Foundation Course on Finite Element Method: Theory and Practical

Target Audience

Design Engineers, CAE Engineers, and Engineers working in Mechanical, Automotive, Aerospace, Railways, Defense, Nuclear Energy, Thermal Power, Wind, Medical Equipment, Electrical and Electronics, and related industries involving design analysis can participate.

Course Outcomes

Participants will be able to:

- Understand and acquire the fundamental knowledge of the finite element method.
- Apply the finite element method to the most relevant problems in the mechanical engineering domain.
- Demonstrate finite element analysis software (ANSYS) for solving a diversity of mechanical engineering problems.
- Identify and rectify errors while solving engineering problems.
- Interpret the results from the analysis with confidence.

Course Objectives

- Build a structured understanding of finite element theory, from numerical methods foundations through 2D/3D element formulations.
- Develop hands-on proficiency in ANSYS for solving structural, dynamic, and heat transfer problems.
- Reinforce theoretical learning through numerical problems, guided tutorials, and industrial case studies.
- Train engineers to understand the underlying theory rather than only software operation, so results can be verified and errors identified rather than taken on faith.
- Prepare participants for advanced customized topics: contact, fatigue, modal, harmonic, thermal-stress, and multiphysics analysis.
- Consolidate learning through a complete industrial project (Vertical Mill) covering geometry through design optimization.

Course Outline

The course comprises **90 hours** in total, divided into **3 modules**: the 30-hour Foundation Course (10 hours theory, 10 hours ANSYS tutorials, 10 hours guided practice), the 36-hour Customized Course covering advanced additional topics, and the 24-hour Industrial Project (Vertical Mill). Timeline is flexible to change as per client's requirement and add-ons. Each

theory topic is paired with hands-on tutorials to reinforce learning and gauge understanding of the concepts covered.

Prerequisites

Participants should have basic knowledge of mathematics, numerical techniques, statics, dynamics, and general mechanical engineering related topics.

Table of Contents (TOC)

Module 1 – Foundation Course: FEM Theory & Practical

Software: ANSYS

Day 1 – FEM Fundamentals & Numerical Methods

- History of the Finite Element Method
- Numerical Methods: Matrix Algebra, Gauss Elimination Method, Gaussian Quadrature Rule
- Basics of Solid Mechanics
- Introduction to Finite Element Method — Infinite to Finite
- Coordinate Systems: Global, Local & Natural, and Coordinate Transformation
- Selection of Basic Unknown Variable & Shape Functions; Pascal Triangle
- Iso-parametric, Sub-parametric & Super-parametric Formulation
- Convergence Criteria, Patch Test, Conforming & Non-Conforming Elements
- C0 and C1 Continuity Elements
- Tutorial: Modelling, Meshing and Viewing Results (ANSYS)

Day 2 – Element Types & 1D Formulation

- Types of Basic and Higher-Order Elements
- Simplex, Complex & Multiplex Elements, and Special Elements
- Finite Element Formulation of 1D Bar/Rod Elements (Basic & Higher Order)
- Tapered and Uniform Bars Under Different Loading & Boundary Conditions
- Thermal / Temperature Stresses
- Plane and Space Trusses
- Torsion Element Formulation
- Treatment of Boundary Conditions: Elimination, Penalty & Multi-Point Constraint Approaches
- Tutorial: Three Problems Using Bar Elements
- Tutorial: Two Problems Using Truss & Torsion Elements

Day 3 – Beams, Frames & Numerical Problems

- Finite Element Formulation of Beam & Frame Structures
- Basics of Beam Theory & Hermite Shape Functions
- Derivation of Elemental Stiffness Matrix & Load Vectors
- Formulation of Plane and Space Frame Structures
- Numerical Problems on Uniform, Tapered & Stepped Bars (Including Thermal Load)
- Numerical Problems on Trusses and Beams
- Tutorial: Three Problems Using Beam, Plane & Space Frame Elements

Day 4 – 2D/3D Elements, Dynamics & Heat Transfer

- Introduction to 2D Elasticity — Plane Stress & Plane Strain
- Axisymmetric Stress-Strain Relations
- Triangular & Quadrilateral Element Formulation (CST, LST, QUAM4, QUAM8, TETRA4, TETRA10, HEXA8, HEXA20)
- Finite Element Formulation for Dynamic Analysis — Eigenvalue & Eigenvector Extraction
- 1D Formulation for Longitudinal, Lateral & Torsional Vibration
- Finite Element Formulation for Heat Transfer — Conduction & Convection
- Tutorial: Two Problems Using Triangular & Quadrilateral Elements
- Tutorial: Two Problems Using Axisymmetric Elements (2D Analysis)
- Tutorial: Three Problems on 3D Analysis
- Tutorial: Three Problems in Dynamics
- Tutorial: Three Problems in Heat Transfer — Conduction & Convection

Day 5 – Guided Practice & Consolidation

- 10 hours of instructor-guided practice consolidating Foundation Course exercises (scope flexible to cohort needs)

Module 2 – Customized Course: Additional Topics

Software: ANSYS

Advanced Topics (duration as marked — sequencing flexible per client requirement)

- FEM Workflow & Complex Geometry — CAD cleanup, preprocessing, meshing strategy, quality evaluation (covered in regular course)
- Structural Analysis — linear & non-linear behaviour, loads & boundary conditions, stress/deformation evaluation (2 Hrs Practice)
- Contact Analysis — bonded, frictional & sliding contacts, convergence and stability considerations (2 Hrs Theory + 4 Hrs Practice)
- Bolted Joint Analysis — pretension effects, load distribution, joint separation behaviour (covered in regular course)
- Welded Structure Analysis — stress concentration regions, weld modelling approaches, critical stress evaluation (2 Hrs Practice)
- Fatigue Analysis — durability assessment, life prediction methods, S-N curves, fatigue failure evaluation (2 Hrs Theory + 4 Hrs Practice)
- Modal Analysis — natural frequencies, mode shapes, identification of resonance conditions (2 Hrs Theory + 2 Hrs Practice)
- Harmonic Response Analysis — frequency response, damping effects, amplification under cyclic loading (2 Hrs Practice)
- Thermal & Thermal Stress Analysis — temperature distribution, thermal expansion, coupled structural response (2 Hrs Practice)
- Multiphysics Analysis — coupled field analysis, Fluid-Structure Interaction (FSI) (2 Hrs Theory + 6 Hrs Practice)

Module 3 – Industrial Project

Software: ANSYS

Industrial Project (Vertical Mill)

- Geometry Preparation, Meshing, Solution, Validation, Reporting & Design Optimization (24 Hrs Practice)