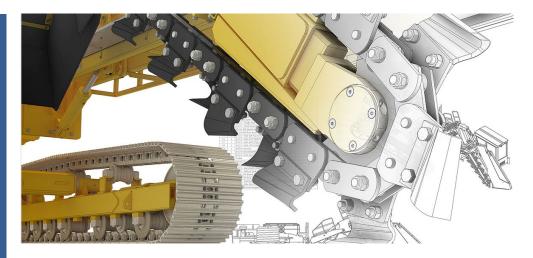
# **Inventor Certified User exam**



# 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.





# 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

Manchester: 0161 526 1456

Sheffield: 0114 3492645

Add and edit dimensions

Add and edit annotations



