Computational Fluid Dynamics (CFD) on an Unstructured Mesh

Short Course

Indian Institute of Technology, Kharagpur, India

Course Objective: The objective of this short course is to teach graduate students, who have already had some exposure to the finite-volume method and developing solvers for the Navier-Stokes Equations, how to develop such a code for an unstructured mesh. The course will be very hands-on and walk the students through all steps involved in the process, ultimately culminating into a solver for solving a benchmark problem in CFD: driven cavity problem in a rhombus.

Course Outcome: Upon completion, students will be confidently able to develop solvers for solving canonical PDEs on an unstructured mesh using the finite-volume method. In particular, they will be able to develop solvers for solving the Navier-Stokes equations on an unstructured mesh.

Instructor

Dr. Sandip Mazumder
Professor, Department of Mechanical and Aerospace Engineering
The Ohio State University

Reading Material

- *Numerical Methods for Partial Differential Equations: Finite Difference and Finite-Volume Methods*, Chapter 7, S. Mazumder, Academic Press, 2016.

Prerequisites

Graduate-level numerical methods course. Some experience in developing CFD codes will be very helpful but not required. The course will require students to develop extensive programs, and students must be very familiar with at least one programming language and software for creating graphs (line plots, contours etc.).

Assessment

Students will be assessed based on their performance on one project, which will have several sub-parts with partial credit.

Syllabus and Course Schedule (Tentative only)

Lecture #	Topics Covered
1	Overview of the finite-volume method for an orthogonal structured Cartesian mesh and its application to an advection-diffusion equation. Treatment of Dirichlet, Neumann and Robin boundary conditions Overview of selected linear algebraic equation solvers
2	Overview of the Navier-Stokes Equations Overview of the SIMPLE algorithm for a co-located Cartesian mesh
3	Finite volume method for an unstructured mesh: processing and storage of geometry and connectivity information Finite volume integration of the generalized advection-diffusion equation on an unstructured mesh
4	Treatment of the viscous terms on an unstructured mesh in 2D and 3D The SIMPLE algorithm for an unstructured mesh
5	Boundary condition treatment Code development and demonstration for the driven cavity problem in a rhombus

The course is designed to be taught over a 5-day period, with one lecture per day of approximate duration 2 hours and 15 minutes. Each lecture will be split into two halves with a 10-minute break between the two halves.

Indian Institute of Technology, Kharagpur, India

Computational Fluid Dynamics (CFD) on an Unstructured Mesh

Short Term Course (31 Mar – 04 Apr, 2025)

Registration Fee

Faculty from academia	**INR 2000/- + GST**
Industry professionals (including research labs)	**INR 5000/- + GST**
Students	**INR 500/- + GST**

Author Bio

Dr. Sandip Mazumder joined the Ohio State University (OSU) in March of 2004. He is currently the Associate Chair of Administration of MAE. Prior to joining OSU, he was employed at CFD Research Corporation in Huntsville, AL for 7 years. He is one of the architects and early developers of the commercial code CFD-ACE+™. His research is computational in nature, and spans three main areas: (1) computational fluid dynamics and heat transfer emphasizing on chemical reactions with applications in combustion, catalytic conversion, fuel cells, batteries and chemical vapor deposition, (2) thermal radiation and its applications, and (3) non-equilibrium transport phenomena as occurring in nanoscale systems. Dr. Mazumder is the author of two graduate-level textbooks, 60+ journal papers, and 60+ peer-reviewed conference publications. He is the recipient of the McCarthy award for teaching and the Lumley award for research from the OSU College of Engineering among other awards and is a Fellow of the American Society of Mechanical Engineers (ASME) since 2011. He is also the recipient of the 2024 Fulbright U.S. Scholar award (specifically the Fulbright-Kalam Climate Fellowship).