Hypermesh Syllabus	
Total duration: 40 hours (Theory 20 Hours + Lab 20 Hours)	
Hypermesh Total duration: 40hrs	
Session	Topics
Session 1	Introduction to FEM
	Brief on Meshing
	Basic interaction with Hypermesh
	User interface, Opening/saving files
	Working with panels
	Model organization
	Display control
	Preparing geometry for meshing
	Creating and Editing Line Data
	Creating and Editing Solid Geometry
	Importing and Repairing CAD
Session 2	Midsurfaces
	Simplifying Geometry
	Refining topology
	Geometry and Mesh Editing using Quick Edit Panel
Session 3	Shell meshing
	Automeshing – meshing on surface geometry
	Meshing without surfaces
	2D Mesh in curved
	QI Mesh Creation
Session 4	Meshing a Model using Shrink Wrap
	Tetra meshing
	Method 1 – standard tetramesh
	Method 2 – volume tetramesh
Session 5	Checking tetra element quality
	Remeshing tetra elements
	Creating hexa and penta mesh
	Creating hex-penta mesh using surfaces
Session 6	Creating hexahedral mesh with the solid map function
	Quality
	Checking and Editing Mesh
Session 7	Assemblies: welding and swapping parts
	Spot connectors, Area connectors, Bolt connectors
	Part replacement
Session 8	Analysis Setup
	Setting up loading conditions
	Formatting Model for analysis
	Obtaining and assigning beam cross section properties using HyperBeam
Session 9	Defining composites
	Working with loads on geometry
Session 10	Hypermesh Solver Interfaces
	Review, Test and Project Discussion