Course Type	Course Code	Name of Course	L	T	P	Credit
DP	NFMC528	CFD of Thermal and Fluid Systems Lab	0	0	3	1.5

Course Objective
The course aimstogiveinformationaboutthe selection of different models and numerical approaches in codes and software for the computation of thermo-fluid systems.
Learning Outcomes
Attheend ofthe course,studentswill gethands-on knowledgeabout • Coding and solving CFD problems on structured grids. • Simulating CFD problems with complex geometries.

Exp. No.	Name of Experiment	Practical Hours	LearningOutcome
1	Introduction to the basics of the CFD lab	3	Basic understanding of the lab manual and lab procedures. Lab guidelines.
2	Determining the numerical solution of anordinary differential equation using the finite difference method.	3	Using finite difference to solve an ODE.
3	Determining the numerical solution of a steady-state 1-D diffusion system using finite volume method (FVM).	3	Using FVM to solve a steady-state 1-D diffusion system.
4	Determining the numerical solution of a steady-state 2-D diffusion system using FVM.	3	Using FVM to solve a steady-state 2-D diffusion system.
5	Determining the numerical solution of a steady-state 1-D convection-diffusion system using FVM.	3	Using FVM to solve a steady-state 1-D convection-diffusion system.
6	Determining the numerical solution of a steady-state 2-D convection-diffusion system using FVM.	3	Using FVM to solve a steady-state 2-D convection-diffusion system.
7	Determining the numerical solution of an unsteady 1-D diffusion system using FVM.	3	Using FVM to solve an unsteady 1-D diffusion system.
8	Solving a LID DRIVEN cavity problem using ANSYSFluent.	3	Knowledge about using ANSYS Fluent software for CFD simulations
9	Solving aLID-DRIVEN cavity problem using OpenFOAM.	3	Knowledge about using OpenFOAM software for CFD simulations
10	Impact of pressure-velocity coupling algorithms on the convergence rates of a LID-DRIVEN cavity problem.	3	Comparing the different pressure-velocity coupling algorithms
11	Simulating a drop tube furnace using ANSYS Fluent.	3	Knowledge about relevant modeling for drop tube furnace simulations

12	Simulating a turbulent premixed flame using ANSYS Fluent.	3	Knowledge about relevant modeling for turbulent premixed flame simulations
13	Analyze transient numerical data obtained from a simulation	3	Methods to analyze transient numerical data obtained from a simulation.
14	Lab Examination	3	Students understanding of the coursewill be gauged based on their capability to numerically solve a thermo-fluid problem.
Total		**42**	

Text Book:

1. An introduction to computational fluid dynamics (the finite volume method): Pearson, *Authors*: H.K. Versteeg and W. Malalasekera.

Reference Book:

1. Lab Manual for CFD of Thermal and Fluid Systems Lab at the FMME Department in IIT ISM Dhanbad.