GUJARAT TECHNOLOGICAL UNIVERSITY

Bachelor of Engineering
Subject Code: 3170109
Semester –VII
Subject Name: Advance Computational Fluid Dynamics

Type of course: Professional Elective course III

Prerequisite: Basic course on Fluid Mechanics, Thermodynamics and Numerical Methods, Basic computational fluid dynamics,

Rationale: This is course in ACFD. In this course, students will be exposed to basics and advance of CFD. Students will gain knowledge on FD/ FV/Turbulence model strategy, formulation of the problem and solution techniques.

Teaching and Examination Scheme:

Teaching Scheme			Credits	Examination Marks				Total Marks
L	T	P	C	Theory Marks		Practical Marks		
				ESE (E)	PA (M)	ESE (V)	PA (I)	
3	0	2	4	70	30	30	20	150

Content:

Sr. No.	Content	Total Hrs
1	**Review of Basic Computational Fluid Dynamics:-** CFD and its application, Governing flow equations, incompressible and compressible flows, Elliptic, Parabolic and Hyperbolic partial differential equations, finite difference and finite volume method, explicit and implicit methods	**06**
2	**Grid Formulation:-** Introduction, Algebric methods, Multi block structured grid generation, Unstructured Grid generation : Delalunay triangulation, Advancing front methods, Bowyer Algorithm, Adaptive grids	**08**
3	**Implementation of Boundary Conditions:-** Introduction, Inlet and Outlet Boundary Condition, Wall Boundary condition, The constant pressure boundary condition, symmetry boundary condition, periodic or cyclic boundary condition	**08**
4	**Turbulence Modeling:-** What is turbulence flow? Characteristics and important features of turbulent flow, The effect of turbulent fluctuations on properties of mean flow, Turbulent flow calculations, Vorticity transport equation, Statistical representation of turbulent flows: Homogeneous turbulence and isotropic turbulence, General Properties of turbulent quantities, Reynolds average Navier stokes (RANS) equation, mixing length model, Reynold stress equation models, Closure problem in turbulence, Necessity of turbulence modeling, Eddy viscosity models, Turbulent kinetic energy and dissipation, The κ-ε model, Advantages and disadvantages of κ-ε model, SST, RNG κ-ε model and κ-ω model, Basics of Reynolds stress model (RSM), Large eddy Simulation (LES), Direct numerical simulation (DNS)	**16**
5	**Some Aspects of Real life CFD analysis: -** How Does CFD code Work? Structure of	**07**

	CFD code, Types of Solver (Pressure based solver and Density based solver)	

Suggested Specification table with Marks (Theory):

Distribution of Theory Marks					
R Level	U Level	A Level	N Level	E Level	C Level
30%	**30%**	**20%**	**10%**	**10%**	**0**

Legends: R: Remembrance; U: Understanding; A: Application, N: Analyze and E: Evaluate C: Create and above Levels (Revised Bloom's Taxonomy)

Note: This specification table shall be treated as a general guideline for students and teachers. The actual distribution of marks in the question paper may vary slightly from above table.

Reference Books:

1. Anderson J.D. (1995) Computational Fluid Dynamics: The Basics with Applications, McGraw-Hill, Inc.
2. Anderson, D.A., Tannehill, J.C. and Pletcher, R.H. (1997).Computational Fluid Mechanics and Heat Transfer. Taylor& Francis. Fluid Mechanics and Hydraulic Machines by R.K. Rajput, S.Chand & Co.
3. Versteeg, H. K. and Malalasekara, W. (2008). Introduction to Computational Fluid Dynamics: The Finite Volume Method. Second Edition (Indian Reprint) Pearson Education.
4. T. J. Chung, Computational Fluid Dynamics, Cambridge University Press.
5. S. V. Patankar, Numerical Heat Transfer and Fluid Flow, McGraw-Hill

Course Outcomes:

Upon completion of this course students should be able:

Sr. No.	CO statement	Marks % weightage
CO1	To understand the underlying theoretical basics of CFD.	30%
CO2	Illustrate various descretization techniques used to solve PDE.	30%
CO3	Apply the various discretization methods, solution procedures to solve flow problems.	20%
CO4	Categorize different numerical techniques used to solve fluid flow problems.	10%
CO5	Determine various quantities of interest such as velocity, pressure etc.analyzing given fluid field.	10%

List of Experiments:

1. Introduction to CFD software in details.
2. To simulate Lid driven cavity flow using different solver.
3. To simulate flow over a Flat Plate using different turbulence model.
4. To simulate flow through C-D Nozzle for compressible flow and Incompressible flow.
5. To simulate Flow over different Airfoil.

.

List of Open Source Software/learning website:
https://nptel.ac.in/