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
c. Generating a Midsurface
d. Simplifying Geometry
e. Midsurface
f. Defeatureing
g. Importing and Repairing CAD Geometry

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. Checking and Editing Mesh
f. Batch Meshing
g. Element quality checks

Chapter 5: 3D Solid Meshing

a. Standard Tetra Meshing
b. Volume Tetra Meshing
c. Tetrameshing
d. TetraMesh Process Manager
e. Using The Tetramesh Process Manager
f. Element quality checks

Chapter 6: 1D Meshing and Connectors

Chapter 7: Morphing

a. Morph Volumes
b. Domains and Handles
c. Map to Geometry
d. Map To Geometry
e. Domains and Handles
f. Morph Volumes

Chapter 8: Analysis Setup and Loading

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

Chapter 9: Exporting and Solving

a. Exporting FE data to various Solvers like Radioss, Optistruct and ANSYS
b. Postprocessing

*********************
IFS Academy, Pune
Phone: +91-20-6400 7296, Email: training@ifsacademy.org,
Visit Us At: www.ifsacademy.org