

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: `$FOAM_APP`
- You will find: `Allwmake solvers test utilities`
(No `test` in `foam-extend-3.1`, but instead a `bin`)
(In 2.3.x, `bin` is instead here: `$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):

<code>basic</code>	<code>discreteMethods</code>	<code>financial</code>	<code>lagrangian</code>
<code>combustion</code>	<code>DNS</code>	<code>heatTransfer</code>	<code>multiphase</code>
<code>compressible</code>	<code>electromagnetics</code>	<code>incompressible</code>	<code>stressAnalysis</code>

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

<code>boundaryFoam</code>	<code>nonNewtonianIcoFoam</code>	<code>pisoFoam</code>
<code>channelFoam</code>	<code>pimpleDyMFoam</code>	<code>shallowWaterFoam</code>
<code>icoFoam</code>	<code>pimpleFoam</code>	<code>simpleFoam</code>

- 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):

<code>errorEstimation</code>	<code>parallelProcessing</code>	<code>surface</code>
<code>mesh</code>	<code>postProcessing</code>	<code>thermophysical</code>
<code>miscellaneous</code>	<code>preProcessing</code>	

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

<code>Co</code>	<code>flowType</code>	<code>Mach</code>	<code>Q</code>	<code>uprime</code>
<code>enstrophy</code>	<code>Lambda2</code>	<code>Pe</code>	<code>streamFunction</code>	<code>vorticity</code>

- 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.