COMPUTATIONAL AERODYNAMICS LABORATORY

VI Semester: AE

Course Code	Category	Hours / Week			Credits	Maximum Marks		
		L	T	P	C	CIA	SEE	Total
AAE109	Core	-	-	3	2	30	70	100
Contact Classes: Nil	Tutorial Classes: Nil	Practical Classes: 36				Total Classes: 36		

I. COURES OVERVIEW:

Computational Aerodynamics laboratory sessions focus on the creation of geometry, meshing (Dis-cretization) and the physics applied to aerodynamics in order to visualize fluid flow and temperature distribution, and estimating the flow parameters around the aerodynamic body. Computational Aero- dynamics laboratory also covers the usage of finite difference methods and necessary coding techniques.In this lab course, the students are trained on conducting simulations using the numerical methods analysis tool of CAD systems. The simulations include fluid, structural, thermal systems in the emerg- ing technologies of interdisciplinary applications such as mechanical, aerospace, refrigeration systems.

II. OBJECTIVES:

The course should enable the students to:

I	The concepts of grid generation techniques for simple and complex domains to model fluid flow problems.
II	The aspects of numerical discretization techniques such as finite volume and finite difference methods.
III	The mathematical modeling of different classes of partial differential equations to show their impact on computational fluid dynamics.
IV	The characteristics of different turbulence models and numerical schemes for estimating the criteria of stability, convergence, and error of fluid flow problem.

III. COURSE OUTCOMES:

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

CO 1	**Choose** the finite difference method at grid points of the domain for understanding discretization technique in solving fluid flow problem.	Apply
CO 2	**Classify** the nature of fluid flow problems for solving the governing equations using computational methods.	Analyze
CO 3	**Make use of** the computational methods and algorithms forobtaining solutions of fluid flow problems using ANSYS.	Apply
CO 4	**Simplify** the parameters of thermo-fluid systems using simulation methods for validating numerical and experimental results.	Analyze
CO 5	**Estimate** the aerodynamic forces on the slender and bluff bodiesfor calculating the lift and drag coefficients.	Evaluate
CO 6	**Assess** the numerical solution of fluid flow problems using discretization methods and convergence criteria for better resultsand minimize the errors.	Evaluate

IV. SYLLABUS:

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 classical aerodynamic's problems.

Week-2	**INTRODUCTION TO GAMBIT**
Introduction to gambit, geometry creation, suitable meshing types and boundary conditions.	
Week-3	**INTRODUCTION TO FLUENT**
Introduction to fluent, boundary conditions, solver conditions and post processing results.	
Week-4	**FLOW OVER A FLAT PLATE**
Flow over a flat plate at low Reynolds numbers, observe the boundary layer phenomena, no slip condition and velocity profile inside the boundary layer.	
Week-5	**FLOW THROUGH PIPE**
Flow through pipe at different Reynolds numbers; observe the velocity changes for laminar and turbulent flows.	
Week-6	**FLOW OVER A CIRCULAR CYLINDER**
Flow over a circular cylinder at different Reynolds numbers, observe the properties at separation region and wake region.	
Week-7	**FLOW OVER A CAMBERED AEROFOIL**
Flow over a cambered aerofoil at different velocities, observe flow properties and compare the computation results with experimental results (consider the model from aerodynamics laboratory).	
Week-8	**FLOW OVER A SYMMETRIC AEROFOIL**
Flow over a symmetric aerofoil at different velocities, observe flow properties and compare the computation results with experimental results (consider the model from aerodynamics laboratory).	
Week-9	**FLOW OVER WEDGE**
Flow over wedge body at supersonic mach number; observe the shock wave phenomena and change of properties across the shock wave.	
Week-10	**FLOW OVER A CONE**
Flow over a cone at supersonic mach number; observe the shock waves and 3D relieving effect.	
Week-11	**CODE DEVELOPEMENT**
Solution for the following equations using finite difference method I. One dimensional wave equation using explicit method of lax. II. One dimensional heat conduction equation using explicit method.	
Week-12	**CODE DEVELOPEMENT**
Generation of the following grids I. Algebraic grids. II. Elliptic grids.	

Reference Books:

1. Anderson, J.D., Jr., Computational Fluid Dynamics The Basics with Applications, McGraw-Hill Inc, 1st Edition 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+Modules
Course Home Page:
SOFTWARE AND HARDWARE REQUIREMENTS FOR A BATCH OF 36 STUDENTS: **SOFTWARE:** ANSYS 16 **HARDWARE:** Desktop Computers with 4 GB RAM 36 nos