Case Study # 2: OpenFOAMLid-Driven Cavity Flow

John Karasinski
Graduate Student Researcher
Center for Human/Robotics/Vehicle Integration and Performance
Department of Mechanical and Aerospace Engineering
University of California
Davis, California 95616
Email: karasinski@ucdavis.edu

1 Problem Description

The flow of an incompressible fluid induced in a square cavity by a moving upper boundary, or "lid-driven" cavity flow, is a classic validation case for Computational Fluid Dynamics (CFD) solvers. The top wall moves in the *x*-direction at a speed of 1 m/s while the other 3 are stationary. This case study involves performing numerical simulations of this flow problem for various flow regimes using an Open Source PDE solver: OpenFOAM [1]. Initially, the flow will be assumed laminar and will be solved on a uniform mesh using the icoFoam solver for laminar, isothermal, incompressible flow. This problem is solved for a 129 x 129 uniform grid with Reynolds numbers of 100, 400, 1000, and 3200, and a 257 x 257 grid with Reynolds numbers of 5000, 7500, and 10000 and are analytically compared to previous results [2].

2 Solver Setup

The installation of OpenFOAM is a nontrivial process. While official online documentation lists installation instructions for choice Linux distributions, there is no official documentation for installing OpenFOAM on OS X. Fortunately, some users have documented their installations and made the steps available online [3].

OpenFOAM 2.3.0 can be built on OS X 10.9 fairly simply by making use of Homebrew (a free/open source software package management system that simplifies the installation of software on the OS X). Homebrew, when used alongside additional command lines tools such as curl and git, greatly simplifies the installation of OpenFOAM. Installation instructions are available in Appendix A. Before running these instructions, however, one must first install Paraview. Paraview has readibly avaialble precombiled binaries which can be installed in a normal fashion.

To confirm that OpenFOAM has been installed correctly, the guide recommends running the lid-driven cavity flow example which comes installed with OpenFOAM [3]. After the files are copied into a local directory, the example can be run via the following terminal commands:

```
$ blockMesh
$ icoFoam
```

which should result in a large amount of text being outputted to the screen. The user can then run the paraFoam command to view the resulting simulation data.

3 Numerical Solution

Numerical solutions were found for the lid-driven cavity flow problem for a 129 x 129 uniform grid with Reynolds numbers of 100, 400, 1000, and 3200, and a 257 x 257 grid with Reynolds numbers of 5000, 7500, and 10000. To do this, the OpenFOAM icoFoam example, 'cavity' was used.

The initial mesh was set to be 129 x 129 grid points (by adjusting line 33 of the blockMeshDict file) and was generated with the blockMesh command. To insure that the simulation runs to convergence (and given the quickness at which the simulation runs), the simulation end time is set to 30 seconds (line 26 of the controlDict file). Other default values were used, and the only values changed between runs were *Re*, the Reynolds number, and the step time, deltaT. This is done by adjusting ν, the kinematic viscosity, which relates to the Reynolds number via:

$$Re = \frac{d|U|}{\nu},\qquad(1)$$

where $d = 0.1$m is the characteristic length, and $|U| = 1$ ms^{-1} is the characteristic velocity. A Reynolds number of 100, for instance, can be achieved by setting $\nu = 0.001$m^{2}s^{-1} (line 18 of the transportProperties file). For both of the 129 x 129 grid cases, a step time of 0.005 was used. For the 257 x 257 grid with a Reynolds number of 5000, a time step of 0.001 was used, while the simulation with a Reynolds number of 10000 required a time step of 0.0001. After generating the grid and adjusting other necessary parameters, the

simulation can be run via the icoFoam command. Once the simulation has finished running, the streamfunction can be generated with the streamFunction command.
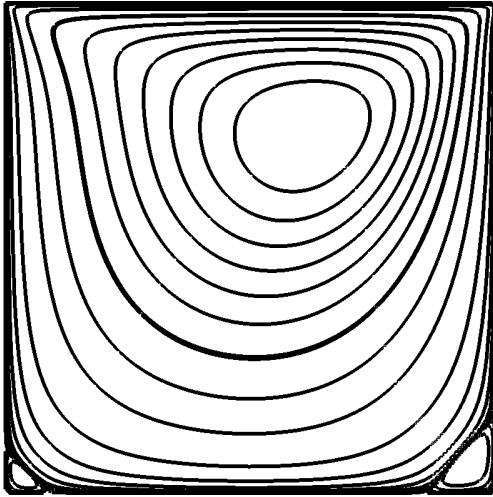

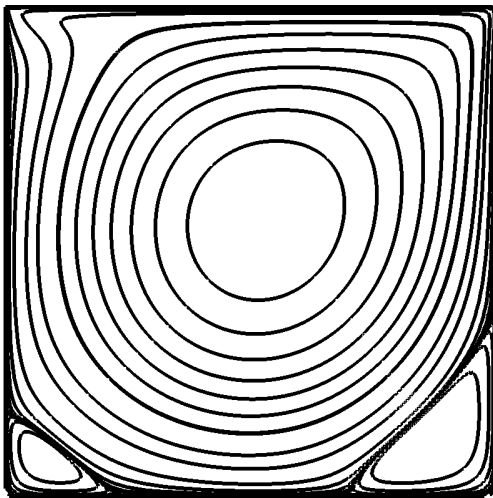Fig. 1. $Re = 100$ at $t = 30$, Uniform Grid (129 x 129)

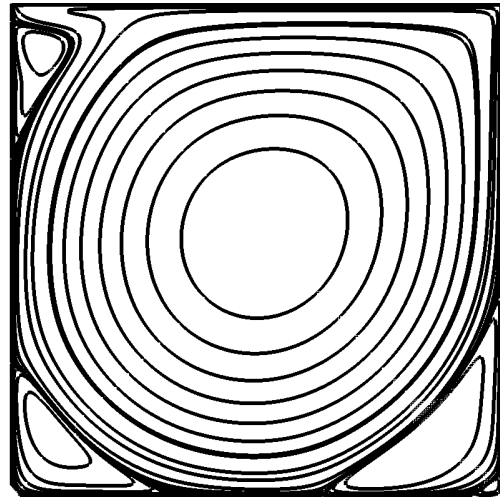Fig. 2. $Re = 1000$ at $t = 30$, Uniform Grid (129 x 129)

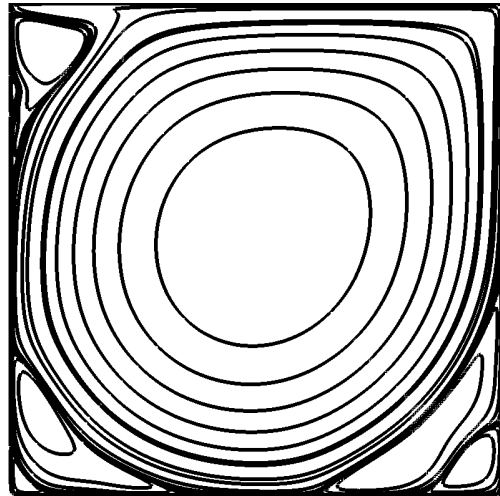
Fig. 3. $Re = 5000$ at $t = 30$, Uniform Grid (257 x 257)

Fig. 4. $Re = 10000$ at $t = 18.7$, Uniform Grid (257 x 257)

4 Discussion and Conclusions

References

[1] Jasak, H., Jemcov, A., and Tukovic, Z., 2007. "Openfoam: A c++ library for complex physics simulations". In International workshop on coupled methods in numerical dynamics, Vol. 1000, pp. 1–20.

[2] Ghia, U., Ghia, K. N., and Shin, C., 1982. "High-re solutions for incompressible flow using the navier-stokes equations and a multigrid method". *Journal of computational physics,* **48**(3), pp. 387–411.

[3] Matveichev, A., 2014. Hopscotch: Building openfoam on os x. http://matveichev.blogspot.com/2014/04/building-openfoam-on-os-x.html.

[4] Bruneau, C.-H., and Saad, M., 2006. "The 2d lid-driven cavity problem revisited". *Computers & Fluids,* **35**(3), pp. 326–348.
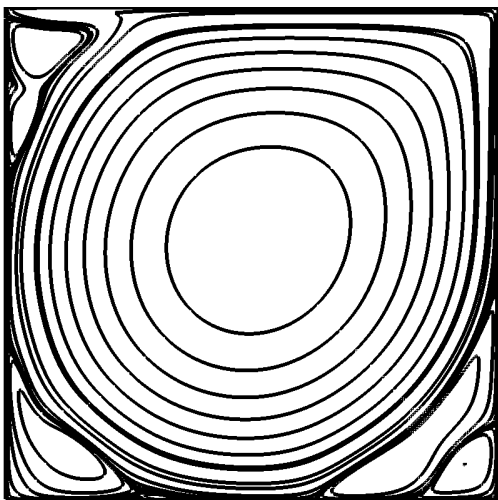
Fig. 5. $Re = 10000$ at $t = 30$, Uniform Grid (257 x 257)

Appendix A: OpenFOAM 2.3.0 installation

```
1   #Download and install prerequisites
2   brew tap homebrew/science
3   brew install open-mpi
4   brew install scotch
5   brew install boost --without-single --with-mpi
6   brew install cgal
7   cd Downloads
8   mkdir OpenFOAM && cd OpenFOAM
9   curl -L http://downloads.sourceforge.net/foam/OpenFOAM-2.3.0.tgz > OpenFOAM-2.3.0.tgz
10  curl -L https://raw.githubusercontent.com/mrklein/openfoam-os-x/master/OpenFOAM-2.3.0.patch > OpenFOAM-2.3.0.patch
11
12  #Build OpenFOAM
13  cd
14  hdiutil create -size 4.4g -type SPARSEBUNDLE -fs HFSX -volname OpenFOAM -fsargs -s OpenFOAM.sparsebundle
15  mkdir OpenFOAM
16  hdiutil attach -mountpoint $HOME/OpenFOAM OpenFOAM.sparsebundle
17  cd OpenFOAM
18  tar xzf ~/Downloads/OpenFOAM/OpenFOAM-2.3.0.tgz
19  cd OpenFOAM-2.3.0
20  cp ~/Downloads/OpenFOAM/OpenFOAM-2.3.0.patch .
21  git apply OpenFOAM-2.3.0.patch
22  source etc/bashrc
23  ./Allwmake > log.Allwmake 2>&1
24
25  #Test to see that everything worked out
26  mkdir -p $FOAM_RUN
27  run
28  cp -r $FOAM_TUTORIALS/incompressible/icoFoam/cavity .
29  cd cavity
30  blockMesh
31  icoFoam
32
33  #Post config
34  alias of230='hdiutil attach -quiet -mountpoint $HOME/OpenFOAM OpenFOAM.sparsebundle; sleep 1; source $HOME/OpenFOAM/OpenFOAM-2.3.0/etc/ba
35  of230
```