

SOLIDWORKS / SOLIDPROFESSOR PRACTICE EXAM (SAMPLE)

STUDY GUIDE



**SAMPLE STUDY GUIDE
WITH LIMITED QUESTIONS**

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

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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

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- 1. What icon is used to exit a sketch while saving changes?**
 - A. Red X icon
 - B. Blue arrow icon
 - C. Checkmark icon
 - D. Save icon

- 2. Where are Spline Handles located in a spline?**
 - A. At the start and end of the spline only
 - B. At every control point and every endpoint
 - C. Only at control points
 - D. Only at endpoints

- 3. What is a primary use of the Helix feature in SolidWorks?**
 - A. Creating straight lines
 - B. Defining circular patterns
 - C. Generating spiral shapes
 - D. Making rectangular features

- 4. In SolidWorks, which of the following describes a "Sketch"?**
 - A. A representation of a 3D object
 - B. A 2D drawing used for creating features
 - C. A collection of parts in an assembly
 - D. A template for sheet metal designs

- 5. After selecting the profile sketches of a loft, dragging the green connectors will:**
 - A. Delete the profiles
 - B. Twist the Loft feature
 - C. Create additional profiles
 - D. Change the color of the sketches

6. True or False: To create a Center Line Loft, the center line must intersect each profile and connect points from each profile together.
- A. True
 - B. False
 - C. Only if the profiles are horizontal
 - D. Only when using sketches
7. True or False: You can select a planar face as the reference plane for mirroring a feature?
- A. True
 - B. False
 - C. Only in assembly mode
 - D. Only for sketches
8. What is the "Extrude Cut" feature used for?
- A. To reshape a solid body into a different profile
 - B. To remove material from a solid body using a 2D sketch
 - C. To create a hole by revolving a sketch profile
 - D. To apply a surface feature over an existing body
9. What should you expect when changing the sheet scale in relation to detail views?
- A. The sheet scale does not affect detail views
 - B. Detail views scale automatically with the sheet
 - C. Detail views only update if linked
 - D. Only the primary view is affected
10. Which command is used to change the scale of a view in SolidWorks?
- A. Modify View
 - B. Edit Scale
 - C. Scale Tool
 - D. Properties

Answers

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

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Explanations

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1. What icon is used to exit a sketch while saving changes?

- A. Red X icon
- B. Blue arrow icon**
- C. Checkmark icon
- D. Save icon

The correct answer is the blue arrow icon, which signifies the action of exiting a sketch while saving any changes made during the editing session. In SolidWorks, the blue arrow icon typically indicates a confirmation action, allowing users to save their modifications to the sketch and return to the previous work environment in one step. Exiting a sketch with the blue arrow ensures that any geometrical adjustments, constraints, or other modifications are properly recorded before leaving the sketch mode. This is a vital step in the design process, as ensuring changes are saved helps maintain the integrity of the part or assembly being created. In contrast, the other icons serve different purposes. The red X icon usually indicates a cancellation action, explicitly used to exit without saving changes, while the checkmark icon is often associated with confirming individual features rather than saving a sketch. The save icon typically relates to general file-saving operations rather than specifically exiting a sketch. Therefore, choosing the blue arrow accurately represents the process of saving changes while exiting a sketch in SolidWorks.

2. Where are Spline Handles located in a spline?

- A. At the start and end of the spline only
- B. At every control point and every endpoint**
- C. Only at control points
- D. Only at endpoints

The correct choice emphasizes that spline handles are located at every control point and every endpoint of the spline. In SolidWorks, splines are defined by control points that determine their shape and curvature. The handles associated with these control points allow users to manipulate the spline more intricately. When you add control points to a spline, each control point will have a handle that can be adjusted to change the direction and extent of the curve at that point. Additionally, the endpoints of the spline also have handles, allowing for adjustments to be made at the very ends of the spline. This capability is crucial because it provides the user with the flexibility to shape the spline in a very specific manner, by both altering the positioning of control points and adjusting the handles to manipulate how the spline curves through space. The other choices do not accurately capture this functionality. For example, stating that the handles are only at the start and end of the spline or only at control points overlooks the fact that both sets of points have handles for manipulation. Understanding this structure is fundamental for effectively using splines in design and ensuring the desired curves and shapes are achieved in SolidWorks.

3. What is a primary use of the Helix feature in SolidWorks?

- A. Creating straight lines
- B. Defining circular patterns
- C. Generating spiral shapes**
- D. Making rectangular features

The Helix feature in SolidWorks is primarily used for generating spiral shapes. This can include creating helical curves or paths that may be used for a variety of applications such as designing screws, coils, or springs. The Helix allows for precise control over the number of turns, diameter, and pitch of the spiral, which makes it an essential tool for modeling complex shapes that follow a helical pattern. In contrast, straight lines can be created using basic sketching tools, while defining circular patterns involves using the Circular Pattern feature which duplicates features around a center point. Making rectangular features is typically achieved through the use of extrudes or revolves, relying on different sketch tools rather than the Helix function. Hence, the usage of the Helix feature is specifically aligned with creating spiraled or coil designs, making it a unique and valuable option in the design process within SolidWorks.

4. In SolidWorks, which of the following describes a "Sketch"?

- A. A representation of a 3D object
- B. A 2D drawing used for creating features**
- C. A collection of parts in an assembly
- D. A template for sheet metal designs

A "Sketch" in SolidWorks is best described as a 2D drawing used for creating features. When working in SolidWorks, a sketch is the fundamental building block for creating 3D models. It consists of lines, arcs, circles, and other 2D shapes that can be defined with dimensions and constraints. These sketches are typically used to define the profiles or outlines for various features such as extrudes, revolves, and cuts, ultimately contributing to the 3D geometry of a part. By utilizing sketches, users can precisely control the shape and size of their models before employing features that convert those sketches into 3D forms. This process is essential in solid modeling because it allows for changes and adjustments to be made at an early stage of design, promoting efficiency and accuracy. Sketches can also be reused and referenced in other sketches or in the creation of assemblies. The other options describe different aspects of SolidWorks. A representation of a 3D object pertains to the final products created through features derived from sketches, while a collection of parts in an assembly refers to how multiple components are brought together in a SolidWorks project. A template for sheet metal designs focuses on specific design settings and standards rather than the foundational sketches used for modeling.

5. After selecting the profile sketches of a loft, dragging the green connectors will:

- A. Delete the profiles
- B. Twist the Loft feature**
- C. Create additional profiles
- D. Change the color of the sketches

When you drag the green connectors of the profile sketches in a loft feature, it causes the loft to twist along the profiles. This twisting occurs because the loft feature is designed to interpolate a surface between the defined profiles, and manipulating the connectors directly affects how the surfaces transition from one profile to the next. The twisting effect can help produce more complex and aesthetically appealing shapes, which is often necessary in design tasks that require unique surfaces. Other options, such as deleting profiles or creating additional profiles, do not apply in this context since dragging the connectors does not remove any existing elements or generate new ones. The ability to change the color of the sketches is also not relevant because dragging the connectors is a geometrical transformation rather than a visual or attribute modification.

6. True or False: To create a Center Line Loft, the center line must intersect each profile and connect points from each profile together.

A. True

B. False

C. Only if the profiles are horizontal

D. Only when using sketches

To create a Center Line Loft in SolidWorks, the center line indeed must intersect each of the profiles to ensure a smooth transition between them. This connection is crucial because the loft feature requires a path to guide the shape from one profile to another. By having the center line connect corresponding points between the profiles, it allows for the proper shaping and design intent, creating a more controlled and predictable lofted surface. In designing lofts, the profiles can be in various orientations, and the center line's ability to connect these points is not limited to horizontal profiles or specific sketching conditions. This flexibility in profile positioning, while still requiring intersection and connectivity, reinforces the statement's accuracy.

7. True or False: You can select a planar face as the reference plane for mirroring a feature?

A. True

B. False

C. Only in assembly mode

D. Only for sketches

When mirroring a feature in SolidWorks, you can indeed select a planar face as the reference plane. Selecting a planar face allows for precise and accurate mirroring of features, as it serves as a clear geometric reference. This capability is fundamental in many design scenarios where symmetry or duplication of features across a plane is required. All planar faces, whether they are part of a part or assembly, can act as effective reference planes. This versatility enhances the modeling workflow, allowing designers to efficiently create mirrored features without needing to create additional reference planes. Utilizing existing planar faces helps in maintaining the design intent and ensuring accuracy in the mirrored features. In contrast, the other options imply limitations that do not exist in SolidWorks. For example, the assertion that mirroring is only possible in assembly mode or limited to sketches doesn't hold true, as the functionality is available in part modeling as well. Thus, the correct answer reflects the fundamental capabilities of SolidWorks concerning the mirroring of features relative to any planar face.

8. What is the "Extrude Cut" feature used for?

- A. To reshape a solid body into a different profile
- B. To remove material from a solid body using a 2D sketch**
- C. To create a hole by revolving a sketch profile
- D. To apply a surface feature over an existing body

The "Extrude Cut" feature in SolidWorks is specifically designed for removing material from a solid body by utilizing a 2D sketch. When a user creates a sketch on a face of a part and then applies the Extrude Cut feature, the sketch is extended in a specified direction to subtract material from the original shape, effectively creating voids or holes within the solid body. This capability is crucial in designing components that require precise cutouts, holes, or internal features, as it allows for efficient modifications to the geometry based on the sketch defined by the designer. The resulting cut can take various forms depending on the original sketch and the parameters set during the operation, offering flexibility in part design. Other options reference different features or methods in SolidWorks. For example, reshaping a solid body or creating holes through revolving sketches involves different operations, such as the Extrude Boss or Revolve features. Additionally, surface features refer to applying surfaces rather than removing material from a solid body, which defines a different functionality altogether. The specificity of the Extrude Cut to material removal distinguishes it clearly from these other processes.

9. What should you expect when changing the sheet scale in relation to detail views?

- A. The sheet scale does not affect detail views
- B. Detail views scale automatically with the sheet**
- C. Detail views only update if linked
- D. Only the primary view is affected

When changing the sheet scale in SolidWorks, it's important to understand how this affects detail views. Detail views are intended to provide a closer look at specific areas of a model, and they are inherently linked to the overall scale of the sheet they reside on. Therefore, when the sheet scale is adjusted, the detail views will scale automatically along with it to maintain their relative proportions and clarity in relation to the other views on that sheet. This automatic scaling ensures that the detail view remains accurate and useful, as it reflects the same scale as the primary view, avoiding any potential confusion that could arise from differing scales. This functionality is designed to improve the ease of interpretation for the end user, ensuring that all views are consistent and coherent. In contrast, the other options do not accurately depict the behavior of detail views in relation to sheet scale changes. For instance, asserting that the sheet scale does not affect detail views overlooks the fundamental linkage between views. Stating that detail views only update if linked doesn't adequately capture the automatic nature of the scaling process inherent in SolidWorks. Finally, suggesting that only the primary view is affected fails to recognize the interconnected nature of all views on a drawing sheet, including detail views.

10. Which command is used to change the scale of a view in SolidWorks?

- A. Modify View
- B. Edit Scale
- C. Scale Tool
- D. Properties**

The command used to change the scale of a view in SolidWorks is found within the "Properties" dialog box. When you access the properties of a drawing view, you will see options to adjust the scale among other things. This allows you to set a specific scale or adjust it to a predefined value, ensuring that your drawing appears at the correct size for presentation or documentation purposes. In contrast, while commands like "Modify View," "Edit Scale," and "Scale Tool" might suggest similar functionalities, they do not accurately reflect the way SolidWorks handles scale adjustments within drawing views. The integration of scale changes into the properties makes it a centralized and coherent method to manage view characteristics efficiently. Thus, utilizing the properties allows users to not only change the scale but also adjust several other attributes of the view simultaneously.

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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://solidworkssolidprofessor.examzify.com>

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

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