COMPUTATIONAL FLUID DYNAMICS / FLOW PHYSICS AND NUMERICAL METHODS – 3ème année cycle ingénieur Estia

Second diplôme : Master of science « Computational Fluid Dynamics » de l'Université de Cranfield

Course details : MODULES

MODULE INTRODUCTION TO FLUID MECHANICS AND HEAT TRANSFER

Aim
To introduce the foundations of fluid mechanics, various formulations of governing equations and their mathematical properties in order to establish a firm basis for other modules.

Syllabus
Introduction to thermodynamics of gases and liquids
Introduction to heat transfer
Compressible flows
Incompressible flows
Dimensional analysis and similarity parameters
Mathematics of governing equations, classification of PDEs
Model equations for fluid dynamics
Introduction to unstable and turbulent flows.

MODULE NUMERICAL METHODS AND HIGH PERFORMANCE COMPUTING

Aim
To introduce the basics of numerical analysis and numerical methods for partial differential and algebraic equations, relevant to Computational Fluid Dynamics, and how to efficiently employ the latest technologies of high performance computing (HPC) for numerically solving these equations.

Syllabus
- Introduction to numerical analysis
- Discretisation approaches: finite difference, finite volume, finite element and spectral methods
- Numerical methods for algebraic equations/systems of equations
- Numerical schemes for hyperbolic, parabolic and elliptic systems and for fluid dynamics.
- Desktop versus distributed computing facilities
- Hardware and software aspects of HPC
- Parallel computing challenges and main issues
- Parallelisation approaches for distributed and shared memory systems. MPI & OPENMP
- Current CFD process with respect to partitioning and distributed computing and related bottlenecks
- Whole HPC product applications

MODULE NUMERICAL MODELLING FOR INCOMPRESSIBLE FLOWS

Module Leader

Dr Laszlo Konozsy

Aim
To understand the state-of-the-art CFD methods used for computing incompressible flows in science and engineering.

Syllabus
- Overview of various formulations of the governing equations and numerical methods for incompressible flows (linear & high-resolution methods)
- Solution approaches: pressure Poisson, projection (approximate and exact), artificial compressibility
- Centred schemes
- TVD and Riemann solvers for incompressible methods
- Second and high-order methods (time and spatial discretise)

MODULE NUMERICAL MODELLING FOR COMPRESSIBLE FLOWS

Module Leader
Dr Panagiotis Tsoutsanis

Aim
To introduce basic concepts in the discretisation and numerical solution of the hyperbolic systems of partial differential equations describing the flow of compressible fluids.

Syllabus
- Mathematical properties of hyperbolic systems
- Conservation Laws
- Non-linearities and shock formation
- WENO schemes
- MUSCL schemes Introduction to the Riemann problem
- Lax-Wendroff scheme
- Introduction to Godunov's method
- Flux vector splitting methods
- Approximate Riemann solvers
- Explicit and implicit time-stepping schemes

MODULE TURBULENCE MODELLING

Aim
To introduce students to closure methods for the Navier-Stokes equations as applied to turbulent and transitional flows, and the classical physical modelling approximations required to achieve this. To introduce the advanced turbulence modelling approaches used in Computational Fluid Dynamics such as Large Eddy Simulations and Direct Numerical Simulations.

Syllabus
- Introduction to Reynolds Averaged Navier Stokes Modelling
- Zero, One and Two equation models
- Reynolds Stress Transport Schemes
- Low-Re Modelling
- Transition Modelling Extensions
- Best Practice Guidelines
- Overview of the basic equations used in LES, including filtered and unfiltered formulations
- Classical LES and sub-grid scale models
- Implicit LES (numerical and physical principles)
- Numerical and physical properties of DNS
- Applications and challenges for LES and DNS

MODULE GRID GENERATION / CAD

Aim

To introduce the concepts of grid generation, including structured and unstructured approaches. To provide hands-on experience using commercial CAD and grid generation packages.

Syllabus
- Computer aided design interface with grid generation
- Geometry Modelling and Surface Grids
- Algebraic Mesh Generation
- Structured Meshes from Partial Differential Equations
- Automatic generation of Unstructured Meshes
- Multiblock Mesh Generation
- Unstructured grids by Delaunay Triangulation
- Mesh Adaptation on Unstructured Grids
- Unstructured Grids for Viscous Flows
- Grid generation tutorials with ANSYS-Icem-cfd and Pointwise commercial software.

MODULE DATA ANALYSIS AND UNCERTAINTY

Aim
To provide an introduction into the use of visualisation, data mining, and interactive human-computer interfaces for the analysis and interpretation of CFD simulations. Visualisation can be a critical component in helping an engineer gain insight into the typically complex optimization problems that arise in design. Through the combination of visualisation and user interaction in computer tools, the engineer's insight can help guide the computer in the process of identifying better, more effective designs. Visualisation can also be combined with automated data mining techniques to improve optimization procedures.

To provide hands-on experience using both commercial and community developed visualisation packages.

To introduce the concepts of error and uncertainty and how they relate to the credible numerical solution of the partial differential equations encountered in computational fluid mechanics.

Syllabus
- Data interchange formats
- Interpretation of data
- Graphical representation of data
- Parallel data visualisation
- Data mining, reduced order modelling, model identification and surrogate models
- Data fusion
- Virtual reality visualisation
- The right answer: consistency, stability and convergence revisited
- Taxonomies of error and uncertainty
- Principles of code verification
- Introduction to the method of manufactured solutions
- Principles of solution verification
- Role of systematic iterative and space-time grid convergence studies
- Richardson extrapolation
- Principles of validation
- Construction of validation hierarchies

MODULE THE ROLE OF EXPERIMENTAL DATA IN CFD

Aim
To provide an introduction into practical techniques for experimental data collection and its subsequent post-processing. To contrast the resultant data representation with that obtained through CFD simulation.

Syllabus
Introduction to the measurement of turbulent flows
Velocity and pressure measurement by aerodynamic probes
Velocity measurement by hot-wires/hot-film

Velocity measurement by optical techniques
Temperature measurement
Simple optical visualisation, Shadowgraph, Schlieren
Laser-based temperature and species measurements
Laser Induced Fluorescence
Skin friction, convective and radiative heat transfer
Error analysis.

Elective modules

A selection of modules from the following list need to be taken as part of this course

CFD FOR AEROSPACE APPLICATIONS

Module Leader
Dr Panagiotis Tsoutsanis

Aim
To understand the key features of CFD methods used for simulating external flows in aeronautical and aerospace applications.

Syllabus
Overview of external flow problems in aeronautical and aerospace applications
CFD methods for subsonic, supersonic and hypersonic regimes
CFD methods for design
Application examples.

CFD FOR MICRO AND NANO FLOWS

Aim
To introduce micro- and nano- scale phenomena and CFD methods used for micro and nanoscale applications.

Syllabus
Introduction to micro- and nano- scale phenomena
Areas of CFD application in micro- and nanoscience
Borderline continuum/molecular models and their domains of applicability
Multiscale modelling.

CFD FOR ROTATING WINGS

Aim
To introduce the numerical approaches required to meet the challenges of flows associated with rotating wings, including rotorcraft, propellers, wind turbines and
turbomachinery.

Syllabus
Introduction to rotary wing aerodynamics
Formulation of the governing equations in a rotating inertial frame of reference
Numerical approaches to vortex capturing
Blade dynamics as an example of fluid-structure interaction
Formulation of the governing equations for moving/deforming grids
Numerical modelling of dynamic stall.

CFD FOR AUTOMOTIVE FLOWS

Aim

To increase the awareness of the students regarding the representative flow physics and flow structures usually encountered in automotive flows, the main challenges that arise for modelling them, and the state-of-the-art approaches for modelling them.

Syllabus

- Application Areas for CFD in automotive engineering
- Choice of CFD technique appropriate to the problem
- CFD as a Complement to Experiment
- High Performance Computing & Design Optimisation
- Analysing Results
- Lattice Boltzmann methods
- RANS, LES, DES and VLES in the context of automotive flows
- Differentiation between Time-Averaged Flow Field and steady-state average solutions
- Unsteady Flow Simulations concepts (Strouhal number, vortex shedding, time averaging, time-step selection, convection times)
- Stopping criteria for unsteady simulations.
- Meshing concept (near wall modelling, Immersed boundary methods, sliding meshes)
- Non-zero yaw modelling and crosswind simulation strategies
- Reference geometries (Ahmed, Drivair, Windsor)

Wide range of commercial/open-source software packages can be used including: Pointwise, ICEM, STAR CCM+, Fluent, OpenFoam, Powerflow.

CFD FOR MUTIPHASE FLOWS AND COMBUSTION

Aim

To introduce physics of multiphase flows and combustion as well as numerical methods for the simulation of multiphase and reacting flows. To provide examples of applications.

Syllabus

Physical insight into multiphase and reacting flows

Governing equations and models for multiphase flows & combustion

Numerical methods for multiphase and reacting flows.

Particle tracking methods

Examples of applications.

CFD FOR ENVIRONMENTAL FLOWS

Aim

To introduce the application of CFD to environmental flows in urban, inland and coastal environments.

Syllabus

Atmospheric boundary layer

Pollution dispersion in the atmosphere

Rivers, estuaries and tidal flows

Sediment transport

Building and urban aerodynamics

Free-surface and shallow-water flows.

CFD FOR FLUID-STRUCTURE INTERACTION

Aim

To introduce Fluid Structure Interaction (FSI) models and associated computational challenges. To provide examples of FSI problems arising in engineering applications.

Syllabus
Introduction to FSI
Physical models (Classical models, Distinction between linear and nonlinear models), Time-linearised models, Nonlinear dynamical models, Reduced-order models
Computational challenges of FSI modelling
Examples of applications.