

# SolidWorks / SolidProfessor Practice Exam (Sample)

## Study Guide



**Everything you need from our exam experts!**

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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 type of tool is the "Smart Dimension" in SolidWorks?**
  - A. A tool for creating complex assemblies**
  - B. A tool for measuring distances in 3D models**
  - C. A tool for formatting documents**
  - D. A tool for adding dimensions to sketches**
  
- 2. What is one way to remove an existing sketch in SolidWorks?**
  - A. Select it and press the Delete key**
  - B. Select it and click Remove**
  - C. Right-click and choose Remove**
  - D. Select and press Backspace**
  
- 3. If there is an existing center-line in a sketch, what will SolidWorks do regarding the axis of revolution?**
  - A. Ignore the center-line**
  - B. Automatically pre-select it**
  - C. Ask for user confirmation**
  - D. Select only visible center-lines**
  
- 4. What feature allows you to create sections in a drawing for detail?**
  - A. Section View**
  - B. Detail View**
  - C. Break View**
  - D. 3D View**
  
- 5. How do you export a SolidWorks model to a PDF?**
  - A. Navigate to File and select Export As**
  - B. Go to File > Save As, select PDF from the file type dropdown**
  - C. Select File > Print and choose PDF as a printer**
  - D. Use the Export to PDF feature in the tool menu**

- 6. Which annotation would you select to label a hole in a drawing?**
- A. Hole Feature**
  - B. Dimension**
  - C. Hole Callout**
  - D. Text Box**
- 7. How is the Shortcut menu activated in SolidWorks?**
- A. Pressing the 'S' key**
  - B. Right-clicking**
  - C. Using the Function keys**
  - D. Clicking on the menu bar**
- 8. Which icon activates the Smart Dimension tool to add dimensions to a sketch?**
- A. The one with two parallel lines**
  - B. The one with a single line**
  - C. The one with a circle**
  - D. The one with a triangle**
- 9. What mode should SolidWorks be in to create two-dimensional geometry?**
- A. Assembly mode**
  - B. Editing Sketch mode**
  - C. Part mode**
  - D. Simulation mode**
- 10. What feature allows you to create holes that follow a specific pattern?**
- A. The "Hole Wizard" combined with "Pattern" features**
  - B. The "Array" feature only**
  - C. The "Rotate" feature in the sketch mode**
  - D. The "Mirror" feature in assembly mode**

## Answers

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

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## **Explanations**

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**1. What type of tool is the "Smart Dimension" in SolidWorks?**

- A. A tool for creating complex assemblies**
- B. A tool for measuring distances in 3D models**
- C. A tool for formatting documents**
- D. A tool for adding dimensions to sketches**

The "Smart Dimension" tool in SolidWorks is specifically designed for adding dimensions to sketches. This tool allows users to easily define the size and position of geometric entities within a 2D sketch. By clicking on entities, users can assign dimensions that control the distance, angle, and size of shapes, ensuring that the sketch is accurately represented. The Smart Dimension tool streamlines the process of dimensioning, making it intuitive and efficient, particularly for creating and modifying sketches in part design. In contrast to the other options, which focus on different functionalities, such as assembly creation, 3D measurement, or document formatting, the Smart Dimension tool is unequivocally centered on sketching, making it an essential function for anyone looking to create precise designs within SolidWorks.

**2. What is one way to remove an existing sketch in SolidWorks?**

- A. Select it and press the Delete key**
- B. Select it and click Remove**
- C. Right-click and choose Remove**
- D. Select and press Backspace**

One method to remove an existing sketch in SolidWorks is to select it and then press the Delete key. This action effectively deletes the sketch from the feature tree, allowing for modifications in the design without that particular sketch present. Using the Delete key is a straightforward and efficient way to manage sketches as it is a common function across many software applications. This method provides a quick removal process, especially when you are working within the context of modifying sketches or managing the feature tree directly. The other options could include alternative ways to manage sketches, but they either do not exist in the SolidWorks interface or aren't standard functions for deleting sketches. Thus, relying on the Delete key is the most widely recognized and accepted method for this task.

**3. If there is an existing center-line in a sketch, what will SolidWorks do regarding the axis of revolution?**

- A. Ignore the center-line**
- B. Automatically pre-select it**
- C. Ask for user confirmation**
- D. Select only visible center-lines**

When there is an existing center-line in a sketch, SolidWorks will automatically pre-select it as the axis of revolution when creating rotational features like Revolve Boss/Base or Revolve Cut. This behavior is designed to streamline the design process, as the software recognizes that the center-line is intended to serve as the axis around which the sketch entities will revolve, making it more intuitive for the user. Pre-selecting the center-line saves time and effort, as users do not have to manually indicate the axis each time they create a revolved feature. Instead, they can focus on defining the profile they want to revolve around that axis. This feature aligns with SolidWorks' user-friendly design principles, where the software tries to predict user intent and offers helpful defaults based on existing sketch entities. The other options do not correctly describe the behavior of SolidWorks when it encounters a center-line in a sketch during the creation of revolved features.

**4. What feature allows you to create sections in a drawing for detail?**

- A. Section View**
- B. Detail View**
- C. Break View**
- D. 3D View**

The feature that allows you to create sections in a drawing for detail is the Section View. This feature enables you to cut through a model to reveal internal components and features that are not visible from the exterior. By using a Section View, you can better illustrate how different parts of an assembly fit together and provide a clearer understanding of the internal structure. This view introduces a cutting plane that effectively slices through the model, presenting a cross-sectional perspective that highlights important details, such as holes, ribs, and other internal features. In engineering and design documentation, this is crucial for conveying information that might not be apparent from external views alone. The other options serve different purposes: the Detail View focuses on enlarging a specific area of a drawing for enhanced clarity, the Break View is used to shorten long parts without providing a cross-section, and the 3D View displays the model in a three-dimensional format without any sectioning.

## 5. How do you export a SolidWorks model to a PDF?

- A. Navigate to File and select Export As
- B. Go to File > Save As, select PDF from the file type dropdown**
- C. Select File > Print and choose PDF as a printer
- D. Use the Export to PDF feature in the tool menu

The correct method to export a SolidWorks model to a PDF is by navigating to File > Save As and selecting PDF from the file type dropdown. This process allows you to directly save the model as a PDF file, which is a common format for sharing and printing documents while preserving the layout and appearance of the original model. When you choose the Save As option, the dialog provides various file formats, including PDF, making it a straightforward and effective method for creating a portable document format version of your SolidWorks model. This way, you can retain the integrity of the design and ensure that others can view it without needing SolidWorks installed on their machines. Other methods mentioned, such as using the Export As function, might not directly support PDF as a format in all instances, and while selecting print options or dedicated features may sound feasible, they are not the standard technique for exporting a model as a PDF in SolidWorks. Using the Save As feature ensures you have a reliable and straightforward file conversion process.

## 6. Which annotation would you select to label a hole in a drawing?

- A. Hole Feature
- B. Dimension
- C. Hole Callout**
- D. Text Box

To label a hole in a drawing, using a hole callout is the most appropriate choice. A hole callout provides specific details about the hole, such as its diameter, depth, and any threading if applicable. This type of annotation consolidates relevant information in a concise manner, making it clear to anyone reading the drawing what the specifications of the hole are. A hole callout automatically pulls data from the hole feature in the 3D model, ensuring that the dimensions and characteristics are accurate and reflect any design changes. This integration streamlines the documentation process and enhances clarity in technical communication, which is crucial in engineering and manufacturing settings. In contrast to this, while dimensions can denote measurements, they do not inherently convey specific information about hole features like a hole callout does. A text box offers flexibility for custom notes but lacks the structured, standardized information provided by a callout. Lastly, a hole feature itself refers to the underlying design element in the 3D model rather than the annotation needed on a 2D drawing. Therefore, a hole callout uniquely serves the purpose of labeling holes effectively in technical drawings.

## 7. How is the Shortcut menu activated in SolidWorks?

- A. Pressing the 'S' key**
- B. Right-clicking**
- C. Using the Function keys**
- D. Clicking on the menu bar**

The Shortcut menu in SolidWorks is activated by pressing the 'S' key on the keyboard. This key provides a quick and efficient way to access frequently used commands and tools without having to navigate through the standard toolbars or menus. This feature streamlines the workflow, especially for users who rely on specific functions regularly, allowing for a faster and more productive experience within the software. In addition to the 'S' key, users can also access various commands using right-clicking, which brings up context-specific options based on the selected item. However, the most efficient and direct method to bring up the Shortcut menu is indeed the 'S' key. Other methods, such as using Function keys or clicking on the menu bar, also exist but serve different purposes and do not directly activate the Shortcut menu in the same way. Therefore, pressing the 'S' key is the most accurate representation of how to bring up the Shortcut menu in SolidWorks.

## 8. Which icon activates the Smart Dimension tool to add dimensions to a sketch?

- A. The one with two parallel lines**
- B. The one with a single line**
- C. The one with a circle**
- D. The one with a triangle**

The icon that activates the Smart Dimension tool to add dimensions to a sketch is represented by two parallel lines. This icon is an intuitive representation of the function, as it suggests the relationship of dimensions between two entities or features, analogous to how dimensions relate to parallel lines in a sketch. When you select this icon, it allows you to click on any sketch entity, such as lines, arcs, or circles, to define their dimensions accurately. The Smart Dimension tool automates the process of setting the appropriate dimension type and helps ensure that the sketch adheres to the defined measurements, which is critical for maintaining design intent and model accuracy. In contrast, the other icons represent different tools or features within SolidWorks. For instance, the single line might relate to line drawing tools, the circle to creating circular features, and the triangle could indicate a different function altogether, which does not involve dimensional annotations. Understanding these distinctions is crucial for effective sketching and modeling in SolidWorks.

**9. What mode should SolidWorks be in to create two-dimensional geometry?**

- A. Assembly mode
- B. Editing Sketch mode**
- C. Part mode
- D. Simulation mode

To create two-dimensional geometry in SolidWorks, the software must be in Editing Sketch mode. This mode is specifically designed for sketching and allows users to create and manipulate 2D shapes using various sketch tools such as lines, circles, and arcs. When in Editing Sketch mode, the workspace is set up to facilitate drawing and dimensioning, which are essential for creating 2D geometry that can later be used to define 3D features. In this mode, users can employ constraints and relations to control the geometry's behavior, ensuring that it meets design requirements. The focus on 2D construction makes it the ideal environment for initial design phases before transitioning to 3D modeling. In contrast, while Assembly mode is used for bringing together different parts in a model, and Part mode allows for the creation and modification of 3D objects, these modes are not specifically tailored for drawing 2D sketches. Simulation mode, on the other hand, is utilized for analyzing models and does not offer tools for creating geometric shapes. Thus, Editing Sketch mode is the most appropriate choice for creating two-dimensional geometry within SolidWorks.

**10. What feature allows you to create holes that follow a specific pattern?**

- A. The "Hole Wizard" combined with "Pattern" features**
- B. The "Array" feature only
- C. The "Rotate" feature in the sketch mode
- D. The "Mirror" feature in assembly mode

The feature that allows you to create holes that follow a specific pattern is the "Hole Wizard" combined with "Pattern" features. The Hole Wizard in SolidWorks is a specialized tool aimed at creating various types of holes, such as countersink, counterbore, and threaded holes, with ease and precision. It provides options to define specifications like diameter, depth, and type of hole, making it an essential tool for designing components that require precise hole placement. Once holes are created using the Hole Wizard, users can employ pattern features—such as linear pattern, circular pattern, or fill pattern—to replicate these holes in specific arrangements. This combined approach allows for quick and efficient design modifications, especially in parts that require multiple holes organized in an array or following a geometrically defined pattern. Utilizing both the Hole Wizard and pattern features streamlines the process and reduces the need for repetitive tasks, ensuring consistency and saving time in the design workflow. Other options, such as the Array feature or features like Rotate and Mirror, do not focus specifically on creating holes or are not directly applicable to forming patterns in the context of hole creation.

## 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](mailto: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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