Introduction to ANSYS Mechanical APDL - Part I

Length: 3 Days

Prerequisites

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

Overview

This course is for users of ANSYS Parametric Design Language (APDL) tools in ANSYS Mechanical APDL, previously available in the traditional ANSYS format. The ANSYS Workbench environment will not be discussed or covered in this course.

Course Description

This course is recommended for anyone who performs finite element analysis (FEA) of mechanical parts or fluids and has little or no ANSYS Mechanical APDL software experience. Introduction to ANSYS Mechanical APDL, Part I is a three-day course that focuses on basic linear and static analyses of structural and/or mechanical parts. After completing the course, attendees should be able to efficiently use ANSYS Mechanical APDL software to build two- and three-dimensional models, apply loads, obtain solutions, and display results.

This training course will cover the following topics:

- FEA and ANSYS Mechanical APDL
- Getting Started
- ANSYS Mechanical APDL Basics
- General Analysis Procedure
- Creating the Solid Model
- Creating the Finite Element Model
- Defining Material Properties
- Applying Loads
- Solution Process
- Structural Analysis
• Thermal Analysis
• Post-processing

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

**Fee:** Rs 15,000/- + Applicable Taxes
**For more information / registration contact:**
**Email:** india-register@ansys.com
**Phone:** +91-20-66522545 ; Fax : +91-20-66522600