Finite Element Analysis (FEA): Linear Static, Nonlinear, Modal and Buckling Studies

20 – 24 September 2027
Barcelona - Eixample district four-star

Finite Element Analysis (FEA): Linear Static, Nonlinear, Modal and Buckling Studies

Ref: 546_17254 **Date:** 20 – 24 September 2027 **Location:** Barcelona - Eixample district four-star **Fees:** 23500 SAR

Introduction:

Finite element analysis FEA now signs off brackets, pressure parts, frames and machine components, yet many models reach approval with idealised supports that over-stiffen the part, peak stresses read at singular corners and no hand calculation to confirm the load path. This Core Concept course trains mechanical, structural and design engineers to build, check and challenge structural FEA studies in Nastran, ANSYS or Abaqus-type solvers, from element choice and meshing through linear, nonlinear, modal and buckling runs to allowable checks. Participants produce an FEA Study Report with a Verification Log for a case component.

Course Objectives:

- Idealise a component as beam, shell or solid elements and justify the simplifications against the stiffness and stress quantities the design decision needs
- Build meshes that pass Jacobian, warpage and aspect ratio checks and prove stress convergence through h-refinement or p-refinement studies
- Apply supports, contact, bolt pretension and distributed loads that reproduce the real load path without artificial restraint or reaction errors
- Interpret linear static results by separating genuine peak stress from singularities and nodal averaging artefacts, then compare linearised stresses with design code allowables
- Run modal, linear buckling and overview nonlinear and thermo-mechanical analyses and judge when a linear answer is no longer credible
- Verify models against hand calculations and validate them against strain gauge or load test data, and review another engineer's FEA report with a structured checklist

Target Audience:

- Mechanical design functions that size brackets, housings, shafts, frames and pressure-retaining parts using simulation
- Structural engineering functions that assess steel, plate and connection details beyond the reach of frame analysis programs
- Product development and engineering analysis functions that run simulation to replace or reduce physical prototype testing
- Engineering review and technical assurance functions that approve FEA submissions from consultants, fabricators and equipment vendors
- Asset integrity and reliability functions that assess cracked, corroded or modified components for continued service

Course Outline:

Day 1: Finite Element Theory, Element Families and Model Idealisation

- Direct Stiffness Method: Nodes, Degrees of Freedom and Global Stiffness Matrix Assembly
- Shape Functions, Integration Points and Linear Versus Quadratic Element Order
- Beam, Plate and Shell and Solid Element Families with Selection Criteria
- Geometry Idealisation: Midsurface Extraction, Defeaturing Fillets and Symmetry Planes
- FEA Study Specification: Decision Question, Load Cases, Result Quantities and Acceptance Limits

Day 2: Mesh Generation, Element Quality Checks and Convergence Studies

- Tetrahedral Versus Hexahedral Solid Meshes and Shell Quad Dominant Meshing
- Element Quality Diagnostics: Jacobian Ratio, Warpage, Taper and Aspect Ratio
- h-Refinement and p-Refinement Convergence Plots for Displacement and Stress
- Local Refinement at Notches, Holes and Weld Toes with Transition Zones
- Mixed-Dimension Connections: Rigid Links, Multi-Point Constraints and Shell-to-Solid Coupling

Day 3: Boundary Conditions, Load Application and Linear Static Stress Interpretation

- Support Modelling: Fixed, Pinned, Elastic Springs and Remote Displacement Choices
- Pressure, Bearing, Bolt Pretension and Gravity Loads with Reaction Force Balance Checks
- Contact at Overview: Bonded, Frictionless and Frictional Interfaces
- Stress Singularities at Re-Entrant Corners and Point Loads Versus Genuine Stress Concentrations
- Von Mises, Principal and Linearised Membrane Plus Bending Stresses with Nodal Averaging Effects

Day 4: Nonlinear, Modal, Buckling and Thermo-Mechanical Analysis

- Material Nonlinearity at Overview: Elastic-Plastic Stress-Strain Curves and Plastic Strain Limits
- Geometric Nonlinearity at Overview: Large Deflection, Stress Stiffening and Newton-Raphson Iteration
- Modal Analysis: Natural Frequencies, Mode Shapes and Effective Mass Participation
- Linear Eigenvalue Buckling Load Factors Versus Nonlinear Collapse with Imperfections
- Steady-State Thermal Runs and Thermal Stress Mapping onto the Structural Model

Day 5: Verification, Validation and Capstone FEA Study

- Hand Calculation Verification with Beam Theory, Roark-Type Formulas and Free-Body Checks
- Validation Against Strain Gauge Readings and Physical Load Test Results
- Design Code Allowable Checks and Utilisation Tables for the Case Component
- FEA Report Peer Review Checklist Applied to a Supplied Consultant Submission
- Capstone: FEA Study Report and Verification Log Presentation and Panel Challenge

Skills You Will Gain:

- Structural Model Idealisation
- Element Type Selection
- Mesh Convergence Assessment
- Load Path and Restraint Modelling
- Stress Singularity Diagnosis
- Modal and Buckling Assessment
- Simulation Verification and Validation
- FEA Report Technical Review

Why Attend This Course:

- Return with an FEA Study Report and Verification Log built on a case component during the capstone
- Stop reading peak colours at sharp corners as failure and explain to a reviewer which stresses belong in an allowable check
- Challenge consultant and vendor FEA submissions with a checklist that exposes restraint, mesh and load errors before approval
- Compare simulation practice with mechanical, structural and integrity engineers from manufacturing, energy, construction and equipment sectors

Conclusion:

An FEA result is only as credible as its idealisation, its restraints and the evidence that the answer holds under refinement and against independent checks. The week moves from stiffness method theory and element families, through meshing and convergence, to boundary conditions and linear static stress interpretation, then to nonlinear, modal, buckling and thermal analysis. The final day verifies and validates a case model, checks it against allowables and assembles an FEA Study Report with a Verification Log for panel challenge.