

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal - 500 043, Hyderabad, Telangana

COURSE CONTENT

COMPUTATIONAL FLUID DYNAMICS LABORATORY								
VI Semester: ME								
Course Code	Category	Hours / Week			Credits	Maximum Marks		
AMED45	Core	L	Т	Р	С	CIA	SEE	Total
		0	0	2	1	40	60	100
Contact Classes: Nil	Tutorial Classes: Nil	Practical Classes: 45				Total Classes: 45		
Prerequisite: Nil								

I. COURSE OVERVIEW:

This laboratory course provides learners, the depth practical knowledge on Ansys Workbench and its different modules. The students will get the hands-on experience to validate their theoretical design ideas to provide solutions to real-world problems. The lab sessions focus on creation of geometry, meshing (Discretization) and the physics applied to fluid thermal systems in order to solve and review results. This helps the learners in interdisciplinary applications such as mechanical, aerospace, automobile, refrigeration systems etc.

II. COURSES OBJECTIVES:

The students will try to learn

- I. The formulation of the problem including material properties and boundary conditions.
- II. The modelling of complex geometries in the best possible manner and identify the domain of interest.
- III. The discretization of the domain in a manner to capture the physics accurately.
- IV. The software to analyse results with different methods like contour plots, vector plot, streamlines, etc.
- V. The skills of using CFD packages to carry out research in interdisciplinary applications.

III. COURSE OUTCOMES:

At the end of the course students should be able to:

- CO 1 Explain the physics of the given problem and identify the appropriate tool to be used.
- CO 2 Make use of different modelling tools of Ansys workbench for modelling the domain of interest.
- CO 3 Utilize the different meshing tools of Ansys workbench for discretizing the domain of interest.
- CO 4 Make use of Ansys CFX software Flow Simulation for analyzing simple fluid flow problems.
- CO 5 Utilize the Ansys steady state- thermal software for analyzing simple heat transfer problems.
- CO 6 Analyze the conjugate heat transfer problems using Ansys Fluent software.

IV. SYLLABUS:

WEEK 1: INTRODUCTION TO ANSYS

Understand and practice the modeling in Ansys workbench Design Molder.

WEEK 2: INTRODUCTION TO ANSYS WORKBENCH MESHING

Understand and practice the meshing techniques in Ansys workbench Mesh.

WEEK 3: CFD ANALYSIS OF FLOW THROUGH DUCTS

Analyze the flow through a straight and converging diverging duct.

WEEK 4: CFD ANALYSIS OF FLOW OVER BLUNT BODIES

Analyze the flow over a cylinder and Building.

WEEK 5: CFD ANALYSIS OF FLOW OVER STREAM LINED BODIES

Analyze the flow over a symmetric and cambered aero foil.

WEEK 6: CFD ANALYSIS OF FLOW THROUGH VALVES

Analyze the flow through a ball valve and gate valve.

WEEK 7: CFD ANALYSIS OF 2D STEADY STATE HEAT FLOW PROBLEMS Analyze the heat flow through 2D simple and composite walls.

WEEK 8: CFD ANALYSIS OF 3D STEADY STATE HEAT FLOW PROBLEMS

Analyze the heat flow through 3D simple and composite walls.

WEEK 9: CFD ANALYSIS OF NATURAL CONVECTION PROBLEMS

Analyze the heat flow from circular fins under natural convection condition.

WEEK 10: CFD ANALYSIS OF FORCED CONVECTION PROBLEMS

Analyze the heat flow from square fins under forced convection condition.

WEEK 11: CFD ANALYSIS OF TUBE IN TUBE HEAT EXCHANGER PROBLEMS

Analyze the heat flow through heat exchanger in parallel flow and counter flow conditions.

WEEK 12: CFD ANALYSIS OF SHELL AND TUBE HEAT EXCHANGER PROBLEMS

Analyze the heat flow through heat exchanger in parallel flow and counter flow conditions.

WEEK 13: CFD ANALYSIS OF FLOW THROUGH CENTRIFUGAL MACHINE

Analyze the flow through centrifugal pump or compressor

WEEK 14: CFD ANALYSIS OF FLOW THROUGH RADIAL MACHINES

Analyze the flow through radial turbine.

V. TEXT BOOKS:

- 1. Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu Computational Fluid Dynamics: A Practical Approach, Elsevier Science, 2017.
- 2. J. D. Anderson, (Jr), "Computational Fluid Dynamics", McGraw-Hill Book Company, 1st Edition, 2017.
- 3. K.A.Hoffman, and S. T. Chiang, "Computational Fluid Dynamics", Vol. I, II and III, Engineering Education System, Kansas, USA, 2004.

VI. REFERENCE BOOKS:

- 1. Chung, T.J., "Computational Fluid Dynamics", Cambridge University Press, 2003.
- D. A. Anderson, J. C. Tannehill, and R. H. Pletcher, "Computational Fluid Mechanics and Heat Transfer", CRC Press, 4th Edition 2020.
- 3. Muralidhar K and Sundararajan. Computational Fluid Flow and Heat Transfer", 2009.

VII. ELECTRONICS RESOURCES:

- 1. https://courses.ansys.com/
- 2. https://www.udemy.com/course/ansys-tutorial/
- 3. https://akanksha.iare.ac.in/index?route=course/player&course_id=1685§ion_id=3005&lesson_id=19706.

VIII. MATERIALS ONLINE:

- 1. Course Content
- 2. Lab manual