Theory and Applications of Computational Fluid Dynamics

11th Feb – 15th February, 2019

Course Overview

The course on Theory and Applications of Computational Fluid Dynamics (CFD) was targeted towards faculty and students with affiliations that are under the TEQIP-Phase 3 program. The course introduced to the participants the basic concepts of CFD, that includes introduction to the governing equations of incompressible flow and their discretization. Furthermore, application of such methods to simulate flow of engineering importance like turbulence, combustion and multiphase flow was demonstrated. Each day, the participants were introduced to the theory associated with such flows, followed by laboratory sessions such that the participants get hands-on experience in using a commercial CFD software to solve engineering problems.

Course Curriculum

The table below gives the detailed schedule of each day of the course

Theory & Applications of Computational Fluid Dynamics

Day wise schedule

Day	**Teaching Slot 9:30 to 11:00**		**Teaching Slot 11:15-12:45**		**Teaching Slot 2:00-3:30**		**Teaching Slot 3:45-5:15**
Day 1 11 Feb., 2019 (Monday)	Introduction to CFD ***(C-LH3)***	**Tea Break 11am-11:15am**	Finite Volume Method ***(C-LH3)***	**Lunch 12:45PM-2PM**	CFD Lab – Heat Conduction ***(CAE-Lab)***	**Tea Break 3:30-3:45**	CFD Lab – Heat Conduction ***(CAE-Lab)***
Day 2 12 Feb., 2019 (Tuesday)	Solution Algorithms-I ***(C-LH8)***		Solution Algorithms-II ***(A-117)***		CFD Lab – Laminar Flows ***(CAE-Lab)***		CFD Lab – Laminar Flows ***(CAE-Lab)***
Day 3 13 Feb., 2019 (Wednesday)	Theory of Turbulence ***(A-620)***		Turbulence Modelling ***(A-620)***		CFD Lab – Turbulent Flow ***(CAE-Lab)***		CFD Lab – Turbulent Flow ***(CAE-Lab)***
Day 4 14 Feb., 2019 (Thursday)	Introduction to Combustion ***(C-LH3)***		Combustion Modelling ***(C-LH3)***		CFD Lab – Combustion Modeling ***(CAE-Lab)***		CFD Lab – Combustion Modeling ***(CAE-Lab)***
Day 5 15 Feb., 2019 (Friday)	Introduction to Multiphase Flow ***(A-620)***		Multiphase Modelling ***(A-620)***		CFD Lab – Multiphase Modeling ***(CAE-Lab)***		CFD Lab – Multiphase Modeling ***(CAE-Lab)***

- **Please make workshop teaching slots of 1.5hrs each.**

Speakers

Prof SP Vanka

Prof Vanka is Research Professor and Professor Emeritus in the Department of Mechanical Science and Engineering of University of Illinois Urbana-Champagne and VARJRA and Adjunct Faculty in the Department of Mechanical and Aerospace Engineering of IIT Hyderabad. Prof Vanka, is a pioneer and long-time researcher in Computational Fluid Dynamics, including multiphase flows, parallel computing, algorithm acceleration, and applications to a wide variety of industrial and basic science flows. Professor Vanka graduated from the Spalding school of CFD at Imperial College, in 1975, which pioneered the basic algorithms currently used in many CFD software including the OpenFOAM software proposed in this effort. Over the past 43 years, Prof. Vanka has published more than 160 technical papers on a variety of areas in CFD, and written more than 20 different academic CFD software based on different methodologies and different computing platforms. His extensive experience in CFD and parallel computing algorithms will significantly benefit the proposed research in developing a GPU-CPU based algorithm for multiphase flows

Prof Raja Banerjee

Dr. Raja Banerjee is a Professor in the Department of Mechanical and Aerospace Engineering of IIT Hyderabad. Before joining IIT Hyderabad, Prof Banerjee worked as a Senior Research Engineer in Mark IV Automotive Inc., USA for close to 8 years. His primary research interest is in multiphase flow with particular focus on liquid spray and atomization, turbulence and parallel computing for CFD applications. He has co-authored several papers on these topics which are reported in leading journals and conference proceedings.

Prof Vinod Janardhanan

Dr Vinod Janardhanan is a Professor in the Department of Chemical Engineering of IIT Hyderabad. Dr Vinod's research interests are primarily focused on electrochemical devices and heterogeneous catalysis. Specifically, their fuel cell research is focused on the development of models for various physical and chemical processes that occur in Solid Oxide Fuel Cells and High Temperature Polymer Electrolyte Membrane Fuel Cells. Fuel cells based on paper support and laminar flow fuel cells for application in micro-nano systems that consume milli-watts of power are other research interests of their group.

Dr Narasimha Mangadoddy

Dr Narasimha Mangadoddy is an Associate Professor and Head of the Department of Chemical Engineering of IIT Hyderabad. The focus of their research has been to develop and validate the multi-phase CFD models for various mineral processing units like dense medium cyclones (DMC), hydrocyclones (HC), feed slurry distributors and flotation devices. In particular, extensive results have been obtained on the detailed multi-phase flow in DMC/HC devices in terms of air-core resolution, mean and turbulence flow field analysis, turbulent dispersion analysis w.r.to particle classification and understanding the classification mechanism.

Day One

Theory: The basic governing equation in fluid mechanics and heat transfer is introduced to the students. The conversation principles were explained. The candidates were introduced to the basic discretization schemes of the governing equations.

Initially the finite difference algorithm was explained followed by the order of accuracy of the discretized equation. The above scheme was explained using the heat conduction equation.

Lab: Steady state 2D heat conduction problem was demonstrated in this session. A heated coil is placed inside a rectangular chamber using the heat transfer model in ANSYS FLUENT. The coil diameter is L/4 placed at location (L/2, L/2). The temperature of the coil is at 310 K. The walls are maintained at a temperature of 298K . The temperature gradient between the coil and the walls is the driving force for the heat transfer

Day Two

Theory: Finite volume method is explained to the students. Approach to solve the various parts of a typical convection diffusion equation is explained. Implementation of the boundary conditions is demonstrated. Various iterative techniques of in terms of line solvers and point solvers was explained.

Lab: In the second day of lab session, a tutorial for solving a laminar flow problems using pressure based solvers in ANSYS Fluent was conducted. The students were instructed for solving a case setup for lid driven cavity and flow over a flat plate. These are very classical problem which is suitable for checking various algorithmic technique for pressure based solver like SIMPLE, PISO, SIMPLEC and COUPLED. Students were also familiarized how to do basic post-processing techniques like creating streamlines and shear stress calculation

Day Three

Theory: Theory of turbulence was explained to the students. This includes the basic flow features of turbulence flow, length and times scales in turbulent flow, production, dissipation and cascading of turbulent kinetic energy. Turbulence closure problem was explained to the students while deriving the RANS equation. Boussinesq approximation based one-equation and two-equation turbulence model for RANS equation was explained.

Lab: Turbulent heat transfer through a coil in a closed container using convective flow of cool air through the inlet was simulated. A heater coil is placed inside a rectangular chamber filled with air and turbulent air inlet is provided at the bottom section is studied using the $k - \varepsilon$ turbulence model in ANSYS FLUENT

Day Four

Theory: Theory of reacting flow was conveyed to the students. This includes conservation of element mass fraction, behaviour of the thermodynamic properties as a function of temperature, chemical thermochemistry and chemical kinetic. As an application to reacting flows, laminar non-premixed combustion was explained.

Lab: In this lab session, the study on mixing of chemical species and combustion of gaseous fuels was studied. A cylindrical combustor burning methane (CH4) in air is studied using the eddy dissipation model in ANSYS FLUENT.

Day Five

Theory: General concepts of multiphase flow was explained followed by the challenges and general definition in multiphase flow. Modelling of bubbles, drops and particles were explained. Key modelling parameters like flow patterns, slip-velocity/pressure drop and phase hold-up was explained.

Lab: Two case studies were: (a) Sloshing of liquid in a rectangular enclosure with two fluids having difference density and viscosity. The sloshing motion was simulated using VOF model (b) tracking of particle laden flow in a U bend tube using Eulerian-Lagrangian approach.