

NIMS COMPUTER NUMERICAL CONTROL (CNC) OPERATOR PRACTICE TEST (SAMPLE) STUDY GUIDE



SAMPLE STUDY GUIDE WITH LIMITED QUESTIONS

**VISIT US ONLINE FOR
THE FULL VERSION STUDY GUIDE**

<https://nimscncoperator.examzify.com>



Copyright © 2026 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 accurate, complete, and timely information about this product from reliable sources.

SAMPLE

Table of Contents

Copyright	1
Table of Contents	2
Introduction	3
How to Use This Guide	4
Questions	5
Answers	8
Explanations	10
Next Steps	15

SAMPLE

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

SAMPLE

- 1. What key restores the feed rate to the full programmed value?**
 - A. 100% SPINDLE
 - B. 5% RAPID
 - C. 100% FEEDRATE
 - D. -10 SPINDLE

- 2. What is the designation for a jog key that rotates machine components around the A-axis in a negative direction?**
 - A. -A
 - B. +A
 - C. -B
 - D. +B

- 3. What function does the auxiliary M code M88 serve?**
 - A. Turns coolant off
 - B. Turns on high-pressure coolant
 - C. Stops the spindle
 - D. Changes cutting tools

- 4. What is the importance of the Tool at Power Down setting?**
 - A. It determines tool wear
 - B. It indicates monitoring parameters
 - C. It specifies the tool retained in the spindle during shutdown
 - D. It controls the machining speed

- 5. What does the display on the Haas control primarily show during operation?**
 - A. Future scheduled programs
 - B. Current program and axis positions
 - C. Historical machining data
 - D. Firmware updates

- 6. Which function key is used to enter work coordinate offsets automatically during part setup?**
- A. SETUP
 - B. PART ZERO SET
 - C. OFFSET
 - D. TOOL SET
- 7. Which operation mode runs part programs from the control's memory?**
- A. Manual Data Input
 - B. MDI/DNC
 - C. MEM
 - D. OFFSET
- 8. What is the input buffer on a CNC machine?**
- A. A temporary location for outgoing messages
 - B. A dedicated space for a program's final output
 - C. A storage area for tool data
 - D. A temporary location that holds incoming information before it continues to the CPU for processing
- 9. What does the Tool at Power Down setting indicate?**
- A. The number of tools available
 - B. The tool to occupy the spindle upon power down
 - C. The current tool in use
 - D. The next tool in the sequence
- 10. Which override key on the Haas control increases the current feed rate in increments?**
- A. +10 SPINDLE
 - B. -10 FEEDRATE
 - C. +10 FEEDRATE
 - D. 25% RAPID

Answers

SAMPLE

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

SAMPLE

Explanations

SAMPLE

1. What key restores the feed rate to the full programmed value?

- A. 100% SPINDLE
- B. 5% RAPID
- C. 100% FEEDRATE**
- D. -10 SPINDLE

The key that restores the feed rate to the full programmed value is the one that specifically indicates a return to the full feed rate settings. In CNC operations, the feed rate defines how quickly the tool advances during cutting, which is crucial for ensuring proper machining performance and surface finish. The correct choice indicates a full restoration of the programmed feed rate, meaning the machine will operate at the intended speed as defined by the CNC program. This is significant in various machining tasks since the feed rate affects material removal, tool life, and overall efficiency. Other options represent different functionalities which do not pertain directly to restoring the feed rate to its full programmed value. For example, the spindle control keys typically affect the rotational speed of the spindle rather than the feed rate. Similarly, rapid settings would typically be faster than programmed feeds, indicating non-cutting movement rather than the outlined cutting operation. Understanding these distinctions is essential for effectively managing machine settings during operation.

2. What is the designation for a jog key that rotates machine components around the A-axis in a negative direction?

- A. -A**
- B. +A
- C. -B
- D. +B

The designation for a jog key that rotates machine components around the A-axis in a negative direction is represented by the negative sign followed by the axis name. In this case, the A-axis indicates movement or rotation around that particular axis. The negative sign indicates that the motion is in the counterclockwise direction when viewed from a specific point in space, generally following the right-hand rule used in machinery and CNC programming. Understanding the designation is crucial for operators because it helps them control the machine accurately and perform movements required for machining operations. For the A-axis, the negative direction implies a backward rotation, ensuring that the operator can make precise adjustments to the workpiece or tool position as needed.

3. What function does the auxiliary M code M88 serve?

- A. Turns coolant off
- B. Turns on high-pressure coolant**
- C. Stops the spindle
- D. Changes cutting tools

The auxiliary M code M88 is used to turn on high-pressure coolant in CNC machining operations. High-pressure coolant improves the cooling of the tool and workpiece, enhances chip removal, and helps maintain a consistent cutting temperature, which can result in longer tool life and improved surface finishes on the machined part. This capability is especially important in machining operations using harder materials or when performing processes that generate significant heat. By utilizing M88, operators can ensure that the coolant system operates effectively during machining, leading to better overall efficiency and productivity.

4. What is the importance of the Tool at Power Down setting?

- A. It determines tool wear
- B. It indicates monitoring parameters
- C. It specifies the tool retained in the spindle during shutdown**
- D. It controls the machining speed

The Tool at Power Down setting is significant because it specifies the tool that should be retained in the spindle when the CNC machine is in a shutdown state. This setting helps ensure that the machine is prepared for the next operation, reducing the risk of errors or misalignment when operations resume. Retaining the correct tool during power down is crucial for maintaining the integrity of the tool and the workpiece, as well as for preventing damage to the machine itself. Monitoring parameters or tool wear do not directly relate to what tool is kept in the spindle after shutting down, and controlling machining speed pertains to operation rather than the state of the machine during shutdown. Thus, specifying which tool is retained during this period is essential for effective machine operation and setup.

5. What does the display on the Haas control primarily show during operation?

- A. Future scheduled programs
- B. Current program and axis positions**
- C. Historical machining data
- D. Firmware updates

The display on the Haas control primarily shows the current program and axis positions because it is essential for operators to have real-time information about the machine's operation. This information allows the operator to monitor the progress of the machining process, verify that the correct program is running, and ensure that the machine's axes are at the desired positions. Additionally, displaying the current axis positions helps in making timely adjustments if necessary, ensuring both accuracy and efficiency during operations. In a CNC environment, knowing the current state of the program and position is crucial for maintaining quality control and ensuring that the machine operates as intended. This allows the operator to have complete situational awareness, which is fundamental for troubleshooting and maintaining productivity on the shop floor.

6. Which function key is used to enter work coordinate offsets automatically during part setup?

- A. SETUP
- B. PART ZERO SET**
- C. OFFSET
- D. TOOL SET

The function key used to enter work coordinate offsets automatically during part setup is the one related to establishing the starting point or reference for the machining process, which is typically referred to in CNC terms as "part zero." When setting up a part, it is crucial to define the coordinate system correctly so that the CNC machine can accurately interpret the locations of various machining operations in relation to the workpiece itself. By using this specific function key, operators can ensure that the machine knows exactly where the origin of the workpiece is located, simplifying the setup process and minimizing potential errors in machining. This function assists in streamlining the operations by automatically inputting the necessary offsets, allowing the operator to proceed with machining with confidence that the setup has been accurately established. In contrast, other keys may be associated with different tasks such as managing tools or general settings rather than directly entering work coordinate offsets. These functionalities, while important to overall operations, do not specifically support the initial step of defining part coordinates within the machine's setup process.

7. Which operation mode runs part programs from the control's memory?

- A. Manual Data Input
- B. MDI/DNC
- C. MEM**
- D. OFFSET

The correct choice, which indicates the operation mode that runs part programs from the control's memory, is "MEM." In this mode, the CNC machine reads the part program that has been stored in its internal memory. This allows for automated operation, where the machine can execute the sequence of instructions without continuous input from the operator during the machining process. In contrast, Manual Data Input (MDI) allows the operator to input commands one at a time directly into the control, which is useful for manual programming or test runs but does not utilize stored programs for full operations. DNC (Direct Numerical Control), often paired with MDI, refers to a method of controlling CNC machines via a computer network or direct connection but does not pertain to memory execution specifically. The OFFSET mode is used for adjusting tool positions or work offsets but does not relate to running stored part programs. Thus, the "MEM" mode is specifically designed for executing the stored part programs, making it the correct answer.

8. What is the input buffer on a CNC machine?

- A. A temporary location for outgoing messages
- B. A dedicated space for a program's final output
- C. A storage area for tool data
- D. A temporary location that holds incoming information before it continues to the CPU for processing**

The input buffer on a CNC machine serves as a temporary location to hold incoming information before it is processed by the CPU. This allows the machine to manage the flow of data more effectively, ensuring that the CPU can process commands without interruption while still accepting new incoming data. The buffering mechanism helps to prevent data loss and ensures smoother operation, especially in scenarios with high-speed communication or complex computation. In contrast, other options refer to different functionalities that do not accurately describe the input buffer's purpose. For example, a location for outgoing messages pertains to the output buffer, which is distinct from the input buffer's role. Similarly, the final output and tool data storage also represent separate functions related to program execution and tools used in machining, rather than the preliminary data handling done by the input buffer.

9. What does the Tool at Power Down setting indicate?

- A. The number of tools available
- B. The tool to occupy the spindle upon power down**
- C. The current tool in use
- D. The next tool in the sequence

The Tool at Power Down setting indicates the tool that is intended to occupy the spindle when the CNC machine is powered down. This setting is important because it helps in preserving the integrity of the tooling and ensures that the machine is in the correct state for the next startup. By having a specific tool designated to remain in the spindle during power down, it minimizes the risk of damage to the tool and the spindle, and helps prevent errors when the machine is next powered up. Proper tool management is essential for maintaining the efficiency and accuracy of CNC operations, and this setting plays a crucial role in that process.

10. Which override key on the Haas control increases the current feed rate in increments?

- A. +10 SPINDLE
- B. -10 FEEDRATE
- C. +10 FEEDRATE**
- D. 25% RAPID

The correct answer reflects the function of the override key that specifically relates to adjusting feed rates. On the Haas control, the "+10 FEEDRATE" key is designed to increase the current feed rate in increments of 10%. This key allows the operator to make precise adjustments to the feed rate during machining operations without having to reprogram or stop the machine. By using this key, CNC operators can adapt the machining parameters to the specific needs of the operation, such as optimizing cutting speeds and ensuring efficient material removal while maintaining tool life and workpiece quality. This feature is valuable in real-time situations where adjustments may be necessary based on material conditions or tooling performance. The other options relate to different functions or percentage settings that do not specifically pertain to increasing the current feed rate in increments like the option that was correctly chosen.

SAMPLE

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://nimscncoperator.examzify.com>

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

SAMPLE