Course Type	Course Code	Name of Course	L	T	P	Credit
DP	NMEC530	Computational Fluid Dynamics Lab	0	0	3	1.5

Course Objective
This is introductory laboratory course on Computational Fluid Dynamics. This course aims to provide fundamentals concepts and their application in Computational Fluid Dynamics. They will be learning different solution methods to handle the complex problem in the course.
Learning Outcomes
Upon successful completion of this course, students will: • have a broad understanding of Computational Fluid Dynamics. • have analytical and mathematical tools to handle complex problems. • be able to provide some basic solution to real life heat transfer and fluid dynamics problems.

Unit No.	Topics to be Covered Lecture	Lab Hours	Learning Outcomes
1	Solution of 1D and 2D unsteady heat conduction equation using FDM (Implementation of Crank-Nicolson scheme/ RK4 schemes with explicit/ implicit methods, Gauss-Seidel, TDMA, line-by-line methods through computer coding)	3	Students will learn about 1D and 2D steady heat conduction equation and its application. Various solving methods will be discussed.
2.	Solution of 1D and 2D unsteady heat conduction equation using FEM	3	Students will learn about 1D and 2D unsteady heat conduction equation and its application. Various solving methods will be discussed.
3.	Solution of 1D and 2D convection/ convection-diffusion equations using FDM	3	This module will enable the students to understand 1D and 2D convection/ convection-diffusion equations using FDM.
4.	Solution of 1D and 2D convection/ convection-diffusion equations using FEM	3	This module will enable the students to understand 1D and 2D convection/ convection-diffusion equations using FVM.
5.	Introduction to the commercial package ANSYS-FLUENT	3	This module is about learning ansys fluent package.
6.	Construction of simple and complex meshes using ANSYS-FLUENT: A few cases	3	Students will learn about constructing meshes using ansys fluent.
7.	Basic CFD problem using laminar flow in ANSYS-FLUENT	3	Students will learn about Basic CFD problems of laminar flow.
8.	Introduction to basic turbulence modeling theory and simple examples in ANSYS-FLUENT	3	Students will learn about turbulence modelling.
9.	Unsteady flow past a fixed circular cylinder ANSYS-FLUENT	3	Students will learn about flow past a circular cylinder.
10.	Unsteady flow past an airfoil at different Reynolds numbers using ANSYS-FLUENT	3	Students will learn about the flow past an airfoil.

11.	Basic Heat transfer problems in ANSYS-FLUENT	3	Students will learn about Basic heat transfer problems.
12.	Structured mesh generation by computer coding - algebraic method and method of transformation	3	Students will learn about constructing structured meshes using ansys fluent.
13.	(a) Structured uniform/non-uniform meshing (by writing codes) for simple shapes like square, rectangular, cylindrical geometries, (b) Generation of structured mesh for regular/irregular shapes using coordinate transformation	3	Students will learn about constructing various meshes using ansys fluent.
14.	Report submission, evaluation	3	Students have to submit the lab report for evaluation and test/viva-voce will be conducted.
	Total	42	

Text Books:

1. John D. Anderson, Computational Fluid Dynamics The basics with applications, McGraw-Hill Education, 1st Edition, 2017.
2. Joel H. Ferziger and M. Peric, Computational Methods for Fluid Dynamics, Springer, 3rd Edition, 2002.

References:

1. Richard H. Pletcher, John C. Tannehill and Dale A. Anderson, Computational Fluid Mechanics and Heat Transfer, CRC Press, 3rd Edition, 2012.
2. Clive A. J. Fletcher, Computational Techniques for Fluid Dynamics, Springer, 1st Edition, 1988.