



Finite Element Analysis using Hypermesh & RADIOSS OR OptiStruct

Course Curriculum (Duration: 110 Hrs.)

Section I: Hypermesh

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. Defeaturing
- 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

Section II: RADIOSS

Chapter 1: Introduction

- a. HyperWorks Overview
- b. HyperWorks Tool Descriptions.
- c. RADIOSS Integration with HyperWorks
- d. RADIOSS Overview
- e. RADIOSS process
- f. Guide Lines

Chapter 2: Linear Static Analysis

- a. What Is a Static Analysis?
- b. Model Definition Structure
- c. Input/output section
- d. Subcase information section
- e. Bulk data section
- f. Linear Static Analysis Setup
- g. Workshop

Chapter 3: Modal Analysis

- a. Definitions
- b. Natural frequency (Eigenvalue)
- c. Mode shape (Eigenvector)

- d. How to Setup a Modal Analysis
- e. Workshop

Chapter 4: Linear Buckling Analysis

- a. Definitions
- b. Linear buckling and offset elements
- c. How to Setup a Linear Buckling Analysis
- d. Workshop

Chapter 5: Inertia Relief Analysis

- a. Definitions
- b. How to Setup an Inertia Relief Analysis
- c. Workshop

Section III: OptiStruct

Chapter 1: Introduction

- a. HyperWorks Overview
- b. OptiStruct Integration with HyperWorks
- c. OptiStruct Overview
- d. Finite Element Analysis
- e. Structural Design and Optimization
- f. Case studies

Chapter 2: Theoretical Background

- a. Optimization, Design Variable and Response
- b. Subcase Independent Response
- c. Objective Function
- d. Constraint Functions
- e. Gradient-based Optimization
- f. Gradient Method
- g. Sensitivity Analysis
- h. Move Limit Adjustments
- i. Constraint Screening
- j. Regions and Their Purpose
- k. Discrete Design Variables

Chapter 3: HyperMesh Optimization Interface

- a. Model Definition Structure
- b. Input/Output Section
- c. Subcase Information Section
- d. Bulk Data Section
- e. Optimization Setup
- f. Optimization GUI
- g. Design Variable [DTPL]
- h. Responses [DRESP1]
- i. Dconstraints [DCONSTR]
- j. Obj. reference [DOBJREF]
- k. Objective [DESOBJ]
- l. Table entries [DTABLE]
- m. Dequations [DEQATN]
- n. Discrete dvs [DDVAL]
- o. Opti. control [DOPTPRM]
- p. Constr. Screen [DSCREEN]
- q. How to Setup an Optimization in HyperMesh

Chapter 4: Concept Design

- a. Topology Optimization
- b. Homogenization method
- c. Density method
- d. Workshop
- e. Topography Optimization
- f. Design Variables for Topography Optimization
- g. Variable Generation
- h. Multiple Topography Design Regions
- i. Free-size Optimization
- j. Design Interpretation – OSSmooth
- k. OSSmooth Input Data
- l. Running OSSmooth
- m. Interpretation of Topography Optimization Results
- n. Shape Optimization Results, Surface Reduction and Surface Smoothing
- o. Fine tuning
- p. Workshops

IFS Academy, Pune

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

Visit Us At: www.ifsacademy.org