Notes on OpenFOAM cases

Daniel Duque
Dep. Ciencias Aplicadas a la Ingeniería Naval
ETSIN, UPM

October 11, 2012

Contents

Chapter 1

Setup

1.1 Installation

This method describes the installation of `openFOAM 2.1.1` onto `openSUSE linux 12.2` from the source code. See [1]. The procedure will be slightly different for other cases.

1. Make sure you have installed the C/C++ development packages during installation, or through the package manager (YaST in openSUSE).

2. Create a new folder named `OpenFOAM` on your home folder.

3. Obtain the tgz source files from the official download site. There are two of them: `OpenFOAM-2.1.1.tgz` and `ThirdParty-2.1.1.tgz`. Save them in the directory created in the previous step.

4. Uncompress both files with some file manager, or running `tar zxvf`.

5. From a terminal window, change to the directory created by the uncompression (`cd ~/OpenFOAM/OpenFOAM-2.1.1`). Then, run `source etc/bashrc`.

6. run `./bin/foamSystemCheck` to check if the system is OK.

7. run `./Allwmake`, then let your PC compile for several hours.

8. This completes the installation of `openFOAM`.

Next, some third party software must also be compiled. For these, one must install the `gnuplot`, `cmake`, and `libqt4-devel` packages (the later actually implies many others). Right after the previous steps, and in the same terminal (otherwise, run `source etc/bashrc` again), do the following

1. Change to proper directory with `cd ~/OpenFOAM/ThirdParty-2.1.1`, or `cd $WM_THIRD_PARTY_DIR`
2. One of the files needs a small modification in openSUSE and fedora, so one must run

   ```
   sed -i -e
   's/ClearAndSelect = Clear | Select /
   ClearAndSelect =
   static_cast<int>(Clear) | static_cast<int>(Select)/'
   ParaView-3.12.0/Qt/Core/pqServerManagerSelectionModel.h
   ```

3. run `./makeParaView`
4. Then compile the plugins by changing directory:

   ```
   cd \$FOAM_UTILITIES/postProcessing/graphics/PV3Readers
   ```

5. run `wmSET`
6. then, `./Allwclean`
7. finally, `./Allwmake`

Check the installation running `~/OpenFOAM/OpenFOAM-2.1.1/bin/foamInstallationTest`.

Finally, for the following we will be using the tutorials included in openFOAM, so make a copy of them first:

```
cd
mkdir -p $FOAM_RUN
cp -r $FOAM_TUTORIALS $FOAM_RUN
```

1.2 Session setup

This command must be issued every time a new session is opened:

```
. /opt/OpenFOAM/OpenFOAM-1.7.1/etc/bashrc
```

(mind the dot, which is equivalent to `source`).

This is convenient for sporadic use of openFOAM. If more the use is more frequent, it is convenient to run the following only once:

```
echo ". ~/OpenFOAM/OpenFOAM-2.1.1/etc/bashrc" >> ~/.bashrc
```

This way, all the setup is carried out whenever a terminal session is opened.

Chapter 2

Lid-driven cavity flow

The main reference is [2].

2.1 Setup

```
cd $FOAM_RUN/tutorials/incompressible/icoFoam
./Allclean
cd cavity
less constant/polyMesh/blockMeshDict
blockMesh
checkMesh
less 0/p
less 0/U
less constant/transportProperties
less system/controlDict
paraFoam
```

2.2 Viewing the mesh

- Run `paraFoam &`. Notice the ampersand to keep the process running in the background. If the later is forgotten, one may type ctrl + z, then **bg** to send the process to the background.
- You can see the case `cavity.OpenFOAM` is already selected in the `Pipeline Browser` tool.
- In the `Object Inspector` tool, `Properties` tab, check the grayed `Mesh Parts` button to select all regions.
- Click `Apply`. Not much is seen, just a bounding box.
- In the `Display` tab, scroll down to find the `Style` menu. Select `Wireframe` for the `Representation` option.

- The mesh can now be inspected. It can be rotated with the mouse left button.

2.3 Running the simulation

Return to the terminal, then type

icoFoam

This just takes a second.

2.4 Post-processing

The new simulated data is automatically piped to paraFOAM. Let us have a look at the pressure. In the Display tab, in the Color section, select the p field in the Color by selector. The grey value we see is just the constant value of the pressure at $t = 0$.

In the green arrows at the VCR controls toolbar (just below the menus), go to the later time, 2 to reveal a beautiful high-pressure area in the upper-right corner, and a low-pressure one in the upper-left one (click on Rescale to Data Range if this is not shown).

Actually, there are two p fields: the one with the cube is the actual values at the mesh (hard science), whereas the one with the dot means interpolated values (nicer plots).

A "movie" of the animation may be played clicking on the right-arrow symbol in the VCR controls toolbar. (Too fast!).

The default color map goes from "cold" to "hot": blue to white to red. A color scale can be switched on and off with the little color scale button in the Active Variable Controls toolbar. Other maps may be chosen with the neighboring button. Notice this toolbar also provides a shortcut to the fields to be plotted.

2.4.1 Velocity field

In order to plot the velocity field, a "Glyph filter" should be created. The easier way is by selecting the Glyph symbol in the Common toolbar — It looks like a sphere with little balls on it. Click Apply in the Properties tab of the Object Inspector. The velocity field is automatically loaded onto the pressure field. Hide the later by clicking on the eye symbol next to cavity. OpenFOAM in the Pipeline Browser. The background color may be changed in Edit, Settings, Colors tab, Background Color (a solid black is quite good for a computer screen).

The arrows are scaled by the "vector" Scale Mode. If this is set to "off" in the Properties tab, all arrows have the same length (changeable in the Set Scale Factor box), which may be better for visualization. Do not forget to click on the green Apply button after any change is made.

The vectors are colored by the p value. One can change to other color in the the Display tab, e.g. “GlyphVector”, “Magnitude” (its module).

2.4.2 Streamlines

Apply a Stream Tracer filter (looks like a cylinder with flow around it in the Common toolbar). Make sure to select cavity. OpenFOAM in the Pipeline Browser, not Glyph1. To seed several tracers starting from a common line, select a Seed **type** given by a Line Source. Drag the two gray balls to the desired value, or click “P” to put them where the mouse is. Make sure z is around 0.005 for both points. Use around 20 point only, and play with either the Maximum100 Step Length or Maximum Steps in the Maximum Stream Tracer section, then click Apply.

Nicer paths may be obtained by applying a Tube filter onto the StreamTracer1 filter. This filter is found in Filters, Filters, Alphabetical (or Recent, in later runs).

Chapter 3

Couette flow

See [3]. For this, you will need the "canal" tutorial files (contact the author in order to get them).

3.1 Setup

```
cd $FOAM_RUN/tutorials/canal/Couette
./clean.sh
less constant/polyMesh/blockMeshDict
```

Notice the mesh definitions are still rather simple, even though we have named a wall "inlet" and the opposing one "outlet".
Next, check:

```
less 0/p
less 0/U
```

The pressure file is rather simple, with zeroGradient at all relevant walls. The velocity one is similar, but with fixed values of the velocity at the two moving walls.
Finally, set up the mesh, and view it:

```
blockMesh
checkMesh
paraFoam &
```

Run the simulation by typing icoFoam.

3.2 Post-processing

View the resulting pressure and velocity fields as before. This time we also have a theoretical prediction for the velocity field we can check.

Select Filters, Data Analysis, Plot Over Line. Drag the two end points around, or click "P" to place them where the mouse pointer is. Make sure both values of z are between 0 and 0.01, e.g. 0.005. Click Apply. The velocity field should be linear in the steady state.

Chapter 4

Poiseuille flow

This is again in the "canal" files.

4.1 Setup

```
cd $FOAM_RUN/tutorials/canal/Poiseuille
./clean.sh
less constant/polyMesh/blockMeshDict
```

Notice the simulation box is a rather simple, and almost identical to the Couette case.

Now, have a look at

```
less 0/p
less 0/U
```

You will see the boundary conditions for p and U are linked [4]:

- **Inlet:** Fixed value $p = 1$, U of type pressureInletVelocity, initially set at 0. According to the guide (sec. 5.2.4 of [5] or sec. 6.2.4 of [6]), the later type applies "when p is known at inlet, U is evaluated from the flux, normal to the patch".

- **Outlet:** Fixed value $p = 0$, U of type zeroGradient.

In a nutshell, we now specify a *pressure drop* across the system.

As usual by now, set up the mesh, and view it:

```
blockMesh
checkMesh
paraFoam &
```

Then, run the simulation by typing `icoFoam`.

4.2 Post-processing

As in the Couette case, select `Filters`, `Data Analysis`, `Plot Over Line`. Drag the two end points around, or click "P" to place them where the mouse pointer is. Make sure both values of z are between 0 and 0.01, e.g. 0.005. Click Apply. The velocity field should be quadratic in the steady state (also, in intermediate ones).

Chapter 5

Dam break

This case is included in the openFOAM tutorial collection [7].

5.1 Setup

By now this lest of commands should be quite clear:

```
cd $FOAM_RUN/tutorials/multiphase/interFoam/laminar
./Allclean
cd damBreak
less constant/polyMesh/blockMeshDict
blockMesh
checkMesh
```

Have a look at the mesh with `interFoam&`. Notice the mesh is *refined* at some parts, like the botton and in the middle. This is specified in the "blocks" section of blockMeshDict, where we can see some hexes get more boxes than others. Also, the top wall is called "atmosphere".

Next, have a look at the files containing the physical constants. We now have a hydrostatic pressure, set to $p_0 = 0$ at the atmosphere:

```
less 0/p_rgh
```

The velocity field is quite clear:

```
less 0/U
```

Of course, we now have gravity:

```
less constant/g
```

There is now an extra field, the volume-of-fluid (VOF), or α field, which is set to 0 at the air, and 1 at the liquid:

```
cp 0/alpha1.org 0/alpha1
less system/setFieldsDict
setFields
```

The first line uses a backup file (org must stand for "original"), the last line calls a script to assign the initial value of the α field. The script is controlled by the `setFieldsDict` file. Have a look at the initial VOF field with `interFoam&` (make sure you select the alpha1 field in the "Volume Fields" area of the Object Inspector – it is not loaded by default). The red zone, with value 1, is the water column that will move.

Another features which we have not discussed thus far are the transport properties, such as the viscosities and densities of the two fluids:

```
less constant/transportProperties
```

There are many turbulence models implemented in openFOAM, but for the time being:

```
less constant/turbulenceProperties
```

we are only modelling lamilar flow.

Finally, run

```
interFoam
```

which is the "Solver for 2 incompressible, isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach", sec. 3.5 of [5].

5.2 Post-processing

In `paraFoam`, be sure to select the `alpha1` field. This way, a movie of "the water" moving can be played. Actually, an "animation" of the action can be saved, but this consists of several jpg snapshots that must be linked into a proper movie with some external software.

Notice this dam break problem has an extra obstacle to spice things up. An exercise would be to remove this obstacle in order to simulate a more standard dam break problem.

Chapter 6

Potential flow around cylinder

This covers only the well-known potential solution for the laminar flow around a cylinder [8, 9].

6.1 Setup

In the latest version of openFOAM, because of security concerns, the following must be run once:

```
mkdir -p  ~/.OpenFOAM/$WM_PROJECT_VERSION/
echo -e "InfoSwitches\n{\n␣allowSystemOperations␣1;\n}" >> \
    ~/.OpenFOAM/$WM_PROJECT_VERSION/controlDict
```

Then,

```
cd $FOAM_RUN/tutorials/basic/potentialFoam
./Allclean
cd cylinder
less constant/polyMesh/blockMeshDict
```

As usual, set up the mesh, and view it:

```
blockMesh
cp -r 0.org/ 0
checkMesh
paraFoam
```

The second line is just to use the back-up directory for time 0. Inspect the resulting mesh carefully. Notice the `symmetryPlane` entries in `blockMeshDict`.

Next, find them in:

```
less 0/p
less 0/U
```

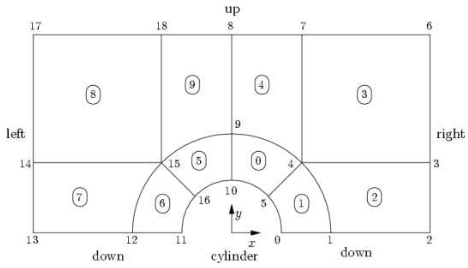

Figure 6.1: Blocks in cylinder geometry, from Ref. [5]

Run the simulation by typing `potentialFoam`. Or, use `./Allrun`, which does all of the above automatically (including the obscure first step).

Chapter 7

Flow around cylinder

7.1 Setup

```
cd $FOAM_RUN/tutorials/canal/cylinder
./clean.sh
less constant/polyMesh/blockMeshDict
blockMesh
checkMesh
paraFoam &
```

View the resulting mesh carefully. This has been borrowed from the previous (potential) example, Figure 6.1. Notice that a symmetry plane is imposed, which may be unrealistic for certain flow patterns (such as vortex shedding). Notice again the `symmetryPlane` entries in `blockMeshDict`.

Notice the pressure is fixed on the right, with zero gradient on the left. Velocity, on the other hand, is $(1, 0, 0)$ on the left, zero gradient on the right — similarly to the Poiseuille case. Other boundary conditions may be explored.

Run the simulation by typing `icoFoam`. Inspect the resulting pressure and velocity fields. Seed tracer particles on the right to inspect the flow pattern and its features (vortices, stagnation points).

Change ν in `constant/transportProperties` to explore the onset of vortex shedding.

7.2 References

The main references for openFOAM are:

- Its User Guide; there are actually two of them, with some differences: Ref. [5] and Ref. [6]
- The Programmer's Guide [10]
- An interesting unofficial wiki [11]

- CFD-online provides an interesting reference wiki and a forums section. In the later, there is a topic devoted to OpenFOAM issues. Subtopics: News & Announcements, Installation, Meshing & Mesh Conversion, Pre-Processing, Running, Solving & CFD, Post-Processing, Programming & Development, Verification & Validation, Bugs. [12].

Bibliography

[1] Opensuse installation notes. online.

[2] Lid-driven cavity in OpenFOAM user guide. online.

[3] Couette flow boundary conditions, discussion in CFD-online. online.

[4] Question about Poiseuille flow in CFD-online. online.

[5] OpenFOAM 2.1 user guide. online.

[6] OpenFOAM 2.3 user guide. online.

[7] Dam break case, user guide. online.

[8] Cylinder case in the OpenFOAM Programmer's Guide. online.

[9] Cylinder exercise in sc'09 education program. online.

[10] OpenFOAM programmer's guide. online.

[11] OpenFOAM wiki. online.

[12] CFD-online OpenFOAM topic in forums. online.