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: `$FOAM_APP`
- You will find: `Allwmake solvers  test tools  utilities`
  (No `test` in foam-extend, but instead a `bin`)
  (In the ESI version, `bin` is instead here: `$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.
- Have a look yourself, and we can discuss it.

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 (version-dependent):

```
adjointShapeOptimizationFoam  nonNewtonianIcoFoam  shallowWaterFoam
boundaryFoam                  pimpleFoam           simpleFoam
icoFoam                       pisoFoam
```

- 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):

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

- In sub directory `postProcessing` you find:

```
dataConversion   lagrangian   miscellaneous   postProcess
graphics         lumped       noice
```

- 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 `noise`:

```
Utility to perform noise analysis of pressure data.
```

- The number of utilities have been greatly reduced in recent versions, since they are being moved into functionObjects.

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.
- The `Make` directory contains two files, `files` and `options`, that specifies how `icoFoam` should be compiled.
- We will have a look at the code later.