **C** When applied to a tolerance of position, MMC allows additional tolerance through the concept of bonus tolerance equal to the departure from MMC
- ☐ **D** LMC and MMC may only be applied to features of size and cannot be referenced in datum feature references

ANSWER A - C

WHY A is correct: per ASME Y14.5-2018, Maximum Material Condition (MMC) is the condition in which a feature contains the maximum amount of material within its stated limits of size (e.g., a shaft at its largest diameter, a hole at its smallest diameter). B is incorrect because LMC (Least Material Condition) is the condition of minimum material, not maximum. C is correct: when MMC is applied to a geometric tolerance such as position, any departure from MMC may be used as bonus tolerance, accumulating with the stated geometric tolerance. D is incorrect because the material condition modifiers are explicitly designed to be referenced in both the geometric tolerance and the datum feature references (e.g., datum feature symbols followed by the MMC or LMC modifier).

QUESTION 2

MULTIPLE CHOICE

Which of the following statements about datum establishment and datum feature simulators per ASME Y14.5-2018 are correct? (Choose two.)

- ☒ **A** A datum is a theoretical reference plane, axis, or point derived from the datum feature simulator
- ☐ **B** The datum feature simulator is the actual physical surface of the part used to establish the datum
- ☒ **C** For a planar datum feature, the datum feature simulator is typically a precision surface plate of sufficient size
- ☐ **D** A datum feature symbol may be applied to a dimension line or extension line when the datum feature is a feature of size

ANSWER A • C

WHY A is correct: per ASME Y14.5-2018, a datum is a theoretical exact reference (plane, axis, or point) derived from the datum feature simulator, not the physical surface itself. B is incorrect because the datum feature simulator is not the actual surface of the part — it is the physical precision tool or fixture (surface plate, gauge pins, etc.) used to simulate the datum. C is correct: for planar datum features, the datum feature simulator is commonly a precision surface plate of adequate size and flatness. D is incorrect because, per Y14.5-2018, the datum feature symbol is applied to the feature outline or surface in view, aligned with the dimension line when the feature is a feature of size — placing it on an extension line alone is not the prescribed placement.

QUESTION 3

SINGLE CHOICE

In ASME Y14.5-2018, what does the term 'tolerance of position' control when applied to a feature at RFS (regardless of feature size)?

- ☐ A Only the orientation of the feature relative to the datums
- ☐ B Only the form of the feature
- ☒ C The location, orientation, and form of the feature within the specified positional tolerance zone
- ☐ D The cylindricity of the feature

ANSWER C

WHY Per ASME Y14.5-2018, the tolerance of position controls the location, orientation, and form of the tolerated feature within the specified tolerance zone. When applied at RFS, the stated tolerance applies regardless of the actual produced size, so no bonus tolerance is gained.

QUESTION 4

SINGLE CHOICE

According to ASME Y14.5-2018, when a datum feature symbol is applied to a feature of size, the datum is established by which of the following?

- ☐ A The two extreme points of the feature of size
- ☒ B The axis or center plane of the feature of size as derived from the datum feature simulator
- ☐ C The actual physical surface of the feature of size
- ☐ D Only the projected surface defined by the tangent plane

ANSWER B

WHY Per ASME Y14.5-2018, when a datum feature symbol is associated with a feature of size, the datum is established as the axis (for a cylindrical feature) or center plane (for a parallel pair of surfaces) derived from the contacting datum feature simulator. The actual physical surface itself is not the datum; it is the simulator that defines the theoretical datum.

QUESTION 5

SINGLE CHOICE

Per ASME Y14.5-2018, circularity (roundness) is a characteristic that controls which of the following?

- ☒ **A** The cross-sectional form of any circular feature at any single cross section
- ☐ **B** The combined form, orientation, and location of the feature
- ☐ **C** The taper of a conical feature along its axis
- ☐ **D** The flatness of a planar surface

ANSWER A

WHY Per ASME Y14.5-2018, circularity (called roundness in earlier editions) is a form tolerance that independently controls the circularity of individual cross sections of a cylindrical, conical, or spherical surface at any single cross section. It does not control the longitudinal form (such as taper or straightness along the axis); those require separate controls such as cylindricity or straightness.

QUESTION 6

SINGLE CHOICE

Per ASME Y14.5-2018, what is the default precedence order for datum features when no modifier is specified on the datum feature symbol?

- ☒ **A** The primary datum is referenced first, then the secondary, then the tertiary
- ☐ **B** All datums are treated equally regardless of order
- ☐ **C** Only the primary datum controls orientation
- ☐ **D** The last datum referenced controls the most degrees of freedom

ANSWER A

WHY Per ASME Y14.5-2018, the order in which datum feature symbols appear in the feature control frame (or on the drawing) establishes the precedence: primary (first) is referenced first and typically restrains the most degrees of freedom, secondary (second) further restrains the part, and tertiary (third) completes the restraint of the part in space. There is no implicit equality of datums when listed in order.

QUESTION 7

SINGLE CHOICE

According to ASME Y14.5-2018, a feature control frame for position must be associated with which of the following to fully specify the location of the feature?

- ☒ **A** The basic dimensions and the appropriate datums
- ☐ **B** Only the geometric tolerance value
- ☐ **C** The title block tolerance and a material specification
- ☐ **D** The coordinate tolerance and surface finish

ANSWER A

WHY Per ASME Y14.5-2018, the position of a feature of size is fully specified using a feature control frame containing the position symbol, the tolerance value (with any material condition modifier), and the datum feature references. The location is then established by basic dimensions shown on the drawing in rectangular coordinates or in notes. A feature control frame without basic dimensions and datums does not fully define position.

QUESTION 8

SINGLE CHOICE

Per ASME Y14.5-2018, what is the difference between circular runout and total runout when applied to a cylindrical surface?

- ☒ **A** Circular runout controls the combined variation of circularity, straightness, taper, and profile at one revolution
- ☐ **B** Circular runout controls only the surface roughness, while total runout controls form
- ☐ **C** There is no difference; both terms describe the same control
- ☐ **D** Total runout applies only to planar surfaces

ANSWER A

WHY Per ASME Y14.5-2018, circular runout is the difference between the maximum and minimum readings of an indicator measuring full-indicator-movement during one revolution of the part, applied at any single circular element of the feature. It therefore controls the combined variation of circularity and any element-to-element variation at that location. Total runout similarly measures FIM but accumulates the variation over the entire surface, additionally controlling taper, straightness along the axis, and profile. Option A captures the definition of circular runout correctly; the other statements misrepresent Y14.5-2018.

QUESTION 9

SINGLE CHOICE

According to ASME Y14.5-2018, what is the meaning of the symbol ' ⌀ ' (circularity) applied as a geometric tolerance to a cylindrical surface?

- ☐ A It controls the taper of the cylindrical surface along its axis
- ☒ B It controls the circularity of individual cross sections of the cylinder
- ☐ C It controls the cylindricity of the entire feature
- ☐ D It controls the perpendicularity of the cylinder to the datum

ANSWER B

WHY Per ASME Y14.5-2018, the circularity symbol applied to a cylindrical surface controls the form of individual cross sections — that is, each circular element must lie between two concentric circles separated radially by the specified tolerance value. It does not control longitudinal elements such as taper (option A) or the entire cylindrical form, which would require cylindricity (option C). It also does not control perpendicularity (option D), which is an orientation tolerance requiring a datum reference.