Download and compile the bug-fixed version

- We will now learn how to download bug-fixed source files and compile them.
- Download the bug-fixed **source files** using the following commands:

  ```
  mkdir -p /scratch/$USER/OpenFOAM
  cd /scratch/$USER/OpenFOAM
  rm -rf OpenFOAM-1.6.x
  git clone git://repo.or.cz/OpenFOAM-1.6.x.git
  ```
  (if needed) applies to the `rm -rf` line.

- Use the pre-installed `ThirdParty-1.6.x`, similar to what we did before.
- Source as before, but now for `1.6.x`.
- Compile by doing:

  ```
  cd /scratch/$USER/OpenFOAM/OpenFOAM-1.6.x
  ./Allwmake >& log_Allwmake &
  ```

  and wait 5 hours... done! Now you have compiled all of OpenFOAM!
  Note that usually all the trouble you have installing OpenFOAM has to do with Third Party products!

Use another compiler

- Change `Gcc` in `etc/bashrc` to something else.
- Point at that compiler in `etc/settings.sh`.
- Modify in `wmake/rules/` for that specific compiler.
- Output directory names will be according to the name decided above, so it will not interfere with the Gcc compilation.

OpenFOAM-extend at SourceForge

- In the OpenFOAM-extend project at SourceForge you can find an extended version of OpenFOAM, the `OpenFOAM-1.5-dev` version.
  `http://openfoam-extend.wiki.sourceforge.net`
  `http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/\`
  `trunk/Core/OpenFOAM-1.5-dev/`
- Check out the source code using the Subversion system:
  ```
  svn co http://openfoam-extend.svn.sourceforge.net/svnroot/\
  openfoam-extend/trunk/Core/OpenFOAM-1.5-dev/
  ```
- Use the same Third Party Products as `1.5`
- Compile similar as `1.5`
- The `dev` version is developed by Hrvoje Jasak, and **not** by OpenCFD.