

COMPUTATIONAL FLUID DYNAMICS

ENAS 412

Lecture : 3
Tutorial : 1
Practical : 3/2

Year : IV
Part : I

Course Objectives:

The objective of this course is to provide students with a comprehensive understanding of the mathematical foundations of Computational Fluid Dynamics (CFD). It aims to develop practical proficiency in commercial and open-source CFD software, as well as MATLAB for implementing and visualizing numerical algorithms.

1 Introduction (3 hours)

- 1.1 Historical development of CFD
- 1.2 Engineering applications of CFD
- 1.3 General CFD workflow
- 1.4 Overview of major CFD software platforms
- 1.5 Verification and validation (V and V)

2 Governing Equations: Classification and Boundary Conditions (6 hours)

- 2.1 Governing equations of fluid flow
- 2.2 Conservation of mass, momentum and energy
- 2.3 Scalar transport: General convection-diffusion equation
- 2.4 Closure relations: Equation of state, Newtonian fluid constitutive relations and thermodynamic assumptions
- 2.5 Mathematical nature and classification of PDEs
- 2.6 Classification of the Navier-Stokes and Euler equations
- 2.7 Domain of dependence and range of influence
- 2.8 Well-posedness: Existence, uniqueness and continuous dependence on data for CFD problems
- 2.9 Types of boundary conditions: Dirichlet, Neumann and Robin
- 2.10 Inlet, outlet, wall, symmetry and periodic boundary conditions
- 2.11 Far-field and non-reflecting boundary conditions for external flow simulations
- 2.12 Initial conditions for steady and unsteady simulations: Influence on convergence behavior and transient accuracy

3 Discretization Schemes

(9 hours)

- 3.1 Finite difference method (FDM)
- 3.2 Taylor series expansion: First and second derivatives
- 3.3 FDM discretization of 1D and 2D steady and unsteady convection-diffusion equations
- 3.4 Explicit and implicit time-marching for parabolic equations
- 3.5 Finite volume method (FVM)
- 3.6 Control volume formulation: Flux balance over cell faces and the integral conservation principle
- 3.7 Cell-centred FVM: Face interpolation, gradient reconstruction
- 3.8 Diffusion discretization: Central differencing and correction for non-orthogonal meshes
- 3.9 Convection discretization: Upwind, central differencing, linear upwind and QUICK schemes
- 3.10 Pressure-velocity coupling: SIMPLE, SIMPLEC and PISO
- 3.11 Time integration in FVM: First- and second-order implicit schemes and dual-time stepping
- 3.12 High-order and flux-based schemes

4 Analysis of Numerical Schemes

(9 hours)

- 4.1 Consistency and truncation error
- 4.2 Definition of consistency: Truncation error and its relationship to the formal order of accuracy
- 4.3 Deriving truncation errors for Finite Difference Method and Finite Volume Methods schemes using Taylor expansion
- 4.4 Modified equation analysis: Identifying numerical diffusion and dispersion errors
- 4.5 Von Neumann stability analysis
- 4.6 CFL (Courant-Friedrichs-Lewy) condition
- 4.7 Stability of the diffusion equation
- 4.8 Lax equivalence theorem
- 4.9 Iterative convergence and grid convergence
- 4.10 Iterative methods for linear systems: Point iterative methods, Krylov subspace methods

5 Turbulence Modeling

(6 hours)

- 5.1 Physics of turbulence: Characteristics of turbulent flow
- 5.2 Turbulent kinetic energy and dissipation rate
- 5.3 DNS cost estimate and practical limitations
- 5.4 Reynolds-Averaged Navier-Stokes (RANS) Modeling
- 5.5 Reynolds decomposition and time-averaging
- 5.6 Zero and one-equation models
- 5.7 Two-equation models: Standard k-epsilon model

- 5.8 k-omega and k-omega SST models
- 5.9 Near-wall treatment and advanced modeling
- 5.10 Large Eddy simulation (LES)
- 5.11 Detached Eddy simulation (DES)
- 5.12 Practical turbulence model selection: Guidelines based on flow type, computational budget and required accuracy

6 Mesh Generation

(6 hours)

- 6.1 Fundamentals of mesh generation
- 6.2 Role of the mesh in CFD
- 6.3 Mesh elements in 2D and 3D
- 6.4 Structured grids: Algebraic generation methods, transfinite interpolation and elliptic grid solvers
- 6.5 Unstructured grids: Delaunay triangulation and advancing front methods
- 6.6 Hybrid and polyhedral meshes
- 6.7 Mesh quality and refinement
- 6.8 Mesh quality metrics: Skewness, orthogonality, aspect ratio and expansion ratio- their effect on solver accuracy and convergence
- 6.9 Boundary layer meshing: Inflation layers, first cell height estimation, stretching ratios and y^+ targeting
- 6.10 Mesh independence study: Systematic refinement, convergence of key output quantities and Richardson extrapolation
- 6.11 Adaptive mesh refinement (AMR): Feature-based, gradient-based and adjoint-based adaptation strategies
- 6.12 Geometry preparation for meshing: Surface cleanup, defeaturing and watertight geometry requirements

7 Errors and Uncertainty in CFD Modeling

(6 hours)

- 7.1 Sources of error in CFD
- 7.2 Classification of CFD errors
- 7.3 Verification of CFD Simulations
- 7.4 Code verification: Method of manufactured solutions (MMS)
- 7.5 Solution verification: Grid convergence studies, Grid convergence index (GCI) and Richardson extrapolation
- 7.6 Iterative convergence verification: Residual monitoring, force and moment convergence and stopping criteria
- 7.7 Validation of CFD simulations
- 7.8 Validation hierarchy: Unit problems, subsystem benchmarks and full system comparisons
- 7.9 NASA/AIAA best practice guidelines for CFD verification and validation in engineering applications
- 7.10 Uncertainty quantification

- 7.11 Sources of uncertainty: Geometric, physical model, numerical and the distinction between aleatory and epistemic uncertainties
- 7.12 Sensitivity analysis: Parameter variation studies and their influence on simulation output quantities

Tutorial

(15 hours)

1. Governing Equation Classification: Identify the mathematical character of potential flow, boundary layer and Navier-Stokes equations to determine appropriate domains of dependence and boundary conditions.
2. Finite Difference Boundary Layer Analysis: Derive finite difference formulas and apply them to 1D laminar flat plate flow, comparing results against the Blasius solution.
3. Finite Volume Convection-Diffusion Discretization: Perform hand-calculated flux balances across central, upwind and hybrid schemes to evaluate Peclet number effects on surface heat transfer.
4. Von Neumann Stability Analysis: Derive amplification factors and CFL limits to determine maximum stable time steps for subsonic and transonic flows.
5. Truncation Error and Numerical Diffusion: Analyze modified equations for upwind and central schemes to quantify numerical diffusion and its impact on shock thickness.
6. Grid Convergence and Uncertainty Estimation: Apply systematic mesh refinement and Richardson extrapolation to evaluate grid independence and numerical uncertainty for aerodynamic forces.

Practical

(22.5 hours)

1. 1D Convection Scheme Implementation: Program upwind, central and Lax-Wendroff schemes to analyze phase, dissipation and dispersion errors against exact wave solutions.
2. 2D Convection-Diffusion Aerodynamic Solver: Assemble and solve finite volume matrices to visualize scalar contours and evaluate grid refinement effects over a heated surface.
3. Stability and Convergence Analysis: Systematically vary CFL numbers and mesh spacing across explicit and implicit schemes to map stability boundaries for unsteady acoustic wave propagation.
4. 2D Subsonic External Flow Simulation: Generate structured boundary layer grids for a subsonic airfoil, apply far-field conditions and extract pressure coefficients and streamlines.
5. Turbulence Model Evaluation at High Angle of Attack: Compare one-equation and two-equation turbulence closure models against experimental lift, drag and boundary layer separation data.
6. Transonic Compressible Flow and Shock Capture: Simulate transonic flow over a supercritical airfoil using a density-based solver to quantify shock wave location, wave drag and discretization uncertainty.

7. Supersonic Nozzle Flow Analysis: Set up a convergent-divergent nozzle to analyze design and off-design conditions, comparing shock position and Mach distributions against 1D isentropic theory.
8. Script-Based Solver Setup and Post-Processing: Configure directory-based case files and boundary conditions for incompressible flow over a flat plate, validating skin friction against analytical theory.
9. Automated 3D Mesh Generation and Aerodynamic Validation: Build an automatically refined 3D volume mesh using surface geometries, simulate high-Reynolds-number flow and document solver performance discrepancies.

Practical sessions use MATLAB (For implementing and analyzing numerical algorithms), Ansys Fluent (For commercial CFD simulation) and OpenFOAM (For open-source simulation and case customization).

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	9	10
3	9	15
4	9	15
5	6	10
6, 7	12	10
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.
2. Versteeg, H. K., Malalasekera, W. (2007). An introduction to computational fluid dynamics: The finite volume method. Pearson.
3. Ferziger, J. H., Perić, M., Street, R. L. (2020). Computational methods for fluid dynamics. Springer.
4. Blazek, J. (2015). Computational fluid dynamics: Principles and applications. Butterworth-Heinemann.
5. Wilcox, D. C. (2006). Turbulence modeling for CFD. DCW Industries.
6. Roache, P. J. (1998). Verification and validation in computational science and engineering (Latest Edition). Hermosa Publishers.