Autodesk Fusion 360: CAM for Multi-Axis Milling (Beginner to Expert Program)

Target Audience

This course is designed for CNC programmers, manufacturing engineers, and CAM professionals ranging from beginners to advanced users who aim to develop expertise in multi-axis milling using Autodesk Fusion 360. It is particularly suitable for individuals working in aerospace, automotive, medical, industrial design, and precision manufacturing sectors who seek to build strong foundational knowledge and progress toward expert-level proficiency. The course is also aligned for candidates preparing for the Autodesk Certified Expert in CAM for Multi-Axis Milling certification, enabling them to validate their skills in advanced toolpath strategies, simulation, and NC code optimization within a structured and exam-focused learning environment.

Course Outcomes

- Understand CNC machining fundamentals and multi-axis concepts
- Apply 3-axis and multi-axis toolpath strategies in Fusion 360
- Create and configure setups, fixtures, and work coordinate systems (WCS)
- Optimize toolpaths for 3+1, 3+2, and simultaneous 5-axis machining
- Analyze and simulate machining operations for collision avoidance
- Troubleshoot and verify NC code for multi-axis environments
- Prepare and validate parts using in-process inspection techniques
- Develop proficiency aligned with Autodesk certification standards

---

Course Objectives

- Provide a structured learning path from basic CNC concepts to advanced multi-axis programming
- Develop proficiency in Autodesk Fusion 360 CAM environment through guided practice
- Reinforce theoretical knowledge with hands-on lab exercises and real-world scenarios

- Build competency in toolpath planning, simulation, and NC code generation
- Enhance problem-solving skills for collision detection and machining optimization
- Prepare learners for Autodesk Certified Expert examination through comprehensive coverage
- Strengthen understanding of inspection and validation processes in manufacturing workflows

---

Course Outline

The course comprises **56-hours** of theory and labs and is divided into 8 different chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Table of Contents (TOC)

Module 0: Fundamentals of CNC & CAM

- Introduction to CNC machining (milling vs turning)
- Basics of axes (2-axis, 3-axis, 4-axis, 5-axis)
- Understanding G-code fundamentals (G00, G01, G02, etc.)
- Cutting tools basics (end mills, ball nose, face mills)
- Basics of feeds, speeds, and material removal
- Introduction to CAD vs CAM workflow
- Overview of Autodesk Fusion 360 CAM workspace

Module 1: 3-Axis Machining & CAM Basics

- Creating basic setups in Fusion 360
- 2D toolpaths (Facing, Pocketing, Contour)
- 3D toolpaths (Adaptive clearing, Parallel, Scallop)
- Tool library management
- Basic simulation and verification
- Post-processing basics (generating NC code)
- Introduction to workholding and WCS

Module 2: Planning and Setup for Multi-Axis Machining

- Interpreting engineering drawings for multi-axis operation
- Planning stock preparation for multi-axis work holding
- Designing fixtures and evaluating cutting forces
- Analyzing geometry for toolpath sequencing
- Assessing if multi-axis machining is suitable
- Performing multi-axis CAM setup in Fusion 360
- Locating the Work Coordinate System (WCS) for center of rotation
- Creating machine configurations for multi-axis environments

Module 3: Machine and Tool Setup

- Digitally planning tool assemblies and holders
- Selecting tools and holders for specific geometry
- Digitally assembling work holding devices
- Choosing work holding based on part specifications
- Defining multi-axis work offsets and operation parameters
- Establishing WCS strategies for multi-fixture setups

Module 4: Programming Multi-Axis Toolpaths

- Choosing machining strategies for complex geometry
- Selecting appropriate toolpaths (e.g., swarf, steep and shallow)
- Defining tool orientation for 3+1, 3+2, and simultaneous 5-axis
- Managing axis of rotation and tilt angles
- Creating wrapped 2D toolpaths using tool axis control
- Defining containment boundaries using edges, sketches, surfaces
- Controlling toolpath slope limits and retraction policies
- Applying collision avoidance with shaft and holder controls

- Trimming toolpaths and adjusting movement strategies
- Optimizing program order and toolpath smoothing

Module 5: Simulation and Verification

- Simulating toolpaths with machine motion
- Validating stock removal strategies using comparison tools
- Detecting tool collisions and holder interference
- Making toolpath adjustments based on simulation outcomes

Module 6: NC Code Output

- Verifying axis setup with posted NC code
- Troubleshooting NC code errors
- Configuring output to match machine capabilities

Module 7: In-Process Part Inspection

- Using probing cycles to verify features and dimensions
- Manually inspecting part dimensions and feature tolerances
- Updating machine parameters based on inspection results
- Proving out programs on a physical machine