

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 OpenFOAM. 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.

## 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?
- List the sequence of Linux commands that are executed through the `Allrun` script of the `icoFoam` tutorials.
- 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.