Computer facilities and OpenFOAM installations

- Hopefully you already have Linux and OpenFOAM on your own laptop.
- Your CID accounts are valid in the MT13 Linux computer lab in the M-building and all other Linux computer labs at Chalmers. OpenFOAM-1.5-dev, 1.6.x, and 1.7.x are installed on those machines. There is no `ssh` access to those computers.
- Remote access can be done through:

```
ssh -XY -l <CID> remote1.student.chalmers.se
ssh -XY -l <CID> remote2.student.chalmers.se
ssh -XY -l <CID> remote3.student.chalmers.se
ssh -XY -l <CID> remote4.student.chalmers.se
ssh -XY -l <CID> remote5.student.chalmers.se
```

  (where `<CID>` is your user name at Chalmers)
- Student accounts and temporary accounts only have 1GB disk. You can use /scratch to temporarily store larger files, but they will automatically be removed after 168 hours.
- You should be able to do an `ssh` to your own computing facilities and work there if you like.

Recommended set-up the OpenFOAM environment at Chalmers

At the end of file `~/.bashrc`, type *on one line per alias*:

```
alias OF15dev='export FOAM_INST_DIR=/chalmers/sw/unsup64/OpenFOAM;
              . /chalmers/sw/unsup/OpenFOAM/OpenFOAM-1.5-dev/etc/bashrc'
alias OF17x='export FOAM_INST_DIR=/chalmers/sw/unsup64/OpenFOAM;
              . /chalmers/sw/unsup/OpenFOAM/OpenFOAM-1.7.x/etc/bashrc'
```

Also make `~/.profile point at ~/.bashrc`:

```
ln -s ~/.bashrc ~/.profile
```

This makes everything work when doing remote login.

Open a new terminal window and type the following:

```
OF17x
which icoFoam
```

You should get:

```
/chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.7.x/applications/bin/linux64GccDPOpt/icoFoam
```

Create the run-directory and find documentation

- In a terminal window, type:
  `mkdir -p $HOME/OpenFOAM/$USER-1.7.x/run`
  This is where you will put your case files.
  They may be located elsewhere, but this is the default location.

- The OpenFOAM documentation can be found by typing either of the following:

```
acroread $WM_PROJECT_DIR/doc/Guides-a4/UserGuide.pdf
acroread $WM_PROJECT_DIR/doc/Guides-a4/ProgrammersGuide.pdf
```

Now we will go through the `icoFoam/cavity` tutorial, in the UserGuide, in detail, to show the basics of using OpenFOAM solvers.