Introduction to ANSYS LS-DYNA

Length: 2 Days

Prerequisites

• A technical education and background is recommended but an engineering degree is not required.

Overview

ANSYS LS-DYNA is a useful tool for those users who need to analyze problems involving contact, large deformations, nonlinear materials, high frequency response phenomena and/or problems requiring explicit solutions.

Course Description

This training course teaches students how to effectively use ANSYS LS-DYNA to build an explicit mechanical simulation model, analyze it, and interpret the results. At the conclusion of the course, students will be able to:

• distinguish problems that should be solved explicitly versus implicitly
• identify and choose element types, materials and commands used in explicit dynamic analyses
• setup, solve, and postprocess an ANSYS LS-DYNA analysis

Course Topics include:

• Elements
• Part Definitions
• Material Definitions
• Boundary Definitions, Loading, and Rigid bodies
• Meshing
• Solution and Simulation Controls
• Postprocessing
• Restarting

Each course chapter is followed by "hands-on" workshops and exercises.

Fee: Rs 18,000/- + Applicable Taxes

For more information / registration contact:
Email: india-register@ansys.com
Phone: +91-20-66522545 ; Fax : +91-20-66522600