NX Design  
(CAD)

NX Manufacturing  
(CAM)
The learning tracks have been provided to help you plan a productive development path through our courseware.

Learning tracks start from the top and progress downward. Select any course below to open the training content.

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For more NX Design (CAD) courses see next page:
NX Design (CAD)

Select the Course for more information:

- PLM Manager Awareness Training
- CAD Hothouse
- CAD Update
- Open API
- Product and Manufacturing Information (PMI)
- NX Render
- Shipbuilding for Admins or Key Users
- Product Template Studio
Designing Parts in NX
(previously known as: Essentials for NX Designers)

Duration: 6 days
Who should attend: Engineers, Designers, Detailers, Checkers
Prerequisites: None
Follow up Training: Designing Parts in NX for experienced users

This course is taught in the Teamcenter Integration environment using Active Workspace embedded in NX.

The Designing Parts in NX as the initial class, covers the essential NX task-based processes that new users will utilize when creating and editing parametric parts. The real-world experience of the Siemens instructors aids students in transferring knowledge gained through this single course to their job resulting in faster time to productivity.

At the completion of the Designing Parts in NX class, the student will be able to develop parametric solid and assembly models as well as drawings using the master model concept. These concepts can be applied in the real world of product development and collaboration.

This training is scheduled over 2 separate weeks in 3 + 3 days.

Day 1 (Modeling)
• Your first day in NX
• Make design changes
• Create basic models from Sketches
  • Sketch workflows
  • Modeling workflows

Day 2 (Modeling)
• Fit the pieces together
• Create and update a Drawing
• Organize Models
• Design parts with Sketches

Day 3 (Modeling)
• Create basic machined parts
• Add finishing details
• Create parts with constant wall thickness
• Create symmetric models
• Create patterns of features
• Create basic sheet metal parts

Day 4 (Assemblies)
• Create a basic assembly
  • Assembly concepts
  • Sketching in an assembly context
• Position Components
  • Move components
  • Constrain components
• Create a pattern of components

Day 5 (Assemblies)
• Manage assemblies
  • Loading assembly structure
  • Loading component internal data
  • Loading component visible data
• Revise components and assemblies

Day 6 (Drafting)
• Master Model drawings and drafting standards
• Drawing Sheets
• Drawing Views
• Annotations
• Dimensions

More information?
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Designing Parts in NX for experienced users

Duration: 4 days
Who should attend: Engineers, Designers, Detailers
Prerequisites: Designing Parts in NX
Follow up Training: Large Assembly Management, Freeform Modeling, Sheet Metal, Etc.

This course is taught in the Teamcenter Integration environment using Active Workspace embedded in NX.

The Designing Parts in NX for experienced users class is designed to advance students further up the productivity curve. This course builds on the tools you deployed as a result of attending the Designing Parts in NX course. This method-based course focuses the student on productive modeling techniques that capture design intent in the context of the Master Model. Delivering on that outcome, this course will incorporate sketching, inter-part modeling, design intent, several assembly and drafting topics as a significant part of the instruction.

This training is scheduled over 2 separate weeks in 2 + 2 days.

Day 1 (NX Modeling)
- Ruled Surface + Variational Sweeps
- Capturing design intent with formulas
- Duplicating features (Cut, Copy and Paste = Reuse Library)
- Blending techniques
- Simplifying geometry for downstream applications

Day 2 (NX Modeling)
- Pattern Techniques
- Offset Face / Offset Region
- Creating Ribs
- Swept Volume
- Reuse Library - 2D sections
- Emboss Body + Unite options

Day 3 (NX Assemblies)
- Assembly Clearance
- Interpart Expressions + WAVE Geometry
- Arrangements + Assembly Cut + Hole Series
- Mirror Assembly
- Capturing part shape variations when assembled (deformable parts)
- HD3D Visual Reporting

Day 4 (NX Drafting)
- Advanced Section + Broken Views
- View Boundaries
- Sketch in Drafting
- Broken Views
- Basics PMI (Product Manufacturing Information)
- Ordinate Dimensions + Hole tables
- Exploded view + Secondary Geometry
- Tabular Note + Parts List
- Track drawing changes

More information?
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Teamcenter Integration for NX Users

Duration: 1 day
Who should attend: NX CAD users who work with Teamcenter
Prerequisites: Designing Parts in NX
Follow up Training: Using Teamcenter

Introducing Product Data Management (PDM) and adjacent systems comes with a certain way-of-working, and creates the need for a new set of skills for the NX CAD users. The Teamcenter Integration for NX training is created just for that specific purpose, from the NX perspective that highlights the NX functionality of the Teamcenter Integration.

This Teamcenter Integration for NX Users course is given in a project form, where it focusses on working together as a team. During the project the engineers work together to create and build an Assembly (concurrent engineering) by using each other’s parts, including modifying and revising. And so during the course they will better understand the usage of the NX functionality of the Teamcenter Integration.

After this course the engineer has to be able to use those “Teamcenter tools” which are required in managing the data of NX in the Teamcenter environment.

Day 1
• What is PDM?
• What is the Teamcenter Rich Application Client?
• Person, Users, Roles en Groups
• What is Teamcenter Integration?
• The Teamcenter Integration Open Part File dialog
• The Teamcenter Integration Create new Part File dialog
• Default Container, Create Folder
• Open Folder as Templates Palette
• Teamcenter Integration Preferences
• What are Items, Item-revisions en Datasets
• Create new modeling files (Master-parts) en related files (Non-Master-parts)
• Search function in Teamcenter Integration
• Create Assemblies in Teamcenter Integration
• Opening Assemblies in Teamcenter Integration
• Implicit and explicit Check-out Check-in mechanism
• Introduction of Revision Rules
• Where-Referenced and Where-used
• Starting a Workflow process (introduction workflow)

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Synchronous Modeling

**Duration:** 1 day  
**Who should attend:** NX CAD, CAM and Simcenter users  
**Prerequisites:** Designing Parts in NX (Modeling)  
**Follow up Training:** Freeform Modeling, Sheet Metal

The NX Synchronous Modeling training is intended for users who want to modify existing 3D models without changing the original NX parameters of existing NX features. If you are working on parts imported in NX, you can perform modifications on the imported (Iges/Step) models using the Synchronous commands.

After this training, the student will be able to edit models quickly, using Synchronous Technology.

**Day 1**
- Introduction to Synchronous Modeling
- Move Face - Pull Face
- Resize Face / Blend / Chamfer
- Label Notch Blend / Chamfer
- Reorder Blends
- Offset Region
- Replace Face – Replace Blend
- Delete Face
- Move Edge – Offset Edge
- Copy / Cut / Paste Face
- Pattern Face
- 2D Synchronous
- Constrain Face (Make Coplanar, Make Parallel, etc.)
- Linear / Angular / Radial Dimension
- Mirror Face
- Optimize Face

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Freeform Modeling

Duration: 4 days
Who should attend: Design Engineers and Manufacturing Engineers
Prerequisites: Designing Parts in NX (Modeling)
Follow up Training: NX Shape Studio

The NX Free Form Modeling class provides the techniques to generate complex models. As today’s products require more and more advanced forms, prismatic elements alone are not enough anymore. NX gives you the tools to create these Free Form Models in a user-friendly and structured way.

This training is scheduled over 2 separate weeks in 2 + 2 days.

Day 1
• Creating Splines
• Ruled Surface
• Mesh Surfaces
• Through Curves
• Through Curves Mesh

Day 2
• Bridge Curves
• Surface Trim operations
• Other Curve operations
• Curve Analysis
• Trim Body
• Patch

Day 3
• Sweep Surfaces
• Other Surface Types
• Reference Geometry
• Sheet Extensions

Day 4
• Detail Operations
• Global Shaping
• Surface Analysis
• Fix, Enlarge, Improve Surface topology
• Changing Surface Topology (Class A Surfacing)
• Realize Shape

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Large Assembly Management

Duration: 2 days  
Who should attend: CAD Users who work with large or complex assemblies  
Prerequisites: Designing Parts in NX (Assemblies) and Designing Parts in NX for experienced users (Assemblies)

Working with Large Assemblies requires a separate process.

This class covers how you can easily, quickly and selectively open and close components from an assembly. With that waiting times are brought to a minimum.

This class is not only a ‘must’ if you work with large assemblies. This training also describes the analysis of an assembly at a distance from the parts relative to each other and the simplified and accelerated appearance of components.

Day 1
- Introduction Large Assemblies
- Set Assembly Load Options
- Representations
  - Load options
  - Update structure
- Component Groups
  - Proximity
  - Zones
  - Bookmarks
- Find Components
- Open by Proximity using True Shape data

Day 2
- Wrap Assembly and Linked Exterior
- Simplify Assembly
- Assembly Display Performance
- Advanced Weight Management
- Assembly Drawings

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Sheet Metal

Duration: 1 day
Who should attend: Sheet Metal Designers
Prerequisites: Designing Parts in NX (Modeling)
Follow up Training: Advanced Sheet Metal

The Sheet Metal training is developed for engineers who create linear sheet metal products.

All Sheet Metal features will be covered in this training. For example Sheet Metal Flanges, Tabs, Contour Flanges etc. are explained. Also some common modeling features such as Sketch, Hole and Patterns are used in this training.

Day 1
- Introduction to Sheet Metal
- Sheet Metal Preferences
- Tab
- Flange
- Bend Taper
- Contour Flange
- Lofted Flange
- Hem Flange
- Closed Corner - Three Bend Corner
- Break Corner / Chamfer
- Jog - Bend
- Unbend / Rebend / Normal Cutout / Extrude
- Edge Rip
- Sheet Metal from solid
- Convert to Sheet Metal
- Resize Bend Radius - Resize Bend Angle - Resize Neutral Factor
- Cleanup Utility - Optimize Face
- Dimple - Drawn Cutout – Louver – Bead – Gusset - Solid Punch
- Flat Solid - Flat Pattern
- Bend Table (Drafting)
- Export Flat Pattern...

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Shape Studio

Duration: 2 days
Who should attend: Designers
Prerequisites: Designing Parts in NX (Modeling) and Freeform Modeling
Follow up Training: NX Render

The NX Shape Studio class is meant for designers who have knowledge of Free Form Modeling in NX. After the training the student will be able to use his creativity and inspiration with the benefits of a universal design platform from concept to finished products.

Day 1
- Introduction Shape Studio
- Introduction
- Raster Images
- Creating Splines
- Spline and Surface Terminology
- Edit Splines, Smooth Splines
- Studio Surfaces (NvN, Swoop, 4 points)
- Sweep Operations (Styed Sweep)

Day 2
- Advanced Blending Operations (Styed Blend, Styed Corner, Blend Corner)
- Manipulating Surfaces (X-form, Refit Face, Match Edge, Global Shaping)
- Surface Trim Operations (Patch, Snip Surface, Trim and Extend)
- Surface Analysis (Section Analysis, Deviation Gauge)
- Surface Rebuild Options (Class A Surfacing)
- Surfacing from Facet Bodies (Fit curve, Fit surface, Rapid surface)
- Rapid Visualization Overview (True Shading, Advanced Studio, Ray Traced Studio)

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Shipbuilding

Duration: 5 days

Who should attend: Users with shipbuilding background

Prerequisites: Designing Parts in NX

Follow up Training: NX Shipbuilding for admins or key-users

The NX Shipbuilding software provides a focused environment for modeling the structural area of a ship, enabling the user to easily conceptualize this structure and model the frames, decks, bulkheads, Stiffeners and Brackets.

NX Ship Structure provides the ability to design all kind of profiles for ship frames, floors and keelsons, all kind of sheets for compartment walls and all kind of edge reinforcements for support structures.

Once the steel structure is complete, NX Ship Structure aids in the assembly of the ship and in the manufacturing of parts.

This training is scheduled over 2 separate weeks in 3 + 2 days.

Day 1
- Overview Ship Structure
- Overview assembly structure for shipbuilding
- Overview Hull modeling in NX
- Ship Structure Concept Model
- Ship Structure Sectioning

Day 2
- Overview Ship Structure Basic Design
- Tools: Planar Ship Grid, Ship coordinates, Ship Container
- Plate Systems
- Stiffener Systems
- Edge Reinforcement Systems
- Seams + Transition Parts

Day 3
- Trace Lines, Longitudinal Stiffener Guides
- Shell expansion (Shell enrollment)
- Overview Ship Structure Detailed Design
- Plates
- Profiles, Stiffeners
- Edge Reinforcements
- Pillars
- Profile Cutouts, Split Plates/Profiles, End Cuts, Corner Cuts, Guide Cuts, Edge Cuts, Cutouts……

Day 4
- Copy Parts between Plates
- Standard Parts
- Overview Ship Structure Manufacturing
- Manufacturing Preparation (Attributes/Pos. No.)
- Cutting Side, Marking Lines, Reference Lines
- Excess Material, Rolling Lines, Pressure Lines

Day 5
- Plate Preparation incl. Shrinking and expansion of single curved and double curved plates
- Manufacturing XML Output (Export Cutparts)
- Material Allowance, Edge Cut Marking, Knuckled Profiles, Inverse Bending Lines, Profile List
- Templates
- Ship drafting functionalities
- Tips and tricks……

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Routing Mechanical

Duration: 2 days  
Who should attend: NX CAD Users and Designers  
Prerequisites: Designing Parts in NX  
Follow up Training: Routing Electrical

The NX Routing Mechanical class teaches you to utilize the tools used to quickly define paths placed around and through assemblies, to assign stock to these paths, and to qualify and place standard parts (for example, flanges, valves, and pipe tees).

These routings typically define the systems that provide process piping, disposal of waste, and structural support. The course also includes a section on developing logical diagrams.

Day 1
- Linear paths
- Routings with Heal path
- Routings with Stock
- Routings with Parts
- Routings with Assembly Components

Day 2
- Adding Parts
- Qualifying Parts
- Routing Systems Diagramming
- Runs and Spools

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Routing Electrical

Duration: 2 days
Who should attend: CAD Users and Designers
Prerequisites: Designing Parts in NX

The NX Routing Electrical class illustrates how to create connection and component lists; how to qualify parts for use in routing assemblies; how to place parts in a wiring assembly, or to create and edit wiring paths; how to assign components and connectors (manually and automatically); and how to create form boards.

Day 1
- Introduction Routing
- Creating and using connection lists and component lists
- Qualifying parts
- Placing parts
- Routing wiring segments

Day 2
- Assigning components, connectors and wire routing
- Adding overstock
- Creating form boards
- Practice project.

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NX Administration

**Duration:** 1 day  
**Who should attend:** System Administrators, CAD/CAM managers and Designers  
**Prerequisites:** Designing Parts in NX

The NX Administration training covers how to modify and configure an existing NX installation with some of the more commonly modified settings.

Topics include how to install/configure the Machinery Library, how to configure the File New dialog to see customer template files, how to convert a Part Family to a Knowledge Enabled Reuse object, and much more.

---

Day 1

- Setup of custom File -> New templates
- Creation of custom NX User and Group Roles
- Customization of the NX interface
- Creation of simple Macros linked to an icon
- Configuration of commonly used Environment Variables
- Configuration of the Machinery Library
- Setup of Reusable Objects
- Review of Customer Defaults used for NX customization and administration

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Teamcenter NX CAD Configuration

Duration: 5 days
Who should attend: Consultants, System Administrators and Key users
Prerequisites: Using Teamcenter, Teamcenter Integration for NX users, Teamcenter Data Model Administration and Teamcenter Installation (optional)

The Teamcenter NX CAD Configuration class is designed for those who have to install, configure or maintain NX CAD in combination with Teamcenter.

During the class we go through different aspects that are needed to know how the integration works. This class focuses on the NX CAD integration and does not(!) focus on any of the CAM or CAE aspects of the NX integration with Teamcenter. Nor do we handle the Teamcenter installation / configuration.

Day 1 and 2
- NX Modeling
  - Basic Modeling/Sketch explanation
  - Select Bundles
  - Roles / Pallets / Pax files
  - Hole feature custom / Reuse library
  - UDF
  - Customer defaults/preferences
  - Macro / Journaling / Tools > Customize / Movie
  - PDF/CGM/ import/export
  - Use of Material
- NX Assembly
  - Basic Assembly explanation
  - Multi CAD
  - Part families
  - Wave / Interpart expression
  - Attributes
  - Demo HD3D Tools / Demo Check mate
  - Demo Product template studio
- NX Drafting
  - Basic Drafting explanation
  - Templates
  - Symbols
  - Reuse library
  - Notes
  - Partlist
  - Tabular note

Day 3, 4 and 5
- Teamcenter NX CAD Configuration
- Adding the NX Manager feature onto a Rich Application Client
  - Add the “NX Manager for Rich Client” –feature to the Rich Application Client
  - Change the FCC cache size for working with large assemblies
- Teamcenter Integration for NX startup techniques
  - Configuring the NX startup
  - Modifying NX start message / Adding switches
  - Working with NX Customer Defaults
- NX Customer Defaults & Teamcenter Preferences
  - View the existing Teamcenter Preferences
  - Create/modify the startupscript for NX
  - Create/modify NX Customer defaults
- Queries
- Import/configuration of NX templates, pax files in Teamcenter
- Attribute Synchronisation
- Wave links configuration options
- UDF & Classification
- NX Assemblies & Structure Manager
- NX Drafting Partlist
- PDF’s en CGM’s
- MultiCAD in NX
- ugConfig options
- Plotting

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PLM Manager Awareness Training

Duration: 2 days
Who should attend: CAD/CAM Managers
Prerequisites: none

The PLM Awareness class aims to give you an overview and understanding of the PLM spectrum.

The Awareness sessions provide managers an overview of multiple PLM topics and how they can be used by End Users.

This will not only be an overview session with time for questions, but there will also be time for interaction within the NX software.

Day 1
- Introduction Siemens PLM software
- Introduction to NX Modeling and Teamcenter
- Opening and working with NX parts / Getting to know the NX interface
- Parametric design
- Creating parts with sketches and extrudes
- Sketch Overview
- Types of (Sketch) Constraints
- Sketch Dimensions
- Synchronous Technology
- Introduction to NX Assemblies
- Master Model principal
- Loading and working with assemblies
- Adding and positioning parts in an assembly / Assembly Constrains
- Revisions in Teamcenter
- Visual Analytics and Validation
- Introduction to NX Drafting Application
- Drawing Sheets
- Drafting Views / Detail Views
- Dimensions / Notes
- Product and Manufacturing Information (PMI)

Day 2
- Introduction Computer-Aided Manufacturing (CAM)
- Integrated Solution / CAD for CAM
- CAM Template
- The Operation Navigator / CAM Views
- Program / Tools / Geometry / Method
- Type Milling operations / Cavity Milling
- Turning
- Visualization / Post processing and Simulation
- CMM-programming

More information?
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CAD Hothouse

**Duration:** 10 days  
**Who should attend:** Design Engineers  
**Prerequisites:** none  
**Follow up Training:** Large Assembly Management, Freeform Modeling and Sheet Metal

The NX CAD Hot House provides you a fast and thorough training in NX CAD.

It is a combined and comprised training containing the following topics: Modeling (Sketch), Assemblies, Drafting, Synchronous Technology and Teamcenter Integration for NX Users.

The learned knowledge will be put into practice in a project.

**Day 1 - 10**
- 2.5 days: Basic Modeling (Sketch)  
- 0.5 day: Synchronous Modeling  
- 2.5 days: Assemblies  
- 1.5 days: Drafting  
- 1 day: Teamcenter Integration for NX Users  
- 2 days: Project

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CAD Update

**Duration:** .. days
**Who should attend:** CAD Users and Designers
**Prerequisites:** Designing Parts in NX

The NX CAD Update training teaches you the new functionalities in NX CAD. The amount of days depends on the topics.

**Objectives / Topics:**
- Fundamentals / Gateway
- Sketch
- Modeling
- Synchronous Modeling
- Assemblies
- Drafting
- PMI
- Freeform modeling
- Sheet Metal
- Routing

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Open API

**Duration:** 1 – 2 days

**Who should attend:** Application developers interested in creating NX Open API programs

**Prerequisites:** Designing Parts in NX

The NX Open API training teaches the basics of interfacing with the Common API through Visual Basic.NET, C#, C/C++, Python and Java. Journaling and ribbon bar customization are also covered.

The number of training days depends on the knowledge of programming and programming needs.

**Day 1 / 2 Objectives**

- Understanding navigating the common API
- Using the journal tool
- Turning journals into applications
- Understanding runtime license control

**More information?**

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Product and Manufacturing Information (PMI)

Duration: 1 day
Who should attend: CAD Users, Designers and CAD/CAM Managers
Prerequisites: Basic NX knowledge

The Product and Manufacturing Information (PMI) training is an NX task environment for attaching non-geometric information to a part file. You can attach information needed by downstream applications such as Drafting, Manufacturing, Inspection, and shipping. The information can be text, dimensions or symbols.

Day 1
- Managing views and preparing your model for PMI
- Creating PMI dimensions and centerlines
- Adding GD&T PMI to your model
- Making adjustments to PMI
- Adding notes, symbols, and section views to your model
- Adding process and proprietary information to your model
- Querying and validating PMI
- Displaying PMI in drawings
- Reusing PMI in an assembly

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NX Render

Duration: 1 days
Who should attend: Designers
Prerequisites: Designing Parts in NX

The NX Render training focuses on rendering techniques in NX.

The class focuses on how to assign specific properties to render model geometry in NX, for example assigning materials, modifying material properties, adding light and setting light variables. Also using shadow techniques and setting backgrounds is discussed during the training. Furthermore the projection of images on model geometry is addressed, as well as different rendering methods like ray traced and photo-realistic.

Day 1
- Introduction Advanced and Ray Traced Studio
- Creating Showroom Environment
- Assigning and Editing Materials Texture and layers
- Shading Methods
- Adding Environment Options
- Assigning and Editing Light sources
- Using Image Based Lightning
- Shadow Control
- Parameter Overview
- Using and editing Visualization Scenes...

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NX Shipbuilding for Admins or Key Users

Duration: 1 day
Who should attend: Training is addressed to users with responsibility for settings, library etc.
Prerequisites: Designing Parts in NX (Modeling)

NX Ship Design software provides a focused environment for modeling the structural area of a ship, enabling the user to easily conceptualize this structure and model the frames, decks and bulkheads.
NX Ship Design provides the ability to design linear and nonlinear profiles for ship frames, linear and nonlinear sheets for compartment walls and linear and nonlinear belts for support structures between walls. Once the steel structure is complete, NX Ship Design aids in the assembly of the ship and in the manufacturing of parts.

Day 1
• Cutting side
• straightbrake parts
• marking lines
• reference lines
• user text
• shrinking
• plate expansion
• profile list
• manufacturing xml output
• Settings in xml files and configs

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Product Template Studio

Duration: 1 day  
Who should attend: Designers and Engineers  
Prerequisites: Designing Parts in NX (Modeling and Assemblies)

The Product Template Studio Author application lets you create a product template from an NX parametric part or assembly. A product template is a parametric model that contains custom NX dialog boxes.

In this class the basics about NX Product Template Studio (PTS) are explained and practiced.

Day 1
- What is PTS?
- Product Template overview
- Use an existing Product Template
- Product Template Studio Layout
- Product Template Studio Creation Workflow
- Create a Basic Product Template
- Add advanced design controls
- Add a visual rule
- Use and build product template assemblies...

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# NX Manufacturing (CAM)

The learning tracks have been provided to help you plan a productive development path through our courseware.

Learning tracks start from the top and progress downward. Select any course below to open the training content.

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1. The [Designing Parts in NX](#) training may be taken in place of the CAD for CAM training.

For more NX Manufacturing (CAM) courses see next page:
NX Manufacturing (CAM)

Select the Course for more information:

- PLM Manager Awareness Training
- CMM Inspection Programming
- Customization for Manufacturing
- Manufacturing Fundamentals Shorttrack
- Fixed Axis Machining Shorttrack
- Open API
- CAM Update
The NX CAD for CAM training is to help everyone who is struggling to get their NX parts prepared for Manufacturing. In this training you will learn to create, modify and simplify geometry for Manufacturing Purposes.

**Duration:** 3 days  
**Who should attend:** CAM users  
**Prerequisites:** none  
**Follow up Training:** Manufacturing Fundamentals

**Day 1**
- Opening and working with parts  
- Getting to know the NX interface  
- Impact of coordinate systems on parts  
- Creating parts with sketches  
- Sweeping geometry to create part features  
- Editing and manipulating sketches  
- Edge operations  
- Hole feature  
- Datum Features  
- Measure

**Day 2**
- Introduction to Assemblies  
- Master model principle  
- Layers / Show and Hide  
- Reference sets  
- WCS-MCS  
- Assembly load options  
- Assembly navigator  
- Modeling top down, bottom up  
- Design in context  
- Replace component  
- Move component  
- Assembly constraints  
- Reuse Library

**Day 3**
- Introduction to Drafting  
- Views – Dimension  
- Section view in Modeling  
- Introduction to Wave  
- Wave geometry linker  
- Synchronous modeling

**More information?**
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NX Manufacturing Fundamentals

Duration: 3 days
Who should attend: NC/CNC programmers; CAD/CAM managers; Manufacturing engineers
Prerequisites: CAD for CAM
Follow up Training: Fixed Axis machining

The NX Manufacturing Fundamentals training provides the NC programmer the ability to gain a detailed insight into the powerful 2 and 2.5-axis drilling and milling modules in NX.

Besides the basics this training also teaches the 2D rest milling and planar milling. These modules enable the programmers to create drilling and milling programs in a structured way.

Furthermore the emphasis will be on creating an NC file and generating documents for the workplace. Previously this training was known as Basic Manufacturing.

Day 1
- Master model principle
- Workflow
- CAM User Interface
- Create manufacturing setup
- Manufacturing process incl.
- Operation navigator
- Create operation
- Tool definition
- Parent group work piece
- Coordinate systems
- Cavity mill essentials
- Cut levels
- Cut patterns
- Visualization (basics)
- Floor Wall milling
- Stock options
- NC assistant

Day 2
- Planar milling
- Boundaries
- Planar milling cut levels
- Hole making / Point to Point (Drilling)
- Drilling tools
- Post Processing basics
- Shop Documentation basics

Day 3
- Floor Wall milling
- Floor Wall milling multi faces
- Planar Milling profile advanced
- Face milling Mixed
- Hole Milling
- Groove milling
- Text engraving / Planar text
- Multiple Part Programming
- Tool library

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Turning

Duration: 2 days  
Who should attend: Manufacturing Engineers, NC/CNC programmers, CAD/CAM managers  
Prerequisites: CAD for CAM and optionally Manufacturing Fundamentals (Basic Manufacturing)

The NX Turning training is the core turning class designed to convey concepts, functionality, and application of the turning module.

Turning is taught from the perspective of an NC/CNC programming session and emphasizes programming concepts and techniques which take advantage of the latest developments in turning equipment and technology.

Day 1 / 2 Objectives

- Introduction & Course Overview
- Define part & blank geometry
- Retrieve & create tools
- Face operations
- Verification
- Common options
- Centerline operations
- Rough operations - OD & ID
- Finish operations - OD & ID
- Groove operations
- Teach mode operations
- Thread operations
- Mill-Turn
- Projects

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Fixed Axis machining

**Duration:** 3 days  
**Who should attend:** NC/CNC programmers, CAD/CAM managers, Manufacturing engineers  
**Prerequisites:** Manufacturing Fundamentals  
**Follow up Training:** Multi Axis Techniques

The NX Fixed Axis Machining training provides the ability to gain a detailed insight into the powerful 3-axis milling modules in NX. The modules enable the programmers to create milling programs in a structured way of prismatic and free form models.

The module for processing residual material makes it possible to work the milling areas with a large milling cutter, after which the residual material will automatically be removed with a smaller tool. The training also elaborates on the High speed milling options within the milling modules. Previously this training was known as Advanced Manufacturing.

**Day 1**
- Advanced cavity milling topics
- Cut levels
- In process work piece
- Pre drill engage points
- Stock options
- Z-level milling
- Z-level geometry

**Day 2**
- Fixed contour (basics)
- Fixed contour geometry
- Area milling options
- Cut region management
- Non cutting moves
- Surface area options
- Streamline area options

**Day 3**
- Fixed contour non steep angle
- Remaining fixed contour driving options
- Z-level steep angle
- Z-level cut between levels
- Plunge milling
- Flow cut
- Flow cut options
- Radial cut
- High speed machining
- Path shape in corners
- Ramp on part
- Mixed cut direction
- Trochoidal cut pattern
- Profile 3D
- Solid profile 3D

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Multi Axis Techniques

Duration: 2 days
Who should attend: NC/CNC programmers, CAD/CAM managers, Manufacturing engineers, NC Planners
Prerequisites: Manufacturing Fundamentals and Fixed Axis Machining

The Multi-axis Milling course is designed for NC/CNC programmers who machine complex parts with variable tool capabilities. Students will learn how to create variable axis tool paths. You will also be introduced to NX workflows for machining contoured parts.

Day 1 / 2 Objectives
• Introducing to four and five axis machining
• Variable Contour - basics
• Contour Profile
• Projection Vectors
• Five Axis Z Level
• Sequential Mill basics
• Variable Contour - advanced
• Variable Streamline
• Generic Motion

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Post Building Techniques

Duration: 4 days
Who should attend: Manufacturing engineers, system managers and NC/CNC programmers
Prerequisites: Knowledge of NC/CNC programming basics, knowledge of NC/CNC machine/controller functionalities, NX Manufacturing Fundamentals

The Post Building Techniques training is designed to acquaint users with tools and techniques that are used for building custom, machine tool specific postprocessors.

This training teaches the necessary skills that are needed to build most machine tool specific post-processors using the Post Builder tool.

For those postprocessors that cannot be fully developed using the Post Builder tool, methods are shown for customization and modification of the Definition and Event Handler files, utilized by POST/execute for specific needs.

Day 1-4 Objectives
• NX Post post-processor
• Building a postprocesser with the Post Builder
• Post Builder for Wire EDM applications
• Post Builder for 5-axis mill and lathe applications
• Creating mill-turn post processors
• Tcl basics for post processors
• Customizing post processors with post builder
• User defined events
• Virtual NC Controller
• A guide to best practices of building a post processor
• Custom command examples
• Advanced post building topics

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CMM Inspection Programming

**Duration:** 3 days  
**Who should attend:** CMM Inspection Programmer  
**Prerequisites:** Designing Parts in NX (Modeling and Assemblies)

NX CMM Inspection Programming is an offline programming application that helps you program faster and more accurately by putting you within the context of a 3D solid model environment including the CMM machine, probing equipment and the part.

Use this application to reduce programming time, visually validate CMM programs, free up expensive CMM machine resources, and respond to design changes rapidly.

**Day 1**
- Introduction to QPPV Quality Management Solution  
- NX CMM/DMIS Program Workflow requirements  
- Review NX CMM User Interface  
- Starting NX CMM  
- Creating Features and Tolerances from Linked PMI  
- Creating Inspection Paths

**Day 2**
- Creating Features and Tolerances (without PMI)  
- Tool path simulation and verification  
- Part alignment  
- Machine Coordinate System (MCS)  
- Save PCS  
- Recall PCS  
- Constructed Features  
- DMIS Program Header parameters  
- Post-processing DMIS output

**Day 3**
- Part Alignment Techniques  
- Constructed Features  
- Complete CMM Program A-Z

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Customization for Manufacturing

**Duration:** 1 day

**Who should attend:** NC/CNC programmers, CAD/CAM managers, Manufacturing engineers, NC Planners

**Prerequisites:** Knowledge of NC/CNC programming basics, knowledge of NC/CNC machine/controller functionalities, NX Manufacturing Fundamentals

This Training is useful for CAM users who want to customize the NX CAM environment.

By using templates the productivity will be increased when using NX Manufacturing.

**Day 1**
- CAM Session Environment
- CAM setup / template parts
- Copy and Paste Operation
- Tool/machine/feed and speeds library
- Fully shade toolassemblies
- Hole making
- Manufacturing wizards
- Master model concept

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Manufacturing Fundamentals Shorttrack

**Duration:** 2 days  
**Who should attend:** NC/CNC programmers; CAD/CAM managers; Manufacturing engineers  
**Prerequisites:** CAD for CAM  
**Follow up Training:** Fixed Axis Machining

This training is the short version meant for the experienced user. Information about the complete training can be found [here](#).

The NX Manufacturing Fundamentals training provides the NC programmer the ability to gain a detailed insight into the powerful 2 and 2.5-axis drilling and milling modules in NX.

Besides the basics this training also teaches the 2D rest milling and planar milling. These modules enable the programmers to create drilling and milling programs in a structured way.

Furthermore the emphasis will be on creating an NC file and generating documents for the workplace. Previously this training was known as Basic Manufacturing.

**Day 1**  
- Master model principle  
- Workflow  
- CAM User Interface  
- Create manufacturing setup  
- Manufacturing process incl.  
- Operation navigator  
- Create operation  
- Tool definition  
- Parent group work piece  
- Coordinate systems  
- Cavity mill essentials  
- Cut levels  
- Cut patterns  
- Visualization (basics)  
- Floor Wall milling  
- Stock options  
- NC assistant

**Day 2**  
- Planar milling  
- Boundaries  
- Planar milling cut levels  
- Hole making / Point to Point (Drilling)  
- Drilling tools  
- Post Processing basics  
- Shop Documentation basics

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Fixed Axis Machining Shorttrack

**Duration:** 1 day

**Who should attend:** NC/CNC programmers, CAD/CAM managers, Manufacturing engineers, NC Planners

**Prerequisites:** Knowledge of NC/CNC programming basics, knowledge of NC/CNC machine/controller functionalities, NX Manufacturing Fundamentals

The NX Fixed Axis Machining training provides the ability to gain an insight if the powerful 3-axis milling modules in NX. The modules enable the programmers to create milling programs in a structured way of prismatic and free form models. The module for processing residual material makes it possible to work the milling areas with a large milling cutter, after which the residual material will automatically be removed with a smaller tool. The training also elaborates on the High speed milling options within the milling modules. Previously this training was known as Advanced Manufacturing.

**Day 1**

- Non cutting moves
- Z-level milling
- Z-level geometry
- Fixed contour basics
- Fixed contour geometry
- Flow cut
- Contour area
- Flow cut options
- Area milling options
- Surface area options
- Remaining fixed contour driving options

**More information?**

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CAM Update

Duration: .. days
Who should attend: CAM Engineers
Prerequisites: NX Basic Manufacturing, Advanced Manufacturing en Multi Axis
Follow up Training: Turning update, ISV update

The NX CAM Update training teaches you the new functionalities in NX CAM. The amount of days depends on the topics.

Objectives / Topics:
• Fundamentals / Gateway
• NX Basic Manufacturing
• Advanced Manufacturing
• Multi Axis

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