EN530.767 - Computational Fluid Dynamics
Assignment 5
Sushrut Kumar

1 Introduction

In this work, solution of Lid Driven Cavity flow was obtained by solving 2-D Navier-Stokes Equation. Lid Driven Cavity flow is considered to be a benchmark problem for validating the fidelity and accuracy of Navier Stokes equations. The results obtained by Ghia et al. [1] was used for validation of results. The schematic of the problem is shown in fig. 1

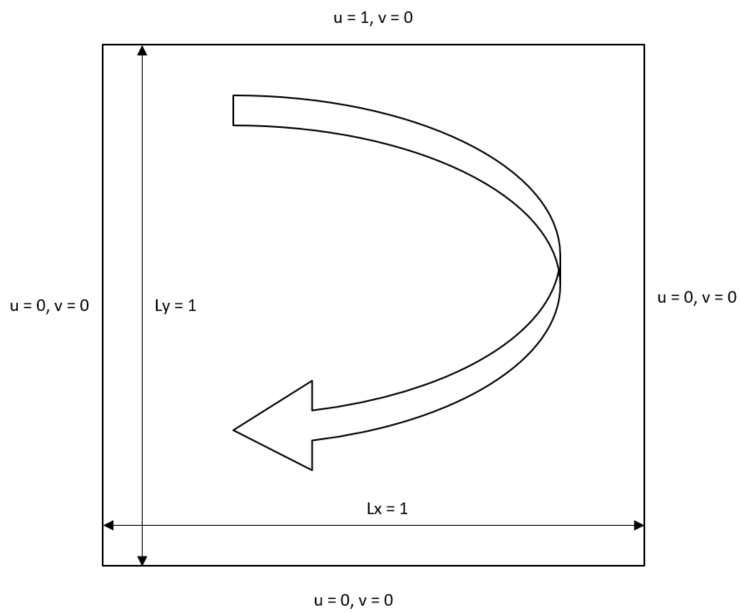


Figure 1: Schematic of Lid Driven Cavity problem

The following sections will elaborate the methods adopted for developing the Navier-Stokes equation solver and the results obtained.

2 Methodology

In this work, non-dimensional 2-D unsteady Navier Stokes equation as shown below was used.

$$\nabla \cdot u = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + \frac{\partial (Uu)}{\partial x} = -\frac{\partial p}{\partial x} + \frac{1}{Re}\frac{\partial^2 u}{\partial x^2} \tag{2}$$

In order to eliminate the time constraint due to viscous stability criteria the the Diffusion term was given an implicit treatment. Then the equations were discretised using a cell-centered, collocated grid arrangement [2] [3] where a separate update of staggered mass flux was done on faces using the face velocity U_j which helps in using a compact stencil without going to a staggered mesh approach. The governing equations are then split using fractional step method.

Mittal et al. [4] observed that an implementation of collocated grid arrangement as shown in figure 2 is easier to implement than the staggered arrangement and a good discrete kinetic energy conservation is achieved when central differencing Scheme is used [5]. The system of equations were solved using a Line SoR algorithm for Advection-Diffusion and Gauss-Seidel for Pressure Poisson Equations. The code was written in Fortran 90. The code can be seen at GitHub

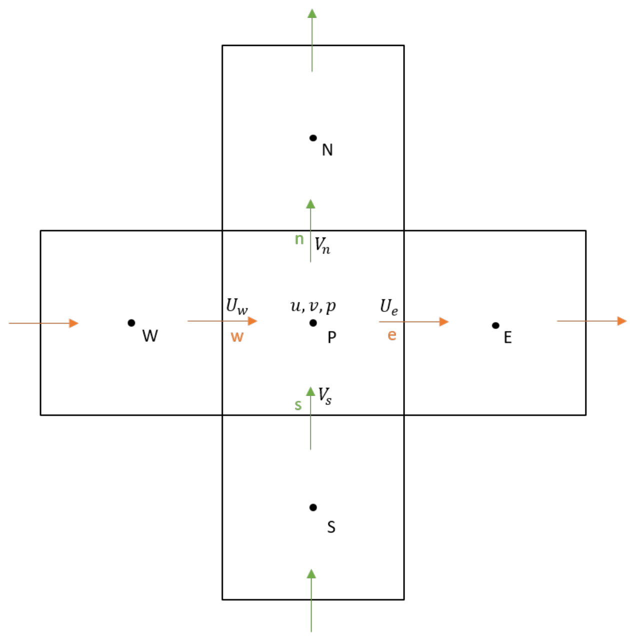
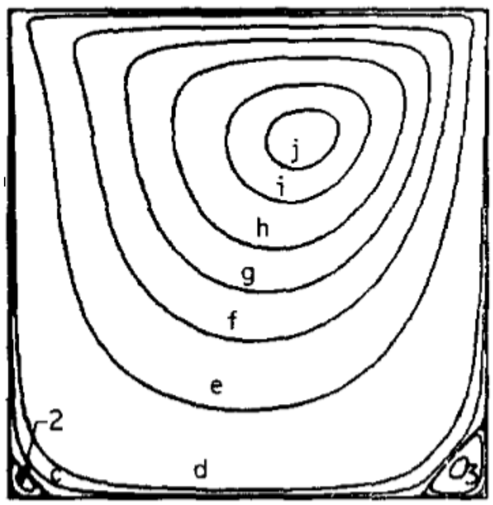


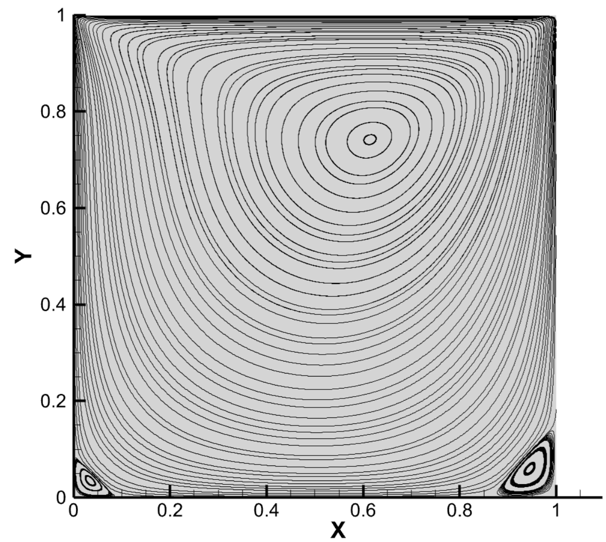
Figure 2: Collocated Grid Arrangement

3 Results and Discussions

The simulations were run untill the steady is achieved and then the results obtained were compared and validated against the ones from Ghia et al. First a qualitative comparison is shown using the streamlines for both cases of Re = 100 and Re = 1000. Fig. 3 shows the streamline for Re=100 and subsequently fig. 4a - 4d shows the streamlines obtained for different grid resolutions. It can be seen that lower grid resolutions is not able to resolve the two corner vortices properly.

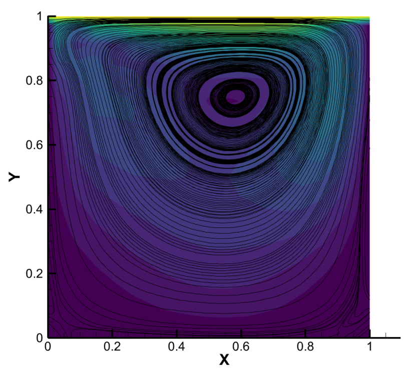


(a)

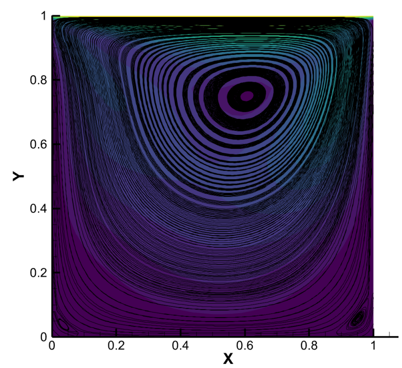


(b)

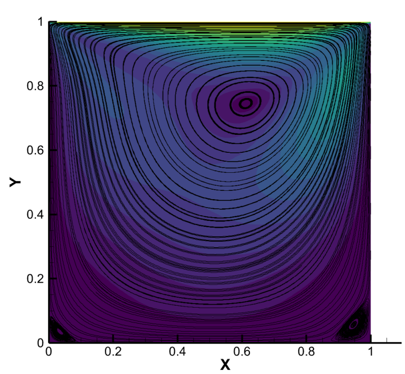
Figure 3: Streamlines by Ghia et al.[1] and present study



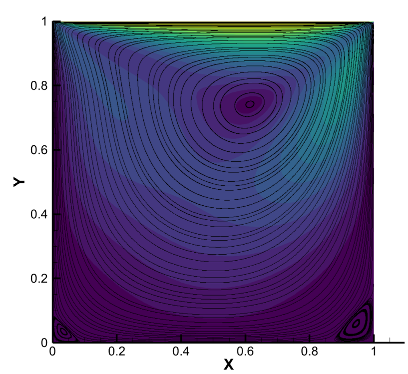
(a)



(b)



(c)



(d)

Figure 4: Velocity magnitude contours along with Streamlines at Re = 100 for a. 16 $\times$ 16 Grid Points, b. 32 $\times$ 32 Grid Points, c. 64 $\times$ 64 Grid Points, d. 128 $\times$ 128 Grid Points

A qualitative comparison is not enough to show the accuracy of the numerical solution. Therefore, presented next is the variation of v/U along a horizontal line through the geometric center of the cavity (fig. 5) and the variation of u/U along a vertical line through the geometric center of the cavity(fig. 6). It can be seen that the results starts to converge as resolution is increased from 16^2 grid points to 128^2 grid points and the data at higher resolution closely matches the validation dataset.

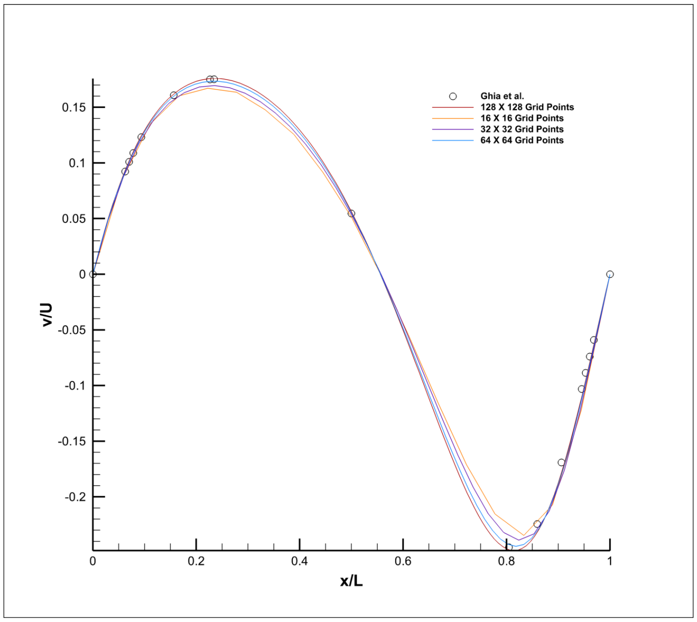


Figure 5: Variation of v/U along a horizontal line through the geometric center of the cavity

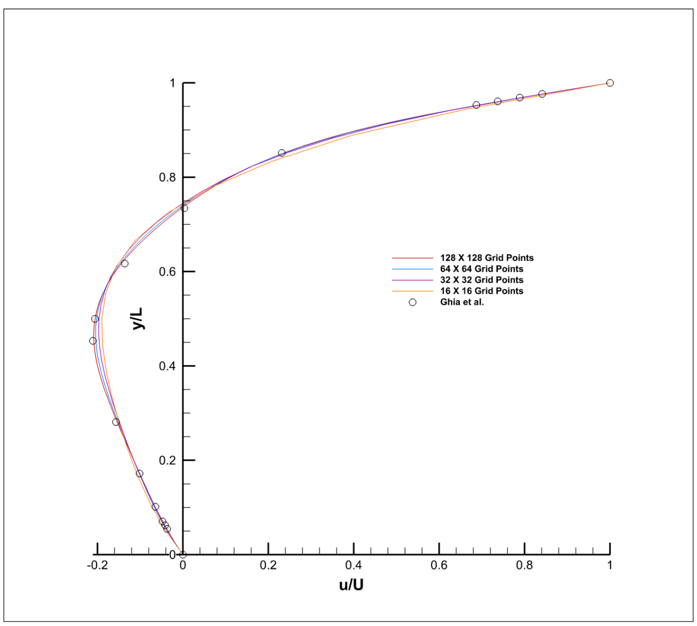
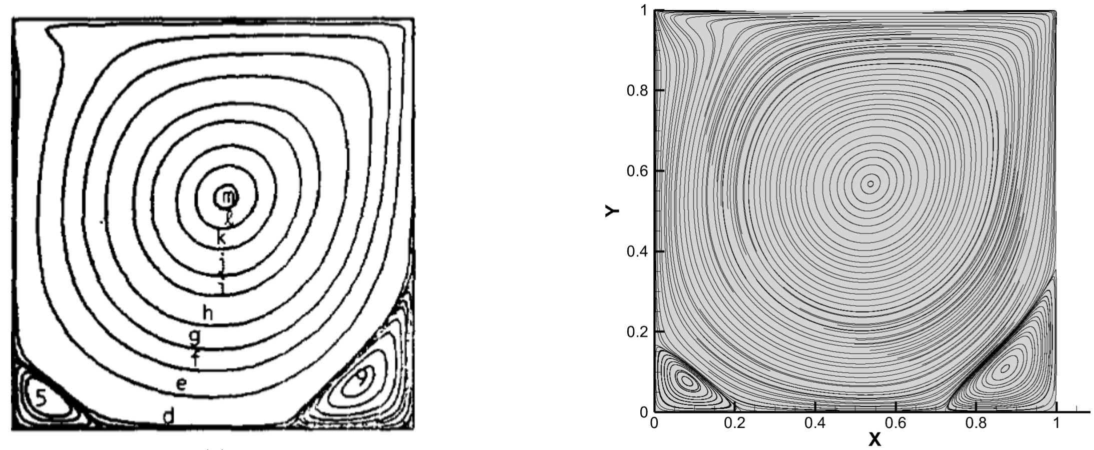


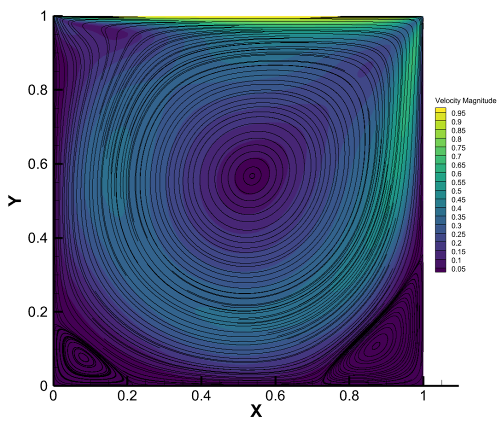
Figure 6: Variation of u/U along a vertical line through the geometric center of the cavity

Presented next is the qualitative comparison of streamlines at Re=1000 for different grid resolutions.

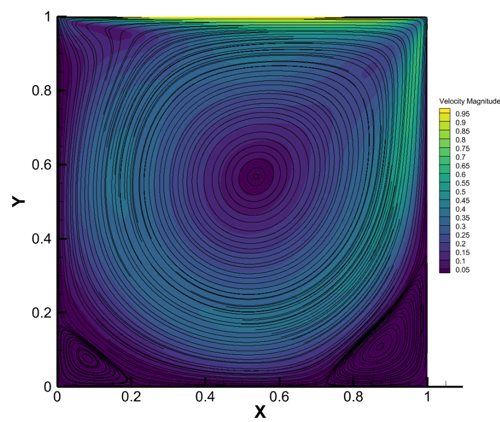


(a) (b)

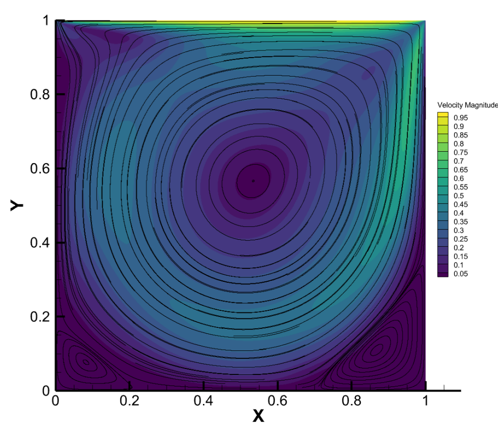
Figure 7: Streamlines by Ghia et al.[1] and present study



(a)



(b)



(c)

Figure 8: Velocity magnitude contours along with Streamlines at Re = 1000 for a. 128 × 128 Grid Points, b. 256 × 256 Grid Points and c. 512 ×512 Grid Points

Below shown are the variation of v/U along a horizontal line through the geometric center of the cavity (fig. 9) and the variation of u/U along a vertical line through the geometric center of the cavity(fig. 10) at Re=1000. Clearly grid resolution of 128^2 is not enough to match the validation data and resolution has to be increased. Resolution of 256^2 gave a better fit that 128^2. The data from 512^2 is the most accurate.

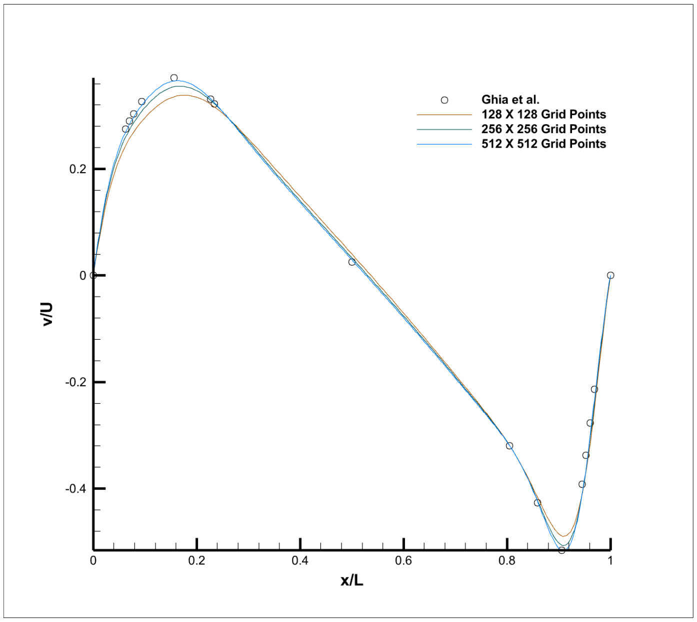


Figure 9: Variation of v along a horizontal line through the geometric center of the cavity

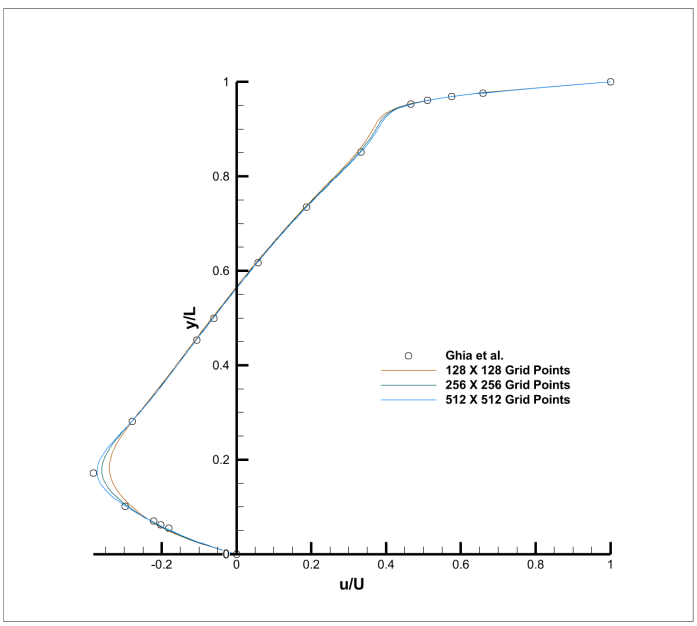


Figure 10: Variation of u along a vertical line through the geometric center of the cavity

References

[1] U. Ghia, K. N. Ghia, and C. Shin, "High-re solutions for incompressible flow using the navier-stokes equations and a multigrid method," *Journal of computational physics*, vol. 48, no. 3, pp. 387–411, 1982.

[2] Y. Zang, R. L. Street, and J. R. Koseff, "A non-staggered grid, fractional step method for time-dependent incompressible navier-stokes equations in curvilinear coordinates," *Journal of Computational Physics*, vol. 114, no. 1, pp. 18–33, 1994.

[3] T. Ye, R. Mittal, H. Udaykumar, and W. Shyy, "An accurate cartesian grid method for viscous incompressible flows with complex immersed boundaries," *Journal of computational physics*, vol. 156, no. 2, pp. 209–240, 1999.

[4] R. Mittal, H. Dong, M. Bozkurttas, F. Najjar, A. Vargas, and A. Von Loebbecke, "A versatile sharp interface immersed boundary method for incompressible flows with complex boundaries," *Journal of computational physics*, vol. 227, no. 10, pp. 4825–4852, 2008.

[5] F. N. Felten and T. S. Lund, "Kinetic energy conservation issues associated with the collocated mesh scheme for incompressible flow," *Journal of Computational Physics*, vol. 215, no. 2, pp. 465–484, 2006.

[6] J. Van Kan, "A second-order accurate pressure-correction scheme for viscous incompressible flow," *SIAM Journal on Scientific and Statistical Computing*, vol. 7, no. 3, pp. 870–891, 1986.