ME 448/548 Applied Computational Fluid Dynamics Winter 2022

Mechanical and Materials Engineering Department
Portland State University

Description

Applied Computational Fluid Dynamics (CFD) is an introduction to the use of commercial CFD codes to analyze flow and heat transfer in problems of practical engineering interest. The course begins with a study of simple finite-difference and finite volume models of one-dimensional partial differential equations. These equations contain important features of more complex CFD models. The SIMPLE algorithm for solution to incompressible flow problems is described.

The bulk of the course involves using a commercial CFD code, StarCCM+. Models of simple flow geometries are developed and studied to learn how to properly apply boundary conditions, set numerical parameters that control the analysis, and to visualize model results. Homework exercises introduce features of the StarCCM+ to simulate more complex flow problems, including the use of turbulence models. StarCCM+ is also used to simulate heat conduction, convection and conjugate heat transfer. In addition to learning how to correctly use the CFD tool, homework and in-class exercises are designed to aid in developing physical insight and intuition about complex flow and heat transfer.

In the last half of the course, students work on a major project to simulate a flow problem of their choosing. In the process, students create small tutorials that explain one or more features of the model used in their project. Creating the tutorial contributes to the final project report and helps students focus on implementing model details correctly. By exchanging these tutorials, students gain exposure to additional modeling features they may not be using in their own project.

At the end of the course. successful students will understand the process of developing a geometrical model of the flow, applying appropriate boundary conditions, specifying solution parameters, and visualizing the results. They will also have an appreciation for the factors limiting the accuracy of CFD solutions.

Time and Place

Remote meeting via Zoom on Mondays and Wednesdays, 6:40 – 8:30 PM.

Instructor

Gerald Recktenwald, Associate Professor, Mechanical and Materials Engineering Department
Engineering Building, Room 402C, 725-4296, `gerry@pdx.edu`
Websites for the course:
`canvas.pdx.edu` and `www.me.pdx.edu/~gerry/class/ME448`
Drop-in Zoom Office Hours: To be determined

Reference Textbooks

The following books cover the finite-volume method used in StarCCM+. Perić developed codes that were very influential in the design of StarCCM+, and for a time he worked for CD-Adapco, the company that developed StarCCM+, and was acquired by Siemens. These books are not required, but may be helpful for self-study. Additional references are listed at the end of the syllabus.

1. Joel H. Ferziger, Milovan Perić and Robert L. Street, *Computational Methods for Fluid Dynamics*, fourth ed., 2019, Springer-Verlag, New York, ISBN 978-3319996912.
2. Jiyuan Tu, Guan Heng Yeoh, and Chaoqun Liu, *Computational Methods for Fluid Dynamics: a Practical Approach*, 2008, Butterworth-Heinemann, Burlington, MA, ISBN 978-0750685634

Lecture Notes

Lecture notes are made available, either has physical handouts or via links on the public web site, `www.me.pdx.edu/~gerry/class/ME448`.

On-line documentation for StarCCM+ is used extensively, and is available on the computers in EB 420.

Learning Objectives

The Learning Objectives are what I expect that you will be able to do at the end of the course.

- Be able to derive the BTCS and FTCS formulation for the one-dimensional heat equation, and perform meaningful computational experiments with sample codes that implement these methods.
- Be able to interpret exact and numerical solutions to the one-dimensional convection-diffusion equation. Be able to explain the relative merits of numerical solutions to this equation obtained with the upwind difference scheme and the central difference scheme.
- Describe the qualitative differences between a physical flow that is turbulent, and the flow field predicted by a numerical solution to the Reynolds Averaged Navier-Stokes equations.
- Starting with a rough sketch of a flow problem, identify all physical data necessary to set up and the solve the velocity, pressure, and temperature fields using a CFD package.
- Use StarCCM+ to solve three-dimensional laminar, and turbulent flow problems.
- Identify whether and when a run of StarCCM+ has converged, and make adjustments to solution parameters necessary to obtain a converged solution.
- Describe the key features of a three-dimensional flow field represented the visualization tools in StarCCM+.

Evaluation

Learning of the course material will be evaluated with homework, on-line quizzes, a tutorial mini-project, and an independent project. Individual learning assessments are used to compute a cumulative score with the following weights

25%	Homework
15%	On-line quizzes
25%	Project Tutorial
35%	Project

Project

Your individual project involves a detailed simulation of a particular flow problem with StarCCM+. You may chose from a list of projects that have already been identified, or, with instructor approval, you may create your own project. Each student will submit a report and, as time allows, will give a brief presentations on their project to the entire class.

Part of your course grade will be based on a tutorial that you develop to demonstrate a feature of the CFD code StarCCM+. Ideally this feature is something that is necessary for you to complete the term project. The goal is to encourage and reward you for exploring details of the CFD code that will not be covered during lecture. You will share a draft of your tutorial with one other student, who will learn from your tutorial and give you feedback for improvement. After that feedback cycle, you will have time to improve your tutorial before submitting it for grading.

Supplemental References

The following textbooks are useful references for CFD.

Anderson, Dale A., Tannehill, John C., and Pletcher, Richard H. *Computational Fluid Mechanics and Heat Transfer*, 1984, Hemisphere, Washington D.C.

Date, A.N., *Introduction to Computational Fluid Dynamics*, 2005, Cambridge University Press, New York.

Elman, H.C., Silvester, D.J., and Wathen, A.J., *Finite Elements and Fast Iterative Solvers: with applications in incompressible fluid dynamics*, Oxford University Press, Oxford, 2005.

Hirsch, C., *Numerical Computation of Internal and External Flows, Volume 1: The Fundamentals of Computational Fluid Dynamics*, 2007, Butterworth-Heinemann, 2nd ed.

Lomax, H., Pulliam T.H. and Zingg, D.W., *Fundamentals of Computational Fluid Dynamics*, 2001, Springer-Verlag, New York.

Patankar, Suhas V., *Numerical Heat Transfer and Fluid Flow*, 1980, Hemisphere, Washington D.C.

The following textbooks provide useful background for the study of fluid mechanics.

Acheson, D.J., *Elementary Fluid Dynamics*, 1990, Clarendon Press, Oxford.

Panton, R.L., *Incompressible Flow*, third edition, 2005, Wiley, New York.

Schey, H.M., *Div, Grad, Curl, and All That*, 1973, Norton, New York.

Tritton, D.J., *Physical Fluid Dynamics*, second edition, 1988, Clarendon Press, Oxford.