

Autodesk Certified User (ACU) Practice Test (Sample)

Study Guide



Everything you need from our exam experts!

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SAMPLE

Questions

- 1. What type of environment facilitates the creation of T-Spline bodies?**
 - A. Assembly Environment**
 - B. Drafting Environment**
 - C. Contextual Environment**
 - D. Simulation Environment**
- 2. What do you call the feature that defines the radius of curvature of an edge within a design?**
 - A. Drawing Border**
 - B. Fillet**
 - C. Feature**
 - D. Drawing Notes**
- 3. What is the primary role of a Split Body tool in CAD applications?**
 - A. To modify a profile's dimensions**
 - B. To create complex surfaces**
 - C. To create new bodies from selected features**
 - D. To enhance visibility of parts**
- 4. What is a Title Block in a drawing?**
 - A. A section where design errors are noted**
 - B. A block of text providing information on a file**
 - C. A graphical representation of the design**
 - D. A summary of tool usage within a project**
- 5. Which tool allows the modification of selected geometry using commands like offset and fillet?**
 - A. Press Pull**
 - B. Mirror**
 - C. Motion Studies**
 - D. Dimension Tool**

- 6. What does a Thread represent in regard to design elements?**
- A. The surface finish of a part**
 - B. The ridges on screws or in holes**
 - C. The degree of curvature in a profile**
 - D. The strength of a material**
- 7. What do Snap Points assist with in a design application?**
- A. Creating measurements automatically**
 - B. Identifying edges or faces for alignment and measurement**
 - C. Automating the creation of sketches**
 - D. Enhancing the visibility of components**
- 8. In CAD techniques, which term best describes a sketch's capability to become a 3D object?**
- A. Fillet**
 - B. Engineering Drawing**
 - C. Extrusion**
 - D. Drive Joint**
- 9. Which workspace allows the use of 2D sketch geometry to create solid, surface, or T-Spline models?**
- A. Design Workspace**
 - B. Data Panel**
 - C. Degrees of Freedom**
 - D. Detail View**
- 10. What is the term for a round edge on a part or face within a design?**
- A. Fillet**
 - B. Drive Joint**
 - C. Extend**
 - D. Exploded View**

Answers

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

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Explanations

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1. What type of environment facilitates the creation of T-Spline bodies?

- A. Assembly Environment**
- B. Drafting Environment**
- C. Contextual Environment**
- D. Simulation Environment**

A Contextual Environment is the correct choice for facilitating the creation of T-Spline bodies. In Autodesk software, particularly in tools like Fusion 360, the Contextual Environment provides a specialized workspace where designers can utilize advanced modeling features specific to T-Splines. T-Splines are a type of surface modeling technique that allows for the creation of complex, organic shapes that are both flexible and controllable. The Contextual Environment supports the manipulation of these surfaces by providing the necessary tools and options that cater specifically to T-Spline geometry. This could include features such as editing control points and adding or removing edges to refine the surface geometry. In contrast, the other environments listed do not offer the specific capabilities required for T-Spline modeling. For instance, the Assembly Environment is focused on the arrangement and interaction of multiple components within a larger assembly, making it unsuitable for standalone surface creation tasks. The Drafting Environment is primarily aimed at creating 2D drawings and annotations, lacking the 3D modeling functionality required for T-Splines. Lastly, the Simulation Environment deals with analyzing designs under various conditions, rather than creating or modifying 3D models. Thus, the Contextual Environment is the most appropriate setting for working with T-Spline bodies,

2. What do you call the feature that defines the radius of curvature of an edge within a design?

- A. Drawing Border**
- B. Fillet**
- C. Feature**
- D. Drawing Notes**

The radius of curvature of an edge in a design is specified by a fillet. A fillet is a geometric feature used in design modeling to create a smooth transition between two intersecting surfaces or edges. This feature helps to eliminate sharp corners, providing a more aesthetically pleasing appearance and enhancing the strength and durability of the component, as it reduces stress concentrations that can occur at sharp edges. In contrast, other options like drawing border and drawing notes refer to elements used in documentation and layout but do not apply to the physical characteristics of edges in the design process. The term "feature" is too broad and does not specifically indicate the process of defining the radius of curvature. Thus, the term "fillet" accurately captures the intent and functionality of the curvature in design.

3. What is the primary role of a Split Body tool in CAD applications?

- A. To modify a profile's dimensions
- B. To create complex surfaces
- C. To create new bodies from selected features**
- D. To enhance visibility of parts

The primary role of the Split Body tool in CAD applications is to create new bodies from selected features. This tool allows users to divide an existing solid or surface body into two or more separate bodies based on defined cutting parameters. This is particularly useful when designers want to manipulate distinct portions of a model independently, whether for further modifications, assemblies, or for producing parts that require individual processing. Creating new bodies via the Split Body tool enables design flexibility, enhances organization within assemblies, and simplifies the workflow by allowing targeted editing or applying features to those split bodies without affecting the original geometry. For example, after splitting a body, each resultant body can be treated as an individual entity for purposes such as applying different materials, creating assemblies, or generating geometry for manufacturing. The other choices relate to different functionalities in CAD software. Modifying a profile's dimensions pertains more to dimensioning tools or editing features rather than splitting geometry. Creating complex surfaces typically involves surface modeling techniques rather than splitting existing solids. Enhancing visibility of parts does not align with the function of the Split Body tool, as it involves manipulation of the model's visibility settings rather than the structural integrity or creation of new geometry.

4. What is a Title Block in a drawing?

- A. A section where design errors are noted
- B. A block of text providing information on a file**
- C. A graphical representation of the design
- D. A summary of tool usage within a project

A title block is an essential component found on technical drawings, architectural plans, and engineering schematics. It provides critical information about the project, such as the title of the drawing, the name of the designer, the date of creation, the scale of the drawing, and relevant project details. This information is crucial for anyone reviewing the drawing, as it helps identify the context, purpose, and authorship of the document. Choosing a section where design errors are noted, a graphical representation of the design, or a summary of tool usage would not accurately reflect the fundamental purpose of a title block. Instead, the title block serves as a standardized means of conveying vital information that is essential for understanding and managing the drawing effectively.

5. Which tool allows the modification of selected geometry using commands like offset and fillet?

- A. Press Pull**
- B. Mirror**
- C. Motion Studies**
- D. Dimension Tool**

The correct choice highlights the Press Pull tool, which is specifically designed to modify geometry in CAD software effectively. This tool allows users to extend or shrink surfaces and solids by using various commands including offset and fillet. The offset command enables users to create parallel copies of selected edges or faces, while the fillet command rounds off corners, enhancing the design's aesthetic and functional quality. By allowing the user to manipulate existing geometry directly, the Press Pull tool streamlines the design process and maintains precision. In contrast, the other options serve different functions that do not involve the direct modification of geometry through offset or fillet commands. For instance, Mirror replicates selected objects across a defined axis but does not alter their shape. Motion Studies analyze the movement of objects within the model rather than altering their geometry. The Dimension Tool is used for annotating the drawing and measuring distances, not for modifying the geometry itself. Overall, Press Pull is the most appropriate tool for the functions described in the question.

6. What does a Thread represent in regard to design elements?

- A. The surface finish of a part**
- B. The ridges on screws or in holes**
- C. The degree of curvature in a profile**
- D. The strength of a material**

A Thread in design elements specifically refers to the ridges or helical cuts found on screws and in holes. It is a crucial feature in mechanical engineering, as threads enable the fastening of components together, allowing for a secure connection. This characteristic allows screws to grip materials effectively, facilitating the assembly of various parts. While surface finish relates to the texture and quality of a part's exterior, it does not encompass what a thread is. Similarly, the degree of curvature in a profile pertains to the shape of a design feature but does not address the specific metric of a thread. Finally, the strength of a material pertains to its ability to withstand forces without failure, but again, this concept is distinct and separate from what defines a thread in a design context. Thus, the identification of threads as the ridges on screws or in holes is accurate and directly tied to their functional role in design.

7. What do Snap Points assist with in a design application?

- A. Creating measurements automatically
- B. Identifying edges or faces for alignment and measurement**
- C. Automating the creation of sketches
- D. Enhancing the visibility of components

Snap Points are essential tools in design applications that help users accurately position and align objects within their workspace. When utilizing snap points, you can quickly identify specific edges, faces, or vertices of existing objects, which facilitates precise alignment and measurement of new elements in relation to those objects. This precision is crucial in ensuring that components are correctly positioned, contributing to the overall integrity and functionality of the design. The functionality of snap points allows designers to focus on accuracy and efficiency, reducing the chances of error that can occur when positioning elements manually. For instance, when drawing or placing components, snap points provide visual cues that indicate where a new object can align or connect with existing elements, enhancing the workflow significantly. Other options may touch on various aspects of design, such as automation of measurements or sketch creation, but they do not specifically address the primary role of snap points, which is to assist with alignment and measurement through the recognition of physical characteristics in the design space.

8. In CAD techniques, which term best describes a sketch's capability to become a 3D object?

- A. Fillet
- B. Engineering Drawing
- C. Extrusion**
- D. Drive Joint

The term that best describes a sketch's capability to become a 3D object is "extrusion." In CAD applications, extrusion is a process that takes a 2D shape—created as a sketch—and extends it along a perpendicular axis to create a three-dimensional object. This technique effectively adds depth to the two-dimensional sketch, allowing it to take on solid form and volume. When a designer creates a profile in 2D, they can utilize the extrusion command to specify how far they want that profile to extend in the third dimension, seamlessly transforming the sketch into a fully-fledged 3D object. This fundamental aspect of solid modeling is crucial in various fields, such as manufacturing and product design. The other terms mentioned relate to different aspects of CAD design: "fillet" involves rounding the edges between two surfaces or lines; "engineering drawing" refers to a technical representation of an object, typically in 2D; and "drive joint" does not directly correspond to the process of converting a sketch to a 3D object. Thus, extrusion is the most relevant term in this context.

9. Which workspace allows the use of 2D sketch geometry to create solid, surface, or T-Spline models?

- A. Design Workspace**
- B. Data Panel**
- C. Degrees of Freedom**
- D. Detail View**

The Design Workspace is where users can effectively utilize 2D sketch geometry to create solid, surface, or T-Spline models. It provides a comprehensive environment for drafting and modeling, integrating tools that facilitate the transformation of sketches into three-dimensional forms. In this workspace, the design process is streamlined, allowing for the manipulation of 2D sketches to build complex geometric representations. The other options do not pertain directly to the creation of 3D models from 2D sketches. The Data Panel is primarily used for file management and does not directly facilitate modeling tasks. Degrees of Freedom relates to constraints and how objects move relative to one another, not specifically to model creation. Detail View refers to a method of enlarging or focusing on specific aspects of a drawing rather than the overall design process.

10. What is the term for a round edge on a part or face within a design?

- A. Fillet**
- B. Drive Joint**
- C. Extend**
- D. Exploded View**

The term for a round edge on a part or face within a design is known as a fillet. A fillet is a significant feature in design and engineering, used to create a smooth transition between two intersecting surfaces. It helps in reducing stress concentrations, improving aesthetics, and allowing easier assembly and manufacturing processes. By rounding the edges, fillets can help prevent sharp edges that may be prone to wear, chipping, or injuries while handling the object. In contrast, other terms listed such as drive joint, extend, and exploded view refer to different concepts in design. A drive joint relates to a connection mechanism in machinery, extend refers to lengthening or increasing the size or reach of an object, and an exploded view is a technique used in technical drawings to show the parts of an assembly spread out but still related in one view. None of these terms involve the specific function of rounding corners like a fillet does.