## Participation opportunities for Students and Faculty Members:

### Learn OpenFOAM using the Spoken Tutorials

The Spoken Tutorials are self-learning tools available in English and other vernacular languages. Learners can use the Spoken Tutorials to get trained in OpenFOAM from the basics.

*For more details please visit- https://cfd.fossee.in/resources*

### Contribute to the Case Study Project

The Case Study Project aims to build a large repository of CFD problems solved using OpenFOAM. It is a repository built 'for-and-by' the CFD Community containing feasible problems of reasonable complexity.

*For more details please visit- https://cfd.fossee.in/case-study-project*

### Contribute in the Research Migration Project

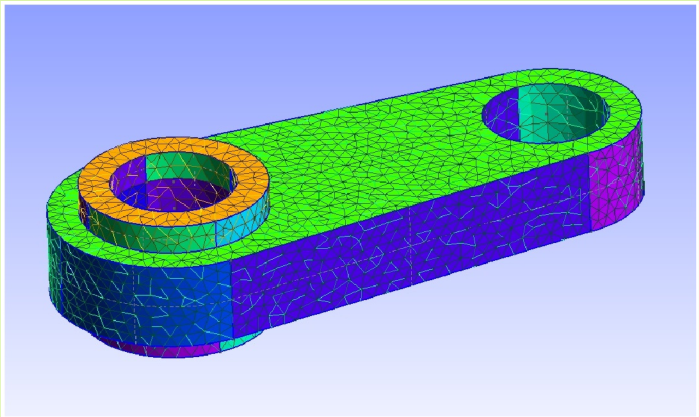
The Research Migration Project encourages students, research scholars and faculty members to contribute by choosing feasible CFD problems, solved using a commercial CFD tool, that are published in a leading journal paper or conference proceedings and solving the same using OpenFOAM.

*For more details please visit- https://cfd.fossee.in/research-migration-project*

### Contribute in the Lab Migration Project

The Lab Migration project aims to convert the labs of engineering institutions that use commercial software to that which will use OpenFOAM. The project invites students and faculty members of engineering institutions to propose OpenFOAM based solutions to the lab courses that use commercial software. The OpenFOAM team of FOSSEE, IIT Bombay, will help to propose solutions to the lab courses that compels engineering institutions to purchase expensive licenses of commercial CFD software.

*Visit https://cfd.fossee.in/lab-migration-project*



*GMSH geometry and meshing.*

**Apply for the Semester-long Internship Program**

**Apply for the Summer Fellowship Program**

Contributors to the Case Study Project / Research Migration Project/ Lab Migration Project/ Semester-long Internship/ Summer Fellowship are awarded with honorarium and e-certificates.

The OpenFOAM Project, FOSSEE, IIT Bombay, invites Faculty Members PAN India to avail partnership opportunities as a '**Faculty Partner**' and contribute to the project.

For details please visit-
***https://cfd.fossee.in/faculty-partners***

**Visit us: https://cfd.fossee.in**

Indian Institute of Technology Bombay

**Contact us:**
**contact-cfd@fossee.in**

fossee better education

# OpenFOAM®

## An Open source alternative to commercial Computational Fluid Dynamics (CFD) Software

### What is CFD?

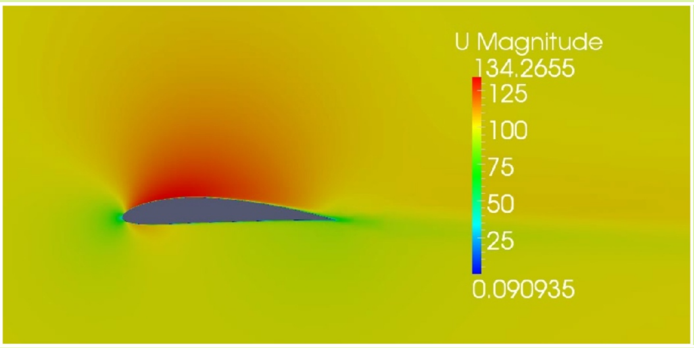
Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows.

### What is OpenFOAM?

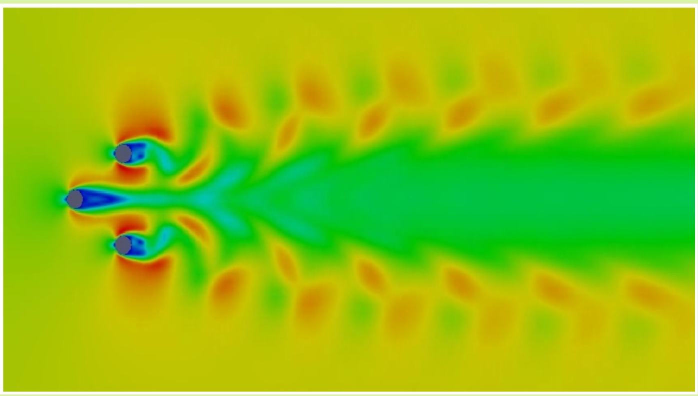
OpenFOAM (Open Source Field Operation and Manipulation) is a free and Open source CFD toolbox. It is used in academia and industry to solve wide variety of computational problems. In contrast to any proprietary Software, the source code here is accessible and modifiable..

### Advantages over other CFD Software

- No License Cost
- Flexible: Code customisation according to problem
- Supports High Performance Computing
- Huge Solver Database
- Import Mesh from other Software




*Flow over an airfoil*



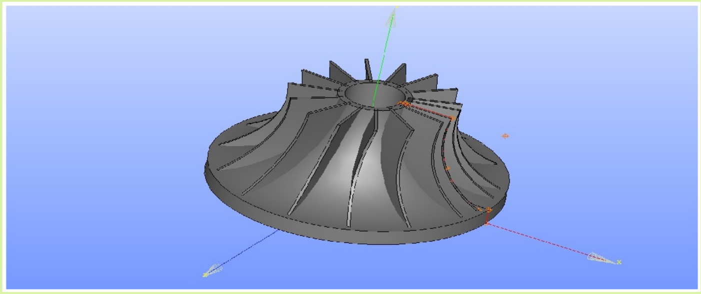
*Flow over circular cylinder in tandem arrangement*

### Capabilities of OpenFOAM

OpenFOAM has massive capabilities. There is a huge solver database which covers the breadth and depth of CFD

### OpenFOAM solver & Applications:

- Incompressible flow
- Compressible flow
- Heat transfer
- Multiphase flow
- Direct Numerical Simulation (DNS)
- Combustion
- Particletracking flows
- Discrete methods
- Electromagnetics
- Stress Analysis



*Impeller design in Salome*

### OpenFOAM Utilities

### Geometry and Meshing:

Utilities to generate geometry like blockMesh and meshing using snappyHexMesh, foamyHexMesh etc. Can import mesh from other meshing Software like Ansys, GMsh, Salome etc.

### Parallel Processing:

Tools to decompose, reconstruct and redistribute the computational case to perform parallel calculations, at times better than other CFD Software.

### Post Processing Utilities:

Post processing can be done using ParaView which comes with OpenFOAM. Can export to other third party visualisation Software.