Course Type	Course Code	Name of Course	L	T	P	Credit
DP	NMEC512	Structural Modeling and Simulation Lab	0	0	3	1.5

Course Objective

The primary objective of the course is to expose the students to:

- Develop 2D/3D numerical mathematical models of different structural components using FEM
- Simulate the models for Vibration, Stress/deformation, and Fracture analyses under static and fatigue loading conditions using FEA/ MATLAB/ SIMULINK.
- Validate the developed Finite Element Models concerning existing analytical/experimental results.

Learning Outcomes

On successful completion of this course, students will be able to:

- Develop FEM-based mathematical models to analyze different structural components through proper selection of finite elements, mesh discretization, and application of boundary conditions.
- Carry out the analyses of the structural components under static/harmonic/fatigue loading conditions.
- Carry out the post-processing for stresses, deformations, failure analyses, fracture analyses, Vibration analyses, Condition monitoring,
- Validate the developed numerical model concerning the existing analytical/experimental results.

Exp. No.	Topics to be Covered	Contact Hours	Learning Outcome
1	Modal Analysis of a Cantilever Beam using ANSYS and its comparison with analytical method	6	Students will learn the modeling of a cantilever beam and the simulation of its modal behavior. They will verify the numerical results with the analytical approach-based results.
2	Time response of a 2-DOF linear oscillator using MATLAB (ODE 45 solvers) and SIMULINK	3	Students will learn to have the time response of a 2 DOF linear oscillator using MATLAB & SIMULINK. The numerical results will be verified by the students using the analytical results.
3	Stress Intensity factor (SIF) at the tips of a sharp crack in a centrally cracked plate under tension.	3	Students will understand the finite element modeling procedure to discover the Mode-I Stress Intensity Factor at crack tips. The students will verify FEA results through the Theory of Elasticity-based formulations.
4	Strain Energy Release Rate (SERR) at the side-edge crack tip plate under mixed mode loading conditions	3	Students will understand the finite element modeling procedure to determine Mode-I /Mode-II Strain Energy Release Rate at crack tips. They will verify the FEA results through VCCT-based formulations.

5	Natural frequencies and mode shapes of a cracked beam	6	The students will learn how to model a cracked beam and its finite element simulation for natural frequencies and mode shapes.
6	Study of variations in modal parameters due to failures such as cracks and bending in a rotor.	3	Students will learn the effect of defects (crack and bending) on the variation of modal parameters in a rotor through finite element analyses.
7	Stress Concentration Factor (SCF) at the periphery of a circular hole for the centrally drilled plate specimen under tension.	3	Students will learn to evaluate the stress concentration factor around a hole through 3D finite element analyses. They will validate FEA-based results concerning the Strength of material-based formulations.
8	Structural analyses of a Truss for support reactions internal forces/stresses and deformation for simply supported boundary conditions.	3	Students will learn how to model a Truss structure using ANSYS. They will understand the post-processing of results regarding support reactions, member stresses, structural deformation, etc. The students will verify the FEA-based results concerning the rigid body mechanics-based solution.
9	Three-dimensional stress (radial/hoop/axial stresses) analyses of a Thick cylinder subjected to external pressure	6	Students will learn the stress analyses of 3D structures through three-dimensional Finite Element Analyses. FEA-based results will be verified concerning the Theory of Elasticity-based formulations.
10	Stress and deformation fluctuation in a DCB subjected to a finite/infinite number of load cycles (Fatigue loading)	6	The students will learn to simulate the behavior of a Double Cantilever Beam (DCB) under fatigue loading conditions. The FE results will be verified in terms of analytical/experimental results.
Total		42	

Text Books:

1. Finite Element Analyses: Theory and Application using ANSYS. Saeed Moaveni, PEARSON Publications.
2. The Finite Element Method and Applications in Engineering using ANSYS. ErdoganMadenci, Ibrahim Guven, Springer Publications

Reference Books:

1. Finite Element Analyses using ANSYS 11.0: P. Srinivas, K.C. Sambana, R.K. Datti. PHI Publishing House.