My First Tutorial in OpenFOAM

Federico Piscaglia *

Dept. of Aerospace Science and Technology (DAER), Politecnico di Milano, Italy

Abstract. The present is a short lab handout with the main commands and steps used to set up and run the OpenFOAM 'cavity' tutorial.

1 Learning outcome

The software used is the open-source CFD software OpenFOAM®-8 by OpenCFD®. In this section you will learn how to:

- Generate a mesh using blockMesh and check the result

- Run a simple solver (`icoFoam`) saving the output for later reference

- Post-process velocity fields with ParaView

- Change setup and run the case

2 Baseline case

2.1 Setup

1. Open a terminal and enter the 'run' folder:

   `user@host:~$ run`

2. Copy the entire tutorials folder into your run

   `user@host:run$ cp -r $FOAM_TUTORIALS .`

3. enter the 'cavity' folder:

   `user@host:run$ cd tutorials/incompressible/icoFoam/cavity/cavity`

4. Generate the mesh (and write screen output on file):

   `user@host:cavity$ blockMesh > log.blockMesh`

5. Check the mesh quality

   `user@host:cavity$ checkMesh > log.checkMesh`

6. See output of 'checkMesh' ('q' to return to shell):

   `user@host:cavity$ less log.checkMesh`

---

*Tel. (+39) 02 2399 8620, E-mail: federico.piscaglia@polimi.it

2.2   Run the solver

1. Compute solution
   `user@host:cavity$ icoFoam > log.icoFoam`

2.3   Postprocessing

1. Open ParaView:

     - if you have compiled ParaView: `user@host:cavity$ paraFoam`
     - if you have not: `user@host:cavity$ paraFoam -builtin`
     - if you wish to use precompiled ParaView for Windows:
         i. create an empty file: `paraFoam -builtin -touch`
         ii. open ParaView for Win
         iii. go to the tutorials folder and select 'cavity/cavity.foam'

2. Generate a vector glyph:

     i select main source, either 'cavity.foam' or 'cavity.OpenFOAM'
     ii Filters → Glyph
     iii In the 'vectors' drop-down list, select U (cell-based)
     iv In the 'scale mode' list, select 'vector'
     v Click 'Apply'
     vi Select wireframe representation

3. Generate streamlines:

     i select main source, either 'cavity.foam' or 'cavity.OpenFOAM'
     ii Filters → Cell centers
     iii Filters → Mask Points
     iv Choose 50 points; activate the 'advanced options' (gear icon) and select 'random sampling'
     v Filters → Alphabetical → Stream tracer with custom source
     vi Select as 'input' the main source and as 'seed source' 'MaskPoints1'
     vii Click 'Apply'

4. Extract data along a line

     i Select main source
     ii Filters → Plot Over Line
     iii Align the line along the desired direction (e.g. Y-axis), by manually dragging points, enter endpoints coordinates or using predefined directions
     iv Click on 'Apply'
     v You can display/undisplay fields using checkboxes

3   Hands-on

It is a good practice to keep a copy of the original case when changing the settings:

1. Go up one level: `user@host:cavity$ cd ../`

2. Copy the folder: `user@host:cavity$ cp -r cavity cavityFiner`

3. Enter the new case folder: `user@host:cavity$ cd cavityFiner`

4. Clear old results: `user@host:cavityFiner$ rm -r 0.?`

3.1 Refine the mesh

1. Open 'system/blockMeshDict' with a text editor

2. Increase the number of cells along X- and Y- direction (e.g. 20 ⇒ 100)

3. Save and close

4. Re-generate the mesh: `user@host:cavityFiner$ blockMesh`

5. Check the new mesh: `user@host:cavityFiner$ checkMesh > log.checkMesh`

6. Run the solver: `user@host:cavityFiner$ icoFoam > log.icoFoam`

The solver has crashed. Why?

Hint: read the log of icoFoam and check the maximum Courant number. It must remain below 1. Co is defined as:

$$\text{Co} = \frac{U\Delta t}{\Delta x}$$

3.2 Make the case 3D

Assignment Modify the case to make it 3D and solve it.

Hint

- Modify the 'blockMeshDict' by putting more than 1 cells in Z- direction
- Set 'frontAndBack' patches to 'wall' instead of 'empty' in blockMeshDict, '0/p' and '0/U' files.