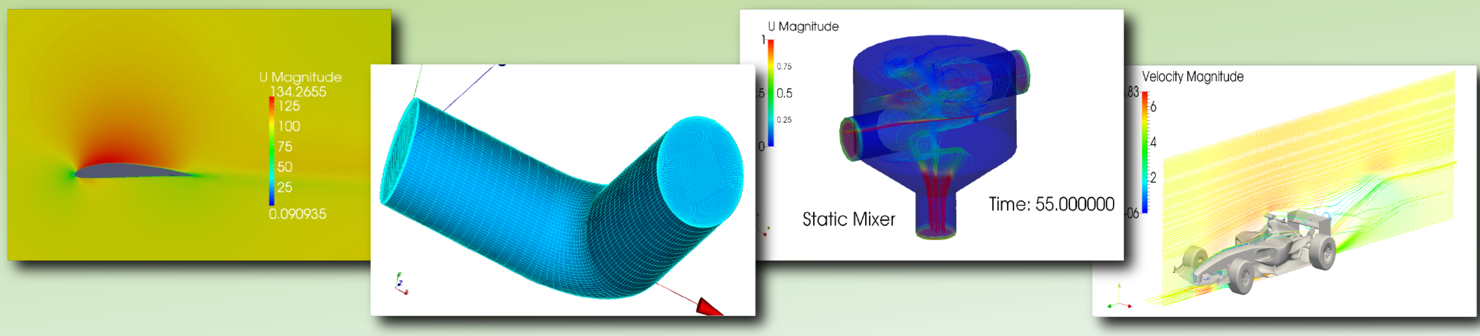
OpenFOAM Symposium

27 February 2016 | LCH, 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 flows
- Multiphase flows
- Combustion
- Buoyancy-driven flows
- Conjugate heat transfer
- Compressible flows
- Particle methods (DEM, DSMC, MD)
- Other (Solid dynamics, electromagnetics)

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.

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

Website

cfd.fossee.in

Last date for online registration:

25 February 2016

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

fossee better education

http://fossee.in

IIT Bombay

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