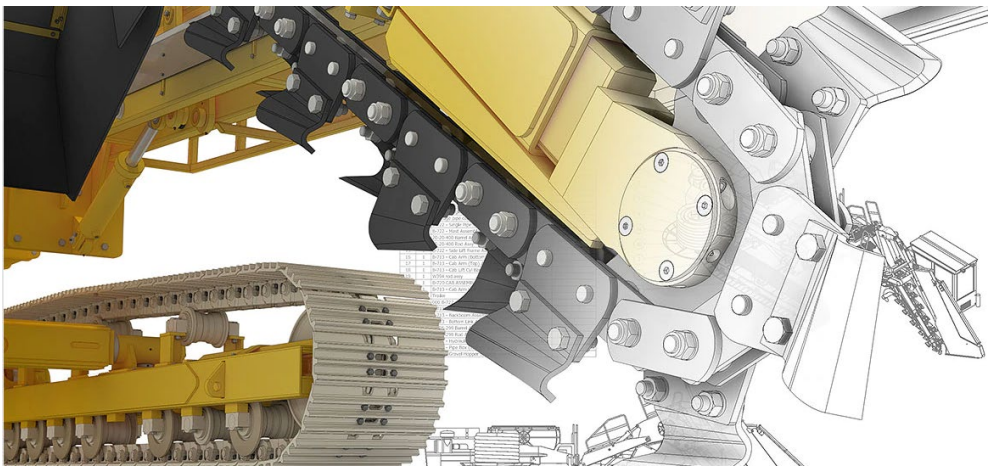
Inventor Certified User exam

Exam guide

Armada is an Autodesk Certification Centre offering exams that lead to industry-recognised qualifications.

For Inventor, we offer an Autodesk Certified User (ACU) exam



Exam summary and preparation

Entry level exam to validate your knowledge of fundamental concepts and procedures in Inventor.

Recognised by industry as proof of competency using Inventor.

Qualification: Autodesk Certified User (ACU) in Inventor.

Sat online, from your place of work or home.

Length of exam: 50 minutes.

Requirements

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

- Relevant workflows, processes and project objectives
- Inventor's user interface and ribbon
- The core tools in the Inventor's Design, Render and Drawing workspaces
- Performing basic Inventor tasks related to component and assembly modelling
- Application options and document settings
- Product documentation, online and written reference materials and the help screens within Inventor
- Correct industry standard terminology
- General concepts associated with technical drawing, drafting and design

For a list of the topics and features you're likely to be tested in, see over.

Recommended preparation

- Attend *Inventor Essentials* training.
- 150 hours' post-course, hands-on experience using Inventor.

Certificate and benefits

Successful candidates receive:

- An e-certificate (PDF) from Autodesk confirming your Inventor-certified status.
- An official Autodesk-Certified badge that you can use to market your skills, e.g. on your business cards, in your email signature, on your website, etc.

Practice test

A practice test is available that reflects the questions you're likely to be asked in your Inventor ACU exam.

Further info, dates, times, price

See armada.co.uk/exams/inventor.

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AUTODESK Certified User

Exam Outline

Workspace and Navigation

Get started (understand/apply)

Describe Inventor file formats –IPT, IAM, IDW, & IPN
- Use templates and set units within a file

Utilize project files (IPJ)

Navigate within the graphics window
- Orbit, Zoom, Pan and Look At
- Slice Graphics
- ViewCube – Change the viewpoint, Set Current View as Home and Reset Front options

Navigate the workspace and environment (understand/apply)

Use the Model Browser (understand/analyze)
- Find components (sketches, work planes, joints, etc.) in the Model Browser
- Identify features in the Model Browser (extrusion, circular patterns, holes, etc.)
- Suppress feature

Toggle visibility

Rename and/or reorder model browser elements (features, sketches, etc.)

Understand the functionality of the end of part marker

Use selection options (WN: no items for this objective)
- Select any or all objects enclosed or crossed by a window or freeform shape
- Select objects in the Browser (multi-select)

Adjust the visual properties of a model
- Adjust the visual style (wireframe vs shading)
- Apply materials and appearance overrides
- View models as a section
- Access Properties, such as surface area, mass, volume

Use the Inspect tools
- Use Measure tool
- Use Region Properties to find the perimeter and area of sketch geometry

Create and use work planes and axes

Create and use work planes
- Offset, Midplane, Tangent

Create and use work axes
- Axis Through a Revolved Face or Feature
- Normal to Plane Through Point

Sketch

Create and modify a sketch

Create a sketch on an origin plane or planar face

Select an appropriate sketch tool(s)
- Rectangle, circles, line, arc, polygon, etc.

Identify sketch element types and their functions
- Construction geometry, line type, centerline type

Project geometry from an existing body onto a sketch
- Project geometry, Project cut edges

Edit a sketch
- Move, Copy, Trim, Extend, Offset, Mirror
- Insert text into an active sketch

Apply sketch constraints and dimensions

Determine which sketch constraints to apply
- Add and remove constraints
- Understand auto-constraints

Add and edit dimensions to sketch geometry
- Create a fully constrained drawing

Use design parameters to create a parametric model

Model

Create and modify sketched features

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

Modify a sketched feature

Create and modify placed features

Create and modify placed features
- Fillet, Chamfer, Shell
- Delete a feature

Create hole features
- Counterbore, Countersink, Spot Face

Create a pattern of features
- Rectangular, Circular

Assemble

Place components into assemblies

Create relationships

Place constraints to components
- Mate, Insert, Angle, Tangent, Offsets
- Ground the base component of an assembly

Place joints to components
- Rigid, Rotational

Analyze components

Determine the degrees of freedom of a component

Manipulate components within an assembly

Check for interference between components

Document

Create drawings

Work with drawing sheets
- Insert a new title block
- Edit the size of the sheet

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

Apply dimensions and annotations

Add and edit dimensions

Add and edit annotations