



# ANSYS Software Training Course Curriculum



## Chapter 1: Before you start using Ansys

- a. Introduction to the Finite Element Method
- b. What is the Finite Element Method?
- c. General Steps of the Finite Element Method
- d. Explanation of 1D, 2D and 3D Elements with examples of ANSYS Elements
- e. Need of FEM
- f. Types of analysis that can be done using ANSYS
- g. Advantages of the Finite Element Method
- h. Limitations of FEA
- i. About ANSYS Inc.
- j. ANSYS Family of products with their capabilities
- k. Types of analysis that can be done with ANSYS.
- l. Introduction to the Ansys GUI
- m. Operation Modes of Ansys
- n. Product Launcher
- o. Launcher Menu Options
- p. The ANSYS GUI
- q. The Icon Toolbar Menu
- r. Quitting Ansys

## Chapter 2: Selection Logic

- a. Pan-Zoom-Rotate
- b. Picking
- c. Coordinate Systems

### **Chapter 3: Solid Modeling**

- a. An Overview of Solid Modeling Operations
- b. Working with Boolean operations
- c. Working Plane
- d. Importing of 3D models

### **Chapter 4: Meshing**

- a. Free meshing or Mapped meshing
- b. Setting Element Attributes
- c. Selecting Element Type
- d. Shape Function
- e. Defining Element Types
- f. Defining Section Properties
- g. Assigning Element Attributes before meshing
- h. Mesh Controls
- i. The ANSYS Mesh Tool
- j. Smart sizing
- k. Meshing
- l. Free Meshing
- m. Mapped Meshing
- n. Hybrid meshing
- o. Mesh Extrusion
- p. Volume Sweeping

### **Chapter 5: Material Properties**

- a. Material Library
- b. Specifying properties

## **Chapter 6: Boundary Conditions**

- a. Types of Loads
- b. Applying loads

## **Chapter 7: Solvers**

- a. Types of Solvers
- b. Solver Setup
- c. Load Step Options
- d. Solving Multiple Load Steps

## **Chapter 8: Post-processing**

- a. Contour Plot Viewing
- b. Time History Postprocessor (POST26)
- c. Report Generator

## **Chapter 9: Introduction to Non-Linear Analysis**

## **Chapter 10: Sample Structural Analysis**

- a. Workshop

## **Chapter 11: Sample Thermal Analysis**

- a. Workshop

## **Chapter 12: Sample Modal Analysis**

- a. Workshop

## **Chapter 13: Tips & Tricks**

- a. Using the Toolbar & Creating Abbreviations
- b. Introduction to APDL
- c. Using Parameters
- d. Using the Start File
- e. Using the Session Editor
- f. Using Input Files

**Chapter 14: ANSYS Workbench**

- a. Introduction to ANSYS Workbench
- b. Graphical User Interface
- c. Static Structural Analysis
- d. Modal Analysis
- e. Thermal Analysis
- f. Contact Recognition

\*\*\*\*\*