FLUID THERMAL MODELING AND SIMULATION LABORATORY

VI Semester: ME

Course Code	Category	Hours / Week			Credits	Maximum Marks		
		L	**T**	**P**	**C**	**CIA**	**SEE**	**Total**
AMEB25	**Core**	-	-	2	1	30	70	100

Contact Classes: Nil	Tutorial Classes: Nil	Practical Classes: 24	Total Classes: 24

I. COURSE OVERVIEW:

The ANSYS software has different modulus (Ansys, CFX, Fluent etc...). The Workbench environment is an intuitive up-front finite element analysis tool that is used in conjunction with CAD systems and/or Design Modeler. ANSYS Workbench is a software environment for performing structural, thermal, and fluid flow analyses. The laboratory sessions are focuses on geometry creation, meshing and how to apply the boundary conditions, attaching existing geometry, setting up the model, solving, and reviewing results. The lab sessions will describe how to create geometry, how to use the basic finite element simulation concepts, as well as Computational Fluid Dynamics concepts and how to do interpretation of results.

II. OBJECTIVES:

The courses should enable the students to:
I.   Analyz the Internal and External fluid flow problems.
II. Apply FEM techniques to fluid flow problems.
III. Evaluate the thermal stresses of real time problems.

III. COURSE OUTCOMES:

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

CO 1	**Demonstrate** the application of finite element method for analyzing 2D fluid flow problems. .	Understand
CO 2	**Make use of** Ansys CFX or Solid Works Flow Simulation for analyzing simple fluid flow problems.	Apply
CO 3	**Develop** the Matlab code for analyzing 2D fluid flow problems.	Apply
CO 4	**Make use of** Ansys or Solid Works Flow Simulation for analyzing simple heat transfer problems.	Apply
CO 5	**Make use of** Ansys Fluent for analyzing conjugate heat transfer.	Apply
CO 6	**Make use of** Ansys for analyzing thermal stress in piston.	Apply

IV. SYLLABUS:

LIST OF EXPERIMENTS

Week-1	**Internal Pipe Fluid Flow - FEM**

Internal Pipe flow problem Using theoretical FEM

Week-2	**Internal Pipe Fluid Flow - ANSYS**

Analyzing Flow in a System of Pipes using ANSYS

Week-3	**Internal Pipe Fluid Flow - MATLAB**

Internal Pipe flow problem using MAT LAB

Week-4	**External Fluid Flow**

Determination of the drag coefficient of a circular cylinder immersed in a uniform fluid stream using ANSYS/Solid Works Flow Simulation

Week-5	**Flow Through Ball Valve**

Flow of water through a ball valve assembly using ANSYS/Solid Works Flow Simulation

Week-6	**Heat Conduction**
Heat Conduction within a Solid using ANSYS	
Week-7	**Temperature Distribution**
Temperature distribution in a fin cooled electronic component using ANSYS	
Week-8	**3D Heat Conduction**
3D Heat Conduction within a Solid-Cell Phone using ANSYS	
Week-9	**Counter Flow Heat Exchanger**
Calculation of the efficiency of the counter flow heat exchanger using ANSYS/Solid Works Flow Simulation	
Week-10	**Conjugate Heat Transfer**
Conjugate heat transfer problem using ANSYS/Solid Works Flow Simulation	
WeeK-11	**3D Thermal Analysis**
3D Thermal Analysis, Finned Pipe using ANSYS	
Week-12	**Thermal Stress Analysis**
Thermal stress analysis of piston	
Week-13	**Review of Fluid Problems**
Week-14	**Review of Thermal Problems**

Text Books:

1. Janna, W.S., "Design of Fluid Thermal Systems", Cengage Learning, 3rd Edition, 2011
2. Jaluria, Y., "Design and Optimization of Thermal Systems", McGraw-Hill, 2nd Edition, 2007.
3. McDonald, A. G., and Magande, H. L., "Thermo-Fluids Systems Design", John Wiley, 2012.
4. Suryanarayana, N. V. and Arici, Ö., "Design and Simulation of Thermal Systems", McGraw-Hill, 2003.

Web References:

1.https://docs.google.com/document/d/1UaDrm0pnHgd8GnN7dAcXM6EikgqAD7BU-0d52VFZz1w/edit
2. http://www.iare.ac.in