

SOLID EDGE CERTIFICATION 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 is the primary function of 'Sheet Metal Commands' in Solid Edge?**
 - A. To design plastic components
 - B. To assist in surface modeling
 - C. To create and modify sheet metal configurations
 - D. To convert models into 2D drawings
- 2. What does the term 'derived part' refer to in Solid Edge?**
 - A. A part created purely for aesthetic purposes
 - B. A part that is based on an existing part or assembly, allowing for design reuse
 - C. A component used to define materials
 - D. A detailed view of individual components in isolation
- 3. What is a 'bending sequence' in the sheet metal environment of Solid Edge?**
 - A. A method to calculate material thickness
 - B. A simulation of the folding process during manufacturing
 - C. An indication of the order of bends required to manufacture a part
 - D. A feature for creating complex sheet metal shapes
- 4. What command would be used to create a profile that cuts through an existing object?**
 - A. Extrude Cut
 - B. Revolve Cut
 - C. Circle Cut
 - D. Segment Cut
- 5. Which option allows the sketch edge to be created at the angle of the y-axis?**
 - A. Horizontal
 - B. Diagonal
 - C. Vertical
 - D. Orthogonal

6. To change the color of an object while in a part file, which tab should be accessed?
- A. Color
 - B. View
 - C. Design
 - D. Properties
7. To use the 3 sketched cross sections (2 rectangles and a circle) to create the solid body as shown, use the _____ command.
- A. Revolve
 - B. Loft
 - C. Extrude
 - D. Sweep
8. How do you suppress one of the features in your model?
- A. Right-click the feature in the path model and click hide
 - B. Select the feature and press delete
 - C. Change the feature's properties to invisible
 - D. Drag the feature to the suppress folder
9. What does it indicate when drawing views have bold gray corners in a draft environment?
- A. A component has been changed outside the assembly and must be updated
 - B. A new design template is applied to the drawing
 - C. The drawing view is currently being edited
 - D. The view is locked for user modifications
10. In what scenario would direct editing be essential in Solid Edge?
- A. When designing from scratch
 - B. When modifying imported models without parametric history
 - C. When creating new parametric features
 - D. When generating assembly drawings

Answers

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

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Explanations

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1. What is the primary function of 'Sheet Metal Commands' in Solid Edge?

- A. To design plastic components
- B. To assist in surface modeling
- C. To create and modify sheet metal configurations**
- D. To convert models into 2D drawings

The primary function of 'Sheet Metal Commands' in Solid Edge is to create and modify sheet metal configurations. These commands are specifically designed to facilitate the design process for sheet metal parts, allowing users to apply features such as bends, flanges, and hems that are characteristic of sheet metal fabrication. This functionality is crucial for accurately modeling components that will be manufactured from flat sheets of material, as it allows for the application of design rules and supports the creation of flat patterns for production. In the realm of design, this command set includes tools that help streamline the workflow from initial design to manufacturing specifications, ensuring that the configurations are not only functional but also manufacturable. This enhances productivity and reduces the potential for errors in the sheet metal fabrication process, making it an essential tool for engineers and designers working in industries where sheet metal is commonly used.

2. What does the term 'derived part' refer to in Solid Edge?

- A. A part created purely for aesthetic purposes
- B. A part that is based on an existing part or assembly, allowing for design reuse**
- C. A component used to define materials
- D. A detailed view of individual components in isolation

The term 'derived part' in Solid Edge specifically refers to a part that is based on an existing part or assembly, facilitating design reuse. This feature allows users to create new parts that are linked to a parent part, inheriting certain properties such as geometry, features, and parameters. The derived part maintains an associative link to the parent, enabling updates in the original to be reflected in the derived part without the need for duplicating effort or creating entirely new designs from scratch. This capability enhances efficiency and consistency within the design process, especially in instances where similar components need to be designed based on a common template. The other options do not accurately define the concept of a derived part. For example, a component created purely for aesthetic purposes does not capture the essence of design reuse or associativity. Similarly, a component used to define materials or a detailed view of individual components in isolation do not represent the fundamental characteristic of a derived part, which is fundamentally about leveraging existing designs to facilitate new creations.

3. What is a 'bending sequence' in the sheet metal environment of Solid Edge?

- A. A method to calculate material thickness
- B. A simulation of the folding process during manufacturing
- C. An indication of the order of bends required to manufacture a part**
- D. A feature for creating complex sheet metal shapes

A 'bending sequence' in the sheet metal environment of Solid Edge is defined as an indication of the order of bends required to manufacture a part. This sequence is critical in sheet metal fabrication because the order in which bends are made can greatly affect the final shape, the ease of manufacturing, and the integrity of the material being worked on. In manufacturing, certain bends must be performed before others to ensure that the component can be formed without interference between tools or equipment. Following the correct bending sequence helps prevent potential issues such as warping or the inability to fit parts together properly. By understanding and configuring the bending sequence in Solid Edge, designers and engineers can optimize the production process and avoid complications during manufacturing. Other options do relate to aspects of sheet metal design or manufacturing, but they do not align with the specific definition of a bending sequence. For instance, calculating material thickness, simulating folding processes, or creating complex shapes are essential tasks but are distinct from specifying the order of bends necessary for assembling a sheet metal part.

4. What command would be used to create a profile that cuts through an existing object?

- A. Extrude Cut**
- B. Revolve Cut
- C. Circle Cut
- D. Segment Cut

The command that would be used to create a profile that cuts through an existing object is the Extrude Cut command. This command allows you to take a 2D sketch profile and extend it into the 3D model, effectively removing material from the existing object based on the geometry of the sketch. This is commonly applied to create holes, slots, or other shapes that penetrate through the object. The Extrude Cut operation is widely used because it provides flexibility in designing various cut shapes and sizes, making it a fundamental tool in parametric modeling within Solid Edge. By defining the profile and the direction of the cut, users can achieve precise and complex modifications to 3D parts, enhancing the overall design process. In contrast, while other options may sound relevant, they do not specifically address the cutting aspect of modifying an existing 3D geometry in the same straightforward manner as the Extrude Cut command does. Revolve Cut, for instance, is typically used for creating cut features as it revolves a profile around a central axis, which is appropriate in specific design contexts but not as versatile for simple cuts through existing parts. Circle Cut and Segment Cut might not even exist as defined commands in Solid Edge, making them less applicable in this context.

5. Which option allows the sketch edge to be created at the angle of the y-axis?

- A. Horizontal
- B. Diagonal
- C. Vertical**
- D. Orthogonal

Creating a sketch edge at an angle to the y-axis is accomplished by using a vertical alignment since the y-axis is oriented vertically in a standard Cartesian plane. When a sketch edge is set to be vertical, it runs parallel to the y-axis, ensuring that any line created in this orientation will align directly with the vertical axis of the design space. This is crucial in drafting and modeling as it allows for precise control over the geometry relative to defined axes, ensuring that components are accurately positioned. The other options do not align with the requirement to create a sketch edge specifically at the angle of the y-axis. Horizontal aligns along the x-axis, diagonal would create an edge at some angle between the horizontal and vertical axes, and orthogonal implies a relationship of right angles, which does not inherently create a vertical edge by itself. Understanding these alignments allows designers to control their sketch geometry more effectively within Solid Edge.

6. To change the color of an object while in a part file, which tab should be accessed?

- A. Color
- B. View**
- C. Design
- D. Properties

To change the color of an object while in a part file, the correct tab to access is the View tab. This functionality allows users to alter the visual representation of their models without modifying the underlying geometry or material properties. The View tab typically includes options for display settings, including colors, which can enhance the visibility and aesthetics of different parts in a model. Accessing the View tab provides tools for managing how objects appear on screen, which is particularly useful in complex assemblies where distinguishing between parts is necessary. Changing colors can also aid in presentations or when preparing for manufacturing, as various colors can signify different materials or components.

7. To use the 3 sketched cross sections (2 rectangles and a circle) to create the solid body as shown, use the _____ command.

- A. Revolve
- B. Loft**
- C. Extrude
- D. Sweep

The Loft command is the appropriate choice for creating a solid body from multiple sketched cross sections, especially when those cross sections are different in shape or size, such as in the case of two rectangles and a circle. Lofting allows for smooth transitions between these profiles, creating a solid that connects them in a 3D space. This is particularly useful when the geometries do not share a common profile but need to be integrated into a continuous form, showcasing the ability to generate complex shapes that align with the specified cross sections. In contrast, the other commands serve different purposes: the Revolve command is primarily used for creating solids by rotating a profile around an axis, which works well for symmetrical shapes like cylinders. The Extrude command extends a 2D shape along a straight line to create a solid, typically useful for uniform thickness scenarios, while the Sweep command follows a defined path with a profile, generating solids along a specified trajectory. Each of these alternatives has its specific application, but they do not effectively combine multiple diverse profiles in the manner that the Loft command does.

8. How do you suppress one of the features in your model?

- A. Right-click the feature in the path model and click hide**
- B. Select the feature and press delete
- C. Change the feature's properties to invisible
- D. Drag the feature to the suppress folder

To suppress a feature in your model, the correct method involves right-clicking the feature in the path model and selecting the hide option. This action effectively prevents the feature from being displayed and taken into account in calculations or visualizations within the design, allowing for a more streamlined model without permanently deleting the feature. This method is advantageous for verifying design changes without permanently altering the model or losing feature information, enabling designers to manage complex assemblies or features easily. The suppression of a feature maintains the integrity of the overall model while also offering flexibility during the design process. Other options do not serve the purpose of suppressing a feature accurately. For example, deleting a feature removes it entirely from the model, which is not reversible unless you undo the action. Changing the feature's properties to invisible might provide some functionality, but it does not truly suppress it as it could still affect other aspects of the model. Dragging the feature to a suppress folder is typically not a standard method in many CAD software options and might not function as expected in Solid Edge. Therefore, the best practice for suppressing features in the context of Solid Edge involves the right-click and hide method.

9. What does it indicate when drawing views have bold gray corners in a draft environment?

- A. A component has been changed outside the assembly and must be updated
- B. A new design template is applied to the drawing
- C. The drawing view is currently being edited
- D. The view is locked for user modifications

When drawing views display bold gray corners in a draft environment, it signifies that a component has been changed outside of the assembly and requires updating. This visual cue is important as it alerts the designer to discrepancies between the drawing and the current state of the model or assembly. When components undergo modifications, the associated drawing views may not automatically reflect these changes until they are updated, ensuring that the designer maintains accurate documentation. Recognizing bold gray corners helps streamline the workflow by prompting designers to refresh their drawings, thus ensuring that any alterations to the component are correctly represented in the drafting environment. The other options depict different scenarios: applying a new design template may not necessarily affect view borders; editing a drawing view typically would not be denoted by bold gray corners; and locking a view for user modifications usually results in a different visual representation, commonly indicated by alternative symbols or changes in color.

10. In what scenario would direct editing be essential in Solid Edge?

- A. When designing from scratch
- B. When modifying imported models without parametric history
- C. When creating new parametric features
- D. When generating assembly drawings

Direct editing is particularly essential when modifying imported models that do not have a parametric history. In scenarios where models are brought into Solid Edge from other CAD systems, it's common that they may lack the parametric data that would typically allow for easy modifications through feature history. Direct editing tools in Solid Edge provide a way to make changes to these geometric entities without the need for a parametric relationship. This approach allows users to perform operations such as pushing, pulling, or twisting surfaces or solid bodies, which helps in fitting the imported geometry into the desired design context. It offers flexibility in adapting existing models to new requirements, especially when starting from non-native files that do not provide the same level of control as fully parametric designs. In contrast, the other scenarios mentioned do not typically necessitate direct editing. For instance, designing from scratch often utilizes parametric features to establish relationships and constraints from the outset. Creating new parametric features involves establishing a design history that can be easily manipulated without the need for direct editing techniques. Lastly, generating assembly drawings does not require changes to the geometry itself, as it is primarily focused on documentation rather than modification of the models.

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://solidedge.examzify.com>

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

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