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: `$FOAM_APP`
- You will find: `Allwmake solvers  test  utilities`
  (No `test` in 1.6-ext, but instead a `bin`)
  (In 2.1.x, `bin` is instead here: `$WM_PROJECT_DIR/platforms/$WM_OPTIONS`)
- `Allwmake` is used to compile all the applications.
- `bin` contains the binaries of the applications after compilation.
- `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.
```

icoFoam/cavity tutorial - The icoFoam solver

- 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)
- In `icoFoam.C` you basically see a `runTime` loop, the specification and solution of the `UEqn` coefficient matrix, and the `PISO` loop.
- In `createFields.H` the kinematic viscosity, the velocity field, and the pressure fields are read from the `startTime` directory. The face convections, `phi` are computed if they are not available in the `startTime` directory. Finally, the reference pressure is set if there are no constant pressure boundary conditions.