



OpenFOAM[®]

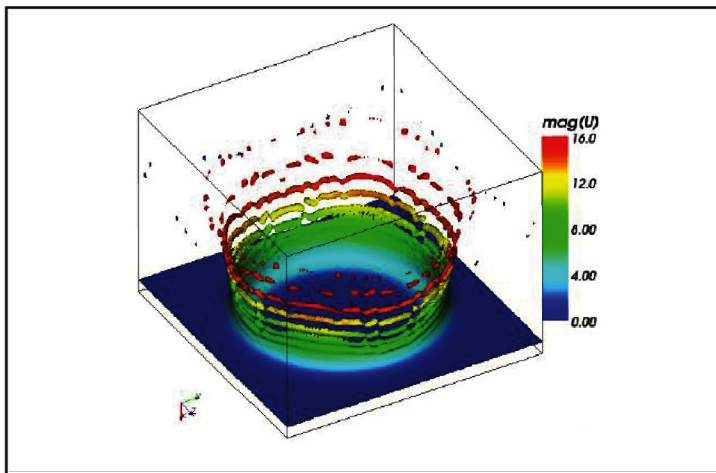
An Open source alternative to commercial Computational Fluid Dynamics (CFD) Software

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.



Drop Splashing

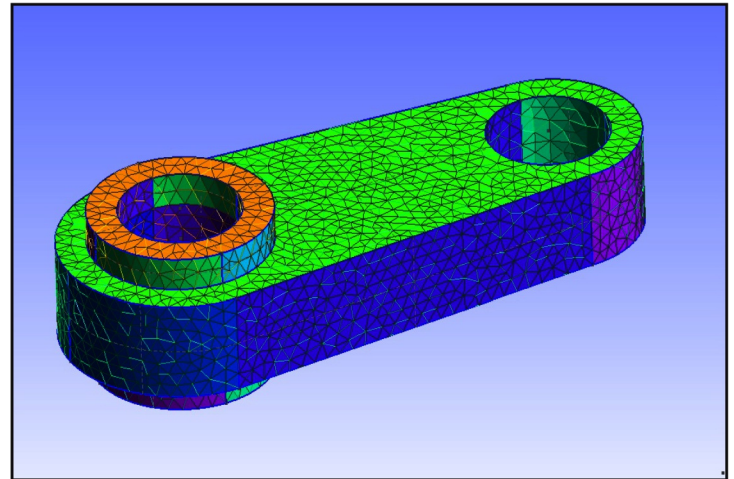
Advantages of using OpenFOAM

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

OpenFOAM solver & Applications

- Incompressible flow
- Compressible flow
- Heat transfer

- Multiphase flow
- Direct Numerical Simulation (DNS)
- Combustion
- Particle-tracking flows
- Discrete methods
- Electromagnetics
- Stress Analysis



Meshed Geometry using Gmsh

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.

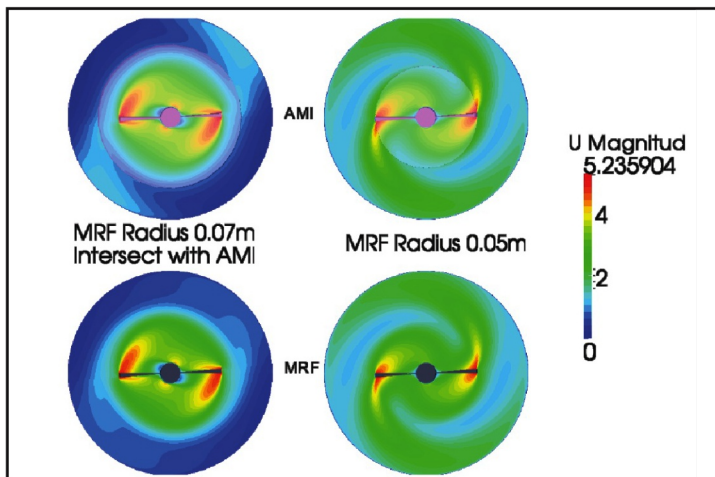
Can export to other third party visualisation Software.

Case Study Project:

The Case Study Project aims to promote OpenFOAM: Open source Computational Fluid Dynamics (CFD) software, by developing case studies based on research journals, projects and thesis using OpenFOAM and making it available to CFD researchers and users for reference and usage. This project is for college students and graduates who are willing to undertake an internship with FOSSEE, IIT Bombay.

Through this project students can:

- Earn honorarium, get certificates and improve job potential
- Available for free download and can be used for academic, research & industrial purpose



Mixing Agitator

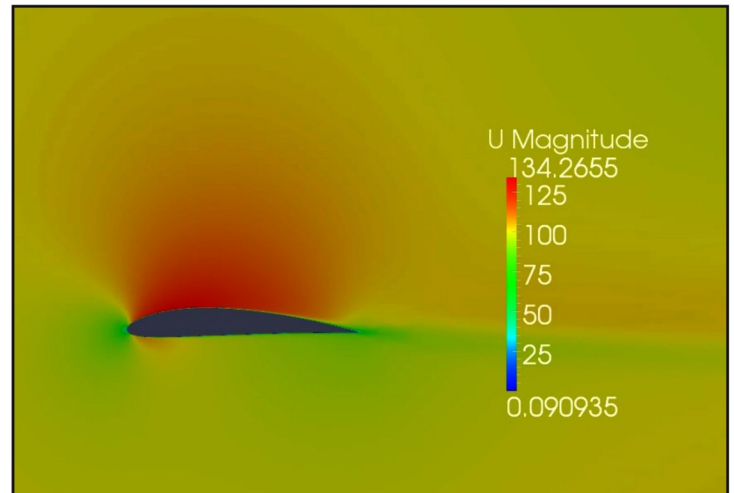
Lab Migration

Lab Migration aims to migrate labs that use proprietary Software to a Free and Open Source Software (FOSS) only lab. Colleges can save a large amount of money that is spent on procuring licenses for commercial Software by switching to FOSS.

The Lab Migration team helps in the following ways:

- Provide suggestions on the different ways FOSS can be implemented in the laboratory
- Coordinate the migration of the lab to a FOSS only laboratory

- Provide solutions to the lab's problem statement



Flow over an airfoil

Spoken Tutorials

Spoken tutorials in OpenFOAM are available free of cost. A spoken tutorial is a screencast of a computer session along with the narration, explaining a FOSS. These tutorials can be used to self learn OpenFOAM.

Forum

Forum is a place where one can post all their doubts and questions which she/he gets while listening to and practicing the spoken tutorials. For this purpose, the CFD team, FOSSEE, IIT Bombay operates it's own forum.

FOSSEE Team, IIT Bombay

FOSSEE (Free and Open Source Software for Education) team aims to provide freedom in education by:

- Motivating and enabling students, faculty & others to use Open Source Software tools instead of proprietary Software.
- Creating educational content around existing Open Source Software.
- Creating documentation, books and courses in the area of Open Source Software.

cfd.fossee.in

Contact us: contact-cfd@fossee.in