Support for OpenFOAM by 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.

Activities of FOSSEE

- Textbook Companion
- Lab Migration
- Spoken Tutorials
- Forums

Textbook Companion

The Textbook Companion (TBC) activity aims to port solved examples from standard textbooks using any Free and Open source software (FOSS).

Objectives:

- Make individuals learn FOSS through a practical approach
- Provide a huge database of Textbook Companions as a learning resource
- To make it easy for users of such textbooks to start using FOSS
- To improve the documentation available for FOSS

This internship is available for the following FOSS supported by FOSSEE:

- OpenFOAM
- Scilab
- FreeEDA
- OR Tools
- Python

http://cfd.fossee.in/Textbook_Companion_Project
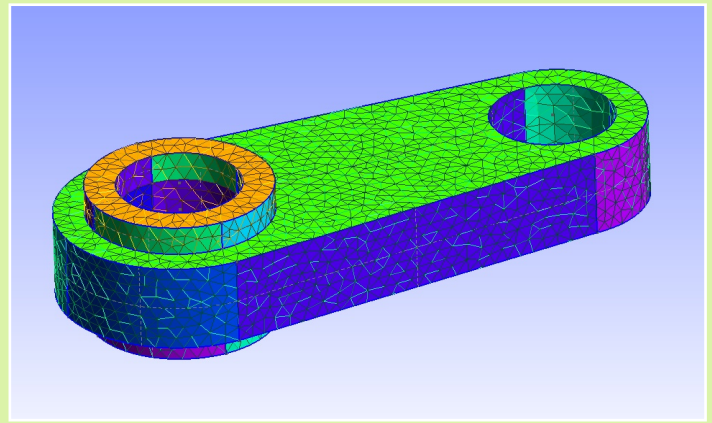
GMSH geometry and meshing.

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 statements

http://cfd.fossee.in/Lab_Migration_Project

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

http://cfd.fossee.in/spoken-tutorial

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 OpenFOAM team, FOSSEE, IIT Bombay operates it's own forum.

http://forums.fossee.in

http://cfd.fossee.in

IIT Bombay

Contact us: Text Book Companion: cfd-tbc@fossee.in
Lab Migration: cfd-lab@fossee.in
General Enquires: contact-cfd@fossee.in

fossee
better
education
http://fossee.in

National Mission on Education through ICT, MHRD h ttp://sakshat.ac.in

OpenFOAM
Open source Field Operation and Manipulation

What is OpenFOAM?

OpenFOAM is a free and open source CFD toolbox. OpenFOAM stands for Open source Field Operation And Manipulation. 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 costs
- Friendly syntax for partial differential equations
- Unstructured polyhedral grid capabilities
- Automatic parallelization of applications written using OpenFOAM high-level syntax
- Wide range of applications and models ready to use
- Code customisation according to problems

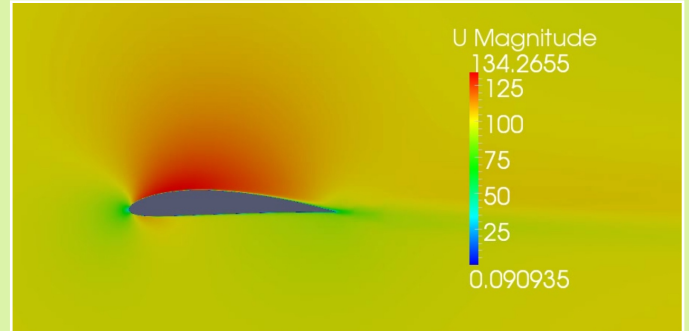## Capabilities of OpenFOAM

OpenFOAM has massive capabilities. There is a huge solver database which covers the breadth and depth of CFD
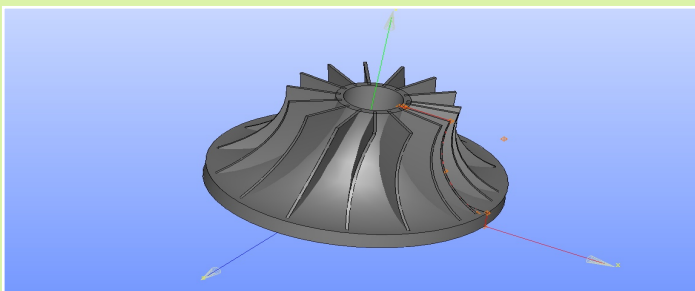
Impeller design in Salome
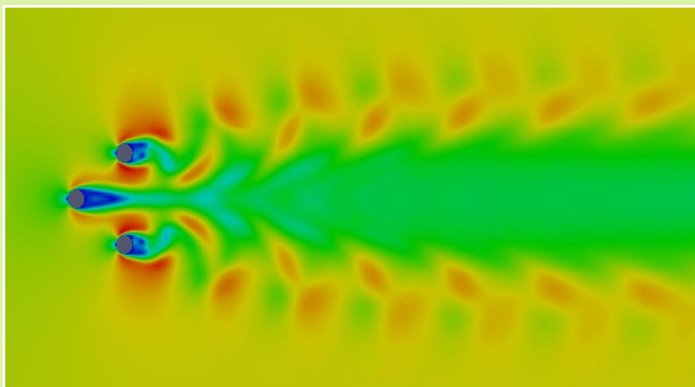
Flow over circular cylinder in tandem arrangement

U Magnitude
134.2655
125
100
75
50
25
0.090935

Flow over an airfoil

OpenFOAM solvers include:

- Incompressible flow with RANS and LES capabilities
- Compressible flow solvers with RANS and LES capabilities
- Buoyancy-driven flow solvers
- DNS and LES
- Multiphase flow solvers
- Particle-tracking solvers
- Solvers for combustion problems
- Solvers for conjugate heat transfer
- Molecular dynamics solvers
- Direct Simulation Monte Carlo solvers
- Electromagnetics solvers
- Solid dynamics solvers

In addition to the standard solvers, OpenFOAM's syntax lends itself to the easy creation of custom solvers and also boundary fields.

OpenFOAM utilities are subdivided into:

Mesh conversion: Utilities to convert mesh generated from third party meshing tools like Ansys, GMSH, etc. to OpenFOAM format.

Parallel processing utilities: Tools to decompose, reconstruct and redistribute the computational case to perform parallel calculations, at times better than other CFD software.

Post-processing utilities: Data can be visualized using paraview, can be exported to other software for visualization such as Ensight, Mayavi.