



## Finite Element Analysis using ANSYS Workbench

### Course Curriculum (60 Hrs.)

#### Chapter 1: Introduction to FEA and ANSYS Workbench

- a. Introduction to the Finite Element Method
- b. What is the Finite Element Method?
- c. History
- d. General Steps of the Finite Element Method
- e. Explanation of 1D, 2D and 3D Elements with examples of ANSYS Elements
- f. Need of FEM
- g. Enlisting different FEM methods and detailed explanation of any one
- h. Derivation of stiffness matrix equation
- i. Types of analysis that can be done using ANSYS
- j. Advantages of the Finite Element Method
- k. Limitations of FEA
- l. ANSYS Workbench Overview
- m. Hosting Applications
- n. Mechanical Overview
- o. Starting Mechanical
- p. The Workbench Environment
- q. The Toolbox
- r. The Project Schematic
- s. Workbench File Management
- t. Working with Units

#### Chapter 2: DesignModeler

- a. Introduction to DesignModeler
- b. Planes and Sketches
- c. Modeling
- d. Geometry Simplification and Repair
- e. CAD Connections
- f. Parameterization
- g. Beams and Shells
- h. Lines and Surfaces

#### Chapter 2: Mechanical Basics

- a. Basic Analysis Procedure
- b. The Mechanical Interface
- c. Menus
- d. Toolbars
- e. Graphics Control and Selection
- f. Outline Tree and Details
- g. Graphics Window
- h. The Mechanical Application Wizard
- i. Scoping Loads and Supports
- j. The Engineering Data Application
- k. Workshop

#### Chapter 3: General Preprocessing

- a. Geometry Branch
- b. Contact
- c. Meshing
- d. Named Selections
- e. Coordinate Systems

- f. Remote Boundary Conditions
- g. Selection Information
- h. Workshop – Mesh Control

**Chapter 4: Meshing**

- a. Global Meshing Controls
- b. Local Meshing Controls
- c. Meshing Troubleshooting
- d. Virtual Topology
- e. Workshops

**Chapter 5: Static Structural Analysis**

- a. Basics of Static Structural Analysis
- b. Geometry
- c. Material Properties
- d. Contact
- e. Analysis Settings
- f. Loads
- g. Supports
- h. Nodal Loads and Supports
- i. Solving Models
- j. Results and Postprocessing
- k. Workshops
- l. Case Studies: Any two

**Chapter 6: Vibration Analysis**

- a. Basics of Free Vibration
- b. Geometry
- c. Contact
- d. Solution Setup
- e. Modal Results
- f. Vibration with Prestress
- g. Workshops
- h. Case Studies: Any two

**Chapter 7: Thermal Analysis**

- a. Basics Steady State Heat Transfer
- b. Geometry
- c. Material Properties
- d. Thermal Contact
- e. Thermal Boundary Conditions
- f. Solution Options
- g. Results and Postprocessing
- h. Workshops
- i. Case Studies: Any two

**Chapter 8: Results and Postprocessing**

- a. Viewing Results
- b. Scoping Results
- c. Exporting Results
- d. Coordinates Systems
- e. Solutions Combinations
- f. Stress Singularities
- g. Error Estimation
- h. Convergence
- i. Workshops

**Chapter 9: CAD & Parameters**

- a. CAD Import
- b. Defining Parameters in Workbench

- c. Using the Parameter Workspace
- d. Updating CAD Parameters
- e. Workshops

**Chapter 10: Advanced Named Selection**

- a. Named Selection Basics
- b. Direct Named Selections
- c. Criteria Named Selections
- d. Named Selection Summary
- e. Workshops

**Chapter 11: Constraint Equations**

- a. Constraint Equations
- b. Constraint Equation Worksheet
- c. Workshops

\*\*\*\*\*

[www.ifsacademy.org](http://www.ifsacademy.org)