

HyperMesh

Introduction

Pre-processing for Finite Element Analysis

This is an introductory course for using HyperMesh to create and set up finite element models for analysis. A combination of lectures and exercises will familiarize students with the HyperMesh environment, process, and suite of tools needed to start using HyperMesh.

Lessons are taught with a process driven approach that challenge the student to learn, use and explore the software in real world situations.

Training Format

- *6 days@ 2hours/day*
- *Instructor led*
- *Presentation and hands-on exercises*

Course Syllabus

- *Basic knowledge of Finite Element Analysis*
- *Basic interaction with HyperMesh*
 - *User interface*
 - *Opening / saving files*
 - *Working with panels*
 - *Model organization*
 - *Display control*
- *Preparing geometry for meshing*
 - *Repairing surface topology*
 - *Midsurfaces*
 - *Defeaturing models*
- *Shell meshing*
 - *Refining surface topology*
 - *Automeshing – meshing on surface geometry*
 - *Checking and editing mesh*
 - *Batch meshing*
- *Creating hexa and penta mesh*
 - *Creating & editing solid geometry*
 - *Creating hex mesh with the solid map panel*
- *Tetra meshing*
 - *Volume TetraMeshing*
 - *Standard TetraMeshing*
 - *TetraMesh Process Manager*