



Curriculum Of NX-CAD

INTRODUCTION TO INTERFACE OF NX CAD

- Mouse function
- Basic commands (Save, Open, Import, Export)
- Detailed Concept of CAD
- Need & Importance Of CAD
- Overview About Actual Designing In Industries, Fundamentals Of Design & Its Implementation Methods
- All Characteristics Of NX To User Friendly Atmosphere
- Superiority Of NX With Its Use And Demand In Industries
- Important Terms and Definitions
- Workbenches In NX
- Adjusting The NX Interface
- Understanding the Functions of the Mouse Buttons
- Graphic User Interface of NX
- Menu And Toolbars
- Opening Files
- Creating New Files
- Keyboard Shortcuts
- Selecting/Moving Objects With Mouse
- Working with Datum/planes
- Properties Toolbar
- Changing The Properties
- Changing The Interface from 3d Modeling To 2d Sketching And Vice-Versa
- Uses & Description About Feature Manager Design Tree & Resource Bar.
- Working with Respect to Ucs.
- Setting Up The Document Options

SKETCHER

- Getting Started With Sketch
- Creating Centerlines
- Constructing Lines
- Constructing Ellipse
- Constructing a Circle
- Constructing an Arc
- Creating Polygon, Conic & Spline
- Equation Driven Curve
- Point
- Creating Text
- Creating Construction Geometry



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EDITING SKETECHES

- Sketch Fillet
- Sketch Chamfer
- Offset Curve
- Offset Move Curve
- Converting Entities
- Trim
- Extending Entities
- Mirror Curve
- Moving Curve
- Copying the Sketch Entities
- Rotating Sketch Entities
- Scaling Sketch Entities
- Stretching Sketch Entities
- Modify Sketch
- Close Sketch Of Model
- Sketch Pattern Curve
- Geometric Constraints
- Automatic Relations
- Conflicts in Relations
- Dimensioning
- Dimension Property Manager
- Exiting The Sketch

PART MODELING

- Terminologies Used In Part Modelling Environment
- Entering The Part Module
- Choosing The Sketch Plane
- Extrude
- Revolve
- Sweep Features
- Mesh Features
- Cut Features
- Selecting Geometrics In NX

REFERENCE GEOMETRY

- Reference Datums
- Creating Datums
- Creating Datum Axes



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- Creating Reference Points
- Creating Datum CSYS
- Editing Reference Geometries
- Creating Curves

PLACED FEATURES

- Creating Simple Holes
- Creating Standard Holes Using The Hole Wizard
- Creating Fillets
- Creating Chamfers.
- Creating Shell Features
- Creating Rib Feathers
- Creating Draft Feature
- Creating Pattern

ASSEMBLY MODELING

- Types of Assembly Design Approaches
- Positioning the Components in Assembly
- Assembly Constraints
- Types of constraints
- Mate Reference
- Replacing The Assembly Components
- Rotating A Component
- Moving Components
- Detecting Interference
- Assembly Pattern
- Assembly Mirror
- Degree of freedom of components.
- Creating Exploded View
- Physical Simulation
- Configuration in Assembly

SURFACE MODELING

- Creating Surface Feature
- Creating Studio Surface
- Creating Extrude Features
- Creating Revolved Surface
- Creating Swept Surface
- Variational Sweep



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- Sweep Along Guide
- Style Sweep
- Creating Through Curves
- Creating Through Curve Mesh
- Creating Fill Surface
- Creating Planar Surface
- Creating Offset Surface
- Creating Variable Offset Surface
- Extending A Surface
- Creating Ruled Surfaces
- Trimming Surface
- Replacing & Deleting Faces
- Un-Trimming & Sew/Unsew surface

SHEET METAL MODELING

- Fundamentals Concepts of Sheet Metal
- Using Sheet Metal Tools
- Creating Base Tab & Secondary Tab
- Creating Flange
- Creating Contour Flange
- Hem Flange
- Creating Jog
- Advanced Flange
- Bridge Bend
- Creating Break Corner/Corner-Trim
- Creating Closed Corners
- Creating Three Bend Corner
- Creating Rip
- Creating Sketched Bend
- Creating Unfold/Fold
- Flattening Sheet Metal Bends
- Dimple, Louver, Drawn Cutout, Bead, Solid Punch, Gusset
- Welded Corner
- Inserting Corner Trim
- Creating Sheet Metal Part by Converting A Solid Body

DRAWING VIEWS & DETAILING

- Introduction of Drawing
- Need & Importance of Drawing



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- Starting The Drawing Workbench
- Defining The Sheet & Sizes
- Adjusting of Drawing Sheet According to Object/Assembly
- Types of Projection
- Using Predefined Drawing Styles

DRAWING VIEWS

- Creating Drawing from Part Or Assembly
- Creating A New Drawing Document
- Generating Standard Views
- Derived Views
- Creating Broken Views
- Working With Assembly Specific View
- Drawing View Properties

DETAILING

- Creating Dimensions
- Creating Model Dimension
- Creating Rapid Dimensions
- Creating Linear Dimension
- Creating Chamfer Dimension

DATA EXCHANGE

- Converting Files For Transferring
- Converting Into IGES, STEP, PARASOLID Etc.
- Convert Into Jpeg, Mpeg, Tiff, Pdf Files