Task Group 142 Tolerances Practice Test (Sample)

Study Guide

Everything you need from our exam experts!

Table of Contents

Introduction

Preparing for a certification exam can feel overwhelming, but with the right tools, it becomes an opportunity to build confidence, sharpen your skills, and move one step closer to your goals. At Examzify, we believe that effective exam preparation isn't just about memorization, it's about understanding the material, identifying knowledge gaps, and building the test-taking strategies that lead to success.

This guide was designed to help you do exactly that.

Whether you're preparing for a licensing exam, professional certification, or entry-level qualification, this book offers structured practice to reinforce key concepts. You'll find a wide range of multiple-choice questions, each followed by clear explanations to help you understand not just the right answer, but why it's correct.

The content in this guide is based on real-world exam objectives and aligned with the types of questions and topics commonly found on official tests. It's ideal for learners who want to:

- Practice answering questions under realistic conditions,
- Improve accuracy and speed,
- Review explanations to strengthen weak areas, and
- Approach the exam with greater confidence.

We recommend using this book not as a stand-alone study tool, but alongside other resources like flashcards, textbooks, or hands-on training. For best results, we recommend working through each question, reflecting on the explanation provided, and revisiting the topics that challenge you most.

Remember: successful test preparation isn't about getting every question right the first time, it's about learning from your mistakes and improving over time. Stay focused, trust the process, and know that every page you turn brings you closer to success.

Let's begin.

How to Use This Guide

This guide is designed to help you study more effectively and approach your exam with confidence. Whether you're reviewing for the first time or doing a final refresh, here's how to get the most out of your Examzify study guide:

1. Start with a Diagnostic Review

Skim through the questions to get a sense of what you know and what you need to focus on. Your goal is to identify knowledge gaps early.

2. Study in Short, Focused Sessions

Break your study time into manageable blocks (e.g. 30 – 45 minutes). Review a handful of questions, reflect on the explanations.

3. Learn from the Explanations

After answering a question, always read the explanation, even if you got it right. It reinforces key points, corrects misunderstandings, and teaches subtle distinctions between similar answers.

4. Track Your Progress

Use bookmarks or notes (if reading digitally) to mark difficult questions. Revisit these regularly and track improvements over time.

5. Simulate the Real Exam

Once you're comfortable, try taking a full set of questions without pausing. Set a timer and simulate test-day conditions to build confidence and time management skills.

6. Repeat and Review

Don't just study once, repetition builds retention. Re-attempt questions after a few days and revisit explanations to reinforce learning. Pair this guide with other Examzify tools like flashcards, and digital practice tests to strengthen your preparation across formats.

There's no single right way to study, but consistent, thoughtful effort always wins. Use this guide flexibly, adapt the tips above to fit your pace and learning style. You've got this!

Questions

1. What is the purpose of MMC in relation to position tolerances?

A. To enforce identical sizes for all holes.

B. To reduce the tolerance zone regardless of size.

C. To set a fixed, unchanging tolerance regardless of size.

D. To permit a bonus tolerance when actual size approaches LMC for holes or MMC for shafts, affecting the position tolerance.

2. What is the monthly compensator placement accuracy tolerance?

A. 0.5 mm

B. 0.75 mm

C. 1 mm

D. 1.5 mm

3. The CT number accuracy tolerance for water is which value?

A. ±2 Hu

B. ±7 Hu

C. ±1 Hu

D. ±5 Hu

4. How does a GD&T callout indicate that a feature must be located within a cylindrical zone rather than a square zone?

A. The positional tolerance is specified with a square symbol.

B. The use of a diameter symbol for the positional tolerance indicates a cylindrical zone.

C. The datum references define the zone shape.

D. A radius symbol indicates a cylindrical zone.

5. What is the monthly asymmetric jaw position indicator tolerance?

A. 0.5 mm

B. 2 mm

C. 1 mm

D. 1.5 mm

6. Which statement best defines a datum in GD&T?

A. A reference point, line, or surface used to establish a datum reference frame for evaluating tolerances.

B. A basic dimension.

C. A tolerance modifier.

D. A projection height.

7. Which statement correctly describes unilateral tolerances?

A. A unilateral tolerance allows variation in only one direction from the basic size.

B. A unilateral tolerance allows equal variation in both directions from the basic size.

C. A unilateral tolerance allows any variation without direction.

D. A unilateral tolerance fixes the dimension exactly.

8. Which statement best describes a datum feature?

A. A datum feature is the target location used for true position measurement.

B. A datum feature is the theoretical plane used for design.

C. A datum feature is the measurement reference used to derive the datum.

D. A datum feature is the actual physical surface used to establish the datum.

9. What is the monthly wedge placement accuracy tolerance?

A. 1 mm

B. 2 mm

C. 2.5 mm

D. 3 mm

10. Which statement best describes a three-datum frame in GD&T?

A. All three datum features must be oriented to create a reference frame.

B. Only two datum features are ever required.

C. A three-datum frame is not used in practice.

D. Datums do not affect the reference frame.

Answers

1. D
2. C
3. D
4. B
5. C
6. A
7. A
8. D
9. B
10. A

Explanations

1. What is the purpose of MMC in relation to position tolerances?

A. To enforce identical sizes for all holes.

B. To reduce the tolerance zone regardless of size.

C. To set a fixed, unchanging tolerance regardless of size.

D. To permit a bonus tolerance when actual size approaches LMC for holes or MMC for shafts, affecting the position tolerance.

The idea behind MMC in relation to position tolerances is to allow a bonus in the positional tolerance based on how far the actual feature size is from its most restrictive material condition. For holes, when the hole size moves toward the largest allowed (toward the least material around the hole), you get extra latitude in how far the axis can be from its true position. For shafts, when the size moves toward the largest allowed (toward MMC), you also gain that additional allowance. In other words, the position tolerance isn't fixed; it expands as the size of the feature approaches the less restrictive extreme, making it easier to assemble parts without violating the tolerance. This explains why a MMC modifier can increase the permissible positional error beyond the basic tolerance, reflecting that more misalignment can be tolerated when the feature size itself is more permissive.

2. What is the monthly compensator placement accuracy tolerance?

A. 0.5 mm

B. 0.75 mm

C. 1 mm

D. 1.5 mm

In compensator-based therapy, exactly where the compensator sits relative to the beam is critical because even small shifts can alter the shaped dose, especially in high-gradient regions. The monthly placement tolerance sets the maximum allowed deviation between where the compensator is actually placed and where the plan expects it to be. One millimeter is the best choice because it balances precision with practical QA realities. Measurement tools and setup procedures in a clinical setting can reliably detect and control about 1 mm of displacement. If the tolerance were much tighter, like 0.5 mm, achieving and verifying that level of accuracy every month would be very demanding and could lead to frequent flagged issues due to normal measurement noise and small mechanical drifts. If it were larger, like 1.5 mm, the delivered dose in the modulated regions could shift enough to breach clinically acceptable dose variations, potentially affecting target coverage or increasing dose to nearby critical structures. So, a 1 mm monthly tolerance provides a realistic, achievable standard that protects treatment accuracy without being overly stringent.

3. The CT number accuracy tolerance for water is which value?

A. ±2 Hu

B. ±7 Hu

C. ±1 Hu

D. ±5 Hu

CT numbers quantify how much a voxel attenuates X-rays relative to water, with water defined at 0 HU. The accuracy tolerance describes how close the measured water value should stay to zero under routine scanning conditions. In practice, about ±5 HU is used as the standard tolerance. This range accounts for typical day-to-day variations from scanner calibration drift, different kVp settings, reconstruction kernels, noise levels, and partial-volume effects when measuring a water phantom. It provides a practical balance between precision and achievable reproducibility across different scanners and QA setups. Smaller tolerances, like ±1 or ±2 HU, are often too stringent for routine QA because of the factors above, while a looser tolerance like ±7 HU is less protective of CT number accuracy. Therefore, ±5 HU best fits common clinical QA expectations for water CT-number accuracy.

4. How does a GD&T callout indicate that a feature must be located within a cylindrical zone rather than a square zone?

A. The positional tolerance is specified with a square symbol.

B. The use of a diameter symbol for the positional tolerance indicates a cylindrical zone.

C. The datum references define the zone shape.

D. A radius symbol indicates a cylindrical zone.

In GD&T, the shape of the tolerance zone for a positional callout is defined by the tolerance symbol that precedes the value. The diameter symbol signifies a circular cross-section, so the allowable deviation forms a right circular cylinder around the true position axis. The feature's axis or centerline must lie inside that cylinder, with the datums constraining where that cylinder sits and how it's oriented in the part. That's why using the diameter symbol indicates a cylindrical zone rather than something like a square zone—the symbol directly tells you the cross section is circular, which creates a cylindrical tolerance zone. The datum references help locate and orient this zone in relation to the part, but they don't change the fact that the zone is cylindrical. The radius symbol or any notion of a square zone aren't used to define the positional tolerance zone shape in this context.

5. What is the monthly asymmetric jaw position indicator tolerance?

A. 0.5 mm

B. 2 mm

C. 1 mm

D. 1.5 mm

Tolerance for a jaw position indicator defines how much deviation from the intended position is allowed during monthly checks. A value of 1 mm is chosen because it strikes a practical balance: it's small enough to detect meaningful misalignment that could affect function, yet large enough to accommodate normal measurement variation, tool wear, and human reading error that occur in monthly inspections. Tighter limits like 0.5 mm would be overly strict for routine monthly verification, likely causing false indications of a problem, while looser limits such as 1.5 or 2.0 mm could miss deviations that impact performance over time. Hence, 1 mm is the appropriate monthly tolerance.

6. Which statement best defines a datum in GD&T?

A. A reference point, line, or surface used to establish a datum reference frame for evaluating tolerances.

B. A basic dimension.

C. A tolerance modifier.

D. A projection height.

In GD&T, measurements are made from a fixed reference system called the datum reference frame, which is defined by a datum. A datum is a real feature used to establish that frame—its surface, edge, or point becomes the reference element that sets the orientation and location for evaluating tolerances. This is why the statement describing a datum as a reference point, line, or surface used to establish a datum reference frame for evaluating tolerances is the best fit: it captures how datums anchor the measurement system. A basic dimension is an exact size used to locate features, not the reference system itself. A tolerance modifier is a symbol that changes how much variation is allowed, not the datum. A projection height relates to how far a tolerance zone is projected away from a feature, which is another concept entirely.

7. Which statement correctly describes unilateral tolerances?

A. A unilateral tolerance allows variation in only one direction from the basic size.

B. A unilateral tolerance allows equal variation in both directions from the basic size.

C. A unilateral tolerance allows any variation without direction.

D. A unilateral tolerance fixes the dimension exactly.

Unilateral tolerances let allowable variation occur in only one direction from the nominal size. That means you choose either a plus or a minus limit, but not both, so the size can move only larger than the basic size or only smaller than it. For example, with a basic size of 10.00 and a unilateral tolerance of +0.05/0, the acceptable range is 10.00 to 10.05. If the tolerance is -0.05/0, the range is 9.95 to 10.00. This is different from a bilateral tolerance, which allows equal variation in both directions (for instance, ±0.05 would give 9.95 to 10.05). The idea of any variation without direction or fixing the dimension exactly doesn't describe unilateral tolerancing.

8. Which statement best describes a datum feature?

A. A datum feature is the target location used for true position measurement.

B. A datum feature is the theoretical plane used for design.

C. A datum feature is the measurement reference used to derive the datum.

D. A datum feature is the actual physical surface used to establish the datum.

Datum features are the real surfaces or features on a part that you use to establish the datum reference frame. The datum itself is a theoretical, exact plane, axis, or point created from that surface for measurement, but the feature—the actual physical surface on the part—is what defines where that datum comes from in the real world. That's why describing a datum feature as the actual physical surface used to establish the datum is the most precise choice. Other ideas mix up roles: a target location relates to where you measure true position, not the surface that defines the datum; a theoretical plane describes the datum itself rather than the physical surface; and a measurement reference used to derive the datum sounds like the datum frame rather than the surface that establishes it. The key is that the datum feature is the tangible surface that anchors the datum.

9. What is the monthly wedge placement accuracy tolerance?

A. 1 mm

B. 2 mm

C. 2.5 mm

D. 3 mm

Wedge placement accuracy tolerance measures how far the wedge can be off from its intended position in the beam during monthly QA. Because the wedge shapes the dose distribution, even a small misalignment shifts the dose gradient across the field, potentially affecting target coverage and normal-tissue sparing. The practical, clinically acceptable limit that aligns with typical mechanical repeatability and QA measurement capabilities is two millimeters. Staying within this tolerance helps ensure the delivered dose closely matches the planned distribution from month to month. If the wedge is off by more than about 2 mm, the resulting dose pattern could become clinically significant, prompting recalibration or service. To check this, a phantom-based measurement compares the wedge edge position against a baseline; deviations beyond 2 mm indicate maintenance is needed. The other values would either be too strict for routine maintenance (1 mm) or would permit larger misplacements that could noticeably impact the dose (2.5 mm or 3 mm).

10. Which statement best describes a three-datum frame in GD&T?

A. All three datum features must be oriented to create a reference frame.

B. Only two datum features are ever required.

C. A three-datum frame is not used in practice.

D. Datums do not affect the reference frame.

A three-datum frame uses three non-parallel datum features to establish a stable, unique reference in space. These three datums define primary, secondary, and tertiary directions, locking all six degrees of freedom—three translations and three rotations. With three independent datums, the part's orientation and location are fixed consistently, which is why this frame is widely used in GD&T for both sizing and inspection. If you only had two datums, one rotational or translational degree of freedom would remain unconstrained, leading to ambiguity in the part's positioning. In practice, three datums are chosen to create a robust, repeatable reference frame.

Next Steps

Congratulations on reaching the final section of this guide. You've taken a meaningful step toward passing your certification exam and advancing your career.

As you continue preparing, remember that consistent practice, review, and self-reflection are key to success. Make time to revisit difficult topics, simulate exam conditions, and track your progress along the way.

If you need help, have suggestions, or want to share feedback, we'd love to hear from you. Reach out to our team at hello@examzify.com.

Or visit your dedicated course page for more study tools and resources:

https://tg142tolerances.examzify.com

We wish you the very best on your exam journey. You've got this!