ANSYS Mechanical – Structural Analysis Professional

Target Audience

This course is designed for mechanical engineering students, design engineers, analysis engineers, and CAD/CAE professionals who want to learn how to perform structural simulation using ANSYS Mechanical. It is suitable for individuals interested in validating product strength, safety, and performance using finite element analysis and applying engineering simulation in real-world design and manufacturing environments.

Course Objectives

- To introduce CAE and FEA and familiarize learners with the ANSYS Workbench and ANSYS Mechanical environment.
- To prepare and simplify CAD geometry for simulation.
- To understand mesh generation techniques and mesh quality considerations.
- To apply loads and boundary conditions for structural analysis.
- To perform static structural and modal analysis using ANSYS Mechanical.
- To simulate assemblies and contact interactions.
- To interpret simulation results and generate engineering reports.

Course Outcomes

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

- Understand CAE principles and the workflow of Finite Element Analysis.
- Work efficiently within the ANSYS Workbench and ANSYS Mechanical environment.
- Prepare geometry and generate high-quality meshes for simulation.
- Apply loads, constraints, and contact conditions in structural analysis.
- Perform static structural and modal analysis of engineering components.
- Interpret simulation results and prepare engineering analysis reports.

Course Outline: The course comprises **40-hours** of theory and labs and is divided into **9** different chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Module 1 – Introduction to CAE & FEA

1.1 What is CAE

- Definition of Computer-Aided Engineering
- Role of simulation in product development
- Benefits of CAE in engineering design

1.2 Role of Simulation in Industry

- Simulation in product validation
- Reducing prototype costs
- Improving design reliability

1.3 FEA Fundamentals

- Concept of Finite Element Analysis
- Nodes and elements
- Boundary conditions and loads

1.4 Types of Structural Analysis

- Static structural analysis
- Modal analysis
- Other structural simulations

1.5 ANSYS Workbench Overview

- Introduction to ANSYS Workbench platform
- Components of Workbench environment
- Overview of simulation systems

1.6 Engineering Simulation Workflow

- Geometry preparation
- Meshing process
- Solving and result evaluation

Module 2 – ANSYS Workbench Interface

2.1 Project Schematic

- Understanding the project schematic
- System components and connections

2.2 Engineering Data

- Material library overview
- Adding and editing materials

2.3 Geometry Setup

- Importing CAD models
- Preparing geometry for simulation

2.4 Model Tree Structure

- Navigating the model tree
- Managing simulation components

2.5 Units and Coordinate Systems

- Unit system management
- Local coordinate systems

2.6 File Management

- Managing project files
- Saving and organizing simulation projects

Module 3 – Geometry Preparation

3.1 CAD Import

- Importing STEP files
- Importing IGES files
- Importing Parasolid geometry

3.2 Geometry Cleanup

- Detecting geometry issues
- Repairing geometry errors

3.3 Suppress Small Features

- Identifying unnecessary features
- Removing small details for simulation

3.4 Named Selections

- Creating named selections
- Using named selections in analysis setup

3.5 Simplification for FEA

- Geometry simplification techniques
- Preparing models for efficient meshing

Module 4 – Meshing Fundamentals

4.1 Types of Elements

- Element types used in FEA
- Element selection considerations

4.2 Tetra vs Hex Mesh

- Characteristics of tetrahedral mesh
- Characteristics of hexahedral mesh

4.3 Mesh Sizing Controls

- Global mesh sizing
- Local mesh refinement

4.4 Inflation Layers

- Creating inflation layers
- Applications of inflation layers

4.5 Mesh Quality Parameters

- Skewness

- Aspect ratio
- Element quality metrics

4.6 Mesh Convergence Study

- Importance of mesh convergence
- Refinement and convergence evaluation

Module 5 – Loads & Boundary Conditions

5.1 Fixed Supports

- Applying fixed support constraints
- Understanding support conditions

5.2 Forces and Pressures

- Applying force loads
- Applying pressure loads

5.3 Remote Loads

- Remote force application
- Remote moment definition

5.4 Moments

- Applying torque and moment loads
- Moment direction control

5.5 Displacement Constraints

- Applying displacement boundary conditions
- Controlling movement in simulation

Module 6 – Static Structural Analysis

6.1 Solving the Model

- Running the simulation
- Monitoring solution progress

6.2 Stress and Strain Results

- Stress distribution analysis
- Strain evaluation

6.3 Deformation Analysis

- Total deformation results
- Directional deformation

6.4 Safety Factor Calculation

- Factor of safety concept
- Evaluating design safety

6.5 Result Probes and Charts

- Using probes for result inspection
- Creating result charts

Module 7 – Contacts & Assemblies

7.1 Bonded Contacts

- Definition of bonded contact
- Applications in assembly simulation

7.2 Frictional Contacts

- Friction contact behavior
- Contact friction parameters

7.3 Contact Behavior

- Contact detection methods
- Contact stiffness and penetration

7.4 Assembly Simulation

- Simulating multi-part assemblies
- Managing interactions between parts

Module 8 – Modal Analysis

8.1 Natural Frequency Concept

- Understanding natural frequencies
- Importance in structural design

8.2 Mode Shapes

- Mode shape visualization
- Interpretation of vibration modes

8.3 Structural Vibration Basics

- Basics of vibration analysis
- Applications of modal analysis

Module 9 – Result Interpretation & Reporting

9.1 Stress Interpretation (Von Mises)

- Understanding Von Mises stress
- Evaluating stress results

9.2 Result Validation

- Checking simulation accuracy
- Identifying unrealistic results

9.3 Engineering Judgement

- Interpreting results for design decisions
- Practical engineering considerations

9.4 Report Generation

- Creating simulation reports
- Organizing analysis documentation

9.5 Exporting Results

- Exporting result images and data
- Sharing simulation reports and outputs