WorkNC Essentials to Advanced: A Hands-On CAM Training

Target Audience

This course is designed for CAM programmers, manufacturing engineers, and CAD/CAM professionals who want to gain proficiency in using WorkNC for machining workflows. It is ideal for learners seeking to understand workzone management, toolpath creation, simulation, postprocessing, and advanced machining strategies in a structured, hands-on environment.

Course Outcomes

By the end of this course, participants will be able to:

- Install, launch, and configure WorkNC for machining projects.
- Navigate the WorkNC interface and manage projects effectively.
- Create and manage workzones with proper coordinate systems.
- Import CAD models and perform part analysis.
- Prepare models for machining, define stock, and set up machines and tools.
- Work with CAM entities including axis systems, views, curves, and points.
- Generate, edit, and optimize toolpaths for roughing, finishing, contouring, and remachining.
- Run stock simulations and analyze machining results.
- Calculate safe tool lengths and configure machining parameters.
- Postprocess toolpaths to generate validated NC code.

Course Objectives

- Provide a structured foundation in WorkNC workflows.
- Train learners in workzone creation, CAD import, and preparation for machining.
- Develop proficiency in toolpath generation, editing, and optimization.
- Enable learners to simulate machining processes and validate results.
- Teach advanced machining strategies including waveform roughing, optimization, and remachining.
- Reinforce learning through practical exercises and documentation workflows.

Course Outline

The course comprises **40 hours** of theory and labs and is divided into **17 different chapters**. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered

Table of Content

Module 1 – Introduction to WorkNC

- Course Overview & Objectives
- What is WorkNC
- Key Features & Capabilities
- System Requirements
- Installing WorkNC
- Launching the Application
- Understanding the WorkNC Interface
- Menu Bar & Toolbars
- WorkNC Launcher
- Project File Structure
- Setting Up User Preferences
- Keyboard Shortcuts & Navigation Tips

Module 2 – Workzones

- What is a Workzone
- Creating a New Workzone
- Defining Workzone Boundaries
- Workzone Coordinate Systems
- Setting the Origin Point
- Editing Workzone Properties
- Copying & Duplicating Workzones
- Deleting Workzones
- Managing Multiple Workzones

- Workzone Organization Best Practices

Module 3 – Basic Functions

- Navigating the Viewing Area
- Zoom, Pan & Rotate Controls
- View Orientation (Top, Front, Iso, etc.)
- Display Modes (Wireframe, Shaded, etc.)
- Importing CAD Models
- Supported File Formats (IGES, STEP, STL, Parasolid, CATIA, etc.)
- Import Settings & Options
- Handling Import Errors
- Print/Plot Functions
- Page Setup & Print Preview
- Taking Screenshots
- Saving Screenshots in Different Formats
- Measurements
- Distance Measurement
- Angle Measurement
- Radius & Diameter Measurement
- Part Analysis
- Surface Analysis
- Draft Angle Analysis
- Curvature Analysis

Module 4 – Preparation Mode

- Entering Preparation Mode
- Preparing the Model for Machining
- Defining Stock Geometry
- Box Stock
- Cylinder Stock

- Custom Stock from CAD Model
- Setting Up the Machine Configuration
- Selecting the Machine Tool
- Tool Assembly Setup
- Tool Library Management
- Creating & Editing Tools
- Tool Holder Definition
- Setting Safety Planes
- Defining Clearance Heights
- Material Selection & Feed/Speed Defaults

Module 5 – CAM Entities

- What are CAM Entities
- Axis Systems
- Creating an Axis System
- Editing Axis Systems
- Aligning Axis Systems to Geometry
- Views
- Creating Views
- Modifying View Parameters
- Using Views for Toolpath Control
- Curves
- Creating Curves from Geometry
- Projecting Curves onto Surfaces
- Editing & Trimming Curves
- Using Curves as Toolpath Boundaries
- Points
- Creating Points
- Importing Points from Files

- Using Points for Drilling & Positioning
- Creating a Simulation Stock Model
- Merging CAM Entities
- Selecting Dependent Toolpaths
- CAM Entity Management Best Practices

Module 6 – Creating Toolpaths — Overview

- Toolpath Concepts & Workflow
- Understanding Toolpath Types
- Toolpath Calculation Process
- Toolpath Parameters Menu
- Standard vs. Specific Parameters
- Toolpath Status Icons
- Interpreting Status Colors & Symbols
- Displaying & Managing Toolpaths
- Showing/Hiding Toolpaths
- Toolpath List Management
- Reordering Toolpaths
- Copying & Pasting Toolpaths
- Deleting Toolpaths
- Toolpath Naming Conventions

Module 7 – 3-Axis Roughing Toolpaths

- Introduction to Roughing Strategies
- Global Rough/Rerough
- Standard Parameters
- Tool Selection for Roughing
- Step Down & Step Over Settings
- Cutting Direction (Climb vs. Conventional)
- Specific Parameters

- Plunge Control & Ramping
- Corner Treatment
- Initializing the Stock Model
- Calculating the Roughing Toolpath
- Updating the Stock Model After Roughing
- Defining Initial Lateral Step
- Defining Lead-in Distance
- Defining Local Stock
- Reroughing
- When to Use Reroughing
- Reroughing with Smaller Tools
- Flat Surface Rough/Rerough
- Parameters Specific to Flat Surfaces
- Waveform Roughing
- Benefits of Waveform Strategy
- Waveform Parameters & Settings
- Tool Engagement Control
- Roughing Best Practices & Tips

Module 8 – 3-Axis Finishing Toolpaths

- Introduction to Finishing Strategies
- Choosing the Right Finishing Strategy
- Flat Surface Finishing
- Parameters & Tool Selection
- Z-Level Finishing
- Constant Z Step Down
- Steep Area Detection
- Scallop Height Control
- Planar Finishing

- Planar Angle Settings
- Path Interval & Overlap
- Parallel Finishing
- Direction Control
- Step Over Optimization
- Combined Z-Level Finishing and Optimization
- Merging Z-Level & Planar Strategies
- Transition Control Between Strategies
- Surface Quality Settings
- Tolerance & Accuracy Parameters
- Finishing Best Practices & Tips

Module 9 – 3-Axis Optimization Toolpaths

- What is Toolpath Optimization
- When to Use Optimization
- Optimized Z-Level Finishing
- Automatic Step Adjustment
- Scallop Height Consistency
- Optimized Planar Finishing
- Adaptive Path Spacing
- Surface Quality Improvements
- Comparing Standard vs. Optimized Results

Module 10 – Contouring Toolpaths

- Introduction to Contouring
- 3D Contouring (Pencil Trace)
- Detecting Internal Corners & Fillets
- Pencil Trace Parameters
- Tool Selection for Contouring
- Parallel Pencil Trace

- Multiple Pass Contouring
- Offset & Overlap Settings
- Contouring Use Cases & Best Practices

Module 11 – 3-Axis Remachining Toolpaths

- What is Remachining
- Automatic Rest Material Detection
- Planar Remachining
- Detecting Unmachined Areas
- Setting Reference Tool Size
- Z-Level Remachining
- Step Down in Remaining Material
- Contour Remachining
- Following Rest Material Boundaries
- Remachining Strategy Selection Guide
- Combining Remachining with Finishing

Module 12 – Editing Toolpaths

- Modifying Toolpath Parameters
- Editing Tool Selection
- Changing Feeds & Speeds
- Adjusting Step Over & Step Down
- Multiple Toolpath Parameter Editing
- Batch Editing Parameters
- Applying Changes to Multiple Toolpaths
- Machining Zone Definition
- Defining Boundaries & Limits
- Including & Excluding Zones
- Using Curves as Machining Boundaries
- Toolpath Trimming & Splitting

- Reordering & Reorganizing Toolpaths

Module 13 – Stock Simulation

- Introduction to Stock Simulation
- Running a Stock Simulation
- Setting Simulation Parameters
- Simulation Display Options
- Displaying the Rest Material Preview
- Color Mapping for Remaining Stock
- Analyzing Simulation Results
- Gouge Detection
- Collision Checking
- Material Removal Verification
- Comparing Simulated vs. Target Geometry
- Generating Simulation Reports
- Troubleshooting Simulation Issues

Module 14 – Calculating the Recommended Safe Tool Length

- Why Safe Tool Length Matters
- Automatic Tool Length Calculation
- Checking for Tool Holder Collisions
- Adjusting Tool Stick-Out
- Verifying Safe Length Across Multiple Toolpaths

Module 15 – Postprocessing Toolpaths

- What is Postprocessing
- Selecting a Post Processor
- Configuring Post Processor Settings
- Generating NC Code (G-Code)
- Reviewing & Validating NC Output
- Understanding G-Code Structure

- Editing NC Code Manually
- Exporting for Different CNC Controllers
- Common Postprocessing Errors & Fixes
- Batch Postprocessing Multiple Toolpaths

Module 16 – Workzone Management & Documentation

- Advanced Management Functions
- Organizing Workzones for Complex Parts
- Archiving & Backing Up Workzones
- Transferring Workzones Between Projects
- Generating HTML Documentation
- Customizing Documentation Templates
- Including Screenshots & Toolpath Details
- Sharing Documentation with Shop Floor

Module 17 – Machining Parameters (Advanced)

- Machining Zone Configuration
- Advanced Zone Strategies
- Multi-Zone Machining
- Zone Priority Settings
- Parameter Templates
- Creating Reusable Templates
- Applying Templates to New Projects
- Feed & Speed Optimization
- Cutting Condition Libraries
- Best Practices for Parameter Management