ANSYS DesignModeler & ANSYS Mechanical Introduction

**Length:** 3 Days

**Prerequisites:** A technical education and background is recommended but an engineering degree is not required.

**Course contents for Day 1:**

**Overview:** The ANSYS DesignModeler training course is for users that want to create and modify geometry in preparation for analysis in ANSYS Mechanical or ANSYS Mechanical APDL. Students who attend this course will learn:

- how to create and modify geometry in preparation for analysis
- how to navigate within the Graphical User Interface
- how to generate 2D sketches and convert them into 2D or 3D models
- how to modify 2D and 3D geometry
- how to import existing CAD geometry
- how to create line bodies and their cross sections in preparation for FE beam analysis
- how to create surface bodies in preparation for FE shell analysis
- how to model assemblies
- how to utilize parameters

**Course Description**

The training course will teach students how to create their geometry in the ANSYS Workbench environment using ANSYS Design Modeler.

**Course Topics Include:**

- Introduction to ANSYS DesignModeler
- Graphical User Interface (GUI)
- Sketch Mode
- 3D Geometry Creation
- Working with Other CAD Geometry
- Parametric Modeling

Each course chapter is followed by "hands-on" workshops and exercises.
Course contents for Day2 and Day 3:

Overview

ANSYS Mechanical provides solutions for many types of analyses including structural, thermal, modal, linear buckling and shape optimization studies.

ANSYS Mechanical is an intuitive mechanical analysis tool that allows geometry to be imported from a number of different CAD systems. It can be used to verify product performance and integrity from the concept phase through the various product design and development phases.

The use of ANSYS Mechanical accelerates product development by providing rapid feedback on multiple design scenarios, which reduces the need for multiple prototypes and product testing iterations.

Course Description

This training course teaches students how to effectively use ANSYS Mechanical to build a mechanical simulation model, analyze it and interpret the results.

Course Topics Include:

- Introduction
- Mechanical Basics
- General Preprocessing
- Static Structural Analysis
- Vibration Analysis
- Thermal Analysis
- Linear Buckling Analysis
- Results Postprocessing
- CAD & Parameters

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

Fee: Rs. 21,000/- + Applicable Taxes
For more information / registration contact:
Email: india-register@ansys.com
Phone: +91-20-66522545 ; Fax : +91-20-66522600