# Basics of Computational Fluid Dynamics using OpenFOAM

| | |
|---|---|
| **Course Title** | Basics of Computational Fluid Dynamics using OpenFOAM |
| **Course Category** | Pedagogy / Specialized Skills |
| **Relevant Discipline(s)** | Mechanical Engineering /Chemical Engineering / Aerospace Engineering/ Energy Science Engineering |
| **Duration of course in equivalent integer no. of days (min 3 days, 1 day = 6 hrs of lectures/hands on sessions)** | 3 days<br>6 hrs of lectures/hands on session on each day |
| **Proposed dates** | 25-27 February 2021 |

| **Brief Course Description and Course Content** |
|---|
| Open source Field Operation And Manipulation, or OpenFOAM, as popularly known, is an Open source Computational Fluid Dynamics software. It's a finite volume based CFD software where the toolbox written in C++ allows for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of computational fluid dynamics or continuum mechanics problems. OpenFOAM is an open source alternative to commercial software, such as Fluent. As OpenFOAM is open source, it is widely used in the industry. In India, OpenFOAM is used in organisations, such as ISRO, BARC, and DRDO labs, apart from the industry.<br><br>The objective of the course is to train students/college teachers/design professionals to solve computational fluid dynamics (CFD) problems while understanding the intricacies involved in such simulations through a robust at the same time easy to use CFD tool box.<br>At the end of this workshop, participants will:<br>● learn how to set up, view the files and run a case in OpenFOAM<br>● become familiar with pre-processing tools and will be able to generate mesh, set the boundary & initial conditions, learn simulation control etc.<br>● learn how to plot and visualise the results in ParaView.<br>● learn how to use a turbulence model in a simulation and to compare the results from a simulation using three different turbulence models.<br>● also learn how to establish grid independence.<br><br>The workshop will be conducted using a mix of pre-recorded spoken tutorial videos with side by side learning and live lectures. A practice problem/assignment will also be provided after every topic followed by discussions on them to get a good understanding of each topic. |

All participants will get the Spoken Tutorials on OpenFOAM, copies of our slides, video recording of all lectures, and about 100 case studies solved using OpenFOAM. Using these, all who are interested in conducting OpenFOAM workshops by themselves can do so. They will also get exposed to the collaborative content creation activity of the FOSSEE Project.

| S. No. | Name of the Instructor | Department | Email |
|---|---|---|---|
| 1. | Prof. Janani Srree Murallidharan | Faculty, Mechanical Engineering | js.murallidharan@iitb.ac.in |
| 2. | Prof. Manaswita Bose | Faculty, Energy Science Engineering | manaswita.bose@iitb.ac.in |

FOSSEE team members will help conduct the hands-on sessions.