COURSE CONTENT

ADVANCED AERODYNAMICS LABORATORY								
I Semester: AE								
Course Code	**Category**	**Hours / Week**			**Credits**	**Maximum Marks**		
BAEE11	Core	**L**	**T**	**P**	**C**	**CIA**	**SEE**	**Total**
		-	-	4	2	40	60	100
Contact Classes: Nil	**Tutorial Classes: Nil**	**Practical Classes: 45**				**Total Classes: 45**		
Prerequisite: Aerodynamics								

I. COURSE OVERVIEW:

The major emphasis of this course is to study complex flow problems in different flow regimes for those methods do not have analytical solutions. Experimental and computational analysis shall be performed to find the aerodynamic characteristics like CL, CD, CM with variation of the Reynolds number and angles of attack. Software's like ANSYS Fluent / CFX will be employed to find aerodynamic efficiency for complex geometries. Modeling of flow help the students to solve realistic problems which are common in industries. Aerodynamic analysis on aircraft models, Rocket and missiles are dealt to obtain the solution for different applied aerodynamic variables.

II. COURSE OBJECTIVES:

The students will try to learn:

I. The utilization of ANSYS Fluent software to obtain the solution for complex flow conditions.
II. The mathematical methods involved in aerodynamic analysis of flight vehicle.
III. Computational aerodynamic modeling using ANSYS software and determine its characteristics.
IV. Experimental Aerodynamic analysis of the complex shapes by using wind tunnel and determine the flight performance and stability criteria.

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.
CO 2 Select the structured, unstructured mesh and multi-blocking strategy in basic, complex geometries and flow domains for computing aerodynamic characteristics.
CO 3 Identify the appropriate physical boundary conditions for attaining the precise results of fluid flow over a body.
CO 4 Choose the suitable numerical modeling and schemes for computational simulations of aerodynamics and thermo-fluid problems using ANSYS.
CO 5 Analyze the numerical solution of fluid flow problems using flow visualization Software's and wind tunnel for recognizing the flow physics in and around the supersonic intake and free jet.
CO 6 Make use of the Wind Tunnel for predicting the profile drag using boundary layer and wake momentum theory.

IV. COURSE CONTENT:

Exercise-1: INTRODUCTION

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

Exercise -2: INTRODUCTION TO ANSYS CFX/ FLUENT

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

Exercise -3: INTRODUCTION TO SUBSONIC WIND TUNNEL

Model a subsonic wind tunnel with 50 m/s velocity in test section and carryout the fluid flow analysis to find the power required and flow characteristics in test section.

Exercise -4: FLOW THROUGH NOZZLE

Design a convergent nozzle with area ratio of 1.58. For inlet pressure of 5bar and temperature of 500K calculate exit velocity and momentum thrust generated by the nozzle.

Exercise -5: FLOW THROUGH SUPERSONIC INTAKE

Design a supersonic intake for Mach 2 free stream velocity and calculate the pressure recovery factor for inlet total pressure of 3 bar and temperature of 300K. Find the shock strength as well.

Exercise -6: SUPERSONIC FREE JET

Design a convergent divergent nozzle to generate Mach 2 free jet and evaluate the decay characteristics of the jet like centerline Mach number, total pressure and vortex propagation.

Exercise -7: SHOCK BOUNDARY LAYER INTERACTION

Design a supersonic flow over a flat plate with free stream Mach number of 2 and evaluate the effect of Shock Boundary Layer Interaction

Exercise -8: FLOW OVER A RE-ENTRY VEHICLE

Design a re-entry vehicle with blunt nose for Mach 10 and calculate the aerodynamic heating on the surface of vehicle and Maximum temperature at the stagnation point.

Exercise -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 shockwave.

Exercise -10: PROFILE DRAG PREDICTION

Calculate the profile drag of a circular cylinder having diameter of 50 mm using wetted area method.

Exercise -11: WAKE MOMENTUM DRAG MEASUREMENT

Locating the Transitional separation bubbles

Exercise -12: BOUNDARY LAYER MEASUREMENT

Predict the profile drag of a flat plate having length of 2000 mm and width of 600 mm for test section velocity of 40 m/s and 50m/s.

Exercise -13: PRESSURE DISTRIBUTION OVER SYMMETRICAL AND CAMBERED AIRFOIL

Predict the lift and drag coefficient of symmetrical and cambered airfoil at test section velocity of 40 m/s and 50m/s.

Exercise -14: FORCE MEASUREMENT

Predict the forces and moments of an aircraft model at different angle of attack for test section velocity of 40m/s, and 50 m/s

V. TEXT BOOKS:

1. Anderson, J.D., Jr., Computational Fluid Dynamics the Basics with Applications, McGraw-Hill Inc,1st Edition, 1998.
2. Mark Drela., "Flight Vehicle Aerodynamics", MIT Press, Cambridge, Massachusetts London, England.

VI. REFERENCE BOOKS:

1. Hirsch, C., "Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics", Vol. I, 2nd Edition, Butter worth-Heinemann (2007).
2. JAF. Thompson, Bharat K. Soni, Nigel P. Weatherill, "Grid Generation", 1st Edition, 2000.

VII. ELECTRONICS RESOURCES:

1. https://www.scribd.com/doc/311680146/eBook-PDF-Cfd-Fluent.

VIII. MATERIALS ONLINE

1. Course template
2. Lab manual