

# CFD with OpenSource software

## Purpose of the course:

- To give an introduction to OpenSource software for CFD
- To give an introduction to OpenFOAM in order to 'get started'
- To introduce how to modify OpenFOAM for specific purposes
- To generate, and make available, new tutorials on OpenFOAM usage
- To increase the use of OpenFOAM in Sweden (and other countries)

## Arrangement

- **First occasion, two days:**  
(For dates, times and location, see course homepage)  
Homework...
- **Second occasion, two days:**  
(For dates, times and location, see course homepage)  
Homework...
- **Third occasion, two days:**  
(For dates, times and location, see course homepage)  
Homework...  
Project work...
- **Project presentations, one or two days:**  
(For dates, times and location, see course homepage)  
**Homework: reviews!!!**
- **Hand-in of review, response to reviewer,  
and updated files**

Remind me to take breaks!

## Syllabus for the first and second days

- An introduction on how to *use* the standard solvers, utilities and libraries of OpenFOAM.
- An introduction on how to post-process using paraFoam (Paraview).
- An introduction to the organization of the tutorials.
- A quick look at Gnuplot, Xmgrace, Python, etc.
- Hands-on exercises using utilities and functionObjects
- Learn how to learn yourself after the course
- Homework for next occasion!

We will use the OpenFOAM User Guide and Programmers Guide, and information that can be found in the OpenFOAM Wiki (such as the work by the OpenFOAM Turbomachinery working group) but I will add my personal experience.

## Syllabus for the third and fourth days

This will now (after 2017) be spread out over both 2nd and 3rd occasions.

- The OpenFOAM source code directory structure.
- High-level programming and compilation – applications (solvers and utilities).
- A crash-course in C++ and object orientation for *reading* Open-FOAM.
- High-level programming and compilation – libraries (turbulence models and boundary conditions)
- Debugging
- Learn how to learn yourself after the course.
- Homework after each occasion!

After these days you should be able to investigate the code and find out what it does. You should also be able to make simple modifications to the code.

## Syllabus for the fifth and sixth days

- See previous slide, since that content will spread out on this occasion as well.
- A project work should be chosen. The project work should end up with a tutorial that should be presented at the final occasion, and peer-reviewed by the other participants. Start thinking about a project already from the beginning!
- Homework.
- The project.

## Syllabus for the presentation day(s)

- The tutorials will be presented by all the participants in workshop format, so that all the participants (and the teacher) can follow the instructions on their computers.  
Slides, written tutorial report, and all needed files must be made available at the course homepage at latest the evening before.
- Sum-up of the course, and course evaluation.

### NOTE: Final homework:

- After this occasion all participants must peer-review the tutorial by another participants in order to pass the course. An example of a good peer-review will be distributed on the course homepage.
- Each participant should update their tutorial according to the peer-reviews in order to pass the course. A short 'reply to the reviewer', pointing out the changes made, should be attached. An example is available at the course homepage.

## Learning outcomes

- Learn how to download, install, compile and run standard OpenFOAM solvers and utilities
- Learn how to implement solvers and utilities
- Learn how to implement a turbulence model
- Learn how to implement a boundary condition
- Learn the basics of C++ and object orientation
- Learn how to do CFD with OpenFOAM together with Python, m4, Gnuplot, Paraview etc.
- Learn basics of Linux (see link on homepage), Doxygen, Compilation procedures, Debugging, Version Control Systems and VTK
- Learn how to use OpenFOAM by doing a project work

### Also:

- Learn how to continue learning
- Help others learn OpenFOAM by writing a tutorial

## References

- OpenFOAM homepage: [www.openfoam.com](http://www.openfoam.com)
- OpenFOAM User Guide, Programmers Guide, Doxygen
- OpenFOAM Wiki: [www.openfoamwiki.net](http://www.openfoamwiki.net)
- OpenFOAM-extend: <http://foam-extend.org>
- OpenFOAM Forum: <http://www.cfd-online.com/Forums/openfoam/>
- OpenFOAM workshop: [www.openfoamworkshop.org](http://www.openfoamworkshop.org)
- C++ direkt, Jan Skansholm, Studentlitteratur  
(C++ from the Beginning, Jan Skansholm, probably the same in english)
- C genom ett nyckelhål, Håkan Strömberg, Studentlitteratur
- An introduction to Computational Fluid Dynamics,  
H K Versteeg & W Malalasekera
- Computational Methods for Fluid Dynamics,  
J.H. Ferziger & M. Peric



## Acknowledgements:

- **Chalmers Centre for Computational Science and Engineering**



<http://www.c3se.chalmers.se>

- **Swedish National Infrastructure for Computing**



<http://www.snic.vr.se/>

- **Hrvoje Jasak at Wikki Ltd. WIKKI** <http://www.wikki.co.uk/>

- **ESI-OpenCFD**, <http://www.openfoam.com/>

- **The OpenFOAM Foundation**, <http://www.openfoam.org/>

- **The OpenFOAM user community**

- **People involved in the development of FOAM-extend**