

NX-CAD

Introduction-

- Introduction to Unigraphics NX
- NX Graphical User Interface
- File management

Sketch-

- Creating Sketches
- Editing sketches
- Constraints

MODELING FEATURES-

- Basic terminologies
- Feature modeling concepts
- About Datum CSYS and Datum Planes
- changing units in NX
- Feature modeling commands

MODELING-

- Feature modeling commands
- Creating Primitives

DATUM. MODIFY TOOLS-

- Creating Datum Features
- Datum Plane
- Datum axis
- Datum CSYS
- Datum Point

- Creating Sweep Features
- Sweep one or more sections along guide curves

- Creating Sweep Features
- Blend Features
- Applying Chamfer

- Feature Modeling Commands
- Hole

- Boss
- Pocket
- Pad
- Emboss
- Slot
- Groove

- Feature Modeling Commands
- Dart
- Thread
- Shell
- Draft
- Feature Operations
- Instance feature
- Mirror feature
- Instance geometry creation
- Divide face
- Trim body
- Split body
- Measure distance between geometries
- find geometric properties
- Synchronous Modeling
- Introduction to Assembly Modeling
- Types of approach
- Placing components
- Assembly Constraints
- Creating component arrays

Assembly Modeling-

- Moving a component
- Replacing component
- Repositioning component
- Mirroring assembly
- Creating a New Component
- Creating new parent
- Finding assembly
- Clearance
- Creating exploded views
- Assembly sequencing with motion
- Creating deformable parts
- Finding degrees of freedom
- Assembly envelopes

Introduction to Drafting. Annotation. Detailing-

- Introduction to drawing
- Graphical User Interface of drafting
- NX drafting methods
- Creating drafting views
- Detail view
- move/Copy a view
- align a view
- hide/show components
- Creating section in view
- Section views
- Adding dimensions
- Adding annotations
- Adding Feature control frame
- Inserting Identification symbol
- Placing tabular note
- Listing parts list
- Creating auto balloon

Surface modeling-

- Surface Modeling commands
- Surface using curves
- Surface generation
- curves from curves
- curve from bodies
- associative copies
- Editing SURFACE geometries

Sheet Metal-

- Sheet metal Design – BASICS
- About NX Sheet Metal Preferences
- creating base feature
- Creating Bend feature
- Corner definition
- Applying punch operations
- Sheet metal cut operations
- Resizing bend radius