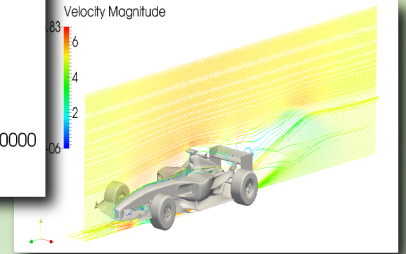
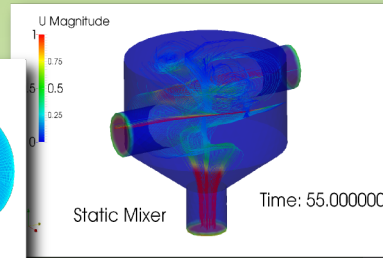
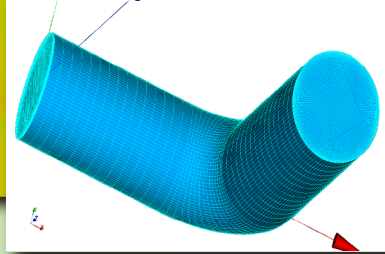
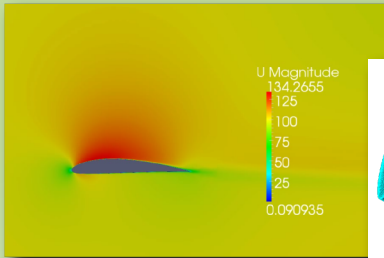




# OpenFOAM Symposium

27 February 2016 | LCH 1&2, IIT Bombay



## OpenFOAM

A free and open source computational toolkit, comprising of several solvers and utility applications. Widely used in the field of engineering, both in industry and academia.

Easily customizable as per the user's requirement.

Used to solve many computational flow problems such as:

- Incompressible / Compressible flows
- Multiphase flows
- Particle methods (DEM, DSMC, MD)
- Combustion
- Buoyancy-driven flows
- Conjugate heat transfer

## About the Symposium

Talks and discussions by eminent users and developers, both from industry and academia.

A hands-on OpenFOAM workshop will be conducted in the symposium.

Complimentary kits containing OpenFOAM live DVDs shall be provided to all the participants.

### Speakers

**Ravikumar Ajjampudi** **Keynote**

(GM - CAE Services at ESI Group)

**Kshitij Neroorkar**

(Development Engineer at CD-adapco)

**Vinay Unnikrishnan**

(ISRO)

## Call for Abstracts

Interested participants can submit abstracts pertaining to their work done strictly using OpenFOAM.

The abstract should be minimum of 250 words and not exceed 500 words.

Please submit the abstracts at : <http://fossee.in/conference/cfd-symposium/#abstracts>

Last date for  
submission of abstracts:  
**19 February 2016**

**Contact us:**  
[contact-cfd@fossee.in](mailto:contact-cfd@fossee.in)  
**Website:** [cfd.fossee.in](http://cfd.fossee.in)

Last date for  
online registration:  
**25 February 2016**

Register at: <http://fossee.in/conference/cfd-symposium/>



<http://fossee.in>



IIT Bombay

National Mission on Education through ICT, MHRD | <http://sakshat.ac.in>

Disclaimer: OPENFOAM® is a registered trade mark of OpenCFD Limited, producer and distributor of the OpenFOAM software.