CFD with OpenFOAM

Purpose of the course:

- 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 advanced courses, see www.openfoam.org
- Manuals and source code are available free from www.openfoam.org
- The D'oxygen manual
- The OpenFOAM Wiki

CFD with OpenFOAM

Time schedule 2007:

- **First occasion:**
  18/10 10.00 - 17.00 in MT11
  19/10 9.00 - 17.00 in MT11
- **Second occasion:**
  Two days in December (we find dates together)
- **Third occasion:**
  Two days in February (we find dates together)
- **Lunches:**
  12-13. You pay for it yourself. Don't leave anything in MT11.

CFD with OpenFOAM

Syllabus for the first occasion:

- 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.
- After these days you should continue practicing installing OpenFOAM on your own system, and run through as many tutorials as we can find. Then you will be prepared for the next stage of the course.

CFD with OpenFOAM

Preliminary syllabus for the second occasion:

- We will have a deeper look into the code and make our own solvers, utilities and libraries. For this we must learn 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.
- 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 third occasion:

- 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.
- Presentations by experienced OpenFOAM users/developers, showing their speciality in OpenFOAM usage.
- Sum-up of the course, and course evaluation.

CFD with OpenFOAM

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 use OpenFOAM by doing a project work

Also:

- Help others learn OpenFOAM by writing a tutorial

CFD with OpenFOAM

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.

CFD with OpenFOAM

OpenFOAM installation:

- To use the pre-installed OpenFOAM, do the following:
  ```
  cp -r /chalmers/sw/unsup/OpenFOAM/OpenFOAM-1.4.1/.OpenFOAM-1.4.1 ~
  ```
  (You have to do this since you need write-permission in that directory)
  Add the following line to your `~/.cshrc` file (if you use tcsh):
  ```
  source $HOME/.OpenFOAM-1.4.1/cshrc
  ```
  Add the following line to your `~/.profile` file AND to your `~/.bashrc` file (if you use bash. The remote nodes use `~/.profile` and the computers in the basement use `~/.bashrc` for some reason.):
  ```
  . $HOME/.OpenFOAM-1.4.1/bashrc
  ```
  Later you will learn how to install OpenFOAM by yourself

- You must now make three directories:
  ```
  mkdir $HOME/OpenFOAM
  mkdir $HOME/OpenFOAM/<CID>-1.4.1
  mkdir $HOME/OpenFOAM/<CID>-1.4.1/run
  ```
  These may be located elsewhere, but this is the default location.
  This is where you will put your case files.

CFD with OpenFOAM

References:

- www.openfoam.org
- OpenFOAM User Guide
- OpenFOAM Programmers Guide
- C++ direkt, Jan Skansholm
- C genom ett nyckelhål, Håkan Strömberg
- An introduction to Computational Fluid Dynamics, H K Versteeg & W Malalasekera

Course homepage:
http://www.tfd.chalmers.se/~hani/kurser/OF_phD_2007