

UT-IMDI HyperWorks Workshop

Summary

HyperWorks is a powerful Computer-Aided Engineering platform developed by Altair®. It consists of several tools to perform all common structural analysis and optimization tasks, as well as simulations involving crush, forming, manufacturing, DOE studies, and much more. It is widely used in automotive, aerospace, biomedical, energy, and electronics sectors to accelerate the design cycle, reduce product costs, and foster innovation through accurate simulations and powerful optimization capabilities.

The focus of this workshop is to provide an introduction to two of the main tools of the HyperWorks package, namely HyperMesh and HyperView, as well as Altair's built-in solver Optistruct. Together, these tools are used to pre-process, solve, and post-process Finite Element models representing a wide range of structural analysis and optimization tasks. The content of the workshop is intended towards students with little or no prior experience with HyperWorks. The interface, tools, concepts, and methods will be introduced by the instructor and the students will be walked through them by following step-by-step examples. The workshop will be wrapped up with a mini-project in which students review the topics and get hands-on experience on model setup, debugging, and post-processing of results.

Workshop Information

Instructor: Mohsen Rahmani (mohsen.rahmani@mail.utoronto.ca), PhD Student at MIE
Work experience: Automotive industry (Multimatic Inc.)
MASc: Mechanical Engineering, University of Toronto
BASc: Mechanical Engineering, Sharif University of Technology

All current UofT engineering students (undergraduate and graduate levels) are welcome to participate. The workshop will take place in one day and will cover the following topics:

1. Introduction
 - a. Computer Aided Engineering
 - b. Finite Element method
 - c. Main steps of a simulation
2. HyperMesh User interface
 - a. Panels
 - b. Menus
 - c. Collectors
3. Geometry
 - a. Classification of geometric features
 - b. Importing CAD
 - c. Sketching/Editing geometries
4. Meshing
 - a. Pre-mesh clean-up and checks
 - b. Element types

- c. Automated meshing
- d. Mesh quality check
- e. Advanced mesh controls
- f. General good practice tips
- 5. Boundary condition and loads
 - a. Boundary conditions
 - b. Forces, moments, pressures
- 6. Loadstep setup and Solver selection
 - a. Analysis types
 - b. Optistruct Solver
 - c. Loadstep definition
 - d. Control cards
 - e. Output request
- 7. Post-processing
 - a. HyperView user interface
 - b. Loading the results
 - c. Contour Plots
 - d. Measurements
- 8. Wrap-up: mini-project

Workshop Timeline – Saturday March 11, 2017

| | |
|--------------------|-------------------------------------|
| 9:00-9:30 | <i>Introduction</i> |
| 9:30-10:00 | <i>HyperMesh user interface</i> |
| 10:00-11:00 | <i>Geometry</i> |
| 11:00-12:00 | <i>Meshing</i> |
| <u>12:00-13:00</u> | <u><i>Lunch Break</i></u> |
| 13:00-13:30 | <i>Boundary condition and loads</i> |
| 13:30-14:30 | <i>Loadstep setup</i> |
| 14:30-15:30 | <i>Post-processing</i> |
| 15:30-17:00 | <i>Wrap-up: mini-project</i> |