CNC THREADING AND MACHINING PRACTICE TEST (SAMPLE)

STUDY GUIDE



SAMPLE STUDY GUIDE WITH LIMITED QUESTIONS

VISIT US ONLINE FOR THE FULL VERSION STUDY GUIDE

https://cncthreadingmachining.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.

SAMPLE

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!

SAMPLE

Questions

SAMPLE

1. What should a part programmer do with a blueprint before using it to create a part on a CNC mill?

A. Blueprint drafting

B. Blueprint conversion

C. Blueprint interpretation

D. Blueprint optimization

2. Which diameter describes the distance between crests of an external thread?

A. Major diameter

B. Minor diameter

C. Pitch diameter

D. Root diameter

3. For outside diameter turning on a lathe, which axis must be touched off and zeroed?

A. X axis

B. Y axis

C. Z axis

D. A axis

4. The forward-and-backward movement of the cross slide aligns with which axis on the engine lathe?

A. X axis

B. Y axis

C. Z axis

D. B axis

5. Which machine type changes axis directions due to the worktable moving rather than the cutting tool?

A. Vertical milling machine

B. Horizontal milling machine

C. Lathe

D. Drilling machine

SAMPLE

6. The phrase 'Indication of feature importance in a blueprint' refers to providing what kind of information about the part?

A. Provide auxiliary, non-crucial information about the part

B. Indicate critical features by notes

C. Show surface finishes

D. Specify material type

7. Which is cut from a workpiece?

A. Chips

B. Swarf

C. Shavings

D. Flakes

8. What does a machinist use to manually check alignment of the tool tip to the spindle?

A. Tailstock

B. Chuck

C. Carriage

D. Tool post

9. When finishing a hole by reaming, how much material should remain?

A. 0.001-0.005 in

B. 0.005-0.010 in

C. 0.010-0.020 in

D. 0.020-0.030 in

10. In CAD/CAM, which feature stores part design information?

A. The database

B. The gage line

C. CAM

D. DXF

SAMPLE

Answers

SAMPLE

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

SAMPLE

Explanations

SAMPLE

1. What should a part programmer do with a blueprint before using it to create a part on a CNC mill?

A. Blueprint drafting

B. Blueprint conversion

C. Blueprint interpretation

D. Blueprint optimization

The main idea is turning the drawing into machine-ready information before you write any toolpaths. A part programmer converts the blueprint into data the CAM/CNC workflow can use—extracting all critical dimensions, tolerances, GD&T, features, hole patterns, and surface requirements and translating them into a concrete plan: which features to machine, in what order, what tools and speeds/feeds, and how the part will be fixtured. This conversion creates unambiguous, actionable instructions that the machine and software can follow to produce the part. While you still interpret the drawing to understand what's required, the step of conversion focuses on translating that understanding into manufacturable, CAM-ready data.

2. Which diameter describes the distance between crests of an external thread?

A. Major diameter

B. Minor diameter

C. Pitch diameter

D. Root diameter

The distance across the crests of an external thread is the major diameter. This is the largest diameter of the thread, measured from crest to crest, and it defines the external size of the screw or bolt. The major diameter is what you use to specify the size of an externally threaded part because it relates to how the bolt will fit into a threaded hole or nut. Other diameters describe different features: the minor (root) diameter is the smallest diameter at the bottom of the thread, and the pitch diameter is the theoretical middle diameter where the thread thickness equals the pitch, used for calculating fit and engagement with the mating internal thread. Root diameter is another term for the smallest diameter at the thread roots.

3. For outside diameter turning on a lathe, which axis must be touched off and zeroed?

A. X axis

B. Y axis

C. Z axis

D. A axis

Outside diameter work is a radial measurement: you're controlling how far the cutting tool is from the spindle centerline. That distance lies along the X axis on a CNC lathe. By touch-offing the tool against the part and establishing a known X reference, you create a baseline from which all OD cuts are defined. Once X is zeroed relative to that surface, moving the tool in X changes the diameter you produce, because X is the radial position that sets the roundness of the part. The Z axis, in contrast, runs along the spindle and governs length along the part, not its diameter. The Y axis (when present) is a secondary lateral axis used on some machines for additional motion, but it isn't the axis that defines the standard outside diameter in turning. The A axis is a rotary axis used for live tooling or rotary tables, not for the basic OD geometry. So, for reliably machining a specific outside diameter, you touch off and zero the X axis.

SAMPLE

4. The forward-and-backward movement of the cross slide aligns with which axis on the engine lathe?

A. X axis

B. Y axis

C. Z axis

D. B axis

The cross slide moves in and out from the center of the spindle, i.e., in a radial direction. On a traditional engine lathe, this radial motion is defined as the X axis. The axis that runs along the length of the spindle is the Z axis, which handles longitudinal travel. Y and B axes aren't part of a standard manual lathe's setup, so they don't describe the cross slide movement. So, the forward-and-backward (in-and-out) motion of the cross slide aligns with the X axis.

5. Which machine type changes axis directions due to the worktable moving rather than the cutting tool?

A. Vertical milling machine

B. Horizontal milling machine

C. Lathe

D. Drilling machine

Positioning the workpiece is done by moving the table, not by moving the cutting tool. In a vertical milling machine, the spindle is vertical, so the cutting action occurs primarily along the vertical axis, while the table shifts in the horizontal X and Y directions to bring different parts of the workpiece into position under the cutter. Because the axis directions that define where you cut are set by how the table moves, this machine type is the one where axis changes come from the worktable rather than the cutting tool. The other machines don't rely on the table to redefine the cutting directions in the same way: in a lathe the tool moves relative to a rotating workpiece across multiple axes, in a drilling machine the drill moves along its own axis to cut and the table mainly serves for basic positioning, and in a horizontal milling machine the spindle orientation is different but the principle of tool-driven axis movement is more integral to the cutting process.

6. The phrase 'Indication of feature importance in a blueprint' refers to providing what kind of information about the part?

A. Provide auxiliary, non-crucial information about the part

B. Indicate critical features by notes

C. Show surface finishes

D. Specify material type

Understanding feature importance on a blueprint is about signaling which aspects of the part are essential to its function and which are auxiliary. When the phrase points to indicating feature importance, it means the drawing conveys information about which features are secondary or non-crucial to performance, helping the maker know what details can be treated as guidance rather than strict constraints. This can include general notes, non-binding suggestions, or context that doesn't directly control form, fit, or function. The idea is to provide helpful background without imposing additional strict requirements on the geometry itself. In contrast, other ways of interpreting feature importance would focus on marking out features that are essential or requiring specific finishes or materials, which shift the emphasis away from auxiliary guidance and toward mandatory constraints or specifications.

SAMPLE

7. Which is cut from a workpiece?

A. Chips

B. Swarf

C. Shavings

D. Flakes

During a cutting operation, material is removed from the workpiece in the form of chips. These are the fragments that curl or break off as the tool shears the metal. The standard term for what is produced by cutting in machining is chips, and their shape can vary with material and cutting conditions. While swarf is sometimes used to refer to metal waste in general, and shavings or flakes might be used in other contexts (like woodworking or for brittle materials), the specific piece cut from the workpiece is best described as a chip. Understanding chips helps with selecting chip-breakers, coolant strategies, and controlling heat, surface finish, and safety.

8. What does a machinist use to manually check alignment of the tool tip to the spindle?

A. Tailstock

B. Chuck

C. Carriage

D. Tool post

The tool tip must run on the same axis as the spindle, so you need a reference that is coaxial with that axis. The tailstock provides that reference: it can hold a center or a precisely machined bar aligned with the spindle axis. By bringing a center (or bar) in the tailstock into alignment opposite the spindle center and checking how the tool tip sits relative to that axis (often with a dial indicator or by observing runout as the spindle turns), you can verify and adjust whether the tool tip is truly on centerline. The chuck, carriage, and tool post each have important roles, but they don't supply the coaxial reference needed to check alignment the way the tailstock does.

9. When finishing a hole by reaming, how much material should remain?

A. 0.001-0.005 in

B. 0.005-0.010 in

C. 0.010-0.020 in

D. 0.020-0.030 in

Finishing a hole by reaming relies on the reamer removing only a small amount of material to bring the hole to its final size while improving roundness and surface finish. Leave about 0.005–0.010 in of stock in the hole after drilling so the reamer has enough material to cut cleanly without rubbing. If you leave less than about 0.005 in, cutting may be inconsistent and the surface finish can be rough due to rubbing. If you leave more than about 0.010 in, the reamer would have to remove more material, which can increase cycle time and risk deflection or loss of concentricity. So, the best practice is to leave 0.005–0.010 in of stock for the finishing pass.

SAMPLE

10. In CAD/CAM, which feature stores part design information?

A. The database

B. The gage line

C. CAM

D. DXF

In CAD/CAM, the central repository for part design information is the database. It stores the model geometry, feature history, constraints and dimensions, material properties, tolerances, and revision history, providing a single source of truth that can be accessed by both design and manufacturing processes. A format like DXF can carry geometry for transfer or viewing, but it doesn't preserve full design intent, relationships, or history. The gage line relates to measurement, not storage, and CAM uses the CAD data to generate toolpaths, not to store the design itself. So the database is the best place to keep all part design information.

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://cncthreadingmachining.examzify.com

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

SAMPLE