# CARLETON UNIVERSITY

## Department of Mechanical & Aerospace Engineering

### MECH5304 – Computational Fluid Dynamics (CFD): Course Outline

**Instructor - ALI BENMEDDOUR**

**Background**

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. It combines aspects of science, applied mathematics and engineering to predict the behaviour of fluid flows through the numerical solution of the equations of motion that govern the flow.

CFD has become a common tool in industrial and research environments and its use has increased rapidly over the last two decades. Numerous sophisticated CFD software packages are available to the users and it is important that engineers and researchers who work in CFD have a solid foundation/knowledge in fluid dynamics, numerical analysis and other CFD related fields/tools such as computer aided design (CAD), grid generation, flow visualization, programming languages and high performance computing (HPC). Without good knowledge, a CFD user can easily produce colourful but meaningless results. Therefore, the main objective of this course is to present the fundamentals of CFD so that students become knowledgeable users of CFD whether to develop/write their own CFD code or to use an existing package.

This course is designed to provide an introduction to the theory and practice of CFD taking into consideration that not all students have the same level of knowledge. It will include CFD terminology and concepts, numerical methods & solution techniques, and simple CFD code development methodology. It will also introduce students to the use of commercial CFD packages (e.g. Ansys CFX, which is available at Carleton).

Again, the main objective of the course is to make you “informed users of CFD” not experts in the field (would take much more time than course allows). Other objectives of the course are:

- Provide an introduction to the theory and practice of CFD
  - CFD terminology and concepts
  - Numerical methods and solution techniques
  - Simple CFD code development
- Introduce students to the use of commercial CFD packages (Ansys CFX)
- Make you “informed CFD users”

**General Outline of the Course –** weekly lesson plan may differ from this general outline

| WEEK | TOPICS |
|---|---|
| 1 | Course overview and introduction to CFD (motivation, history & methodology) |
| 2 | Governing equations of fluid flows/dynamics |
| 3 | Turbulence, Closure problem and Turbulence modelling (eddy viscosity models) |
| 4 | Mathematical classification of partial differential equations (PDEs) |
| 5 | CAD models and grid/mesh generation methods – discretization of geometry and computational domain |
| 6 | Spatial discretization of governing equations – Finite difference method (FDM) |
| 7 | Spatial discretization of governing equations – Finite volume method (FVM) |
| 8 | Temporal discretization – explicit and implicit time integration schemes; initial and boundary conditions |
| 9 | Solution of linear algebraic equation systems – direct and iterative methods |
| 10 | Verification and validation (V&V) of numerical models' implementation – discretization errors; consistency, accuracy and stability of numerical schemes |
| 11 | Solution of coupled and non-linear systems of equations – application to Navier-Stokes equations |
| 12 | Advanced/Emerging topics in CFD |

- Lectures (3hrs/wk) will be a combination of slides and hand-written notes on the chalk board.
- Course webpage – cuLearn MECH5304W Winter 2017
- Course material (slides, handouts and lecture notes) will be posted on cuLearn. Please refer to cuLearn in order to keep up-to-date with the course material that is posted there.

**Assignments**

Three assignments will be assigned throughout the term. Each assignment is worth 15% of the final mark for a total of 45%. The first assignment will involve a simple flow simulation in ANSYS CFX to help you get familiar with the software for your course project. The other two assignments will require you to write a program in your preferred programming language.

All assignments are to be submitted in hard-copy at the beginning (first 15 min) of the class in which they are due. Late submissions will receive a 10% penalty and emailed copies are not accepted. All papers must include appropriate references and citations following an appropriate style.

**Project**

A course project is assigned instead of a final exam. The project will be worth 55% of your final grade (more info on cuLearn).

- See Project Description on cuLearn for details
- Select one topic from the following options
  - Internal flows: convergent-divergent nozzles, channel flows, diffusers, etc.
  - External flows: around simplified bodies (e.g., spheres, cylinders, airfoil, blunt bodies, bluff bodies etc.)
  - Your thesis topic ... see me for approval
- Single-phase fluid
- Non-hypersonic (Mach < 5)
- Ansys CFX (or equivalent ... see me for approval)

**Recommended literature**

- Anderson, J. D., Computational Fluid Dynamics - The Basics with Applications, McGraw-Hill
- Hoffmann, K. A. and Chiang, S. T., Computational Fluid Dynamics for Engineers, Engineering Education System
- Tannehill, J.C.,Anderson, D.A., and Pletcher, R.H., Computational Fluid Mechanics and Heat Transfer, Taylor & Francis
- Fletcher C.A.J., Computational Techniques for Fluid Dynamics, Vol. 1: Fundamental and General Techniques, Springer-Verlag
- Hirsh C., Numerical Computation of Internal and External Flows, Vol. 1: Fundamentals of Numerical Discretization, John Wiley & Sons
- Versteeg, H. K. and Malalasekera, W., An Introduction to Computational Fluid Dynamics - The Finite Volume Approach, Longman
- Blazek, J., Computational Fluid Dynamics: Principals and Applications, Butterworth-Heinemann
- Wilcox, D. C., Turbulence Modeling for CFD, DCW Industries

**Other resources**

- http://www.grc.nasa.gov/WWW/wind/valid/resources.html
- www.cfd-online.com
- www.nas.nasa.gov
- www.cfdreview.com
- www.eng.warwick.ac.uk/staff/cts/cfdbook
- www.erc.msstate.edu/publications/book

**My Background**

- Ph. D. in Mechanical Engineering (CFD) – École Polytechnique of Montreal
- Currently work for the NRC-Aerospace Aerodynamics Laboratory – Senior Research Officer

**Note:** I work full-time for the NRC so I do not have an office on campus and I am not on campus on days when I'm not teaching. If your question can't be answered by email please make arrangements to meet with me before class.

- **Email:** ali.benmeddour@carleton.ca (best way to reach me)

**Accommodations**

Students with disabilities requiring academic accommodations in this course are encouraged to contact the Paul Menton Centre for Students with Disabilities (https://carleton.ca/pmc/about-us/contact/) to complete the necessary forms. After registering with the Centre, make an appointment to meet me in order to discuss your needs at least two weeks before the first assignment is due. This will allow for sufficient time to process your request.

**Plagiarism**

If I see two identical papers, I will treat them as one submission and leave it to the students to decide how that one grade will be split between them. If I see anything copied word for word from a publication or website, that section of the assignment or project receives a zero. https://carleton.ca/slals/credit-esl/plagiarism-cheating/