ANSYS Course Content

Overview : ANSYS Autodyne is computer simulation tool for simulating the response of materials to short duration severe loadings from impact, high pressure or explosions

Day 1

Session 1 : Introduction to

- ANSYS ICEM CFD Overview
- GUI & Layout
- ICEM CFD File Management
- Menu Introduction
- Work Flow and Meshing Process
- Workbench ICEM Link

Session 2 :Geometry Basics

- Geometry Handling
- Importing / editing geometry and mesh
- Geometry creation, repair and simplification
- Faceted Geometry Handling
- 3. Meshing
- Shell meshing
- Local & Global Mesh Settings
- Mesh Methods, Types and Computation
- Auto-Volume meshing
- Unstructured mesh creation
- Prism Meshing
- Introduction to structured mesh

Day 2

Session 1 :

- b. Output Options
- c. File and directory structure
- d. Accessing ICEM CFD from Workbench
- ANSYS FLUENT:•
- Introduction to FLUENT & ANSYS Products
- Basic fluid flow and CFD (Theory topics are as mentioned above.)

Session 2 :

- h. Fluid Flow Modeling (Pressure & Velocity)
- Turbulence modeling
- j. Heat transfer modeling
- k. Theory of Discretization
- l. Convergence settings and monitoring
- m. Post processing

Section 3:

- Hand on Workshops, Exercises and Live Case Studies
- Live Case Study CAD Models with Problem Description shall be given to the students

Session 4 :

- Certificate Distribution