**OpenFOAM: O**pen **S**ource **F**ield **O**peration **A**nd **M**anipulation, is a free and Open Source CFD (Computational Flid Dynamics) software package promoted by the ESI Group. The Development of OpenFOAM, started in late 1999 by Dr. Hrvoje Jasak and Henry Weller in Imperial College of Engineering, with a vision to create a C++ based CFD solver for future research purposes.

OpenFOAM is a collection of C++ libraries which can be used to create executable applications which fall under two categories: Solvers and Utilities. Solvers, which are used to solve the continuum mechanics problems, and Utilities, which perform the task of data manipulation.

The syntax of the solver application in OpenFOAM, is similar to that used in writing a Partial Differential Equation.
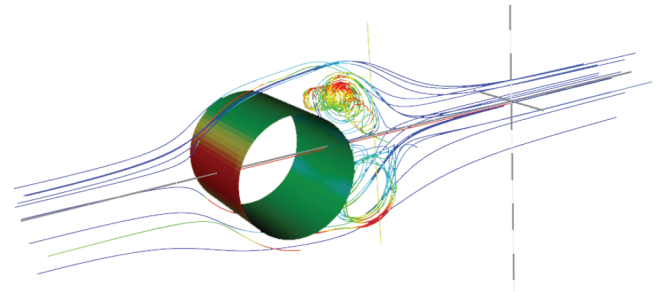
Flow Over 3-D Bluff Body

A snippet of the OpenFOAM code:
Navier-Stokes equation:

$$\frac{\partial \rho U}{\partial t} + \nabla.\rho U - \nabla.\mu\nabla U = -\nabla p$$

Writing the equation in OpenFOAM:

```
solve
  {
    fvm ::ddt (rho, U)
  +fvm :: div (rho, U)
   -fvm :: laplacian(nu, U)
   ==
   - fvc::grad (p)
   };
```
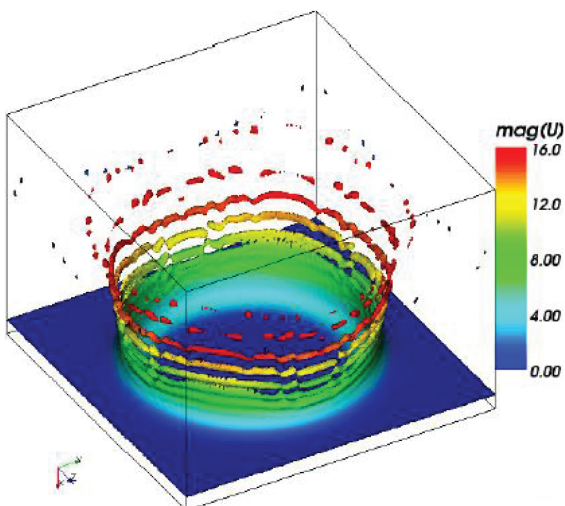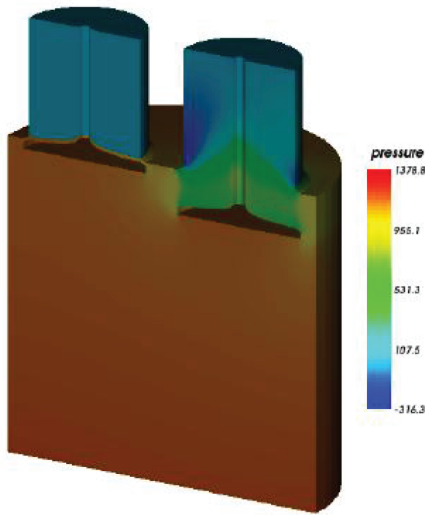

Drop Splashing

Advantages of using OpenFOAM
- Free and Open Source
- Released under GPL
- Flexible: User can change the code according to his requirements
- Supports OpenMPI
- Contains vast number of Solvers
- Abile to import data from other meshing software
- Results can be exported to different post-processing data formats

IIT BOMBAY

fossee
better
education

I. C. Engine Simulation

Application areas of OpenFOAM

- Incompressible Flows
- MHD flows
- Compressible Flows
- DNS
- Multiphase Flows
- DSMC
- Financial Problems
- Combustion
- Particle tracking
- Heat Transfer

Industrial users of OpenFOAM:

- AUDI
- TATA STEEL
- BARC
- VOLKSWAGEN

Textbook Companion

OpenFOAM textbook companion is one of the major activities supported by the CFD team at FOSSEE, IIT Bombay. This work involves generating an online collection of OpenFOAM cases/solvers for worked out examples and select exercise problems from standard textbooks. Participate and earn attractive honorarium and Certificate of Internship from FOSSEE, IIT Bombay.

Contact Us:

Text Book Companion: cfd_tbc@fossee.in   General Queries: contact_cfd@fossee.in

'Learn OpenFOAM the easy way through Spoken Tutorial'

Spoken tutorials for self learning of OpenFOAM are available at
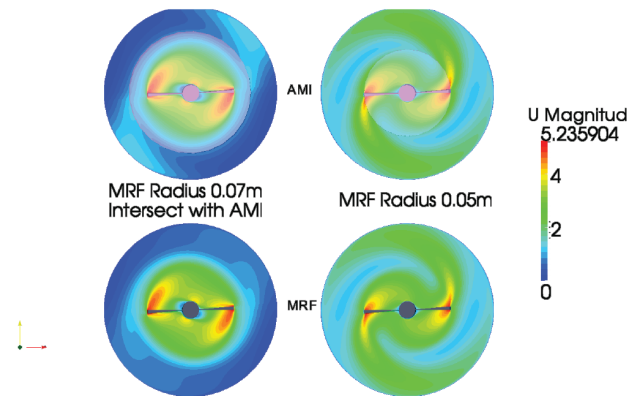http://spoken-tutorial.org
- ▶ Select FOSS Category: OpenFOAM
- ▶ Select Language: English
- ▶ Locate tutorial
  Practise them in the given sequence.

SELF Workshops

Enroll with us to become a Workshop Organiser, and help conduct OpenFOAM and other FOSS workshops like Linux, Scilab, Python, PHP-MySQL, Java, C, and many others, in your college and neighbouring colleges. You can also help conduct these workshops at the organisation that you work in.

About FOSSEE

FOSSEE (Free and Open Source Software for Education) project is part of the National Mission on Education through ICT (NMEICT) with the thrust area being 'Adaptation and deployment of Open Source simulation packages equivalent to proprietary software', funded by MHRD, based at the Indian Institute of Technology Bombay (IITB).

Mixing Agitator