The University of British Columbia

MECH 479/587 - Computational Fluid Dynamics FAQs

2022-W1 (Sept.-Dec. 2022)

1) **Who are the instructors for this course?**
The primary instructor for the course for the 2020W session is Dr. Rajeev Jaiman. In addition, there are two TAs for this course - Nihar Bhardwaj and Xiaoyu Mao.
2) **Class Timings**
The course for the 2021W session consists of weekly lectures from *9.30AM-11.00AM (Pacific Time)* on Tuesdays and Thursdays. There will lab sessions on Mondays at 1PM.
3) **What is CFD and why mechanical engineers should study?**
A precise definition of CFD depends whom you ask. CFD is a process of computing fluid flow and heat transfer using partial differential equations (e.g., Navier-Stokes/Euler, Lattice-Boltzmann equations) with the goal of dynamical predictions and physical insight. CFD is a multidisciplinary subject and lies at the intersection of fluid mechanics, numerical analysis and computer science.
CFD is used by many different people as a tool for mechanical, aerospace, marine/offshore, biomedical, chemical, civil and among others. Beyond engineering applications, CFD gives a general problem solving and algorithmic skills for your professional career. In this course, you will learn to use simple CFD codes and commercial tools to solve academic and industry-level engineering problems.
4) **What should I know before hand for learning CFD effectively?**
This course will assume that students have gone through some undergraduate-level coursework on fluid mechanics and basic numerical analysis. Having said that, a review of fluid mechanics equations and the numerical solution of ordinary differential equations will be provided in the first two lectures.
5) **What will I learn after this course?**
You will learn how to solve the incompressible Navier-Stokes and Euler equations for engineering problems using both customized academic codes and a commercial code. You will also get exposure to various numerical discretization techniques and fundamental concepts of CFD.
6) **What will be teaching strategy or learning activities during the course?**
CFD relies on mathematical and computing-based experiences (i.e., coding and testing). You get better as you practice and take an active role in the learning process. For an effective and higher-level learning, the overall

goal is to motivate you to actively prepare such that simple materials and explanation are not required during the live lecture.

7) **How will I be graded/assessed in this course?**
   Evaluation will comprise of Problem sets = 20%, Mid-term = 20%, Projects = 30%, Final Exam = 30%.

   The scope of projects for MECH479 and MECH587 will be different. While MECH479 students will learn more applied CFD using a commercial software and by modifying MATLAB codes, there will be the projects based on advanced C/C++ programming for MECH587 students.
   The remaining assessment strategy for this course will be identical for both MECH479 MECH587 students. Based on the lectures, students will undertake projects to have hands-on experience of theory and applied aspects of computational fluid dynamics. Problem sets will allow students to practice and apply newly learned lecture materials. They will include some derivations or calculations. Since students have the ability to work with others and seek help during office hours, the emphasis on the grading of homeworks will be on achieving the correct solutions for the given problems. While the mid-term will include short conceptual questions, the final exam will be comprehensive and will assess your mastery of the covered course materials.
8) **How do I contact the instructor/TAs for this course? What are the office hours?**
   The instructor/TAs can be reached via email or by sending a direct message on Canvas (please allow up to 24 hours for a response). Instructors/TAs will also be available at specified office hours and can be contacted at their respective Zoom meeting rooms - please drop a note in advance to set up an appointment. All contact details can be accessed at the course "Syllabus" tab on Canvas.
9) **How can I know the specific topics to be covered in the course?**
   Course outline can be viewed in the "Syllabus" tab on the course Canvas page. Further insights into the topics covered in individual modules can be had on the "MECH_479_Module_Layout.pdf" document on the Canvas landing page for the course.
10) **Do I need any specific equipment (hardware/software) to be able to participate in this course?**
   Some of the course assignments will have to be handwritten/typed-out. If you choose to write it by hand, you should be able to scan/photograph it and upload to Canvas. The course requires certain activities to be performed on specific software (MATLAB, ANSYS). These can be installed locally on your computer for the duration of the course (for learning purposes only). If installing on your local computer is not feasible, you can access machines in the RH 123 lab, the ICICS X060 (PACE) lab, and the NAME lab remotely through LabStats.
11) **Are there any textbooks that I need to refer to for this course?**
   All the course material, including lecture handouts and sample codes will be shared via the Course Canvas page. Additional textbooks are not necessary for the purpose of the course, but the following are the recommended readings for those interested

   a) Anderson, D.A., Tannehill, J.C., Pletcher, R.H. *Computational Fluid Mechanics and Heat Transfer* McGraw Hill.
   b) Charles Hirsch, Numerical *Computation of Internal and External Flows. Volume I and II* Wiley Series in Numerical Methods, 1988.
   c) Joel Ferziger and Milovan Peric, *Computational Methods for Fluid Dy-*

namics Springer Verlag.

d) Wesseling *Principles of Computational Fluid Dynamics* Springer Verlag

e) Chapra, S.C., Canale, R.P. *Numerical Methods for Engineers* 3rd Ed., McGraw-Hill, 1998.

f) Smith, G.D. *Numerical Solution of Partial Differential Equations: Finite Difference Method* Oxford University Press.

12) **Submission of homework/codes**
All submissions will be through Canvas. Submissions via email are not encouraged except for exceptional circumstances. Reports should be submitted in PDF format (not image-based). For code submissions, please combine and zip into a single submission if multiple subroutines are present.

13) **Discussions/collaborations with other student participants on submissions**
Homeworks and timed quizzes must be independently solved without any discussion.
For projects, students are encouraged to discuss with other student participants of the course. However, the work itself, including results, reports and observations should be the independent output of each participant. UBC policies on Academic Integrity and Plagiarism will apply to all submissions.

14) **What are general tips to students from previous year pass on about the course**
(a) Start the home-works and projects early and manage your time, (b) Attend classes and lab sessions regularly and read materials in advance, (c) consider forming a study group.

15) **Why class notes and handouts don't have textbook like details?**
The lecture slides and handouts are meant to provide a framework for you to organize your notes around, hence they don't contain full details of the lectures.

16) **Why there is need for learning the mathematical derivations and discretization procedures in CFD (as opposed to just applications of codes and software tools)?**
CFD sits on the foundations of calculus and numerical mathematics. For a proper usage of any CFD codes and software, it's important the basic CFD theory and underlying numerical properties (e.g., accuracy, stability, convergence) are being understood.
CFD education needs a proper balance of underlying mathematics and applications. One cannot use CFD software without understanding the mathematics and intuition behinds the partial differential equations (PDEs) of fluid flow and hear transfer, and different numerical methods and algorithms.

17) **Why we need to learn so much theoretical details about solving elliptic, parabolic and hyperbolic PDEs?**
Fluid flow equations possess the character of all three fundamental types of PDEs and it's necessary to understand the discretization process and the numerical properties of solution methods for each PDE type.

18) **Why we need to learn different discretization methods such as finite difference, finite volume and finite element?**
There is no universal best technique which can be successfully applied to all physical conditions and geometric requirements (e.g., CAD and meshing). Hence it's required to learn different methods (tools) to understand their strengths and weaknesses for a particular "job" while considering "cost" and other practical aspects.