

NIMS Level I Job Planning, Benchwork, & Layout Practice Test (Sample)

Study Guide



Everything you need from our exam experts!

Copyright © 2025 by Examzify - A Kaluba Technologies Inc. product.

ALL RIGHTS RESERVED.

No part of this book may be reproduced or transferred in any form or by any means, graphic, electronic, or mechanical, including photocopying, recording, web distribution, taping, or by any information storage retrieval system, without the written permission of the author.

Notice: Examzify makes every reasonable effort to obtain from reliable sources accurate, complete, and timely information about this product.

SAMPLE

Questions

SAMPLE

- 1. When machining hard materials, which technique might a machinist use?**
 - A. High-speed machining to reduce tool wear**
 - B. Using only cutting fluids**
 - C. Increasing the machine's vibration**
 - D. Permanently binding the tools to the material**
- 2. How can you identify the highest point of a workpiece during layout?**
 - A. Using a laser measurement tool**
 - B. Using a surface plate and dial indicator or height gauge**
 - C. By visual inspection**
 - D. Using only a caliper**
- 3. What does "G-code" refer to in CNC machining?**
 - A. A type of machine lubricant**
 - B. A language used to instruct CNC machines on how to perform specific tasks**
 - C. A geometric calculation in programming**
 - D. A measurement scale used in machining**
- 4. What does the term "setup" refer to in machining?**
 - A. The assembly of materials for final inspection**
 - B. The process of preparing the machine, tools, and workpiece for machining**
 - C. The adjustment of tolerances in design**
 - D. The maintenance checks performed on machines**
- 5. What is the primary reason for maintaining a clean workspace while operating bench tools?**
 - A. To avoid distractions**
 - B. It helps in reducing the risk of accidents**
 - C. It's a requirement for tool operation**
 - D. To impress supervisors**

- 6. What does "dimensioning" refer to in technical drawings?**
- A. The process of creating 3D models**
 - B. The process of indicating the measurements and specifications of features on a drawing**
 - C. The outlining of boundaries for manufacturing processes**
 - D. The sketching of component placements**
- 7. To calculate the Small Dia of Taper, which formula is used?**
- A. $(\text{Big Dia} - \text{Small Dia} / \text{Length}) * 12$**
 - B. $\text{Small Dia} + 12$**
 - C. $\text{Big Dia} - \text{Length}$**
 - D. $(\text{Big Dia} + \text{Small Dia}) / \text{Length}$**
- 8. When using the included angle formula, which tool is designated for accuracy in measurement?**
- A. Protractor**
 - B. Tape measure**
 - C. Caliper**
 - D. Micrometer**
- 9. The first thing to do when constructing an inspection plan is to?**
- A. Select the critical and important dimensions to inspect**
 - B. Choose the measuring tools**
 - C. Schedule inspection intervals**
 - D. Train inspection personnel**
- 10. What is the included angle for a taper of 0.550 inches?**
- A. 2.626 degrees**
 - B. 3.145 degrees**
 - C. 1.487 degrees**
 - D. 4.329 degrees**

Answers

SAMPLE

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

SAMPLE

Explanations

SAMPLE

1. When machining hard materials, which technique might a machinist use?

A. High-speed machining to reduce tool wear

B. Using only cutting fluids

C. Increasing the machine's vibration

D. Permanently binding the tools to the material

High-speed machining is a common technique used when working with hard materials because it helps to reduce tool wear and improve efficiency. This method involves increasing the cutting speed, which not only accelerates the machining process but also minimizes the heat generated at the cutting edge of the tool. When cutting hard materials, excessive heat can lead to rapid tool degradation and failure. By increasing the speed, the material is removed before excessive heat can build up, thus prolonging the life of the cutting tool. Additionally, high-speed machining often utilizes optimized cutting tools designed specifically for use with hard materials. This combination results in a more efficient and effective machining process, ensuring better surface finishes and tighter tolerances while also minimizing downtime due to tool changes. The other techniques listed do not effectively address the challenge of machining hard materials. For example, relying solely on cutting fluids does not inherently solve the issue of tool wear. Increasing machine vibration would likely be detrimental, leading to poor machining quality and further tool wear. Permanently binding tools to the material is not a standard or feasible method in machining practices as it would complicate or hinder the machining process.

2. How can you identify the highest point of a workpiece during layout?

A. Using a laser measurement tool

B. Using a surface plate and dial indicator or height gauge

C. By visual inspection

D. Using only a caliper

Identifying the highest point of a workpiece during layout is critical for ensuring precise machining and accurate assembly. Using a surface plate paired with a dial indicator or height gauge is a reliable method. The surface plate provides a flat and stable reference surface, ensuring that measurements are taken from a consistent baseline. When employing a dial indicator or height gauge, you can precisely measure variations in height across the workpiece. The gauge can be moved along the surface, and as it travels, it indicates changes in elevation, allowing you to locate the highest point accurately. This method is important in machining and layout work because it minimizes errors that could arise from inconsistently measuring from uneven surfaces. While other techniques, such as a laser measurement tool or visual inspection, can provide a general idea about height, they often lack the specificity and accuracy required in detailed layout tasks. Similarly, relying solely on a caliper may limit the measurement capabilities to particular dimensions and does not provide a comprehensive view of the entire workpiece's structure. Thus, using the combination of a surface plate and measuring tools provides the best accuracy for identifying the highest point on a workpiece.

3. What does "G-code" refer to in CNC machining?

- A. A type of machine lubricant
- B. A language used to instruct CNC machines on how to perform specific tasks**
- C. A geometric calculation in programming
- D. A measurement scale used in machining

G-code, also known as "Geometric code," is indeed a language used to instruct CNC (Computer Numerical Control) machines on how to perform specific tasks. It comprises a series of commands or instructions that dictate the movements and actions of the machine, such as the speed, feed rate, and path of the tool. By using G-code, operators can control the precision of machining operations, allowing for complex and detailed workpiece fabrication. In CNC programming, the G-code commands enable the machine to understand exactly what movements to make, which tools to use, and how to execute the machining process efficiently. This instruction set is essential for translating design plans into actionable steps, ensuring accuracy and repeatability in the manufacturing process. Understanding G-code is crucial for anyone involved in CNC machining, as it forms the foundational language for machine operation, allowing for a wide variety of tasks from drilling to milling and turning.

4. What does the term "setup" refer to in machining?

- A. The assembly of materials for final inspection
- B. The process of preparing the machine, tools, and workpiece for machining**
- C. The adjustment of tolerances in design
- D. The maintenance checks performed on machines

The term "setup" in machining specifically refers to the process of preparing the machine, tools, and workpiece for machining. This involves several crucial steps, including selecting the appropriate tools, configuring the machine settings, and positioning the workpiece securely in the machine. A proper setup is essential for ensuring accurate machining operations, as it impacts the quality of the finished product, the efficiency of the machining process, and the safety of the operators. It also includes aspects like ensuring that the tools are sharp and suitable for the task, calibrating machine parameters according to required specifications, and aligning the workpiece correctly to avoid errors during production. This understanding of "setup" highlights its importance in the overall manufacturing process, underscoring how a well-prepared setup leads to high-quality outputs and reduces the likelihood of mistakes or machine downtime during operations.

5. What is the primary reason for maintaining a clean workspace while operating bench tools?

- A. To avoid distractions**
- B. It helps in reducing the risk of accidents**
- C. It's a requirement for tool operation**
- D. To impress supervisors**

Maintaining a clean workspace while operating bench tools is primarily important for reducing the risk of accidents. A clean environment allows for better visibility and easier access to tools, materials, and workpieces, minimizing the chances of trips, slips, and other mishaps. When work surfaces are cluttered, it can lead to confusion or oversight, increasing the likelihood of tools being misplaced or materials being handled incorrectly, which can result in injury or damage to the workpiece. In addition, a clean workspace facilitates precision in work, as clutter can obstruct necessary movements and hinder proper focus. This practice not only ensures personal safety but also creates a more efficient workflow, allowing operators to concentrate on their tasks without the worry of hazards associated with a disorganized area.

6. What does "dimensioning" refer to in technical drawings?

- A. The process of creating 3D models**
- B. The process of indicating the measurements and specifications of features on a drawing**
- C. The outlining of boundaries for manufacturing processes**
- D. The sketching of component placements**

Dimensioning refers to the process of indicating the measurements and specifications of features on a technical drawing. This is critical for ensuring that everyone involved in the manufacturing process understands the exact size, shape, and location of parts as intended by the designer. Accurate dimensioning provides the necessary information for machining, assembly, and inspection, allowing for precise construction and fit of components. In contrast, creating 3D models involves using software to simulate objects in three dimensions and does not specifically convey the necessary measurements that dimensioning provides. Outlining boundaries for manufacturing processes relates to establishing limits for operations rather than specifying detailed measurements. Sketching component placements focuses on layout but lacks the formalized and standardized approach that dimensioning employs to convey essential measurements and specifications on a drawing.

7. To calculate the Small Dia of Taper, which formula is used?

A. (Big Dia - Small Dia / Length) * 12

B. Small Dia + 12

C. Big Dia - Length

D. (Big Dia + Small Dia) / Length

The formula for calculating the Small Diameter of a taper is based on the relationship between the big diameter, small diameter, and the length of the tapering section. The correct formula expresses the taper's angle in terms of the change in diameter over the length. The first option outlines the change in diameter as a function of the length, multiplied by 12 to convert the units typically from feet to inches or another appropriate unit. It reflects how the taper decreases over the specified length. This formula effectively allows for the calculation of the small diameter by providing a clear relationship between the large diameter and the taper's length. Precision in this calculation is crucial in machining, as it affects how workpieces are shaped and fit during assembly. Thus, this formula is an essential concept in job planning and machining processes where tapering operations are involved.

8. When using the included angle formula, which tool is designated for accuracy in measurement?

A. Protractor

B. Tape measure

C. Caliper

D. Micrometer

The protractor is specifically designed for measuring angles with a high degree of accuracy. Its circular scale allows users to determine the degree of an angle by aligning the angle's arms with the protractor's measurement lines. This makes it an ideal tool for applications requiring precision in angle measurement, such as layout practices in machining and woodworking. While other tools like the tape measure, caliper, and micrometer are essential for measuring length, depth, and thickness, they are not tailored for angle measurement. The tape measure is great for measuring straight distances, the caliper is used for inside, outside, and depth measurements, and the micrometer provides very precise linear measurements, typically millimeters or inches. However, none of these tools can effectively measure angles like a protractor can, which is why it is the correct answer in this context.

9. The first thing to do when constructing an inspection plan is to?

- A. Select the critical and important dimensions to inspect**
- B. Choose the measuring tools**
- C. Schedule inspection intervals**
- D. Train inspection personnel**

When constructing an inspection plan, the first step should be to select the critical and important dimensions to inspect. This is essential because it establishes the foundation of the inspection process. By identifying these dimensions, you ensure that your inspection focuses on the aspects of the product or process that significantly impact its quality and functionality. Identifying critical dimensions helps in prioritizing efforts and resources, ensuring that the inspection plan is effective and targeted towards preventing defects that could affect performance. This step sets the direction for the entire inspection plan, influencing the subsequent choices, such as which measuring tools to use, how frequently inspections should occur, and what training may be necessary. In addition, once the important dimensions are determined, it becomes easier to select appropriate measuring tools, schedule inspection intervals, and ensure that personnel are adequately trained. This structured approach ultimately leads to a more efficient and effective inspection strategy that aligns with quality standards and customer expectations.

10. What is the included angle for a taper of 0.550 inches?

- A. 2.626 degrees**
- B. 3.145 degrees**
- C. 1.487 degrees**
- D. 4.329 degrees**

To determine the included angle for a taper of 0.550 inches, it is important to understand the relationship between the taper dimension and the angle measurement, specifically in reference to the geometry involved in creating a taper. The included angle can be calculated using trigonometric principles, particularly the tangent function, which relates the opposite side (half the taper dimension) to the adjacent side. For a taper, the formula often used involves the taper thickness divided by the distance over which the taper is measured. For instance, if we assume that the taper measurement of 0.550 inches represents the total taper over a certain length, halving this dimension provides the rise per side of the taper. Once this half taper magnitude is divided by the distance and the arctangent function is applied, it leads to the calculation of the angle. The resulting angle reflects the geometry of the taper, forming an understanding of how slight adjustments can impact the diameter or target conical shape needed in machining practices. The value of 2.626 degrees makes sense in this context as it presents a realistic measurement for a taper that's not excessively steep, validating standard practices in taper machining where precision in angle greatly affects fit and performance. This aligns with machining and manufacturing standards, allowing