Download at www.openfoam.org and prepare patch

- Follow the instructions at www.openfoam.org to download and unpack the *Source pack* and the *third party binary packs*. Skip the OpenFOAM binary packs, since we will compile all of OpenFOAM ourselves.

- Check at `http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/branches/OpenCFD_Release/` if there is a patched version of the code.

- If there is a patched version, open `http://openfoam-extend.wiki.sourceforge.net/Subversion_Guidelines`, click on `Guidelines for using OpenFOAM-extend` and scroll down to `Use Cases`. Do the instructions under
`"User only wants to patch his OpenCFD distribution of OpenFOAM with bug-fi`
In this case go to `http://sourceforge.net/projects/openfoam-extend/` and click `Code/SVN Browse/branches/OpenCFD_Release/OpenFOAM-1.4.1_patch`.
Now click on `updateFromRepositoryAndCompile.sh` and download it to your `OpenFOAM/OpenFOAM-1.4.1` directory. Make sure that it has this name and that it is executable (otherwise `chmod +x updateFromRepositoryAndCompile.sh`)

Patch and compile

- Make sure that you have the correct `WM_PROJECT_INST_DIR` path in your `OpenFOAM/OpenFOAM-1.4.1/.OpenFOAM-1.4.1/cshrc` (or `bashrc`) file.
- `cd` to `OpenFOAM/OpenFOAM-1.4.1`and type: `./updateFromRepositoryAndCompile.sh &` to patch and compile. (approx 5h)
- `svnPatch.log` shows what has been patched.
- `recompile.log` shows the output from the compilation
- You can re-do this procedure every now and then to keep updated with the recent patches. Then only the patched files will be re-compiled.

Make paraFoam work

- Unfortunately `paraFoam` does not work if you compiled from scratch.
- `paraFoam` needs the `libPVFoamReader.so` dynamic library to be compiled, and to compile that you need `cmake`, and if you are unlucky you need to recompile all of `paraview` as well. We will not do that here.
- Here we use a quick solution: copy the pre-compiled `libPVFoamReader.so` from the OpenCFD distribution to make paraFoam work. You can find it in `OpenFOAM/OpenFOAM-1.4.1/lib` in the central installation here at Chalmers.

Make FoamX work

- Unfortunately `FoamX` does not work if you compiled from scratch.
- `FoamX` uses `MICO`, which does not compile automatically in our system for some reason.
- Here we use the `MICO` version distributed with OpenFOAM-1.2, which worked perfectly. In the central installation you can see that there are two `mico` versions in `OpenFOAM/OpenFOAM-1.4.1/src`. To use version `2.3.11`, copy that directory to your own installation and modify the `mico` version number in `OpenFOAM/OpenFOAM-1.4.1/.cshrc`. Re-source!
- If you copy `MICO` from OpenFOAM-1.2 yourself you have to do some more things to make it work:
  Rename `src/mico-2.3.11/platforms/linuxGcc4Opt` to `linuxGccDPOpt` (1.4.1 style)
  Remove `$FOAMX_CONFIG/ns.ref` if it exists.
  Remove `$FOAMX_CONFIG/HostBrowserLog.xml` if it exists.
  Remember to modify the version number in the `.cshrc`-file (see above), and to re-source!
- If someone knows how to compile `MICO` in our system, please tell me!