Compilation of all, or part, of OpenFOAM

Compilation of all, or part, of OpenFOAM

The compilation process for all of OpenFOAM consists of the following:

- Source the `bashrc` file to set up the OpenFOAM environment.
- Type `foam` to go to the installation directory.
- Type `./Allwmake` to compile all of OpenFOAM.

Problems are usually related to Qt and Paraview.

The compilation process for an application consists of the following:

- Locate the application you want to compile.
- Go to that directory.
- Make sure that there is a `Make` directory.
- Copy the application directory to your user directory.
- Go to your copy.
- Rename the executable name and location in `Make/files`.
- Type `wmake` to compile the application.