



Course: NX Essentials for CATIA Users

Total Duration: 30 hours

Mode: Hybrid

Software: Siemens NX Ver. 2412

Module I. Introduction and Design of components in NX (18 hours)

This module focuses on the differences and similarities between CATIA and NX, allowing CATIA users to quickly adapt to the NX environment.

- Introduction to the software (user interface, tabs, visualization, comparison with CATIA)
- Model manipulation (keyboard and mouse shortcuts in NX vs CATIA)
- NX CAD Methodology used by OEMs in their design process.
- Correct application of selection filters and rules
- Role configuration and UI customization (icon size, colors, shortcuts, differences with CATIA)
- Creation of datums (planes, axes, points) and their equivalence in CATIA.
- Sketch positioning and definition (comparison with CATIA Sketcher)
- Solid creation and editing (extrude, revolve, swept, shell, offset), highlighting command differences with CATIA
- Solid detailing (edge fillet, drafts, sheet)
- Solid patterns (holes, patterns, threaded holes)
- Boolean operations and comparison with CATIA Boolean functions.
- Use of Reference Sets and Feature Groups (comparison with CATIA Body and Geometrical Set commands)
- Editing basic parts using Synchronous Modeling.
- Measurement analysis (methods, center of gravity, mass).
- Material assignment.
- Creation of basic parts using surfaces and differences versus CATIA.
- Best practices for flexible and reusable modeling based on OEMs design processes.

Module II. Assembly Context Design (6 hours)

This module emphasizes NX tools for working with assemblies and how they differ from CATIA.

- Assembly generation (add/replace/find components, manipulation, visualization)
 - Assembly constraints (constraints, degrees of freedom, comparison with CATIA constraints)
 - Geometry creation in context using WAVE Geometry Linker.
 - Assembly analysis (section cuts, interference analysis)
 - Creation of Arrangements
-

Module III. Drawing Generation (6 hours)

This module compares NX drawing functions with CATIA Drafting to ease the transition.

- Product drawing creation.
- View creation (automatic, projected, front, side, isometric)
- Dimension generation (linear, radial, angular)
- Dimensions with symbols, notes, and tolerances
- Assembly drawing creation (show and hide components)
- Bill of Materials (BOM) creation
- Balloon creation

Suggested Mode:

- Online, asynchronous with doubt resolution sessions.
- Each participant will use their own software license for practice, ensuring access to professional tools and continuous model updates.
- Evaluation: Participation 80%, Assessments 10%, Final Project 10%.