COURSE TITLE	**COMPUTATIONAL FLUID DYNAMICS**
COURSE CODE	**01ME1733**
COURSE CREDITS	**4**

Objective:

1 This course aims to introduce numerical modelling and its role in automotive field; it will enable the students to understand the various discretization methods and solving methodologies and to create confidence to solve complex problems in the automotive field with the knowledge of Heat transfer and fluid dynamics. Further students can able to develop finite difference and finite volume discretized forms of the CFD equations and to formulate explicit & implicit algorithms for solving the Euler Equations & Navier Stokes Equations

Course Outcomes: After completion of this course, student will be able to:

1 Students will be able to apply the principles of conservation equations and boundary conditions to solve fluid flow problems.

2 Students will be able to assess the effect of discretization types on the accuracy of numerical solution in a finite difference method.

3 Students will be able to apply mesh generation methods and coordinate transformations for transport problems.

4 Students will be able to analyze steady and unsteady diffusion and convection-diffusion problems using the finite volume method to examine the influence of various discretization schemes.

5 Students will be able to apply CFD solution procedures to solve fluid flow and heat transfer problems.

Pre-requisite of course: Fluid Mechanics, Heat Transfer, Higher Order Engineering Mathematics

Teaching and Examination Scheme

Theory Hours	Tutorial Hours	Practical Hours	ESE	IA	CSE	Viva	Term Work
3	0	2	50	30	20	25	25

Contents : Unit	Topics	Contact Hours
1	**Introduction to Computational Fluid Dynamics and Principles of Conservation** Continuity Equation, Navier Stokes Equation, Energy Equation and General Structure of Conservation Equations, Classification of Partial Differential Equations, Types of boundary conditions	8

Contents : Unit	Topics	Contact Hours
2	**Basic aspects of Discretization** Introduction, Discrete grid points, Taylor series expansion: first-second order forward-backward-central differential, grid points at boundary, 1-D and 2-D heat conduction, Explicit and Implicit, Errors and stability	12
3	**Grid Generation and Transformation** Overview of mesh generation, Structured and Unstructured mesh, Guideline on mesh quality and design, Mesh refinement and adaptation, General transformation equation, matrices and Jacobean	6
4	**Finite Volume Method** Finite volume discretization of convection-diffusion problems: 1-D steady state diffusion, 2-D diffusion problems, 1-D steady state convection-diffusion, Central difference scheme, Upwind scheme, Hybrid scheme, Power law scheme, QUICK scheme, SIMPLE, SIMPLER, SIMPLEC and PISO algorithms	10
5	**CFD Solution Procedure** Problem setup – creation of geometry, mesh generation, selection of physics and fluid properties, initialization, solution control and convergence monitoring, results reports and visualization	6
Total Hours		**42**

Suggested List of Experiments:

Contents : Unit	Topics	Contact Hours
1	**To prepare the 2-D geometry of the flat plate** To prepare the 2-D geometry of the flat plate	2
2	**To apply mesh condition in 2-D problem** To apply mesh condition in 2-D problem	2
3	**To prepare the 3-D geometry of the pipe** To prepare the 3-D geometry of the pipe	2
4	**To apply mesh condition in 3-D problem** To apply mesh condition in 3-D problem	2
5	**To perform the numerical analysis on flow through a pipe** To perform the numerical analysis on flow through a pipe	2
6	**To perform the numerical analysis on a flat plate boundary layer** To perform the numerical analysis on a flat plate boundary layer	2
7	**To perform the numerical analysis on compressible flow in nozzle** To perform the numerical analysis on compressible flow in nozzle	2
8	**To perform the numerical analysis on convective heat transfer** To perform the numerical analysis on convective heat transfer	2
9	**To perform the numerical analysis on steady flow past a cylinder** To perform the numerical analysis on steady flow past a cylinder	2

Suggested List of Experiments:

Contents : Unit	Topics	Contact Hours
10	**To perform the numerical analysis on unsteady flow past a cylinder** To perform the numerical analysis on unsteady flow past a cylinder	2
11	**To perform the numerical analysis on flow over an airfoil** To perform the numerical analysis on flow over an airfoil	2
12	**To perform the numerical analysis on heat conduction through the wall** To perform the numerical analysis on heat conduction through the wall	2
Total Hours		**24**

Textbook :

1 Computational Fluid Dynamics, J. D. Anderson Jr., McGraw-Hill International Edition, 2017

2 An introduction to computational fluid dynamics: The finite volume method, H. K. Versteeg and W. Malalasekera, Pearson Education, 2007

References:

1 Fundamentals of Computational Fluid Dynamics, Fundamentals of Computational Fluid Dynamics, Patrick J. Roache, Hermosa Publishers, 1998

2 Introduction to Computational Fluid Dynamics: Development, Application, and Analysis, Introduction to Computational Fluid Dynamics: Development, Application, and Analysis, Atul Sharma, Wiley, 2021

3 Computational Fluid Dynamics, Computational Fluid Dynamics, Jiyuan Tu, Guan Heng Yeoh, and Chaoqun Liu, Butterworth-Heinemann, 2018

4 Computational Fluid Flow and Heat Transfer, Computational Fluid Flow and Heat Transfer, K. Muralidhar, and T. Sundararajan, Narosa Publishing. House, 1995

5 Computer Simulation of Flow and Heat Transfer, Computer Simulation of Flow and Heat Transfer, P.S. Ghosdastidar, Tata McGraw-Hill, 1998

Suggested Theory Distribution:

The suggested theory distribution as per Bloom's taxonomy is as follows. This distribution serves as guidelines for teachers and students to achieve effective teaching-learning process

Distribution of Theory for course delivery and evaluation					
Remember / Knowledge	**Understand**	**Apply**	**Analyze**	**Evaluate**	**Higher order Thinking / Creative**
10.00	15.00	25.00	25.00	15.00	10.00

Instructional Method:

1 PPT and Animations

Supplementary Resources:

1. Open FOAM and SCILAB
2. www. Cfd-online.com
3. https://fluids.ac.uk/talks
4. http://www.efluids.com/