ANSYS Workbench

Duration: 20 Days

ANSYS Workbench:

Chapter 1: Introduction to FEA and ANSYS Workbench

- Introduction to the Finite Element Method
- What is the Finite Element Method?
- History
- General Steps of the Finite Element Method
- Explanation of 1D, 2D and 3D Elements with examples of ANSYS Elements
- Need of FEM
- Enlisting different FEM methods and detailed explanation of any one
- Derivation of stiffness matrix equation
- Types of analysis that can be done using ANSYS
- Advantages of the Finite Element Method
- Limitations of FEA
- ANSYS Workbench Overview
- Hosting Applications
- Mechanical Overview
- Starting Mechanical
- The Workbench Environment
- The Toolbox
- The Project Schematic
- Workbench File Management
- Working with Units

Chapter 2: Design Modeler

- Introduction to Design Modeler
- Planes and Sketches
- Modeling
- Geometry Simplification and Repair
- CAD Connections
- Parameterization
- Beams and Shells
- Lines and Surfaces

Chapter 2: Mechanical Basics
- Basic Analysis Procedure
- The Mechanical Interface
- Menus

- Toolbars
- Graphics Control and Selection
- Outline Tree and Details
- Graphics Window
- The Mechanical Application Wizard
- Scoping Loads and Supports
- The Engineering Data Application
- Workshop

Chapter 3: General Preprocessing
- Geometry Branch
- Contact
- Meshing
- Named Selections
- Co ordinate System
- Remote Boundary Conditions
- Selection Information
- Workshop – Mesh Control

Chapter 4: Meshing
- Global Meshing Controls
- Local Meshing Controls
- Meshing Troubleshooting
- Virtual Topology
- Workshops

Chapter 5: Static Structural Analysis
- Basics of Static Structural Analysis
- Geometry
- Material Properties
- Contact
- Analysis Settings
- Loads
- Supports
- Nodal Loads and Supports
- Solving Models
- Results and Postprocessing
- Workshops
- Case Studies: Any two

Chapter 6: Vibration Analysis
- Basics of Free Vibration
- Geometry
- Contact
- Solution Setup
- Modal Results

- Vibration with Prestress
- Workshops
- Case Studies: Any two

Chapter 7: Thermal Analysis
- Basics Steady State Heat Transfer
- Geometry
- Material Properties
- Thermal Contact
- Thermal Boundary Conditions
- Solution Options
- Results and Postprocessing
- Workshops
- Case Studies: Any two

Chapter 8: Results and Postprocessing
- Viewing Results
- Scoping Results
- Exporting Results
- Coordinates Systems
- Solutions Combinations
- Stress Singularities
- Error Estimation
- Convergence
- Workshops