

# Autodesk Certifications Autodesk Certified User certification in Fusion 360 Test PDF

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## Question 1

Which command in Fusion 360 is used to create joints between components in an assembly?

**Options:**

- A. Assemble
- B. Combine
- C. Extrude
- D. Revolve

**Answer: A**

**Explanation:**

The "Assemble" command in Fusion 360 is specifically used to create joints between components in an assembly, allowing users to position and connect parts accurately. This feature is essential for creating complex assemblies and ensuring proper fit and function of the final design.

## Question 2

Which of the following is a feature of direct modeling in Fusion 360?

**Options:**

- A. Parametric modeling
- B. History-based modeling

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C. Real-time collaboration

D. Geometry editing

**Answer: D**

**Explanation:**

Direct modeling allows users to edit geometry directly without the need for a history tree or parametric constraints. This feature provides flexibility and ease of use when making design changes.

### Question 3

Which tool in Fusion 360 is used to create 2D sketches?

**Options:**

A. Extrude

B. Sketch

C. Revolve

D. Fillet

**Answer: B**

**Explanation:**

The Sketch tool in Fusion 360 is specifically used to create 2D sketches before extruding or revolving them into 3D objects.

### Question 4

Which advanced sketching technique in Fusion 360 allows you to create a pattern of an existing sketch geometry?

**Options:**

A. Offset

B. Mirror

C. Circular Pattern

D. Rectangular Pattern

**Answer: C**

**Explanation:**

The Circular Pattern tool in Fusion 360 allows you to create a circular pattern of an existing sketch geometry around a center point. It is a powerful tool for creating symmetrical designs and repeating patterns in your sketches.

### Question 5

Which feature in Fusion 360 allows users to create models based on parameters and design intent?

**Options:**

- A. Sculpting
- B. Assemblies
- C. Parametric modeling
- D. Mesh modeling

**Answer: C**

**Explanation:**

Parametric modeling in Fusion 360 enables users to create designs that can easily be modified by changing parameters, ensuring design intent is maintained throughout the design process.

### Question 6

When creating components in assemblies in Fusion 360, what is the purpose of using constraints?

**Options:**

- A. To add colors to components
- B. To define the position and orientation of components in relation to each other
- C. To delete unnecessary components
- D. To save the components as different file types

**Answer: B**

**Explanation:**

Constraints are used to define the spatial relationships between components in an assembly, ensuring they are positioned and oriented correctly relative to each other.

### Question 7

What is the purpose of constraints in Fusion 360?

**Options:**

- A. To limit the movement of components
- B. To increase the file size
- C. To add unnecessary complexity

D. To make the design process faster

**Answer: A**

**Explanation:**

Constraints in Fusion 360 are used to restrict the movement of components, ensuring that they behave as intended in the design. This helps in creating accurate models and assemblies.

### Question 8

When creating a sketch in Fusion 360, what is the purpose of the Line tool?

**Options:**

- A. To create circles
- B. To create straight lines
- C. To add text
- D. To apply colors

**Answer: B**

**Explanation:**

The Line tool in Fusion 360 is used to draw straight lines when creating sketches. It allows users to create precise shapes and designs by connecting points with straight lines.

### Question 9

What type of sketch constraint is used to ensure lines are parallel in Fusion 360?

**Options:**

- A. Horizontal
- B. Vertical
- C. Tangent
- D. Parallel

**Answer: D**

**Explanation:**

The Parallel constraint in Fusion 360 is used to ensure that lines or edges in a sketch remain parallel to each other, maintaining consistency and precision in the design.

### Question 10

How can you create a sketch in Autodesk Inventor?

**Options:**

- A. Press Ctrl + S on your keyboard
- B. Go to the "Sketch" tab and select "Create Sketch"
- C. Right-click on the workspace and choose "Start Sketch"
- D. Click on the "3D Model" tab

**Answer: B**

**Explanation:**

To create a sketch in Autodesk Inventor, you need to go to the "Sketch" tab and select "Create Sketch" to begin drawing your design on the workspace. This is the correct method for initiating a new sketch in the software.

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