CFD with OpenSource software, 2010

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 increase the use of OpenFOAM in Sweden (and other countries)

How to learn more after this course:

- Learn by doing
- Exchange knowledge with other OpenFOAM users at the Forum
- For more advanced courses, see www.openfoam.com
- Manuals and source code are available free from www.openfoam.com
- The D'oxygen manual
- The OpenFOAM Wiki (www.openfoamwiki.net)

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/

- **OpenCFD Ltd.**, http://www.opencfd.co.uk/

- **The OpenFOAM user community**

Time schedule

- **First occasion:**
  1-2/9 9.15 - 17.00 in ?? (see course homepage)
  Homework...

- **Second occasion:**
  8-9/9 9.15 - 17.00 in ?? (see course homepage)
  Homework...

- **Third occasion:**
  15-16/9 9.15 - 17.00 in ?? (see course homepage)
  Project work...

- **Project presentations:**
  Two days at the end of the course
  (see course homepage)
  Homework: reviews!!!

Remind me to take breaks!

Preliminary syllabus for the first and second days

- How to *use* OpenFOAM.
- An introduction to 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.
- Advanced mesh generation and manipulation.
- Advanced usage for hydraulic turbomachinery.
- We will use the OpenFOAM User Guide and Programmers Guide, and the work by the OpenFOAM Turbomachinery working group as a base, but I will add my personal experience.
- Homework for next occasion!

Preliminary syllabus for the third and fourth days

- How to *modify* OpenFOAM.
- 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 and project work.
- On fifth day: See contents at the course homepage when we get closer to that day.
- On sixth day: Supervised project work - start thinking about a project already from the beginning!
- A project work should be chosen. The project work should end up with a tutorial that should be presented at the third occasion, and peer-reviewed by the other participants.
- Homework: The project.

Preliminary syllabus for the presentation days

- 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, and all needed files must be available at the course homepage.
- 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.

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 and Paraview
- 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:

- 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/`
- OpenFOAM Forum: `http://www.cfd-online.com/Forums/openfoam/`
- 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_2010`