Siemens NX: CAD Design to CAM Manufacturing – 9-Day Course

From Engineering Essentials and Part Design to Multi-Axis CNC Programming

Course Description

This course takes participants through the complete Siemens NX product-development chain, from the first 2D sketch to the final CNC program. The first four days build a strong CAD foundation: the NX interface, parametric sketching, 3D solid modeling, advanced part design, assemblies and technical drawings. The remaining days move into NX CAM (Manufacturing): setups, coordinate systems, cutting tools, milling and hole-making operations, feature-based machining, simulation and post processing. Through hands-on exercises and real-world design-to-manufacturing workflows, learners will gain practical skills to model, document, program and verify a part ready for the shop floor.

The course comprises approximately 72 hours (9 days x 8 hours) of theory and laboratory sessions, divided into 22 modules. Each module is followed by hands-on exercises, and the final day is a capstone project that takes one part from CAD model to NC code.

Audience Profile

This course is ideal for:

- Mechanical engineers, product designers and CAD professionals
- Manufacturing engineers, CNC programmers, CAM engineers and process planners
- Tool designers and production engineers
- Engineering students or graduates entering CAD/CAM-based industries
- Professionals transitioning to Siemens NX from other CAD/CAM platforms
- Anyone involved in product development, prototyping or CNC manufacturing

Prerequisite Knowledge/Skills

- Basic understanding of mechanical design and drafting principles
- Basic knowledge of CNC machining concepts (milling, drilling, tooling)
- Familiarity with general CAD concepts (any CAD software experience is helpful)
- Fundamental computer literacy (Windows environment)

Course Objectives

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

- Navigate the Siemens NX interface and workspace
- Create and manage 2D sketches and parametric constraints
- Build and modify 3D parts using solid modeling techniques
- Design assemblies and apply proper constraints and relationships
- Generate professional technical drawings from models and assemblies
- Apply productivity tools and design best practices within NX
- Understand the Siemens NX Manufacturing environment and configure setups and machine coordinate systems
- Create machining operations for milling and drilling using cavity, planar and contour strategies
- Implement feature-based machining techniques

- Verify toolpaths using simulation tools and optimize machining results
- Generate shop documentation and machine-specific NC code using post processors
- Prepare industrial CNC manufacturing programs from a finished CAD model

Course Outline

PART A: NX CAD – ENGINEERING ESSENTIALS AND PART DESIGN (DAYS 1–4)

Day 1: NX Fundamentals and Sketching

Module 1: Introduction to Siemens NX

- 1.1 Overview and industrial applications of Siemens NX
- 1.2 Interface layout and command navigation
- 1.3 Customizing the NX environment
- 1.4 Understanding NX file structure and data management
- 1.5 Starting a new design project

Module 2: Sketching Fundamentals

- 2.1 Sketch environment and tools
- 2.2 Drawing basic 2D geometry
- 2.3 Applying constraints and dimensions
- 2.4 Managing sketch planes and references
- 2.5 Sketching best practices

Day 2: 3D Modeling and Advanced Part Design

Module 3: 3D Modeling Basics

- 3.1 Feature-based modeling concepts
- 3.2 Extrude, Revolve, Sweep and Loft features
- 3.3 Editing and modifying features
- 3.4 Working with solids and surfaces
- 3.5 Understanding model history and design intent

Module 4: Advanced Part Design

- 4.1 Creating patterns: linear, circular, mirror
- 4.2 Hole wizard and placement strategies
- 4.3 Fillets, chamfers and shell operations
- 4.4 Boolean operations and feature groups
- 4.5 Parametric modeling and expressions

Day 3: Assemblies and Technical Drawings

Module 5: Assembly Design Essentials

- 5.1 Working in the assembly environment
- 5.2 Inserting and positioning components
- 5.3 Assembly constraints and relationships
- 5.4 Exploded views and assembly navigation
- 5.5 Component reuse and management

Module 6: Drafting and Technical Drawings

- 6.1 Creating drawing views: base, projected, section
- 6.2 Adding dimensions, notes and symbols
- 6.3 Customizing title blocks and drawing sheets
- 6.4 Tolerances and annotations
- 6.5 Exporting and printing drawings

Day 4: Productivity Tools and Introduction to NX CAM

Module 7: Design Best Practices and Productivity Tools

- 7.1 Creating and using design templates
- 7.2 Managing layers and model display
- 7.3 Feature reusability and libraries
- 7.4 Performing model checks and error fixes
- 7.5 Time-saving shortcuts and tips

PART B: NX CAM – MANUFACTURING FUNDAMENTALS TO MULTI-AXIS PROGRAMMING (DAYS 4–8)

Module 8: Manufacturing Process and Setup

- 8.1 Manufacturing environment overview
- 8.2 Manufacturing setup creation
- 8.3 Part and blank definition
- 8.4 Machine selection
- 8.5 Manufacturing preferences
- 8.6 Preparing CAD models for manufacturing (CAD-to-CAM transition)

Module 9: Operation Navigator

- 9.1 Operation Navigator interface
- 9.2 Program groups
- 9.3 Geometry groups
- 9.4 Tool groups
- 9.5 Method groups

Day 5: Coordinate Systems, Tooling and Toolpaths

Module 10: Coordinate Systems

- 10.1 Manufacturing Coordinate System (MCS)
- 10.2 Work Coordinate System (WCS)
- 10.3 Part orientation
- 10.4 Datum definition
- 10.5 Coordinate management

Module 11: Cutting Tools

- 11.1 Tool library
- 11.2 Tool creation
- 11.3 Tool parameters
- 11.4 Holder definition
- 11.5 Tool management

Module 12: Operations and Tool Paths

- 12.1 Creating operations
- 12.2 Operation parameters
- 12.3 Toolpath generation
- 12.4 Editing operations
- 12.5 Regenerating toolpaths

Day 6: Milling and Hole Making

Module 13: Cavity Milling

- 13.1 Cavity milling basics
- 13.2 Cut pattern selection
- 13.3 Step-over settings
- 13.4 Entry and exit motions
- 13.5 Toolpath optimization

Module 14: Planar Milling

- 14.1 Planar milling operations
- 14.2 Region machining
- 14.3 Vertical wall machining
- 14.4 Floor machining
- 14.5 Boundary control

Module 15: Hole Making

- 15.1 Drilling
- 15.2 Peck drilling
- 15.3 Counterboring
- 15.4 Reaming
- 15.5 Tapping

Day 7: Contouring, Prismatic Operations and Feature-Based Machining

Module 16: Fixed Axis Contouring

- 16.1 Fixed-axis contouring
- 16.2 Surface finishing
- 16.3 Area machining
- 16.4 Cut direction
- 16.5 Surface quality optimization

Module 17: Additional Prismatic Operations

- 17.1 Face milling
- 17.2 Text engraving
- 17.3 Chamfer milling
- 17.4 Slot machining
- 17.5 Profile machining

Module 18: Feature-Based Machining

- 18.1 Feature recognition
- 18.2 Automatic feature creation
- 18.3 Feature machining
- 18.4 Process templates
- 18.5 Manufacturing automation

Day 8: Verification, Non-Cutting Moves and Post Processing

Module 19: Visualization and Verification

- 19.1 Toolpath replay
- 19.2 Material removal simulation
- 19.3 Collision detection
- 19.4 Gouge checking
- 19.5 Toolpath validation

Module 20: Non-Cutting Moves

- 20.1 Engage motion
- 20.2 Retract motion
- 20.3 Transfer motion
- 20.4 Clearance settings
- 20.5 Motion optimization

Module 21: Shop Documentation and Post Processing

- 21.1 Tool list generation
- 21.2 Operation sheets
- 21.3 NC output
- 21.4 Post processing
- 21.5 Program verification

PART C: INTEGRATED CAPSTONE (DAY 9)

Day 9: Capstone Project – From CAD Model to CNC Program

Module 22: End-to-End Design-to-Manufacturing Project

- 22.1 Project briefing and design requirements
- 22.2 Modeling the part (sketches, features, patterns, holes and fillets)
- 22.3 Creating the technical drawing with tolerances and annotations
- 22.4 Manufacturing setup: blank, MCS, tools and program groups
- 22.5 Creating cavity, planar, hole-making and finishing operations
- 22.6 Simulation, collision and gouge checking, and toolpath optimization
- 22.7 Post processing, NC output and shop documentation
- 22.8 Project presentation, review and feedback
- 22.9 Course wrap-up, best practices and Q&A;

Course Summary

- Duration: 9 days (approximately 72 hours)
- Days 1–4: NX CAD – interface, sketching, solid modeling, advanced part design, assemblies, drafting and productivity tools

- Days 4–8: NX CAM – setups, coordinate systems, tooling, milling, hole making, feature-based machining, verification and post processing
- Day 9: Capstone project integrating CAD and CAM
- Delivery: instructor-led theory with hands-on lab exercises after every module