

Siemens NX Product Design and Manufacturing Essentials

Target Audience

This course is designed for mechanical engineers, product designers, manufacturing engineers, CAD/CAM professionals, design consultants, and engineering students who want to develop practical skills in Siemens NX for product design and manufacturing. It is suitable for professionals involved in mechanical product development, tooling, industrial equipment design, and manufacturing workflows, as well as individuals seeking to build a strong foundation in integrated CAD, CAM, and basic CAE using Siemens NX.

Course Objective

The objective of this course is to provide participants with the knowledge and practical skills required to design, modify, validate, and prepare mechanical components and assemblies for manufacturing using Siemens NX. Through hands-on exercises and industry-oriented projects, learners will explore parametric modeling, assembly design, engineering drawings, surface and sheet metal design, manufacturing fundamentals, simulation basics, and collaborative engineering workflows to develop efficient and production-ready product designs.

Course Outcome

Upon successful completion of this course, participants will be able to:

- Understand the Siemens NX interface, design environment, and integrated product development workflow.
- Create fully parametric 3D parts using feature-based modeling techniques and design intent principles.
- Develop and manage complex assemblies using industry-standard assembly design practices.
- Produce professional engineering drawings with dimensions, annotations, GD&T, and bills of materials.
- Design advanced surface and sheet metal components for manufacturing applications.
- Apply direct editing techniques and synchronous modeling to modify native and imported CAD models efficiently.
- Perform basic engineering validation using simulation tools and interpret analysis results for design improvement.
- Generate CNC machining toolpaths and understand the fundamentals of manufacturing preparation using NX CAM.
- Exchange design data, collaborate with multidisciplinary teams, and manage engineering revisions effectively.
- Complete an end-to-end product development project by integrating design, documentation, validation, and manufacturing workflows.

Course Outline

The course comprises **80 hours** of instructor-led theory sessions and hands-on laboratory exercises and is divided into **16 comprehensive modules**. Each module includes practical activities designed to reinforce core concepts and develop real-world proficiency in Siemens NX.

Module 1: Getting Started with Siemens NX

- Introduction to Siemens NX Platform
- User Interface and Navigation
- Interface Customization
- File Organization
- Design Workflow Overview
- Productivity Tips

Module 2: Sketch-Based Design

- Creating 2D Sketches
- Sketch Constraints
- Dimensional Control
- Editing Sketch Geometry
- Reference Geometry
- Sketch Best Practices

Module 3: Solid Modeling Fundamentals

- Feature-Based Modeling
- Extrude and Revolve
- Sweep and Loft
- Hole and Thread Features
- Patterns and Mirror
- Boolean Operations
- Parametric Design Concepts

Module 4: Advanced Feature Modeling

- Thin Features
- Shell and Draft
- Rib Features
- Helix and Curves
- Expressions and Parameters
- Feature Reordering
- Model Optimization Techniques

Module 5: Assembly Design

- Bottom-Up Assemblies
- Top-Down Design
- Assembly Constraints
- Component Positioning
- Assembly Analysis
- Exploded Views
- Large Assembly Performance

Module 6: Engineering Drawings

- Creating Manufacturing Drawings
- Standard Drawing Views
- Section and Detail Views
- Dimensions and Annotations
- GD&T Fundamentals
- BOM Generation
- Drawing Templates

Module 7: Surface Design

- Surface Modeling Concepts
- Sweep and Through Curves
- Surface Trimming
- Surface Extension
- Surface Offset
- Surface Quality Analysis
- Surface Repair Techniques

Module 8: Sheet Metal Design

- Base Features
- Flanges
- Bends
- Corner Treatment
- Flat Pattern Generation
- Manufacturing Considerations
- Design Validation

Module 9: Direct and Flexible Modeling

- Direct Editing
- Face Operations
- Imported Model Modification
- Live Rules
- Feature Recognition
- Design Reuse
- Flexible Engineering Changes

Module 10: Simulation Essentials

- Introduction to CAE
- Material Properties
- Boundary Conditions
- Mesh Generation
- Linear Static Analysis
- Result Evaluation
- Basic Design Optimization

Module 11: Manufacturing with NX CAM

- CAM Workspace
- Machine Setup
- Cutting Tools
- Toolpath Planning
- Milling Operations
- Hole Machining
- Verification and Simulation
- Post Processing Basics

Module 12: Data Management & Collaboration

- CAD Data Exchange
- Neutral File Formats
- Revision Control
- Collaborative Engineering
- Product Data Management Basics
- Design Review
- Workflow Integration

Module 13: Design Verification

- Geometry Inspection
- Model Validation
- Interference Detection
- Clearance Analysis
- Mass Properties
- Engineering Review
- Quality Assurance

Module 14: Productivity Enhancement

- Templates
- Reusable Components
- Expressions
- Basic Automation
- Design Reuse Strategies
- Workflow Optimization
- Productivity Best Practices

Module 15: Practical Industry Applications

- Mechanical Component Project
- Consumer Product Modeling
- Assembly Documentation
- Sheet Metal Application
- Surface Modeling Exercise
- Manufacturing Documentation
- Final Design Review

Module 16: Integrated Engineering Project

- Product Planning
- Complete Part Modeling
- Assembly Development
- Drawing Creation
- Manufacturing Preparation
- Design Verification
- Documentation
- Final Project Presentation