

Training Agenda for Design and Detailing using Nx (Unigraphics)

Introduction of machine drawing with Geometrical Dimensioning & Tolerance	
Basic features (Sketcher)	Drawing sketches for solid models
	Snap points
	Dimensioning
	Constraints
	Degrees of freedom
	Mirroring
	Patterning
Basis features (part modelling)	Basic Part Modelling (Boolean operations)
	Treatment features
	Shelling and Ribs
Advanced features (part modelling)	Editing and repairing
	Reference elements (Datum planes, Datum axis & CSY)
	Holes
	Arrays
	Threads (Symbolic & detailed)
	Slots (Ball end, U, T, Dove tail)
	Boss, Pocket, Pad features
Creating part models using DIN standards	Under cuts, Keyway cuts, Standard hole sizes, etc..
Assembly modelling	Placing components
	Assembly constraints
	Component arrays
	Replacing component
	Mirroring a component
	Modifying a component in the assembly file
Advanced assembly modelling	Top-Down Assembly design
	Creating sub-assemblies
	Replacing assembly constraints

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Advanced assembly modelling	Interference checking
	Clearance checking
	Exploded views (Automatic & Manual method)
Sheetmetal	Modelling Sheet Metal Parts
	Sheet Metal Forming Tools
	Additional Sheet Metal Features
	Converting Parts to Sheet Metal
Detailing	Drawing views creation
	Modifying the view properties
	Dimensioning
	Modifying the sheet size
	Updating views
	Creating BOM & Balloons
	Using GD&T
	Exploded view creation
	Flat-patten view creation
	Surfaces
Advanced Surface Features	
Converting Surface to Solid	
Queries resolution, Tool test for the topics covered	

First half will be theory session and second half will be hands on,
for the topics covered on that particular day.