Final Project Report
2.29 Spring 2017

Yukino Nagai

May 23, 2017

1 introduction

1.1 Turbulence Modeling

Not much is known about the emergence of turbulence in fluid flow. Turbulent flow is characterized by vortices of varying sizes. Unlike laminar flow, an exact flow profile is difficult to create due to random fluctuations from the mean velocity. It is known that turbulence can be observed in flows above a critical Reynolds number, which is different for different geometries of flow. For example, the critical Reynolds number for pipe flow is around 2300 [1], whereas for flow over a flat plate it is between 10^5 and 10^6 [2]. Most flows observed in nature are turbulent, thus it is useful to understand the nature of turbulent and their effects.

The Reynolds number is defined as

$$Re = \frac{\rho v L}{\mu}$$

where ρ is the density, v is the characteristic velocity, L the characteristic length, and μ is the viscosity. The Reynolds number is derived by taking the ratio of the inertial forces to viscous forces. As ρ, v, and/or L increase and μ decreases, the Reynolds number increases.

As was discussed in 2.29 lecture [3], there are a number of ways to model turbulence. From least computationally expensive to most computationally expensive, the methods are:

1. Correlations
2. Integral equations
3. Averaged equations
4. Large-eddy simulations
5. Direct numerical simulation

Each of these methods have their own strengths and weaknesses, and the "best" method is dependent on the specific value in question. Direct numerical simulation solves the governing Navier-Stokes equations directly to find the velocity profile. Direct numerical simulation at sufficiently high resolution is able to resolve all sizes of eddies, and thus produces the closest velocity profile to the true velocity solution. However, running a DNS at "sufficiently high resolution" requires supercomputers and a lot of time. At lower resolutions, the DNS is not able to resolve the smaller eddies. This leads to small errors in the solution, which propagate over time to cause large errors in the final solution. Although the less computationally expensive methods cannot fully resolve the smaller eddies, they solve modified or additional equation(s) that "fix" the solution. For example, the k-ϵ model is an averaged equation model that also takes into account the conservation of kinetic energy of the flow. The k-ϵ model is a very common method used for turbulence modeling.

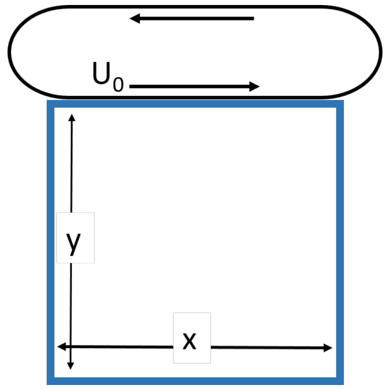


Figure 1: Lid-driven cavity model

1.2 2.29 Finite Volume Framework

The 2.29 Finite-Volume Framework is the code distributed by the MSEAS lab as a part of the 2.29 class. It is also used in the MSEAS lab for running test cases. The code is in Matlab, with some of the loops written in C++. The 2.29 Finite-Volume Framework solves Reynolds-Averaged Navier-Stokes equations.

1.3 Lid-Driven Cavity Flow

This project uses lid-driven cavity flow as a test case for testing how well the 2.29 code handles flow at high Reynolds numbers. The illustration for the lid-driven cavity model is shown in Figure 1. A looped "conveyor belt" touches the top surface of a square cavity filled with a fluid. The conveyor belt moves at a constant velocity U_0. By the no-slip condition, the top layer of the fluid also moves at the same velocity. The Reynolds number of this flow is given by

$$Re = \frac{vL}{\nu}$$

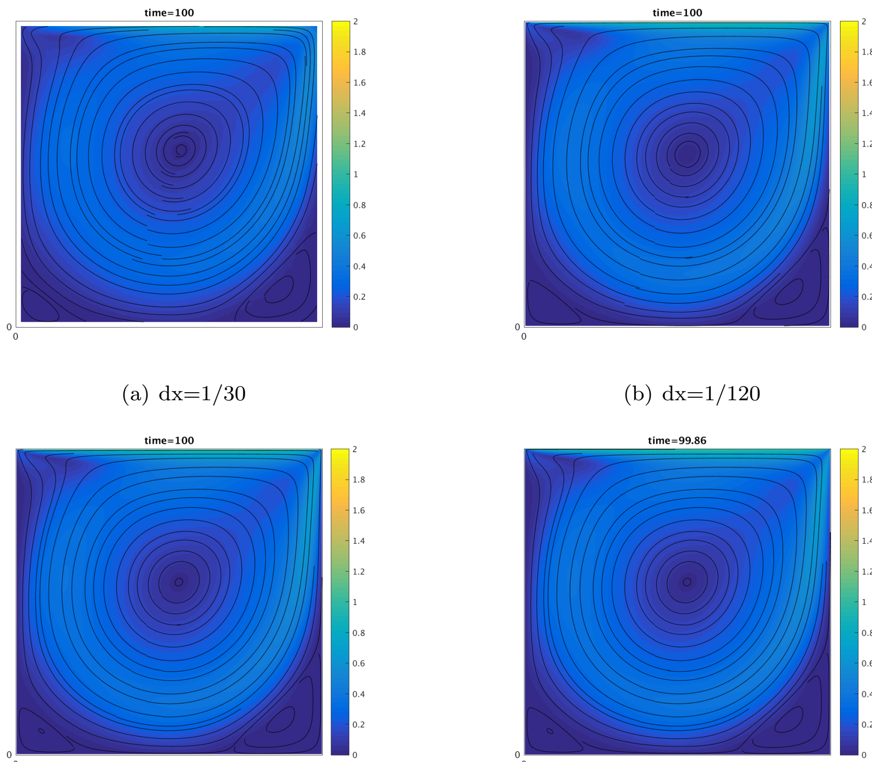
where ν is ρ/μ. In this project, the size of the cavity (x, y) as well as the velocity of the conveyor belt are kept constant at unity across all Reynolds numbers. The Reynolds number is changed by varying ν only. According to literature, the critical Reynolds number is around 7700, at which point Hopf bifurcation occurs. Below the critical Reynolds number, the lid-driven cavity flow reaches a steady-state solution, whereas above the critical Reynolds number, the flow reaches a periodic solution [4]. However, literature has also reported a steady solution at higher Reynolds numbers–as high as 21,000–at higher resolutions [5].

2 Method

The 2.29 FV Framework Example script was run with Reynolds numbers of 1000, 7000, 8000, and 10^8. These numbers were achieved by changing the value of ν. For each Reynolds number, Example was run with dx=dy=1/30, 1/60, 1/120, and 1/240. To ensure that the CFL condition is met, dt was set to dx/2 for all cases. As the Reynolds number increased, the time to a steady solution increased, thus the final time was increased.

3 Results and Discussion

The steady-state solutions of the Example case at Reynolds number 1000 is shown in Figure 2. At all four of the dx values shown, steady-state solutions were observed and the solutions look very similar to each other. Qualitatively, the solutions match those of literature.



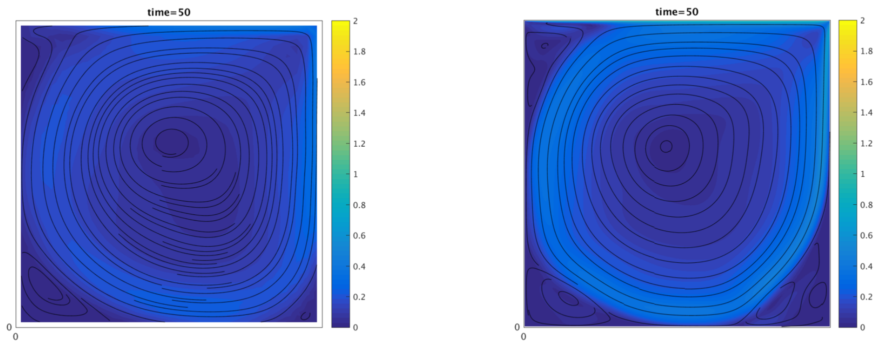
(a) dx=1/30

(b) dx=1/120

(c) dx=1/240

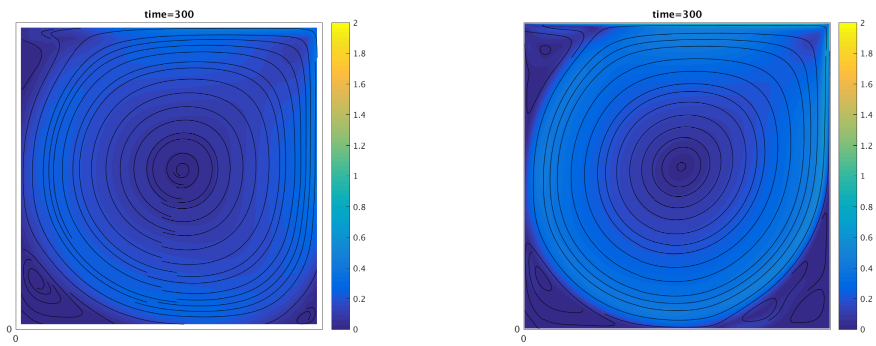
(d) dx=1/350

Figure 2: Steady-state solutions at Re=1000. The lines are the streamlines, and the shading is vorticity.



(a) dx=1/30 (b) dx=1/240

Figure 3: Solution for Re=7000 at time=50. The lines are the streamlines, and the shading is vorticity.



(a) dx=1/30 (b) dx=1/240

Figure 4: Steady-state solutions at Re=7000. The lines are the streamlines, and the shading is vorticity.

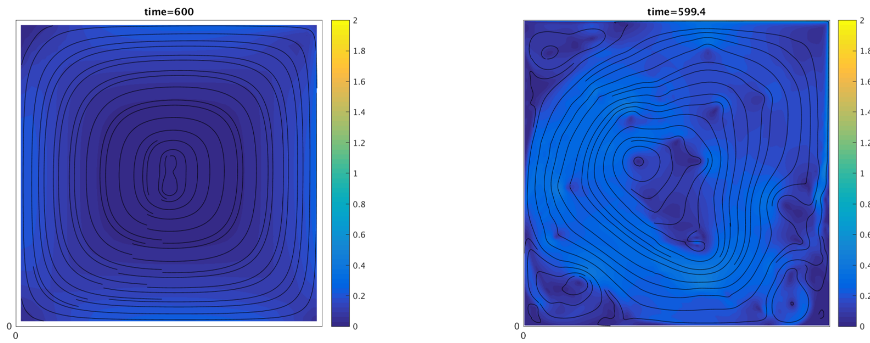
Figure 3 shows the velocity field at time 50. Qualitatively, one can see that the values are different but that the general form is somewhat similar.

The steady-state solutions of the Example case at Reynolds number 7000 is shown in Figure 4. At both of the dx values shown, the solution reached steady state. Qualitatively, the solution for dx=1/240 matches that of literature whereas dx=1/30 does not. This suggests that there is an error as a result of the coarse grid.

Figure 5 shows the streamlines and vorticity at time 600. Compared to the relationship seen in Figure 3, one can see that these two are very different. This is interesting, because physically, both of these runs are modeling the exact same thing. However, these solutions look like two very different phenomena. At dx=1/30, the solution is very close to steady-state, whereas at dx=1/240, the solution is not nearing steady-state (or periodic, for that matter). At dx=1/30, a steady-state solution was observed, contrary to what is suggested by literature. The run for dx=1/240 could not be completed fully due to time and memory constraints.

4 Conclusion

The largest constraint for this project was time and memory. When dx is doubled, the computational cost becomes 8 times as much. While the example case with default values took an order of seconds, the case



(a) dx=1/30

(b) dx=1/240

Figure 5: Solution for Re=8000 at time=600. The lines are the streamlines, and the shading is vorticity.

of Reynolds number 7000 with a dx=1/240 took 11 hours. Further, with the current 2.29 setup of saving the output matrix of each time step for post-processing, computer memory is filled very quickly. The results of this project suggest that as Reynolds number is increased, the errors resulting from decreasing grid size becomes greater. In the 2.29 FV Framework, at turbulent Reynolds numbers, the smaller eddies are washed out, and only larger eddies survive. At larger grid spacing, the size of the eddies that are washed out are larger, thus leading to larger errors. This project only compared output on a surface, qualitative level. Future work should involve a closer look at the output matrices to compare the differences in values.

5 Acknowledgments

I would like to thank the MSEAS group, especially Prof. Lermusiaux, Chinmay, Arkopal, Abhinav, Jing, and Jonathan for their help with my project.

References

[1] Pierre F.J. Lermusiaux. *Lecture notes for 2.005: Thermal-Fluids Engineering I.* 2017.

[2] Jeff Moran. *Lecture notes for 2.006: Thermal-Fluids Engineering II.* 2016.

[3] Pierre F.J. Lermusiaux. *Lecture notes for 2.29: Numerical Fluid Dynamics.* 2017.

[4] Yih-Ferng Peng, Yuo-Hsien Shiau, and Robert R. Hwang *Transition in a 2-D lid-driven cavity flow.* 2003: Computers & Fluids.

[5] E. Erturk, T.C. Corke, and C. Gokcol *Numerical solutions of 2-D steady incompressible driven cavity flow at high Reynolds numbers.* 2005: International Journal for Numerical Methods in Fluids.