



NX CAD for Mechanical Engineers

Duration: 50 Days

Syllabus Content

Module 1: Introduction to NX Fundamental

- Opening And Working with Parts
- Co-ordinate systems
- NX Interface
- Sketcher window
- Save the component
- Sketcher tool And Use
- Manipulating Commands for Sketcher
- Extrude And Revolve feature
- Draft And Shell Feature
- Blend and chamfer option
- Holes with types
- Split and Add body
- Coping and mirroring Part Segments
- Modifying geometry of imported parts
- Examining the structure of a model

Module 2: Introduction to Assembly

- Top Down and bottom up type
- Add the Parts in Assembly
- Constraints the Parts
- Cross-section view
- Hidden parts
- Manipulating the parts
- Assembly Navigator
- Apply Material and check his properties
- Modifying existing part in an assembly
- Moving parts
- Cross-section of an assembly

Module 3: Introduction to Drafting

- Drafting overview
- Part Navigator
- Master model drawings and drafting standards
- Drawing sheets
- Drafting views
- Custom views
- Move, copy, and align views
- Hiding geometry in drafting views

- Updating drawings and drafting views
- Centerline symbols
- Dimensions
- Broken views
- Break-out section views
- Notes and labels
- Parts lists
- Balloon symbols
- GD&T symbols
- Surface finish, weld, and custom symbols
- Section views
- Editing section lines
- Maintaining associativity
- Detail views
- View boundaries
- Exploded views
- Converting drawings to master model

Module 4: NX Synchronous Modeling Fundamentals

- Basic concepts of Synchronous Modeling
- Modify Face
- Detail Feature
- Delete Face
- Adaptive Shell
- Edit Cross Section and Edit Section
- Reuse commands

Module 5: NX Sheet Metal

- Sheet Metal workflow
- Define the basic shape of the part
- Constructing base features
- Sheet Metal corners
- Sheet Metal cutouts
- Sheet Metal deform features
- Flat Solid and Flat Pattern
- Advanced Sheet Metal commands
- Analyze Formability – One step
- Aerospace Sheet Metal
- Working with non-sheet metal data
- Saving Files

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