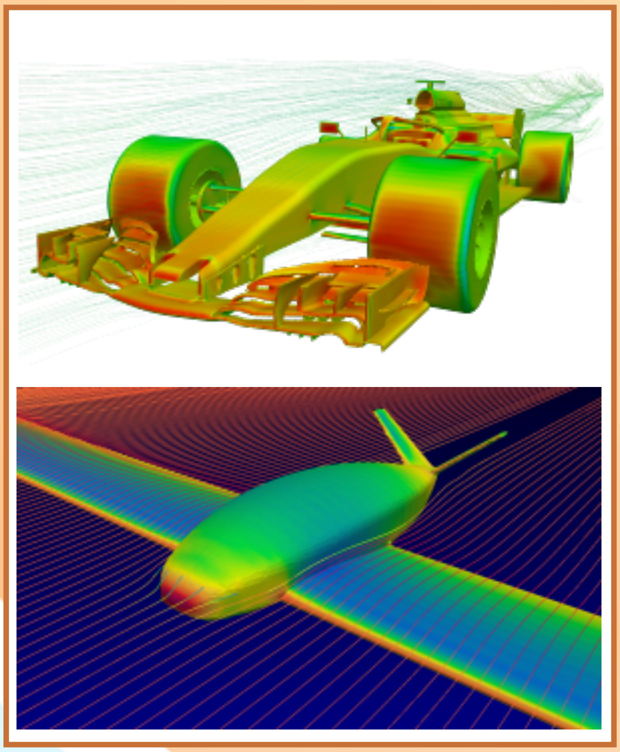
2-day workshop on OpenFOAM

[for beginners]

Date: 13 & 14 November 2021
Time: 10.00 am - 5.00 pm

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. In contrast to any proprietary software, the source code here is accessible and modifiable.



Who can attend?
Anyone with basic fluid dynamics and CFD knowledge

The workshop will cover following topics:

- Setting up mesh using blockMesh
- Setup case and solve using OpenFOAM
- Post-Processing in ParaView
- Grid Resolution and Convergence in OpenFOAM
- Interaction with IITB Faculty with CFD expertise

Benefits:

- Learn fundamentals of CFD using OpenFOAM
- Learn to simulate problem in OpenFOAM
- Learn essentials of post-processing
- E-certificate for attending Workshop

Indian Institute of Technology Bombay

fossee better education
https://fossee.in

NMEICT
National Mission on Education Through ICT

NEW INDIA ASSURANCE
दि न्यू इन्डिया एश्योरन्स कंपनी लिमिटेड
The New India Assurance Co. Ltd

The FOSSEE project is funded by the National Mission on Education through ICT, MoE, Government of India.
The FOSSEE-CFD- OpenFOAM project is partially funded by the CSR initiative by New India Assurance Co. Ltd.