COMPUTATIONAL STRUCTURAL ANALYSIS LABORATORY

VII Semester: AE

Course Code	Category	Hours / Week			Credits	Maximum Marks		
		L	T	P	C	CIE	SEE	Total
AAE111	Core	-	-	3	2	30	70	100
Contact Classes: Nil	Tutorial Classes: Nil	Practical Classes: 36				Total Classes: 36		

OBJECTIVES:

The course should enable the students to:

I. To apply the basic principles learnt from pre-requisites subjects to solve the structural problem.
II. To adopt any computational structural analysis software and learns how to perform analysis.
III. Analyze structural problems related to Aerospace industry.
IV. Interpret the results and how to apply them on the real-life structure.

COURSE LEARNING OUTCOMES (CLOs):

The students should able to:

1. Understand the basic features of an analysis package.
2. Understand how to apply the theoretical process to solve the problem computationally.
3. Build the mathematical model using modern tools to formulate the problem.
4. Computationally solve the 2D and 3D trusses, beams, plates under various loadings.
5. Determined different stresses, deflections, shear force and bending moment diagrams.
6. Demonstrate the modal analysis on different structural members of different materials.
7. Calculate the natural frequencies under various boundary conditions and analyze with forcing functions.
8. Analyze the non-linear behavior of the material to determine the large deflections.
9. Illustrate the harmonic responses of the spring-mass systems and interpret them for real time problem.
10. Model the 3D components and execute the results for the applied loads to measure the results.
11. Apply the static analysis results to assess the dynamic behavior of the structure.
12. Identify the forces acting on landing gear and analyze the basic landing gear to find the stresses.
13. Use the ANSYS ACP application for building the composite structure.
14. Examine the composite behavior of the structure and evaluate the results.
15. Work on ANSYS APDL & Workbench platforms to evaluate the results for basic aerospace structure.

LIST OF EXPERIMENTS

Week-l	**INTRODUCTION AND BASIC FUCTIONS**
a. Starting up of ANSYS/Nastran b. Description of user interface	
Week-2	**STATIC ANALYSIS: TRUSS AND FRAME STRUCTURES**
a. 2-D truss structures b. 3-D truss structures	
Week-3	**STATIC ANALYSIS: BEAMS**
a. Straight beams b. Tapered beams	
Week-4	**STATIC ANALYSIS: TWO DIMENSIONAL PROBLEMS**

a. 2-D structure with various loadings b. 2-D structures with different materials c. Plate with hole	
Week-5	**DYNAMIC ANALYSIS: MODAL AND TRANSIENT ANALYSES**
a. Modal analysis b. Transient Response (spring-mass system)	
Week-6	**THERMAL ANALYSIS**
a. Bars and beams b. 2D structures	
Week-7	**NON LINEAR ANALYSIS**
a. Nonlinear behavior (Large deflections) b. Nonlinear behavior (Materials)	
Week-8	**HARMONIC RESPONSE ANALYSIS**
a. Random Vibration Analysis of a Deep Simply-Supported Beam b. Harmonic Response of a Spring-Mass System	
Week-9	**ANALYSIS OF AIRCARFT STRUCTURE: WING**
a. Static analysis of Aircraft wing structure b. Modal analysis of aircraft wing structure	
Week-10	**ANALYSIS OF AIRCARFT STRUCTURE:FUSELAGE**
a. Static analysis of Aircraft Semi monoque fuselage structure b. Modal analysis of aircraft Semi monoque fuselage structure	
Week-11	**ANALYSIS OF AIRCARFT STRUCTURE:LANDING GEAR**
a. Static analysis of main landing gear b. Modal analysis of main landing gear	
Week-12	**ANALYSIS OF COMPOSITE STRUCTURES**
a. Static analysis of composite bar and beam b. Static analysis of composite plate	
Text Books:	
1. Huei-Huang Lee, "Finite Element Simulations with ANSYS Workbench 16", SDC publications, 2nd Edition, 2016. 2. Anderson, William J "MSC/Nastran: Interactive Training Program" Wiley 1st Edition 2015. 3. "ANSYS Mechanical APDL Basic Analysis Guide", ANSYS,Inc Release 16.0.	
Web References:	
1. https://www.ansys.com/services/learning-hub 2. https://caeai.com/ansys-software-support/ansys-software/mechanical-simulation-software	