

ANSYS Mechanical for Advanced Vibration & Dynamic Analysis

Target Audience

This course is designed for mechanical engineers, simulation analysts, and advanced engineering students who already possess a foundational understanding of finite element analysis and structural mechanics. It is particularly suited for professionals aiming to deepen their expertise in vibration and dynamic analysis using ANSYS Mechanical, with applications in rotating machinery, thermal-structural coupling, and industry-specific case studies.

Course Outcomes

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

- Apply fundamental vibration concepts to engineering problems.
- Perform modal, harmonic, and transient dynamic analyses in ANSYS Mechanical.
- Conduct coupled thermal-structural vibration simulations.
- Analyse rotodynamic behaviour in machinery systems.
- Automate workflows using APDL scripting.
- Interpret and apply vibration analysis to real-world industry applications.

Course Objectives

The objectives of this course are to:

- Build a strong theoretical foundation in vibration and dynamic analysis.
- Provide hands-on experience with ANSYS Mechanical for various dynamic simulations.
- Enable learners to model, simulate, and interpret vibration responses under different loading conditions.
- Introduce advanced topics such as rotor dynamics and thermal-structural coupling.
- Develop automation skills using **APDL** for efficient simulation workflows.
- Bridge theory with industry applications through case studies and capstone projects.

Course Outline

The course comprises **40 hours** of theory and labs and is divided into **8 modules**. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Table of Contents

Module 1 – Fundamentals of Vibrations

- Vibration basics
- Free vs forced vibration
- Natural frequency & resonance
- SDOF and MDOF systems
- Mass, stiffness & damping
- Introduction to vibration in ANSYS
- Exercise: Simple vibration simulation in ANSYS

Module 2 – Modal Analysis in ANSYS

- Modal analysis fundamentals
- Geometry preparation
- Meshing for modal analysis
- Boundary conditions
- Mode shapes & natural frequencies
- Modal participation factors
- Lab: Modal analysis of a machine component

Module 3 – Harmonic Response Analysis

- Harmonic vibration basics
- Frequency response analysis
- Resonance behaviour
- Damping models
- Result interpretation
- Lab: Harmonic response of rotating machinery

Module 4 – Transient Dynamic Analysis

- Transient dynamics basics
- Time-dependent loading
- Impact and shock loads
- Displacement and stress response
- Lab: Transient vibration simulation

Module 5 – Thermal-Structural Coupling

- Thermal expansion basics
- Thermal stress concepts
- Coupled field analysis
- Temperature effects on vibration
- Lab: Thermal-vibration coupling in a shaft

Module 6 – Rotor dynamics for Machinery

- Rotor dynamics fundamentals
- Critical speed analysis
- Rotor imbalance effects
- Bearing stiffness influence
- Case Study: Rotating shaft vibration

Module 7 – APDL Automation

- Introduction to APDL
- Basic APDL commands
- Parametric modelling
- Automation workflow
- Lab: APDL scripting for modal analysis

Module 8 – Industry Applications & Capstone

- Vibration in pumps & compressors
- Turbines and rotating equipment
- Failure case studies (expanded with sub-pointers such as fatigue, misalignment, resonance-induced failures)
- Capstone project integrating multiple analysis methods