APPLIED COMPUTATIONAL FLUID DYNAMICS
ENME 413

Lecture	**: 3**	**Year : IV**
Tutorial	**: 1**	**Part : I**
Practical	**: 3/2**	

Course Objectives:
The objective of this course is to provide principles and practices of Computational Fluid Dynamics (CFD), bridging the gap between theoretical fluid mechanics and industrial simulation. Students will be able to discretize and solve governing equations while learning to construct high-quality meshes and correctly apply physical boundary conditions.

1 Introduction to CFD and Fluid Mechanics Review (6 hours)

1.1 History and role of CFD in modern mechanical engineering design
1.2 Review of fluid kinematics: Velocity field, streamlines, pathlines and flow classification
1.3 Laminar versus turbulent flow, Reynolds number and its engineering significance
1.4 CFD workflow: Geometry, meshing, boundary conditions, solver and post-processing
1.5 Advantages, limitations and validation of CFD against experiments and analytical solutions
1.6 Interface layout, file structure and basic navigation of CFD software interface

2 Governing Equations and Numerical Foundations (9 hours)

2.1 Continuity equation in differential and integral forms
2.2 Navier-Stokes equations and physical meaning of each term
2.3 Enthalpy formulation and simplified heat transfer equation
2.4 Application of Euler equations and stokes flow
2.5 Boundary conditions: Inlet, outlet, wall (No-slip), symmetry and periodic
2.6 Matrix systems, direct and iterative solution methods
2.7 Numerical errors: Truncation error, round-off error, consistency and stability
2.8 Convergence concepts: Residuals, iteration and physical convergence indicators

3 Discretization Methods (12 hours)

3.1 Finite difference method (FDM): Taylor series expansion, forward, backward and central differences
3.2 Accuracy and order of approximation: First-order and second-order schemes

3.3 Finite volume method (FVM): Control volume formulation, flux balance, cell-centered approach
3.4 Discretization of the diffusion term and convection term
3.5 Upwind, central differencing and hybrid schemes: Numerical diffusion and stability
3.6 Time discretization: Explicit and implicit schemes and CFL stability condition
3.7 Pressure-velocity coupling: Physical motivation and SIMPLE algorithm
3.8 Iterative linear solvers used in CFD: Gauss-Seidel, conjugate gradient and algebraic multigrid
3.9 Practical application of FVM

4 Mesh Generation and Solver Setup (9 hours)

4.1 Types of meshes: Structured, unstructured and hybrid
4.2 2D and 3D mesh elements: Triangles, quads, tetrahedra, hexahedra and prism layers
4.3 Mesh quality metrics: Skewness, aspect ratio and orthogonality
4.4 Near-wall mesh requirements: Boundary layer resolution and the y+ parameter
4.5 Geometry preparation and meshing workflow
4.6 Mesh independence study: Systematic refinement and solution sensitivity analysis
4.7 Defining boundary conditions: Velocity inlet, pressure outlet, wall and symmetry
4.8 Solver configuration: Steady-state vs. transient, under-relaxation factors and convergence monitoring
4.9 Post-processing: Contour plots, vector fields, streamlines and surface integrals

5 Turbulence Modeling (3 hours)

5.1 Physical nature of turbulence: Chaotic fluctuations, energy cascade and engineering implications
5.2 Reynolds-Averaged Navier-Stokes (RANS) approach: Time-averaging and the closure problem
5.3 Common turbulence models: k-ε and k-ω SST
5.4 Selection criteria of a turbulence model
5.5 Wall functions: Role in near-wall treatment and their relationship to mesh y+ requirements

6 CFD Applications (6 hours)

6.1 Internal flow: Pipe flow, duct flow and pressure drop prediction in pipe networks

6.2 External aerodynamics: Flow over bluff bodies and streamlined profiles with lift and drag evaluation

6.3 Heat transfer applications: Forced convection in channels, cooling fins and heat exchangers

6.4 HVAC and ventilation: Airflow distribution, thermal comfort and indoor environment simulation

6.5 Fluid machinery: Flow through pump impellers and turbine passages using rotating reference frames

6.6 Case studies and result interpretation: Identifying flow features, validating against benchmarks

Tutorial (15 hours)

1. Fluid properties and flow classification: Worked examples and Reynolds number calculations
2. Finite difference exercises: Deriving approximations and applying them to 1D diffusion problems
3. Finite volume method: Hand-worked flux balance problems for 1D convection-diffusion
4. ANSYS Workbench walkthrough: Setting up a 2D internal flow problem from scratch (geometry to results)
5. Mesh quality assessment: Identifying and correcting poor-quality meshes in provided geometries
6. Application problems: Interpreting CFD output for a heat exchanger and a duct flow case

Practical (22.5 hours)

1. Orientation session: ANSYS Workbench interface, units and file management
2. Laminar pipe flow simulation: Velocity profile and comparison with the Hagen-Poiseuille solution
3. 2D lid-driven cavity: Mesh convergence study and visualization of recirculation zones
4. Flow over a flat plate: Boundary layer growth, skin friction and comparison with Blasius theory
5. External flow over a circular cylinder: Pressure distribution and drag coefficient measurement
6. 2D airfoil simulation (NACA 0012): Lift and drag at varying angles of attack
7. Forced convection in a channel with heated walls: Temperature contours and Nusselt number estimation
8. Heat exchanger geometry: Inlet/outlet temperature difference and pressure drop analysis

9. Centrifugal pump impeller flow: Rotating reference frame setup, pressure rise and velocity vectors
10. Mini-project: Student-chosen mechanical engineering flow problem, report covering geometry, mesh, results and validation

Final Exam

The questions will cover all the chapters in the syllabus. The evaluation scheme will be as indicated in the table below:

Chapters	Hours	Marks distribution*
1, 2	15	20
3	12	20
4, 5	12	15
6	6	5
Total	**45**	**60**

* There may be minor deviation in marks distribution.

References

1. Anderson, J. D. (1995). Computational fluid dynamics: The basics with applications (Latest Edition). McGraw-Hill Education.
2. Versteeg, H. K., Malalasekera, W. (2007). An introduction to computational fluid dynamics: The finite volume method . Pearson Education.
3. Munson, B. R., Young, D. F., Okiishi, T. H. (2013). Fundamentals of fluid mechanics. John Wiley & Sons.
4. Ferziger, J. H., Perić, M., Street, R. L. (2020). Computational methods for fluid dynamics. Springer.
5. Ansys, Inc. (2023). ANSYS Fluent user's guide. Ansys, Inc.