CEPE3002 COMPUTATIONAL FLUID DYNAMICS (3-0-0)

Course Objective:

The overall objective of Computational Fluid Dynamics (CFD) is to simulate and analyze fluid flow and heat transfer problems by using computers to solve the governing mathematical equations, enabling engineers to predict system performance, optimize designs, and understand complex flow phenomena of Heat and mass transfer operations.

Course Outcomes:

After the completion of course students will be able to

CO1: Explain the fundamental principles of CFD, governing equations (continuity, Navier-Stokes), boundary conditions, and their applications in chemical engineering

CO2: Apply discretization techniques (FDM, FVM, FEM) and numerical schemes to solve fluid flow and heat transfer problems.

CO3: Use turbulence models, grid generations, and commercial CFD tools to simulate and analyze engineering systems.

Module - I: (10hrs)

Introduction to CFD and principles of conservation; continuity equation, Navier-Stokes equation. Simplifications for incompressible flow.

Classification and overview of numerical methods; Classification of partial differential equations and physical behaviour i.e., elliptic, parabolic, hyperbolic. CFD application in Chemical Engineering.

Boundary conditions: Boundary layer equations and review of flow dynamics essentials.

Module - II: (10hrs)

Discretization techniques: Finite differential method, Finite volume method, Finite element method (basic introduction), stability, residuals and convergence.

Central difference scheme, Upwind scheme, exponential scheme, hybrid scheme, QUICK scheme, Peclet number influence, Total Variation Diminishing (TVD) schemes (introductory).

Finite volume discretization; Finite element method; Finite differential method.

Module - III: (10hrs)

Turbulence modelling: Laminar vs turbulent flows, Eddy Viscosity concept, k-ε and k-ω models (introductory treatments), RANS equation.

Grid generation: Staggered grids, structured vs unstructured grids, Mesh quality, Grid independence.

Introduction to commercial CFD tools (ANSYS Fluent, Open FOAM). Case studies of pipe flow and heat exchanger segment.

Textbooks:

1. Ferziger, J. H. & Peric, M. — Computational Methods for Fluid Dynamics
2. Versteeg, H. K. & Malalasekera, W. — An Introduction to Computational Fluid Dynamics: The Finite Volume Method

Reference books:

1. Anderson, J. D. — Computational Fluid Dynamics: The Basics with Applications
2. Muralidhar, K. & Sundararajan, T. — Computational Fluid Flow and Heat Transfer
3. Jiri Blazek — Computational Fluid Dynamics: Principles and Applications
4. Computation Fluid Dynamics, by Prof. Suman Chakraborty, IITKharagpur (https://onlinecourses.nptel.ac.in/noc21_me126/preview)