CFD with OpenSource software, 2008

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

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.org
- Manuals and source code are available free from www.openfoam.org
- 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 - Need any changes?

- **First day:**
  7/11 9.15 - 17.00 in MT13

- **Second day:**
  10/11 9.15 - 17.00 in MT13

- **Third day:**
  21/11 9.15 - 17.00 in MT13

- **Fourth day:**
  24/11 9.15 - 17.00 in MT13

- **Fifth day:**
  1/12 9.15 - 17.00 in MT13

- **Project presentations:**
  Two days after Christmas,
  dates decided together right now!

- **Lunches:**
  12-13. You pay for it yourself. Don't leave anything in MT13!

CFD with OpenFOAM

Syllabus for the first and second days:

- An introduction to how to install and use the standard solvers, utilities and libraries of OpenFOAM.
- An introduction to the organization of the code and cases.
- We will use the OpenFOAM User Guide and Programmers Guide as a base, but I will add my personal experience.

CFD with OpenFOAM

Preliminary syllabus for the third and fourth days:

- 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.

- Presentations by experienced OpenFOAM users/developers, showing their speciality in OpenFOAM usage.

CFD with OpenFOAM

Preliminary syllabus for the fifth day:

- Programming in OpenFOAM.
- Examples of different ways of using OpenFOAM.
- Presentations by experienced OpenFOAM users/developers, showing their speciality in OpenFOAM usage.
- 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.

CFD with OpenFOAM

Preliminary syllabus for the presentation days:

- The tutorials will be presented by all the participants.
- After this occasion all participants must peer-review and grade all the tutorials by the other participants in order to pass the course.
- Each participant should update their tutorial according to the peer-reviews in order to pass the course.
- Sum-up of the course, and course evaluation.

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

Computer facilities

- Your CID accounts are valid in the student Linux computer labs in the M-building. There is no `ssh` access to those computers.

- Remote access can be done through:

```
ssh -l <CID> remote1.student.chalmers.se
ssh -l <CID> remote2.student.chalmers.se
ssh -l <CID> remote3.student.chalmers.se
ssh -l <CID> remote4.student.chalmers.se
```

- Student accounts and temporary accounts only have 1GB disk. You can use /local to temporarily store larger files, but they will automatically be removed after 7 days.

- You should be able to do an `ssh` to your own computing facilities and work there if you like.

How to use the pre-installed OpenFOAM-1.5.x

- To use the pre-installed OpenFOAM-1.5.x, do the following:

  BASH users: Add the following line to your `~/.profile` file AND to your `~/.bashrc` file (The remote nodes use `~/.profile` and the computers in the basement use `~/.bashrc` for some reason):
  ```
  . /chalmers/sw/unsup/OpenFOAM/OpenFOAM-1.5.x/etc/bashrc
  ```

  CSH users: Add the following line to your `~/.cshrc` file:
  ```
  source /chalmers/sw/unsup/OpenFOAM/OpenFOAM-1.5.x/etc/cshrc
  ```

  Later you will learn how to install OpenFOAM by yourself

- You must now make three directories:
  ```
  mkdir -p $HOME/OpenFOAM/<CID>-1.5.x/run
  ```
  This is where you will put your case files.
  They may be located elsewhere, but this is the default location.

References

- www.openfoam.org
- OpenFOAM User Guide
- OpenFOAM Programmers Guide
- OpenFOAM Doxygen
- 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_2008`