



ANSYS Workbench & Mechanical APDL

Duration: 40 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
- Coordinate 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

ANSYS Mechanical APDL:

Chapter 1: Before you start using ANSYS

- 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
- About ANSYS Inc.
- ANSYS Family of products with their capabilities
- Introduction to the ANSYS GUI
- Operation Modes of ANSYS
- Product Launcher
- Launcher Tasks
- Use Custom Memory Settings

- Launcher Menu Options
- The ANSYS GUI
- The Icon Toolbar Menu
- Quitting Ansys

Chapter 2: Selection Logic

- Plotting
- Pan-Zoom-Rotate
- Picking
- Coordinate Systems
- Select Logic

Chapter 3: Solid Modeling

- An Overview of Solid Modeling Operations
- Working with Boolean operations
- Working Plane
- Importing of 3D models

Chapter 4: Meshing

- Free meshing or Mapped meshing
- Setting Element Attributes
- Selecting Element Type
- Shape Function
- Defining Element Types
- Real Constants
- Defining Section Properties
- Assigning Element Attributes before meshing
- Mesh Controls
- The ANSYS MeshTool
- Smartsizing
- Meshing
- Free Meshing
- www.ifsacademy.org
- Mapped Meshing
- Hybrid meshing
- Mesh Extrusion
- Volume Sweeping

Chapter 5: Material Properties

- Material Library
- Specifying properties

Chapter 6: Boundary Conditions

- Types of Loads
- Applying loads

Chapter 7: Solvers

- Types of Solvers
- Solver Setup
- Load Step Options
- Solving Multiple Load Steps

Chapter 8: Post-processing

- Contour Plot Viewing
- Path Operations
- Estimating Solution Error
- Time History Postprocessor (POST26)
- Report Generator

Chapter 9: Static Structural Analysis

- Workshops, Exercises and Case Studies

Chapter 10: Modal Analysis

- Workshops, Exercises and Case Studies

Chapter 11: Thermal Analysis

- Workshops, Exercises and Case Studies

Chapter 12: Tips & Tricks

- Using the Toolbar & Creating Abbreviations
- Introduction to APDL
- Using Parameters
- Using the Start File
- Using the Session Editor
- Using Input Files

Copyright 2021, Radiance Technology, Bhubaneswar

Mob. No.: +91-70381 72715, Email: training@radiancetech.org

Visit us At: www.radiancetech.org