NX Nastran Syllabus

Total duration: 72 hours (Theory 36 Hours + Lab 36 Hours)

NX Nastran Total Duration : 80 Hours	
Session	**Topics**
Session 1	Introduction to Finite Element Analysis. Areas of application. Theories behind FEA. FEM Vs FEA. About Pre-Processing, Solver, Post-Processor. Introduction to NX Nastran. Overview on Advanced Simulation.
Session 2	About Simulation File Structure. Advanced Simulation Workflow. Simulation Navigator. Understanding geometry abstraction process. About Polygon geometry. Fillet Identification Process. Model Cleanup commands - Auto Heal Geometry, Split Edge, Splitface, Merge edge, Merge face, Stitch Edge, Unstitch edge, Collapse edge, Face repair, Reset.
Session 3	About geometry idealization process. Comparison of geometry idealization and abstraction. Model Preparation commands – Idealize geometry, Defeature geometry, split body, mid-surface, user defined method, sew, divide face.
Session 4	Overview on materials. Units and Dimension, Unit Management, Units Manager, Unit Converter. Simulation Co-ordinate systems, Nodal coordinate systems. Meshing overview, Mesh points, 3D Tetrahedral Mesh, 3D Swept Mesh, Solid from shell Mesh
Session 5	Overview on 2D meshing, Understanding free mapped meshes and creation, 2D mapped meshing,2D Dependent mesh, Local Refinement for associative 2D meshes, surface coat of 2D elements. 1D meshing, defining beam cross section and orientation, 0D meshing and controlling mesh density on edges & faces
Session 6	Creating and applying physical property tables. Defining element and mesh attributes. Material Orientation, Determining Nastran 2D element material orientation, Determining Nastran 3D element material orientation, Activity.
Session 6	Comprehensive check, Model setup, Element Shapes, Element outlines, Nodes, 2D Element Normal, 2D Element Normal by seed, Node proximity to CAD geometry, Element Material Orientation, CAE Model consistency, Element Shapes Threshold values. Boundary Condition, Creating loads, Creating constraints, Activity

	NX Nastran Total Duration : 80 Hours	
Session	**Topics**
Session 7	Solving the model, Analysis job monitor, Batch Solving, Checking analysis quality, NX Nastran Solution Monitor, Solution information monitor, Sparse Matrix Solver monitor, Iterative Solver Convergence monitor, Contact Analysis convergence monitor, Eigenvalues Extraction monitor, DOF curve monitor, Nonlinear history monitor, Nonlinear history monitor, Load step convergence monitor
Session 8	Overview on Post processing ,Importing results, Post View, Master post view and multiple viewports, Managing display of deformed models, Contour plots, Post View Templates, Post View Layouts, Overlay Plots. Combining load cases, Animation, Identify, Generating Reports, Exporting Reports, Activity. Importing Finite Element model data, Exporting Simulation file, Activity.
Session 9	Introduction to Linear Static Analysis. Analysis on 1D element and over link component
Session 10	Introduction to Contact analysis and understanding contact pressure between shaft & wheel
Session 11	Introduction to gluing and understanding natural frequency in a golf club assembly
Session 12	Introduction to Modal Analysis and analysing mode shapes in a speaker cabinet with & without connection
Session 13	Introduction to buckling analysis and finding the critical load on a strap
Session 14	Introduction to thermal analysis and performing thermal & structural analysis
Session 15	Introduction to Non linear analysis and studying geometric nonlinearity
Session 16	Introduction to Assembly FEM, Assembly FEM Vs multi body FEM, Working with sub-assemblies
Session 17	Finding natural frequency in an assembly by mapping sub assemblies
Session 18	Introduction to Optimization and study to obtain an optimized design for a cross member