Hyper Mesh-Time Duration-60 Hours

Hyper Mesh Window

The Files Panel

The Collectors Panel

Default HyperMesh Files

Creating an FEA Model

Creating, Solving, and Analyzing an FEA Model

Retrieving a Hyper Mesh Database

Creating Material Collectors

Creating and Editing Component Collectors

Creating 2-D Elements

Creating 3-D Elements with the Line Drag Panel

Cleaning up the Model

Creating Load Collectors

Applying Constraints to the Model

Creating Forces

Creating Load Steps

Geometry Clean Up and Tetra meshing

Geometry Cleanup

Importing and observing the model

Cleaning up geometry

Surface Meshing

Checking the Element Quality and Tetra meshing

Cleaning up and verifying the model

Chapter:1 Auto meshing

Exercise 1.1: Model Preparation

Exercise 1.2: Using the Automesh Panel / Interactive Mode

Exercise 1.3: Using the Automesh Module / Density Sub-panel

Exercise 1.4: Using the Automesh Module / Algorithm & Checks Sub-panels

Exercise 1.5: Using the Automesh Module/Type and Biasing Sub-panel

Exercise 1.6: Equivalencing Nodes

Exercise 1.7: Using chordal

Exercise 1.8: Understanding mesh parameters.

Exercise 1.9: Using the Automesh Panel / Automatic Mode

Exercise 1.10: Remeshing surfaces

Chapter: 2:Creating a 2-D Mesh

Exercise 2.1: Importing IGES Data

Exercise 2.2: Setting up Material Properties

Exercise 2.3: Creating Component Collectors

2-D Modeling

Exercise 2.4: Editing Geometry

Exercise 2.5: Trimming A Surface

Exercise 2.6: Using the Spin Panel

Exercise 2.7: Meshing a Surface

Exercise 2.8: Using the Ruled Panel

Exercise 2.9: Using the Skin Panel

Exercise 2.10: Using the Spline Panel

Exercise 2.11: Checking Elements and Models

Exercise 2.12: Splitting Elements

Exercise 2.13: Checking Normals

Chapter 3: Creating a 3-D Mesh

Exercise 3.1: Using the Linear Solid Panel

Exercise 3.2: Using the Solid Map Panel

Exercise 3.3: Using the Element Offset Panel

Exercise 3.4: Checking Element Connectivity

Exercise 3.5: Reflecting Elements

Exercise 3.6: Creating Load Collectors

Exercise 3.7: Creating Pressures

Exercise 3.8: Creating Constraints

Exercise 3.9: Creating Load Steps

Exercise 3.10: Writing the Analysis Deck

Exercise 3.11: Running the OptiStruct Solver

Chapter 4: Post-Processing

Analyzing Results

Exercise 4.1: Using Visualization Tools

Exercise 4.2: Using the Deformed Panel

Exercise 4.3: Viewing a Replay File

Chapter 5: Post-processing with HyperView

Hand-on Domain Projects

1. Assembly connections with 1D Elements.

2. Weld connections for Automotive, heavy engineering industries.