Title: Computational Thermal Engineering

Brief:

- *Understand the governing equations, discretizations, and their implementations*
- *Study of the conduction and convection algorithms*
- *Gain skill in Octave and understand C++ to code in OpenFOAM*
- *Simulate and validate real world problems with practical learning examples in OpenFOAM*
- *Group and individual Projects using Octave and OpenFOAM*

Instructor: Mr. Kumaresh Selvakumar

- Google scholar: https://scholar.google.com/citations?user=AghOa44AAAAJ&hl=en

Grading distribution:

- All exercises (equal weight): **20%**
- Project-I: **10%**, Project-II: **10%**, Project-III: **20%**, Project-IV: **40%**

Duration: 15 weeks, 1 session (3 hrs)/week, Total 45 hours

References:

1) Patankar, S. (1980). Numerical Heat Transfer and Fluid Flow (1st ed.). CRC Press.

2) Versteeg, H. K. and Malalasekera, W. (1995) An Introduction to Computational Fluid Dynamics the Finite Volume Method. Longman Scientific and Technical.

Course curriculum

Weeks	Topic(s)	Brief
Week 1	**Ch. 1 Introduction – Computational Thermal Engineering** * CFD with OpenFOAM an overview * Setting up the system (Exercise 1)	1. Discussing syllabus and projects related to Computational Thermal Engineering subject 2. A brief introduction showing the capabilities of OpenFOAM. 3. Configuring the system to use OpenFOAM and Octave to develop CFD algorithms.
Week 2	**Ch. 2 Mathematical description of physical phenomena**	1. Introduction to conservation laws. 2. Governing equations, derivations, and their significance.
Week 3	**Ch. 3 Discretization methods** * Introduction to programming in Octave * Exercises 2 and 3 (Octave) * Introduction to C++ for OpenFOAM	1. Numerical discretization (FDM and FVM) 2. Presenting Taylor series expansion 3. Stability analysis in numerical methods 4. Sample test case OpenFOAM (steady flow)
Week 4	* Introduction to C++ for OpenFOAM (contd.) * Exercise 4 (Octave)	1. Stability analysis in numerical methods (contd.) 2. Rate of convergence 3. Truncation error analysis 4. Sample test case OpenFOAM (steady flow)
Week 5	**Ch. 4 Heat Conduction** * FDM using diffusion equation * Introduction to C++ for OpenFOAM (contd.) * Exercise 5 (Octave)	1. Introducing finite difference method using heat equation. 2. A template code will be provided to help students fill in required parts to finish this assignment. 3. Solving Diffusion in Octave (steady flow)
Week 6	* FVM using diffusion equation	1. Introducing finite volume method using heat equation.

	* Introduction to C++ for OpenFOAM (contd.) * Exercise 6 (Octave, OpenFOAM) * **Project – I** (in C++)	2. A template code will be provided to help students fill in required parts to finish this assignment. 3. Solving Diffusion in Octave and OpenFOAM (steady flow)
Week 7	**Ch. 5 Convection and Diffusion** * FDM using convection equation * Introduction to C++ for OpenFOAM (contd.) * Exercise 7 (Octave)	1. Introducing finite difference method using advection equation. 2. A template code will be provided to help students fill in required parts to finish this assignment. 3. Solving convection in Octave (steady flow)
Week 8	* FVM using advection equation * Introduction to C++ for OpenFOAM (contd.) * Exercise 8 (Octave, OpenFOAM)	1. Introducing finite volume method using advection equation. 2. A template code will be provided to help students fill in required parts to finish this assignment. 3. Solving convection in Octave and OpenFOAM (steady flow)
Week 9	* FVM convection-diffusion equation * Introduction to C++ for OpenFOAM (contd.) * Exercise 9 (Octave, OpenFOAM) * **Project – II** (in C++)	1. Introducing students to OpenFOAM discretization 2. A template code will be provided to help students fill in required parts to finish this assignment. 3. Solving convection-diffusion in Octave and OpenFOAM (unsteady flow)
Week 10	* Programming in OpenFOAM * Explanation of **Project – III** (Group project)	Working with OpenFOAM.
Week 11	**Ch. 6 Calculation of the flow field** * Numerical algorithms to solve pressure-velocity equations- I, II * Review of **Project – III** (Group project)	1. Unsteady flow with demonstration on boundary conditions 2. Working with OpenFOAM
Week 12	* Staggered and collocated grid explanation * SIMPLE and SIMPLER algorithm * Review of **Project – III** (Group project)	1. Unsteady flow with demonstration on boundary conditions 2. Working with OpenFOAM
Week 13	* Explanation of **Final Project – IV** (Group project)	OpenFOAM simulations
Week 14	* Review of CFD * Review of **Final Project – IV** (Group-project) * Presentations of **Final Project – IV** (Group project)	1. OpenFOAM simulations and project reviews 2. Course review and project queries 3. Final project presentations
Week 15	* Review of CFD * Review of **Final Project – IV** (Group-project) * Presentations of **Final Project – IV** (Group project)	1. OpenFOAM simulations and project reviews 2. Course review and project queries 3. Final project presentations