

Autodesk Certified User Exam Objectives

Inventor



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Target Audience

The Autodesk Certified User (ACU) Inventor exam is designed for an individual who has a foundational level of proficiency and hands-on experience with Inventor (approximately 150 hours), has proven competency at an industry entry-level, and is ready to enter the job market.

Candidate Description

A successful candidate for this certification is familiar with Inventor features and capabilities, as well as relevant workflows, processes, and project objectives. They are competent using the core tools in the Inventor Design, Render, and Drawing workspaces. They can perform basic Inventor tasks related to component and assembly modeling with limited assistance from peers, product documentation, and support services. They can answer routine conceptual and applied questions about how to use Inventor without referring basic questions to others. Additionally, they can comprehend a technical drawing and have a basic concept of drafting standards.

Prerequisites

It is expected that all candidates will have a general understanding of:

- Demonstrate proficiency with basic computer skills and digital literacy
- Apply correct industry standard terminology
- Understand the concepts associated with general technical drawing and design
- Access online and written reference materials including the help screens within Inventor
- Understand application options and document settings
- Know how to use the ribbon

Certified User

Objective Domain

Some of the topics and features of the software that may be covered in the exam are listed below each objective.

Note: Within the context of this exam series, all references to “create, select, manage, etc.” indicate “know how to create, select, manage,

etc.”

1. Workspace and Navigation

1.1 Get started (understand/apply)

1.1.a Describe Inventor file formats –IPT, IAM, IDW, & IPN

i. Use templates and set units within a file

1.1.b Utilize project files (IPJ)

1.1.c Navigate within the graphics window

i. Orbit, Zoom, Pan, and Look At

ii. Slice Graphics

iii. ViewCube - Change the viewpoint, Set Current View as Home, and Reset Front options



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- 1.2 Navigate the workspace and environment (understand/apply)
 - 1.2.a Use the Model Browser (understand/analyze)
 - i. Find components (sketches, work planes, joints, etc.) in the Model Browser
 - ii. Identify features in the Model Browser (extrusion, circular patterns, holes, etc.)
 - iii. Suppress feature
 - iv. Toggle visibility
 - v. Rename and/or reorder model browser elements (features, sketches, etc.)
 - vi. Understand the functionality of the end of part marker
 - 1.2.b Use selection options (WN: no items for this objective)
 - i. Select any or all objects enclosed or crossed by a window or freeform shape
 - ii. Select objects in the Browser (multi-select)
 - 1.2.c Adjust the visual properties of a model
 - i. Adjust the visual style (wireframe vs shading)
 - ii. Apply materials and appearance overrides
 - iii. View models as a section
 - iv. Access iProperties, such as surface area, mass, volume
 - 1.2.d Use the Inspect tools
 - i. Use Measure tool
 - ii. Use Region Properties to find the perimeter and area of sketch geometry
- 1.3 Create and use work planes and axes
 - 1.3.a Create and use work planes
 - i. Offset, Midplane, Tangent
 - 1.3.b Create and use work axes
 - i. Axis Through a Revolved Face or Feature
 - ii. Normal to Plane Through Point

2. Sketch

- 2.1 Create and modify a sketch
 - 2.1. Create a sketch on an origin plane or planar face
 - a. Select an appropriate sketch tool(s)
 - 2.1. i. Rectangle, circles, line, arc, polygon, etc.
 - 2.1.c Identify sketch element types and their functions
 - i. Construction geometry, line type, centerline type
 - 2.1.d Project geometry from an existing body onto a sketch
 - i. Project geometry, Project cut edges
 - 2.1.e Edit a sketch
 - i. Move, Copy, Trim, Extend, Offset, Mirror
 - ii. Insert text into an active sketch
 - 2.2 Apply sketch constraints and dimensions
 - 2.2.a Determine which sketch constraints to apply
 - i. Add and remove constraints
 - ii. Understand auto-constraints
 - 2.2.b Add and edit dimensions to sketch geometry
 - i. Create a fully constrained drawing
 - 2.2.c Use design parameters to create a parametric model



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3. Model

3.1 Create and modify sketched features

- 3.1.a Create a 3D feature from 2D geometry
 - i. Features: Extrude, Revolve, Sweep, Loft
 - ii. Boolean Operations: Join, Cut, New Solid, Intersect

3.1.b Modify a sketched feature

3.2 Create and modify placed features

- 3.2.a Create and modify placed features
 - i. Fillet, Chamfer, Shell
 - ii. Delete a feature

3.2.b Create hole features

- i. Counterbore, Countersink, Spot Face

3.2.c Create a pattern of features

- i. Rectangular, Circular

4. Assemble

4. Place components into assemblies

1 Create relationships

4.2.a Place constraints to components

- i. Mate, Insert, Angle, Tangent, Offsets
- ii. Ground the base component of an assembly

4.2.b Place joints to components

- i. Rigid, Rotational

4.3 Analyze components

4.3. Determine the degrees of freedom of a component

a Manipulate components within an assembly

4.3. Check for interference between components

5. Document

5.1 Create drawings

5.1.a Work with drawing sheets

- i. Insert a new title block
- ii. Edit the size of the sheet

5.1.b Place and edit drawing views

- i. Base, section, detail, break, and projected views
- ii. Create a drawing view based on a part, assembly, or presentation file
- iii. Set the scale, visual style, and label visibility

5.2 Apply dimensions and annotations

5.2. Add and edit dimensions

a Add and edit annotations

5.2.

b



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