How to find solver and utility tutorials in the source code, and how to figure out how to use them

Find solver and utility tutorials in the source code, and figure out how to use them

- There are only written tutorials to some of the applications and utilities in Open-FOAM. Be aware of the *documentation that actually does exist* in the UserGuide and ProgrammersGuide!

- We will now learn how to search the installation for information on how to use solvers and utilities.

- A documentation might be lagging, but the source code is not!

How to search for solver tutorials in the source code

- Type `tut` to go to the `$FOAM_TUTORIALS` directory. Here you find many case-setups for the solvers in OpenFOAM.

- Type:
  `tree -d -L 2 $FOAM_TUTORIALS`
  to get a list of for which solvers there are tutorial cases available.

- Type:
  `tree -d -L 1 $FOAM_TUTORIALS/incompressible/icoFoam`
  to get a list of which tutorial cases are available for the icoFoam solver.

- All the solver tutorials have `Allrun` scripts that describe the use of those tutorials. We will now have a look at the `Allrun` script of the `$FOAM_TUTORIALS/incompressible/icoFoam` tutorials. This is actually what you will do manually when you do the `cavity` tutorials in the UserGuide. In other words, you can use the `Allrun` script as a short summary of the description in the UserGuide.

Run the icoFoam cavity tutorials using the Allrun script (1/7)

(Note that the following description shows the principle. There might be small differences in exactly what is done by the Allrun script between versions.)

In the `icoFoam` tutorial directory there is an `Allrun` script.
When running this script it is preferred to copy the entire directory to your run directory, so that you keep a clean version of the tutorials in the installation directory. Type:
```
cp -r $FOAM_TUTORIALS/incompressible/icoFoam $FOAM_RUN
cd $FOAM_RUN/icoFoam
./Allrun >& log_Allrun&
```

Looking in the Allrun script, you can see a list of cases that will be executed:
```
cavityCases="cavity cavityFine cavityGrade cavityHighRe cavityClipped"
```
Some of those cases are actually created by the script.
At the end of the script it also runs the `elbow` case.
The script contains Linux commands and calls for OpenFOAM applications in order to set up and run the simulations.

Run the icoFoam cavity tutorials using the Allrun script (2/7)

The `Allrun` script for the icoFoam cavity tutorials actually
first runs the cavity case

```
#Running blockMesh on cavity:
blockMesh
#Running icoFoam on cavity:
icoFoam
```

Run the icoFoam cavity tutorials using the Allrun script (3/7)

The `Allrun` script for the icoFoam cavity tutorials actually
then runs the cavityFine case:

```
#Cloning cavityFine case from cavity:
mkdir cavityFine
cp -r cavity/{0,system,constant} cavityFine
    [change "20 20 1" in blockMeshDict to "41 41 1"]
    [set startTime in controlDict to 0.5]
    [set endTime in controlDict to 0.7]
    [set deltaT in controlDict to 0.0025]
    [set writeControl in controlDict to runTime]
    [set writeInterval in controlDict to 0.1]
#Running blockMesh on cavityFine
blockMesh
#Running mapFields from cavity to cavityFine
mapFields -case cavity -sourceTime latestTime -consistent
#Running icoFoam on cavityFine
icoFoam
```

Run the icoFoam cavity tutorials using the Allrun script (4/7)

The `Allrun` script for the icoFoam cavity tutorials actually
then runs the cavityGrade case:

```
#Running blockMesh on cavityGrade
blockMesh
#Running mapFields from cavityFine to cavityGrade
mapFields -case cavityFine -sourceTime latestTime -consistent
#Running icoFoam on cavityGrade
icoFoam
```

Run the icoFoam cavity tutorials using the Allrun script (5/7)

The `Allrun` script for the icoFoam cavity tutorials actually
then runs the cavityHighRe case:

```
#Cloning cavityHighRe case from cavity
mkdir cavityHighRe
cp -r cavity/{0,system,constant} cavityHighRe
#Setting cavityHighRe to generate a secondary vortex
    [set startFrom in controlDict to latestTime;]
    [set endTime in controlDict to 2.0;]
    [change 0.01 in transportProperties to 0.001]
#Copying cavity/0* directory to cavityHighRe
cp -r cavity/0* cavityHighRe
#Running blockMesh on cavityHighRe
blockMesh
#Running icoFoam on cavityHighRe
icoFoam
```

Run the icoFoam cavity tutorials using the Allrun script (6/7)

The `Allrun` script for the icoFoam cavity tutorials actually
then runs the cavityClipped case:

```
#Running blockMesh on cavityClipped
blockMesh
#Running mapFields from cavity to cavityClipped
cp -r cavityClipped/0 cavityClipped/0.5
mapFields -case cavity -sourceTime latestTime
    [Reset the boundary condition for fixedWalls to:]
    [        type            fixedValue;            ]
    [        value           uniform (0 0 0);       ]
    [      We do this since the fixedWalls got       ]
    [   interpolated values by cutting the domain    ]
#Running icoFoam on cavityClipped
icoFoam
```

Run the icoFoam cavity tutorials using the Allrun script (7/7)

The `Allrun` script for the icoFoam cavity tutorials actually
finally runs the elbow case

Now, open each case with `paraFoam` and have a look.

Run ALL the tutorials using the Allrun scripts

We will not do this now!

- You can also run another `Allrun` script, located in the `$FOAM_TUTORIALS` directory. This script will run through ALL the tutorials (calls `Allrun` in each solver directory).

- You can use this script as a tutorial of how to generate the meshes, how to run the solvers, how to clone cases, how to map the results between different cases etc.

- Again, I suggest that you copy the files to your run directory:

```
cp -r $FOAM_TUTORIALS $FOAM_RUN
```

Finding tutorials for the utilities in OpenFOAM

- There are no 'case' tutorials for the utilities, but we can search for examples:

```
find $WM_PROJECT_DIR -name \*Dict | grep -v blockMeshDict | grep -v controlDict
```

You will get a list of example dictionaries for the utilities that use a dictionary. Some of those examples can be found next to the source code of each particular utility, and some are also used in the solver tutorials. The ones that don't use a dictionary are usually easier to learn how to use, in particular when using the `-help` flag.

Now you should be ready to go on exploring the applications by yourself.

More tutorials can be found in

- The UserGuide

- The ProgrammersGuide, chapter 3

- The OpenFOAM Wiki
  (e.g. the Turbomachinery Working Group)

- The OpenFOAM Forum

- The OpenFOAM Workshop trainings

Questions

- Where can you find the UserGuide and ProgrammersGuide, in your installation and on the Internet?

- For which solvers are there detailed written tutorials available in the UserGuide and ProgrammersGuide? Search in the list of contents.

- Where in the installation can you find examples of how to set up cases for the solvers? For the utilities?

- What does this Linux command mean?

```
find $WM_PROJECT_DIR -name \*Dict | grep -v blockMeshDict | grep -v controlDict
```

- The `foamDataToFluent` utility requires a dictionary. What Linux command helps you find examples of such dictionaries? Hint: All dictionaries end with `Dict`. Use RegExp (Google it: Wikipedia) to make Linux narrow your search results as much as possible.