ANSYS Workbench for Finite Element Analysis (FEA)

Target Audience

This course is designed for mechanical engineering students, design engineers, analysis engineers, and CAD/CAE professionals who want to understand the fundamentals of Finite Element Analysis and apply simulation techniques using ANSYS Workbench. It is suitable for learners interested in engineering simulation, **structural analysis, vibration analysis,** and **thermal analysis** for evaluating product performance and design behavior.

Course Objectives

- To introduce the basic concepts and principles of Finite Element Analysis.
- To familiarize learners with the ANSYS Workbench environment and workflow.
- To develop part models suitable for simulation and analysis.
- To understand how to define and assign material properties for engineering analysis.
- To learn mesh generation techniques and mesh quality considerations.
- To perform structural, vibration, and thermal analysis using ANSYS Workbench.
- To interpret simulation results for engineering applications and design validation.

Course Outcomes

By the end of this course, learners will be able to:

- Understand the principles and workflow of Finite Element Analysis.
- Work efficiently within the ANSYS Workbench simulation environment.
- Create and prepare part models for simulation.
- Define material properties and generate appropriate meshes.
- Perform static structural, vibration, and thermal analysis.
- Interpret analysis results for engineering decision-making.

Course Outline

The course comprises **40-hours** of theory and labs and is divided into **11** different chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Chapter 1: Introduction to FEA

1.1 Fundamentals of Finite Element Analysis

- Definition and concept of Finite Element Analysis
- Purpose of numerical simulation in engineering
- Difference between analytical methods and FEA
- Overview of simulation-driven design

1.2 Basic Terminology in FEA

- Nodes and elements
- Degrees of freedom
- Boundary conditions
- Loads and constraints

1.3 FEA Workflow

- Geometry preparation
- Material definition
- Mesh generation
- Solution and result interpretation

1.4 Applications of FEA

- Structural analysis applications
- Thermal analysis applications
- Vibration analysis applications
- Use of FEA in product design and validation

Chapter 2: Introduction to ANSYS Workbench

2.1 Overview of ANSYS Workbench

- Introduction to ANSYS simulation platform
- Components of ANSYS Workbench
- Types of analysis available in Workbench

2.2 Workbench Interface

- Project schematic
- Toolbox panel
- Engineering data section
- Model and solution modules

2.3 Project Setup

- Creating a new analysis system
- Importing geometry
- Organizing project files

2.4 Basic Simulation Workflow

- Connecting system components
- Updating project data
- Managing simulation workflow

Chapter 3: Part Modeling - I

3.1 Introduction to Geometry Creation

- Role of geometry in simulation
- Geometry creation methods in ANSYS
- Preparing geometry for analysis

3.2 Sketching Basics

- Creating sketches
- Sketch constraints

- Dimensioning sketches

3.3 Creating Basic Features

- Extrude operations
- Revolve operations
- Creating simple 3D geometry

3.4 Editing Sketches and Features

- Modifying sketch geometry
- Updating feature dimensions
- Managing design changes

Chapter 4: Part Modeling - II

4.1 Creating Complex Geometry

- Creating multiple features
- Combining geometric operations
- Managing geometry structure

4.2 Feature-Based Modeling

- Feature hierarchy
- Editing and modifying features
- Updating model features

4.3 Geometry Modification

- Fillet creation
- Chamfer creation
- Pattern creation

4.4 Geometry Preparation

- Simplifying geometry
- Removing unnecessary features
- Preparing models for simulation

Chapter 5: Part Modeling - III

5.1 Advanced Geometry Operations

- Boolean operations
- Split body operations
- Surface operations

5.2 Geometry Cleanup

- Detecting geometry errors
- Repairing geometry issues
- Ensuring model continuity

5.3 Geometry Validation

- Checking model integrity
- Verifying geometry readiness for meshing
- Preparing final geometry for analysis

5.4 Geometry Optimization

- Simplifying complex features
- Improving model performance for analysis

Chapter 6: Defining Material Properties

6.1 Introduction to Engineering Materials

- Importance of material properties in simulation
- Common engineering materials used in analysis

6.2 Material Property Types

- Mechanical properties
- Thermal properties
- Elastic properties

6.3 Assigning Material Properties

- Using material libraries
- Assigning materials to geometry
- Modifying material parameters

6.4 Custom Material Definition

- Creating new materials
- Editing material properties
- Managing material database

Chapter 7: Generating Mesh - I

7.1 Introduction to Meshing

- Purpose of mesh in FEA
- Relationship between mesh and accuracy

7.2 Mesh Generation Methods

- Automatic meshing
- Manual meshing
- Mesh generation workflow

7.3 Element Types

- 1D elements
- 2D elements
- 3D elements

7.4 Basic Mesh Controls

- Global mesh size
- Element sizing
- Mesh generation settings

Chapter 8: Generating Mesh – II

8.1 Advanced Mesh Controls

- Local mesh refinement
- Edge and face sizing
- Inflation layers

8.2 Mesh Refinement Techniques

- Adaptive mesh refinement
- Targeted mesh improvement

8.3 Mesh Quality Evaluation

- Element quality parameters
- Skewness and aspect ratio
- Checking mesh statistics

8.4 Improving Mesh Quality

- Adjusting mesh parameters
- Resolving poor-quality elements
- Optimizing mesh for analysis

Chapter 9: Static Structural Analysis

9.1 Introduction to Static Structural Analysis

- Concept of static loading
- Applications of static analysis

9.2 Applying Boundary Conditions

- Fixed supports
- Displacement constraints
- Defining supports

9.3 Applying Loads

- Force loads

- Pressure loads
- Moment loads

9.4 Solving the Model

- Running the simulation
- Monitoring solution progress

9.5 Interpreting Results

- Stress distribution
- Strain results
- Deformation results

Chapter 10: Vibration Analysis

10.1 Introduction to Vibration Analysis

- Importance of vibration analysis
- Basic vibration concepts

10.2 Natural Frequency Analysis

- Understanding natural frequency
- Mode shapes

10.3 Setting Up Vibration Analysis

- Applying constraints
- Defining analysis parameters

10.4 Interpreting Vibration Results

- Mode shape visualization
- Frequency results interpretation

Chapter 11: Thermal Analysis

11.1 Introduction to Thermal Analysis

- Heat transfer fundamentals
- Types of thermal analysis

11.2 Applying Thermal Loads

- Temperature boundary conditions
- Heat flux loads

11.3 Steady-State Thermal Analysis

- Setting up steady-state analysis
- Solving thermal simulations

11.4 Interpreting Thermal Results

- Temperature distribution
- Thermal gradients
- Heat flow analysis