



Finite Element Modeling using Hypermesh

Course Curriculum (Duration: 70 Hrs.)

Chapter 1: Introduction to FEA

- a. Introduction to Finite Element Analysis
- b. Basic steps in Finite Element Analysis
- c. FEM methods and detailed explanation of any one method
- d. Types of analysis
- e. Basic element study: 1D, 2D, 3D
- f. Shape Function
- g. Derivation of Stiffness Matrix Equation

Chapter 2: Introduction to Hypermesh

- a. About Altair HyperWorks and its products
- b. About Hypermesh
- c. Consistent Units
- d. Getting Started With HyperMesh
- e. Opening and Saving Files
- f. Working With Panels
- g. Organizing a Model
- h. Controlling the Display
- i. Workshop

Chapter 3: Geometry Clean-up

- a. Hypermesh Geometry Terminology
- b. Importing and Repairing CAD Geometry
- c. Generating a Midsurface
- d. Simplifying Geometry
- e. Midsurface
- f. Defeaturing

Chapter 4: Shell Meshing

- a. Automeshing
- b. Checking and Editing Mesh
- c. 2D Shell Meshing and Topology Refinement
- d. Refining Topology to Achieve a Quality Mesh
- e. Batch Meshing
- f. Element quality checks

Chapter 5: 3D Solid Meshing

- a. Standard Tetra Meshing
- b. Volume Tetra Meshing
- c. Tetrameshing

- d. Using The Tetramesh Process Manager
- e. Element quality checks

Chapter 6: 1D Meshing and Connectors

Chapter 7: Analysis Setup and Loading

- a. Setting up Loading Conditions
- b. Analysis Setup and Loading

Chapter 8: Exporting and Solving

- a. Exporting FE data to various Solvers like ANSYS, LS Dyna and Radioss
- b. Postprocessing

Chapter 9: Introduction to HyperView

- a. Basic interaction with HyperView
- b. Animation, Plot, Video and Text Windows
- c. Animating data in Animation, Plot and Video Windows
- d. Loading Animation Files
- e. Model display attributes and organization
- f. Window display and mouse options
- g. Results Visualization: Contour, Deformed, Iso Surface, Tensor and Vector Plots

IFS Academy, Pune

Phone: +91-20-6400 7296, Email: training@ifsacademy.org,

Visit Us At: www.ifsacademy.org