NX CAD Training Program

TCA

TRAINING & DEVELOPMENT

GET TRAINED BY

EXPERIENCED TRAINER

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- NX Graphical UI
- Modelling Commands
- Sweep Feature
- Types of Configuration
- Sheet Metal Design

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TCA Training & Development

NX CAD Training Program

NX Total Duration: 80 Hour	
Session	**Topics**
Session 1	**Introduction to Unigraphics NX 8.0, About NX Gateway, Getting Started** **NX Graphical User Interface -** Title bar, Menu bar, Toolbar, Radial toolbar, Selection bar, Cue and status line, Dialog rail, Resource bar, Navigators,HD3D tools, Integrated browser, Palettes, Roles, Full screen, View orientation- trimetric, isometric, View commands, Rotate ,Pan, Zoom in/out, Quick pick, Quick pick, categories, Coordinate system- absolute coordinate system, WCS, Absolute coordinate, Work coordinate system. View triad, Multiple graphics window, Information window, Keyboard accelerators, Dialog box **File management -** Creating new files and about templates, Opening files, Saving files
Session 2	**Creating Sketches -** Profile, Line, Arc, Circle, Fillet, Chamfer, Rectangle, Polygon, Studio Spline, Fit spline, Ellipse, Conic **Editing sketches -** Quick trim, Quick extend, Make corner, Offset curve, Pattern curve, Mirror curve, Intersection point, Derived lines **Constraints -** Geometric constraints, Auto constraint, Inferred constraint, Dimensional constraints, Auto dimension, Animate dimension, Continuous auto dimension
Session 3	**Basic terminologies -** Feature, Body, Solid body, Sheet, Face, Section curves, Guide curves **Feature modeling concepts** **About Datum CSYS and Datum Planes** **Changing units in NX** **Feature modeling commands** **Creating Extrude features** o Extrusion by selecting a section of edges o Limits option o Boolean operation o Applying draft o Offset sketches
Session 4	***Feature modeling commands -*** **Creating Revolve features** o Revolve sketch about an axis

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	○ Limits option ○ Offset sketches **Creating Primitives -** Block, Cylinder, Cone, Sphere
Session 5	***Creating Datum Features* -** Datum Plane, Datum axis, Datum CSYS, Datum Point
Session 6	***Creating Sweep Features* -** Sweep a section along a guide ○ Adding thickness Sweep one or more sections along guide curves ○ Section sweep using spine ○ Section location ○ Interpolation ○ Alignment ○ Orientation method ○ Scaling method
Session 7	***Creating Sweep Features*** Creating Variational Sweep ○ Limits option ○ Adding secondary sections Creating Tube feature **Blend Features** Creating Edge blend ○ Constant radius ○ Variable radius Creating Face blend ○ Types of blend **Applying Chamfer**
Session 8	***Feature Modeling Commands* -** **Hole -** General hole, Drill size holes, Screw clearance holes, Threaded holes **Boss**

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	Pocket - Cylindrical, Rectangular, General **Pad -** Rectangular, General **Emboss -** Offset emboss **Slot -** Rectangular, Ball end, U-Slot, T-Slot, Dove tail **Groove -** Rectangular, Ball end, U-groove
Session 9	***Feature Modeling Commands -*** **Dart, Thread, Shell** **Draft -** From plane, From edges, Tangent to faces, To parting edges, Draw direction, Variable draft points **Draft body, Scale**
Session 10	***Feature Operations -*** **Instance feature -** Rectangular array, Circular array, Pattern face **Mirror feature, Mirror body** **Instance geometry creation -** From bodies, From faces, From faces, From edges, From curves, From points
Session 11	***Feature Operations -*** To Divide face, Trim body , Split body, Boolean commands, User defined feature, Creating Feature group, Editing Expressions, Knowing Feature replay command, Knowing Layer settings, To measure distance between geometries, To measure angle between geometries, To measure bodies and face geometries, To find geometric properties
Session 12	***Feature operations and smart tools*** Synchronous Modeling ***Introduction to Assembly modeling*** **Types of approach** Bottom up assembly modeling Top down assembly modeling **Placing components**
Session 13	**Assembly Constraints**

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	Angle, Bond, Centre, Concentric, Distance, Fit, Parallel, Perpendicular, Touch align **Creating component arrays** Linear array, Circular array, Feature instance array
Session 14	***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
Session 15	***Introduction to drawing*** **Inserting new sheets, Editing sheets** **Setting up standards, Knowing Graphical User Interface of drafting, NX drafting methods** **Creating drafting views** Base view Drawing view Projected view o Orthographic view o Auxiliary view
Session 16	***Creating drafting views*** **Detail view** **Section view** Simple section, Stepped section , Half section, Revolved section, Folded section, Unfolded section, Pictorial section, Half pictorial section, Break out section, Creating Broken view, Cropping view boundary, Standard view **How to move/Copy a view** **How to align a view** **How to hide/show components** **Creating section in view**
Session 17	**Adding dimensions -** Inferred Dimension, Horizontal Dimension, Vertical Dimension, Parallel Dimension, Perpendicular dimension, Angular dimension,

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	Cylindrical Dimension, Hole dimension, Diameter Dimension, Chamfer Dimension, Radius or Radius of Curvature Dimension, Radius to Centre, Folded Radius, Thickness Dimension, Arc Length, Horizontal Chain Dimension, Vertical Chain Dimension, Horizontal Baseline Dimension, Vertical Baseline Dimension, Ordinate Dimension **Adding annotations** **Adding Feature control frame** **Adding Datum Feature symbol** **Adding Datum Target** **Inserting Identification symbol** **Inserting Surface Finish symbol** **Placing target point symbol** **Placing Intersection symbol** **Applying cross hatch and area fill** **Creating centrelines** **Placing table** **Placing tabular note** **Listing parts list** **Creating auto balloon**
Session 18	***Surface Modeling commands*** Creating extrude surface Creating revolved surface Creating ruled surface **Surface using curves** Surface by Through curves Surface by Through curve mesh Creating Studio surface

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	Surface from Section Surface Surface creation by N-Sided surface
Session 19	**Surface generation** Creating surface using Styled Sweep, Surface from Four Point Surface, Swoop feature , Transition feature, Extension feature, Bounded plane, Sheet from curves, Ribbon builder, Patch openings, Law extension, Silhouette flange **Creating curves from curves** **Creating curve from bodies** **Creating associative copies** Extract body, Composite curve **Editing geometries** Emboss sheet, Sew and Unsew, Patch, Trim body, Split body, Trimmed sheet, Trim and extend, Untrim, Offset surface, Variable offset surface, Offset face, Scale body, Thicken, Divide face
Session 20	***Sheet metal Design -*** **About NX Sheet Metal Preferences** **Creating base feature** Tab **Creating Bend feature** Attaching flange, Attaching Contour flange, Creating Lofted flange, Inserting Hem flange, Apply Bend, unbend, rebend, Apply Jog, Creating Sheet metal from solid **Corner definition** Applying Closed corner, Break corner, Applying chamfer **Applying punch operations** Dimple, Louver, drawn cutout, bead, Solid punch, gusset **Sheet metal cut operations** **Resizing bend radius**

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	Resizing bend angles **Resizing Neutral Factor** **Bend operations** **Converting solid body to sheet metal** **Converting to sheet metal** **Creating flat pattern**