A first look at the source code of applications

Finding the source code of the applications in OpenFOAM

- The source code for the applications is arranged in a structure that is useful for finding the application you need.
- Use the pre-defined alias `app` to go to the applications directory: `$FOAM_APP`
- You will find: `Allwmake solvers test utilities`
  (No `test` in foam-extend-3.1, but instead a `bin`)
  (In 2.4.x, `bin` is instead here: `$WM_PROJECT_DIR/platforms/$WM_OPTIONS`)
- `Allwmake` is used to compile all the applications.
- `solvers` contains the source code of the solvers.
- `utilities` contains the source code of the utilities.
- `test` contains source code for testing specific features of OpenFOAM.

Solvers in OpenFOAM

- In `$FOAM_SOLVERS` (use alias `sol` to go there) you find the source code for the solvers arranged according to (version-dependent):

```
basic           discreteMethods    financial        lagrangian
combustion      DNS                heatTransfer     multiphase
compressible    electromagnetics   incompressible   stressAnalysis
```

- In sub directory `incompressible` you find the solver source code directories:

```
boundaryFoam    nonNewtonianIcoFoam   pisoFoam
channelFoam     pimpleDyMFoam         shallowWaterFoam
icoFoam         pimpleFoam            simpleFoam
```

- Inside each solver directory you find a `*.C` file with the same name as the directory. This is the main file, where you will find the top-level source code and a short description of the solver. For `icoFoam`:

```
Transient solver for incompressible, laminar flow of
Newtonian fluids.
```

  For a more complete description, you have the source code right there.

Utilities in OpenFOAM

- In `$FOAM_UTILITIES` (use alias `util` to go there) you find the source code for the utilities arranged according to (version-dependent):

```
errorEstimation  parallelProcessing  surface
mesh             postProcessing      thermophysical
miscellaneous    preProcessing
```

- In sub directory `postProcessing/velocityField` you find:

```
Co         flowType  Mach  Q               uprime
enstrophy  Lambda2   Pe    streamFunction  vorticity
```

- Inside each utility directory you find a `*.C` file with the same name as the directory. This is the main file, where you will find the top-level source code and a short description of the utility. For `vorticity`:

```
Calculates and writes the vorticity of velocity field U.
The -noWrite option just outputs the max/min values without writing
    the field.
```

A quick look at the icoFoam solver directory

- The `icoFoam` solver source code is located in `$FOAM_SOLVERS/incompressible/icoFoam` where you can find two files, `createFields.H` and `icoFoam.C`, and a `Make` directory. (There is also a `icoFoam.dep` file, which is generated when compiling)
- The `Make` directory contains two files, `files` and `options`, that specifies how `icoFoam` should be compiled. (The `linux*` directories are generated when compiling)
- We will have a look at the code later.