CFD with OpenSource software, 2013

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)

CFD with OpenSource software, 2013

How to learn more after this course:

- Learn by doing
- Exchange knowledge with other OpenFOAM users at the Forum (http://www.cfd-online.com/Forums/openfoam)
- Manuals and source code are available free from www.openfoam.com
- The D'oxygen manual at www.openfoam.com
- The OpenFOAM Wiki (www.openfoamwiki.net)
- The OpenFOAM Portal (www.extend-project.de)
- The CoCoons project (http://www.cocoons-project.org)
- The OpenFOAM workshop (www.openfoamworkshop.org)

Acknowledgements:

- **Svenskt VattenkraftCentrum**

  http://www.svc.nu

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

  http://www.c3se.chalmers.se

- **Hrvoje Jasak at Wikki Ltd.**

  http://www.wikki.co.uk/

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

- **The OpenFOAM user community**

Time schedule

- **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)
  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!

Preliminary syllabus for the first and second days

- Get access to computers and software installations.
- A short general discussion about CFD.
- 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 code and cases.
- 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.
- Homework for next occasion!

Preliminary syllabus for the third and fourth days

- An introduction on how to *modify* the standard solvers, utilities and libraries of OpenFOAM.
- A crash-course in C++, from an OpenFOAM perspective
- We will have a deeper look into the code and make our own solvers, utilities and libraries. For this we must know how to compile all, or part, of the code. In particular we will have a look at turbulence models and boundary conditions.
- We will use the OpenFOAM User Guide and Programmers Guide as reference material. It is good to invest in a book in C++, like 'C++ Direkt' by Jan Skansholm (Studentlitteratur).
- 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.
- Homework for next occasion!

Preliminary syllabus for the fifth and sixth days

- Other useful tools (see schedule).
- Advanced OpenFOAM usage and development (see schedule).
- 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.

Preliminary 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 distributen 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`
- OpenFOAM User Guide, Programmers Guide, Doxygen
- OpenFOAM Wiki: `www.openfoamwiki.net`
- OpenFOAM-extend: `http://sourceforge.net/projects/openfoam-extend/`, and `http://extend-project.de`
- CoCoons project: `http://www.cocoons-project.org/`
- OpenFOAM Forum: `http://www.cfd-online.com/Forums/openfoam/`
- OpenFOAM workshop: `www.openfoamworkshop.org`
- C++ direkt, Jan Skansholm, Studentlitteratur
- 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
- **Course homepage:** `http://www.tfd.chalmers.se/~hani/kurser/OS_CFD`(`_YEAR`)