Finite Element Modeling using Hypermesh

Duration: 40 Days.

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