ADVANCED COMPUTATIONAL AERODYNAMICS LABORATORY

VI Semester: AE

Course Code	Category	Hours / Week			Credits	Maximum Marks		
		L	T	P	C	CIA	SEE	Total
BAEB09	Core	-	-	4	2	30	70	100
Contact Classes: Nil	Tutorial Classes: Nil	Practical Classes: 36				Total Classes: 36		

I. COURSE OVERVIEW:

The major emphasis of this course is to solve a complex geometrical structures under a given loads, these methods does not have analytical solutions. Software's like ANSYS and NASTRAN is utilized to interpret results for complex geometries. Modeling of crack and composite structures help the students to solve realistic problems which are common in industries. Structural analysis on aircraft structures and Rocket components are delt to obtain the solution for bending and torsion under the applied aerodynamic loads.

II. COURSE OBJECTIVES:

The course should enable the students to:

I. Experience in computing aerodynamic problems and understanding flow physics over the objects.
II. Knowledge in estimating flow analysis for different mach numbers.
III. Determining the aerodynamic forces like mainly lift and drag.
IV. Analyze the errors and cause of errors in computational analysis.

III. COURSE OUTCOMES:

After successful completion of the course, students will be able to:

CO 1	**Apply** the philosophy behind the computational fluid dynamics for recognizing flow properties in solving fluids and heat transfer problems.	Apply
CO 2	**Select** the structured, unstructured mesh and multi-blocking strategy in basic, complex geometries and flow domains for computing aerodynamic characteristics.	Apply
CO 3	**Identify** the appropriate physical boundary conditions for attaining the precise results of fluid flow over a body.	Understand
CO 4	**Choose** the suitable numerical modeling and schemes for computational simulations of aerodynamics and thermo-fluid problems using ANSYS.	Understand
CO 5	**Analyze** the numerical solution of fluid flow problems using flow visualization Software's for recognizing the flow physics in and around the supersonic intake and free jet.	Analyze
CO 6	**Develop** the numerical code for one dimensional heat and wave equation using explicit finite difference method.	Apply

LIST OF EXPERIMENTS

Week-1	INTRODUCTION

Introduction to computational aerodynamics, the major theories, approaches and methodologies used in computational aerodynamics. Applications of computational aerodynamics for classicalaerodynamic's problems.

Week-2	INTRODUCTION TO ANSYS CFX

Introduction to gambit, geometry creation, suitable meshing types and boundary conditions.

Week-3	INTRODUCTION TO ANSYS FLUENT
Introduction to fluent, boundary conditions, solver conditions and post processing results.	
Week-4	FLOW THROUGH NOZZLE
Flow Through Nozzle	
Week-5	FLOW THROUGH SUPERSONIC INTAKE
Flow Through Supersonic Intake	
Week-6	SUPERSONIC FREE JET
Flow over a Supersonic Free Jet	
Week-7	SHOCK BOUNDARY LAYER INTERACTION
Shock Boundary Layer Interaction).	
Week-8	FLOW OVER A RE-ENTRY VEHICLES
Flow over a re- entry vehicle	
Week-9	SUPERSONIC FLOW OVER A CONE
Flow over wedge body at supersonic Mach number; observe the shock wave phenomena and change of properties across the shock wave.	
Week-10	THERMAL TESTING TURBINE BLADE
Flow over a Missile body	
Week-11	CASCADE TESTING COMPRESSOR BLADE
Solution for the following equations using finite difference method I. One dimensional wave equation using explicit method of lax. II. One dimensional heatconduction equation using explicit method.	
Week-12	HEAT CONDUCTION
I. One dimensional heat conduction equation using explicit method.	

Reference Books:

1. Anderson, J.D., Jr., Computational Fluid Dynamics The Basics with Applications, McGraw-Hill Inc, 1stEdition 1998.
2. Hoffmann, K. A. and Chiang, S. T., “Computational Fluid Dynamics for Engineers”, 4th Edition, Engineering Education Systems (2000).
3. Hirsch, C., “Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics”, Vol. I, 2nd Edition, Butterworth-Heinemann (2007).
4. JAF. Thompson, Bharat K. Soni, Nigel P. Weatherill “Grid generation”, 1st Edition 2000.

Web References:

1. https://www.scribd.com/doc/311680146/eBook-PDF-Cfd-Fluent.
2. https://cfd.ninja/tutorials/ansys-fluent
3. https://confluence.cornell.edu/display/SIMULATION/FLUENT+Learning+UNITs