OpenFOAM: Free and Open Source Alternative to FLUENT/ StarCD

Spoken Tutorial

OpenFOAM: Open Source Field Operation And Manipulation, is a free, open source CFD software package developed by the OpenFOAM Team and the recent new owners of the company ESI Group. The Development of OpenFOAM started in the 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 are to create executable applications which fall under two categories, Solvers and Utilities. Solvers, which are used to solve the continuum mechanics problems and similarly 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.

Advantages of using OpenFOAM:

- Free, Open source released under GPL
- Flexibility : User can change the code according to his needs
- Supports OpenMPI
- Contains vast number of solvers
- Can import data from other meshing software
- Export to different post-processing data formats

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)
      };
```

Application areas of OpenFOAM:

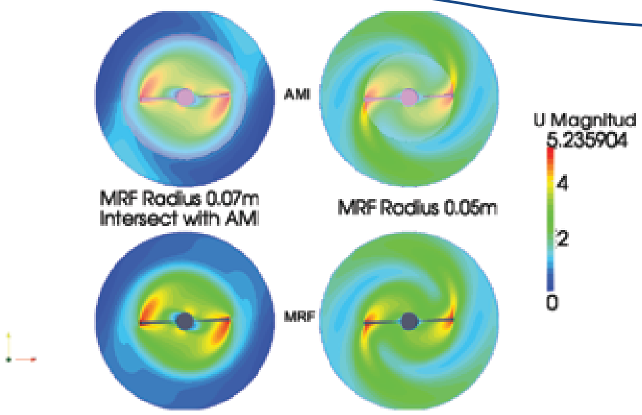
- Incompressible Flows
- Compressible Flows
- Multiphase Flows
- MHD flows
- DSMC
- Combustion
- Financial Problems
- Particle tracking
- Heat Transfer
- DNS, etc...

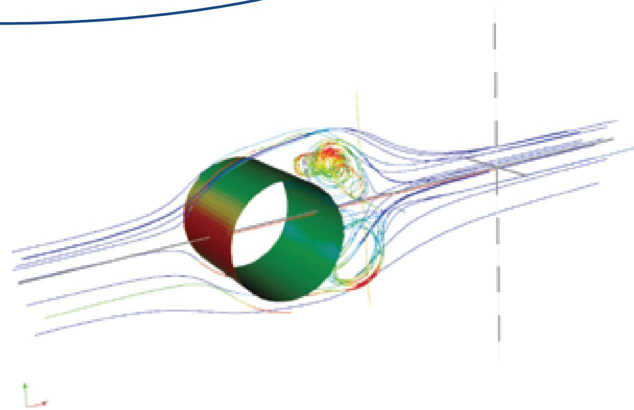
"Learn OpenFOAM the easy way through Spoken Tutorial CD"

Spoken tutorials for self learning of OpenFOAM are available at:
http://spoken-tutorial.org

- Select FOSS Category:- OpenFOAM
- Select Language:- English
- Locate tutorial

and practise them in the given sequence.

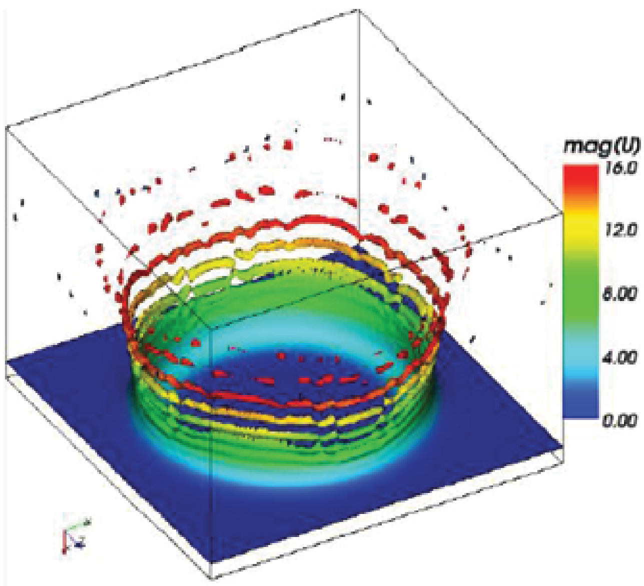




The Spoken Tutorial Project offers:

Computational Fluid Dynamics software "OpenFOAM" workshops to 4th year students of Mechanical, Civil or Chemical Engineering, who have had Fluid Mechanics, Heat Transfer and Maths subjects till the 3rd year.

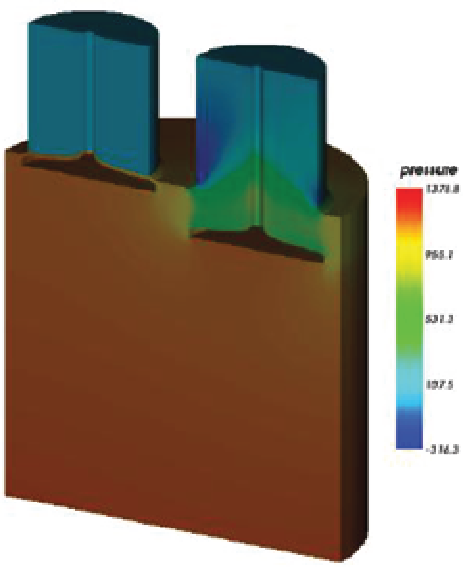
As a pre-requisite, you need to know how to use Linux OS. If not, we offer Linux workshops, too, through the Spoken Tutorial Project.



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.

For more information, write to us at info@fossee.in



Industrial users of OpenFOAM:

- AUDI
- BARC
- CERN LABS
- TATA STEEL
- VOLKSWAGEN

The Spoken Tutorial project is the initiative of the ‘Talk To A Teacher’ activity of the National Mission on Education through Information and Communication Technology launched by the Ministry of Human Resource Development, Government of India.