AUTODESK. Make anything.

Autodesk Certified Associate in CAM for 2.5 Axis Milling

Exam Objectives

Target audience

The Autodesk Certified Associate (ACA) certification is designed for candidates with essential knowledge and skills in Autodesk software who are ready to enter the job market or improve their skills in pursuit of a new career path. Becoming an Autodesk Certified Associate is an excellent way for individuals with approximately 150 hours of Autodesk software experience to validate their product or workflow skills.

Candidates who obtain this certification will have demonstrated CAM skills for 2.5 axis milling and are prepared for jobs as a CNC programmer machinist in a competitive environment. This exam covers common skills that can be applied across a wide range of industries, including aeronautical, aerospace, defense, automotive, general manufacturing, and medical.

Prerequisite skills

It's expected that candidates will already know how to:

- Navigate the user interface.
- Identify areas of the browser.
- Transition through various workspaces.
- Know the available file types.
- Display a part or assembly.
- Create fully constrained sketches.
- Use Parameters in a sketch.
- Use the extrude, hole, loft, and patch tools.
- Identify various planes and axes.
- Understand detailed drawing intent.
- Identify GD&T Symbols and their meanings.
- Identify work holding devices for CNC Milling.
- Create a distributed design.
- Fully constrain assembly parts.
- Use Direct Modeling.
- Create a CAM setup for CNC Milling.
- Create and manage a tool library.
- Create 2.5axis toolpaths for roughing and finishing.
- Create drilling and tapping operations.
- Simulate toolpaths.

- Create a setup sheet.
- Export NC code for a single setup.

Exam objectives

Here are some topics and software features that may be covered in the exam.

1. Manufacturing Process Planning

1.1. Review Detailed Drawings

1.1.a. Identify and Explain GD&T Symbols
1.1.b. Identify required tool type, size and projection
1.1.c. Identify required surface finish
1.1.d. Identify tolerance-controlled features

1.2. Identify Machine and Work Holding Requirements

1.2.a. Identify and select an appropriate CNC machine
1.2.b. Identify or define work holding device requirements

1.3. Create a Tool List

1.3.a. Identify and source applicable tools for manufacture
1.3.b. Identify and source tool holders for manufacture

1.4. Calculate appropriate feeds and speeds

1.4.a. Calculate appropriate feed rate and tool speed

1.5. Populate a Process Plan Form

1.5.a. Classify required machining operations
1.5.b. Infer information from a process plan form

2. CAD Modeling & Model Preparation

2.1. Create and Define Sketches

2.1.a. Create a new sketch on a plane or face
2.1.b. Create a new construction plane
2.1.c. Edit a sketch
2.1.d. Modify sketch display options
2.1.e. Apply dimensions to a sketch (may include horizontal, vertical, aligned, diameter, radius and angular)
2.1.f. Apply and remove sketch constraints
2.1.g. Link user parameters and sketch dimensions
2.1.h. Create user parameters
2.1.i. Apply math operators in user parameters

2.2. Create and Modify Solid and Surface Features

2.2.a. Create extrude features using driven height options (may include up to object and dimensions)
2.2.b. Create a patch surface
2.2.c. Apply fillets and chamfers to a model
2.2.d. Split bodies and faces

2.3. Apply Direct Modeling Tools

2.3.a. Recognize the application of Press/Pull to modify a feature
2.3.b. Recognize the application of Move/Copy to move a feature
2.3.c. Identify the use of Delete to remove a feature

2.4. Create and Manage Assemblies

2.4.a. Create a component
2.4.b. Apply Joints to create motion
2.4.c. Manage an assembly

3. Create and Simulate CNC Milling Toolpaths

3.1. Create and Manage a Digital Tool Library

3.1.a. Create a digital tool library
3.1.b. Create a custom digital tool
3.1.c. Copy and modify a digital tool
3.1.d. Define tool parameters

3.2. Define a digital CNC setup

3.2.a. Define a digital CNC machine
3.2.b. Create stock (From Solid,
3.2.c. Select a box point to locate a working coordinate system
3.2.d. Define a coordinate reference (G54)

3.3. Import and Locate Work Holding Devices

3.3.a. Import a CAD file
3.3.b. Locate digital work holding

3.4. Create Toolpaths to Rough Cut Parts

3.4.a. Create a basic pocket toolpath for roughing
3.4.b. Create an adaptive toolpath for roughing

3.5. Create Toolpaths to Finish Cut Parts

3.5.a. Create a facing toolpath
3.5.b. Create a 2D contour toolpath
3.5.c. Create Chamfer and 2D contour chamfer toolpaths
3.5.d. Create a drilling and tapping toolpath

3.6. Simulate Toolpaths and Operations

3.6.a. Simulate a single toolpath

3.6.b. Simulate a setup

4. Create Required Documents to Set Up and Run a CNC Mill

4.1. Create Supporting documentation

4.1.a. Create a setup sheet with tool list

4.2. Export NC Code for a CNC Mill

4.2.a. Export NC Code for a single coordinate system with an appropriate post processor (G54)

4.2.b. Export NC Code for multiple coordinate systems (G55, G56, G57)

4.2.c. Identify code snippets (may include which coordinate system is used, spindle speed, tool number or tool rotation direction)