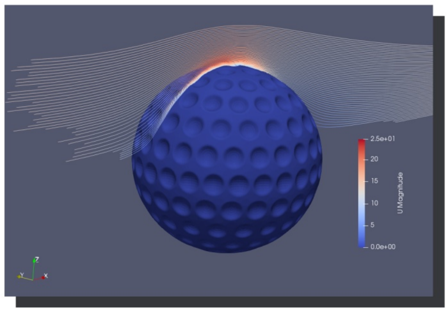
3 DAY ONLINE

Date: 11-13 January, 2023
Time: 10:00 a.m. to 5:00 p.m

Workshop on the Basics of CFD & OpenFOAM

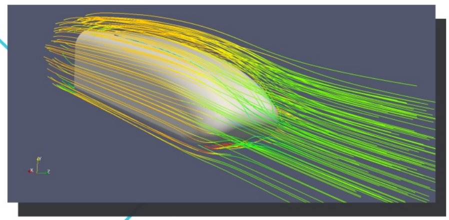
What is CFD?

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows.



What is OpenFOAM?

OpenFOAM (Open Source Field Operation and Manipulation), is a free and open source CFD toolbox. It is used in academia and industry to solve a wide range of computational problems. A big advantage of OpenFOAM is that its source code is accessible and modifiable. Who can attend- Anyone with basic fluid dynamics and CFD knowledge



The workshop will cover the following topics

- Basics of CFD
- Meshing
- Laminar flow through a pipe
- Turbulent flows
- K Omega
- Residuals
- Experiencing a GUI for OpenFOAM for block mesh
- Heat transfer problem for flow through a pipe
- Convection
- scalar TransportFoam
- Combining two solvers icoFoam and scalarTransportFoam
- Code structure in OpenFOAM

Benefits:

- Learn fundamentals of CFD using OpenFOAM
- Learn to simulate problem in OpenFOAM
- Learn to create your own solver
- Learn essentials of post-processing
- e-certificate for attending the workshop

Any queries related to R:
contact-cfd@fossee.in
Any queries related to spoken tutorials on R
Spoken Tutorial Forums
For general queries:
eoutreach@cse.iitb.ac.in

For more details visit:
http://www.it.iitb.ac.in/nmeict/announcements.html

OpenFOAM

fossee better education
https://fossee.in

Indian Institute of Technology Bombay

The FOSSEE project is funded by the National Mission on Education through ICT, Ministry of Education, Government of India.