ZENUS INFOTECH INDIA PVT. LTD. CURRICULUM OF NX-CAD

Course Duration: 6/8 Weeks*

Introduction to CAD, CAE, PDM

Features of UG-NX,

Various products available in UG-NX for Product Design, Simulation, Communication

NX-CAD User Interface - Feature manager design tree, Callouts, Handles, Confirmation corner, mouse buttons, keyboard shortcuts, Command Manager,

Hardware and Software requirements, UG-NX Task Scheduler.

Sketch Entities – Inference line, Centreline, Line, Circle, Arc, Ellipse, Rectangle, Slots, Polygon, Parabola, Ellipse, Partial Ellipse, Spline, Spline tools, Spline on surface, Equation driven curve, Points, Text, Construction geometry, Snap, grid,

Sketch Tools - Fillet, Chamfer, Offset, convert entities, Trim, Extend, Split, Jog, Mirror, Dynamic Mirror, Move, Copy, Rotate, Scale, Stretch, Sketch pattern

Make path, Close Sketch to Model, Sketch picture, Check Sketch for Feature, Area hatch/Fill

Blocks – Make block, edit block, insert block, Add/Remove Entities, Rebuild, Save, Explode

Relations - Adding Sketch Relation, Automatic relations,

Dimensioning - Smart, Horizontal, Vertical, Ordinate, Horizontal ordinate, Vertical ordinate, Align ordinate, Fully define sketch.

Sketch Diagnosis, SketchXpert, 3D Sketching, Rapid Sketch

Part Modeling Tools

Creating reference planes

Creating Extrude features – Direction1, Direction2, From option, Thin feature, Applying draft, Selecting contours

Creating Revolve features – Selecting Axis, Thin features, Selecting contours

Creating Swept features-Selecting, Profile and Path, Orientation/twist type, Path Alignment, Guide Curves, Start/End tangency, Thin feature

Creating Loft features – Selecting Profiles, Guide curves, Start/End Constraints, Centerline parameters, Sketch tools, Close loft.

Selecting geometries – Selection Manager, Multiple Body concepts

Creating Reference - points, axis, coordinates

Creating curves -

Split curve, Project curve, Composite curve, Curve through points, Helix and Spiral

Creating Fillet features

Inserting Hole types

Creating Chamfer

Creating Shell

Creating Rib

Creating Pattern - Linear pattern, Circular pattern, Sketch driven pattern, Curve driven pattern, Table driven pattern, Fill pattern, mirror

Advanced Modeling Tools- Dome, Free form, Shape feature, Deform, indent, Flex

Inserting Fastening features- Mounting boss, snap hook, Snap hook groove, Vent

Environment & Utilities - Working with views and manipulating views, Trouble shooting

Inserting Library feature, Adding Configuration, Inserting Design table, System options, Measuring Geometries, Calculating Mass Properties, Feature Statistics, Working With Equations

Assembly Modeling Tools

Introduction to Assembly Modeling & Approaches – Top down and Bottom up approach

Applying Standard Mates- Coincident, Parallel, Perpendicular, Tangent, Concentric, Lock, Distance, Angle.

Manipulating Components - Replacing Components, Rotating Components, Move Components, Collision Detection, Physical Dynamics, Dynamic Clearance, Detecting Interference

Creating Pattern - Assembly Pattern, Mirror

Creating Explode Views

Top Down Design – Layout Sketch, Work Part In the Context of an assembly.

Smart Components, Smart Fasteners, Physical Simulation

Surface Modeling tools

Creating Extrude, Revolve, Swept, loft, Boundary surface.

Inserting Planar Surface, Offset Surface, Radiate Surface.

Extending a surface, Surface fill, Ruled Surface, Trimming Surface, Mid surface, Replace Face, Delete face, Untrim surface, Knit surface, Thickening a Surface, Move Face

Generating Drawing Views

Introduction To Angle Of Projection

Generating Views - Generating Model View, Projected Views, Inserting Standard 3 View

View creation relative to model, Inserting predefined views, empty views, Auxiliary Views, Detailed Views, Crop view, Broken –Out Section, Broken Views, Section View, Aligned Section View, Alternate Position View, Working assembly specific view, Drawing properties, Manipulating views

Creating Dimensions – Smart, Horizontal, Vertical, Baseline, Ordinate, Horizontal Ordinate, Vertical Ordinate, Chamfer, Attach Dimensions, Align Collinear/Radial, Align Parallel/Concentric, Model Dimensions, Auto dimension, DimXpert, Annotations, Spell check

Inserting Annotations - Datum Features, Geometric Tolerance, Surface Finish, Jog Leaders, Hole Callout, Datum Target, Dowel Pins, Area Hatch, Cosmetic Thread, Balloon, Centre Mark, Centre Lines, Layers, Working With Tables, Bill Of Materials, Hole Table, Sheets And Templates, Sheet Format.

Sheet Metal Design

Concepts in Sheet metal design bend allowance bend deduction, K-factor

Inserting Base Flange, Sheet Metal Tab, Edge Flange, Miter Flange, Hem, Jog.

Creating Break Corner/Corner Trim, Closed Corners, Rip.

Inserting Sketched Bend, Fold/Unfold, Forming Tools.

Inserting Cross Break, Welded Corner.

Adding Corner Trim, Lofted bend

Conversion Of Solid Body To Sheet Metal.

Working with import data - Importing In UG-NX, Editing Imported Features, Feature Recognition, 2d To 3d Conversion

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