

Autodesk Associate in CAM for 2.5 Axis Milling

Target Audience

This course is designed for CNC programmers, machinists, and manufacturing professionals who want to develop and validate their foundational skills in computer-aided manufacturing (CAM) using Autodesk software for 2.5-axis milling operations. It is ideal for individuals preparing to enter roles in aerospace, automotive, general manufacturing, defense, or medical device production.

Course Objective

This course provides participants with essential knowledge and hands-on skills in CAM workflows for 2.5-axis milling using Autodesk Fusion 360. Learners will gain practical experience in preparing CAD models, planning machining strategies, creating and simulating toolpaths, and generating the necessary documentation and NC code for CNC milling operations. The course is aligned with the competencies required for the Autodesk Certified Associate in CAM for 2.5 Axis Milling certification, helping learners build industry-relevant skills and prepare for certification success.

Course Outcome

- **Process Planning Mastery** – Gain expertise in reviewing drawings, selecting machines, defining tooling, and calculating feeds and speeds for efficient production.
- **CAD Preparation Skills** – Learn to create and prepare models, apply direct modeling techniques, and manage assemblies for machining.
- **Toolpath Creation & Simulation** – Understand how to create, simulate, and optimize 2.5-axis toolpaths for roughing, finishing, drilling, and tapping operations.
- **Documentation & Code Generation** – Develop the ability to create setup sheets and generate accurate NC code for CNC milling machines.
- **CAM Workflow Proficiency** – Master the end-to-end process from design to machine-ready code, including work holding, stock definition, and coordinate system setup.

Course Outline: The course comprises **40 hours** of **theory and practical labs** and is divided into **4** comprehensive chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and assess understanding of the topics covered.

Chapter 1. Manufacturing Process Planning

Reviewing Drawings and Requirements

- Identifying and explaining GD&T symbols
- Identifying required tool types, sizes, projections
- Determining surface finishes and tolerance-controlled features

Machine and Work Holding Considerations

- Selecting appropriate CNC machines
- Identifying or defining work holding devices

Tool Selection and List Creation

- Identifying and sourcing applicable tools and holders

Feeds and Speeds Calculations

- Calculating feed rates and spindle speeds

Process Plan Preparation

- Classifying required machining operations
- Inferring information from process plans

Chapter 2. CAD Modeling and Model Preparation

Sketch Creation and Definition

- Creating sketches on planes or faces
- Creating construction planes
- Editing sketches and modifying display options
- Applying dimensions (horizontal, vertical, aligned, diameter, radius, angular)
- Applying and removing constraints
- Linking user parameters and sketch dimensions
- Creating and applying user parameters, using math operators

Solid and Surface Feature Creation

- Creating extrudes (using driven height options)
- Creating patch surfaces
- Applying fillets and chamfers
- Splitting bodies and faces

Applying Direct Modeling Tools

- Using Press/Pull, Move/Copy, Delete to modify features

Assembly Creation and Management

- Creating components
- Applying joints to create motion

- Managing assemblies

Chapter 3. CNC Milling Toolpath Creation and Simulation

Digital Tool Library Management

- Creating and managing digital tool libraries
- Creating and modifying custom tools
- Defining tool parameters

CNC Setup Definition

- Defining digital CNC machines
- Creating stock and selecting box points
- Defining coordinate references (G54)

Work Holding Integration

- Importing CAD files for fixtures
- Locating and positioning work holding devices

Toolpath Creation for Roughing

- Creating pocket and adaptive toolpaths

Toolpath Creation for Finishing

- Creating facing, 2D contour, chamfer, drilling, and tapping toolpaths

Toolpath Simulation

- Simulating single toolpaths and full setups

Chapter 4. Documentation and NC Code Generation

Supporting Documentation Creation

- Creating setup sheets with tool lists

NC Code Export

- Exporting NC code for single coordinate systems (G54)
- Exporting NC code for multiple coordinate systems (G55, G56, G57)
- Identifying code snippets (coordinate system, spindle speed, tool numbers, rotation direction)