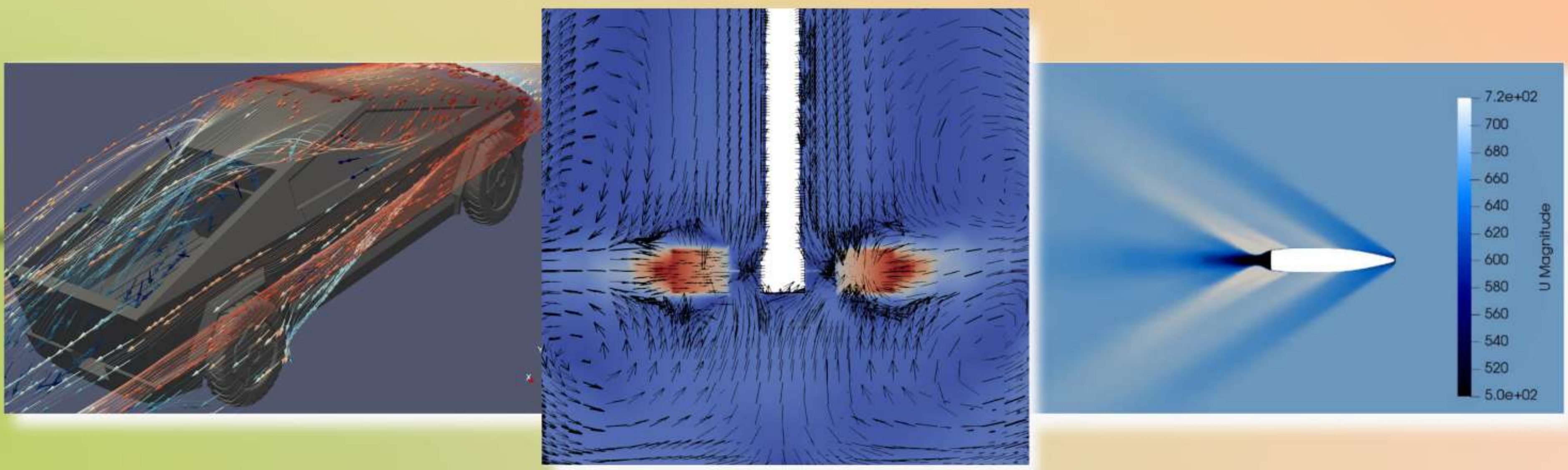
OpenFOAM workshop

Tackling Computational Fluid Dynamics and Heat Transfer problems using OpenFOAM

Date: 19-24 Feb 2024 | Mode: Online



What is CFD?
Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyse 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

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

Scan QR code for more details

Topics covered:
- Basics of CFD
- Meshing basics
- 3D meshing
- Physics of flow through a pipe
- Basics of turbulence
- Convection heat transfer
- Modelling of compressible flow
- Natural convection and conduction
- Multiphase flow and interface tracking
- Magnetohydrodynamics
- Eulerian & Eulerian model approach
- Discrete Element Method
- External flow / aerodynamics
- Laminar and turbulent flows
- Dynamic meshing
- Conjugate heat transfer in an enclosed domain

For any queries related to OpenFOAM contact us at: contact-cfd@fossee.in

For general enquiries contact: eoutreach@cse.iitb.ac.in

*For details regarding the workshop please **CLICK HERE** ⏎*

Indian Institute of Technology Bombay

fossee better education

NMEICT
National Mission on Education Through ICT

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