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

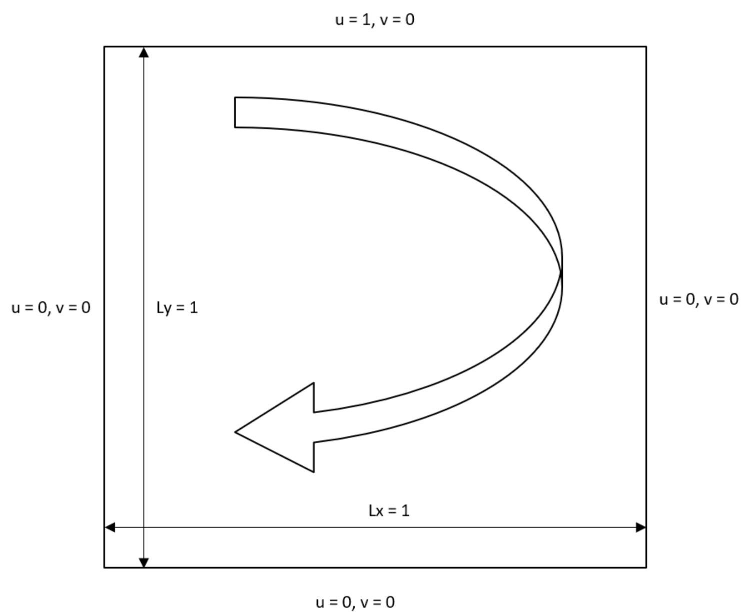


Figure 1: Schematic of Lid Driven Cavity problem

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.

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial (U_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_j} + \frac{1}{Re}\frac{\partial^2 u_i}{\partial x_i x_j} \tag{2}$$

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 a 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. Here the Navier-Stokes equations is split into an Advection-Diffusion equation (Eq. 3) and a Pressure Poisson equation (Eq. 4), whose finite difference forms are shown below.

$$\frac{(u^* - u^n)}{\Delta t} + \frac{(Uu)^n_{i+1,j} - (Uu)^n_{i-1,j}}{2\Delta x} + \frac{(Vu)^n_{i,j+1} - (Vu)^n_{i,j-1}}{2\Delta y}$$
$$= \frac{1}{Re}\left(\frac{u^*_{i+1,j} - 2u^*_{i,j} + u^*_{i-1,j}}{\Delta x^2} + \frac{u^*_{i,j+1} - 2u^*_{i,j} + u^*_{i,j-1}}{\Delta y^2}\right) \tag{3}$$
$$\frac{\boldsymbol{\nabla} \cdot u^*}{\Delta t} = \nabla^2 p \tag{4}$$
$$\frac{U^*_e - U^*_w}{\Delta x} + \frac{V^*_n - V^*_s}{\Delta y} = \nabla^2 p \tag{5}$$

An equation for v can also be written similar to equation 3 where equation 4 is obtained using the constraint that u^{n+1} is divergence free. Where U^* and V^* can be defined as

$$U^*_e = \frac{u^*_P + u^*_E}{2} \tag{6}$$
$$V^*_n = \frac{v^*_P + v^*_N}{2} \tag{7}$$

Now, $u*$ the intermediate velocity is corrected after solving the Pressure Poisson equation along with face velocities (Similarly v^* and V^* can be corrected). The correction is shown below.

$$u^{n+1} = u^* - \Delta t(\nabla p)_{cc} \tag{8}$$
$$U^{n+1} = U^* - \Delta t(\nabla p)_{fc} \tag{9}$$

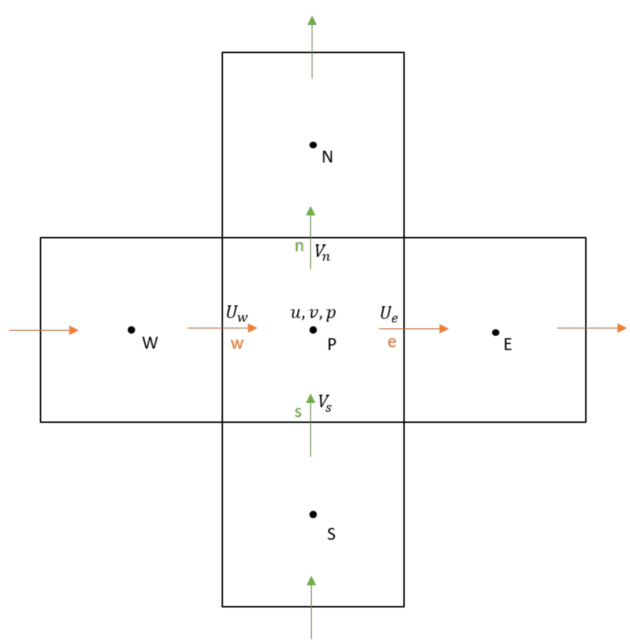


Figure 2: Collocated Grid Arrangement

Mittal et al. [4] observed that an implementation of collocated grid arrangement is easier that 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 both Advection-Diffusion and Pressure Poisson Equations. The code was written in Fortran 90. Attempt was also made for second order time discretisation using Adam-Bashforth for advection terms and Crank-Nicolson for Diffusion terms. The splitting was done using Fractional-Step method of Van-Kan [6]. So far the results were not satisfactory and improvements are being made. The code can be seen at GitHub

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.

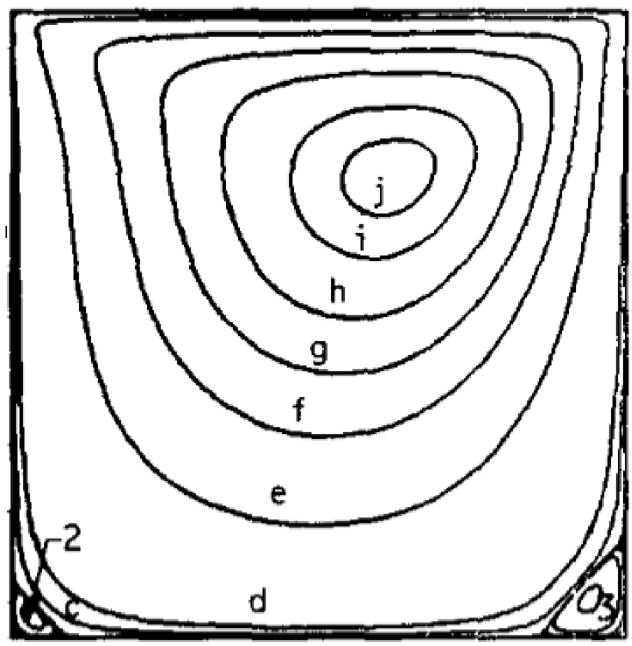
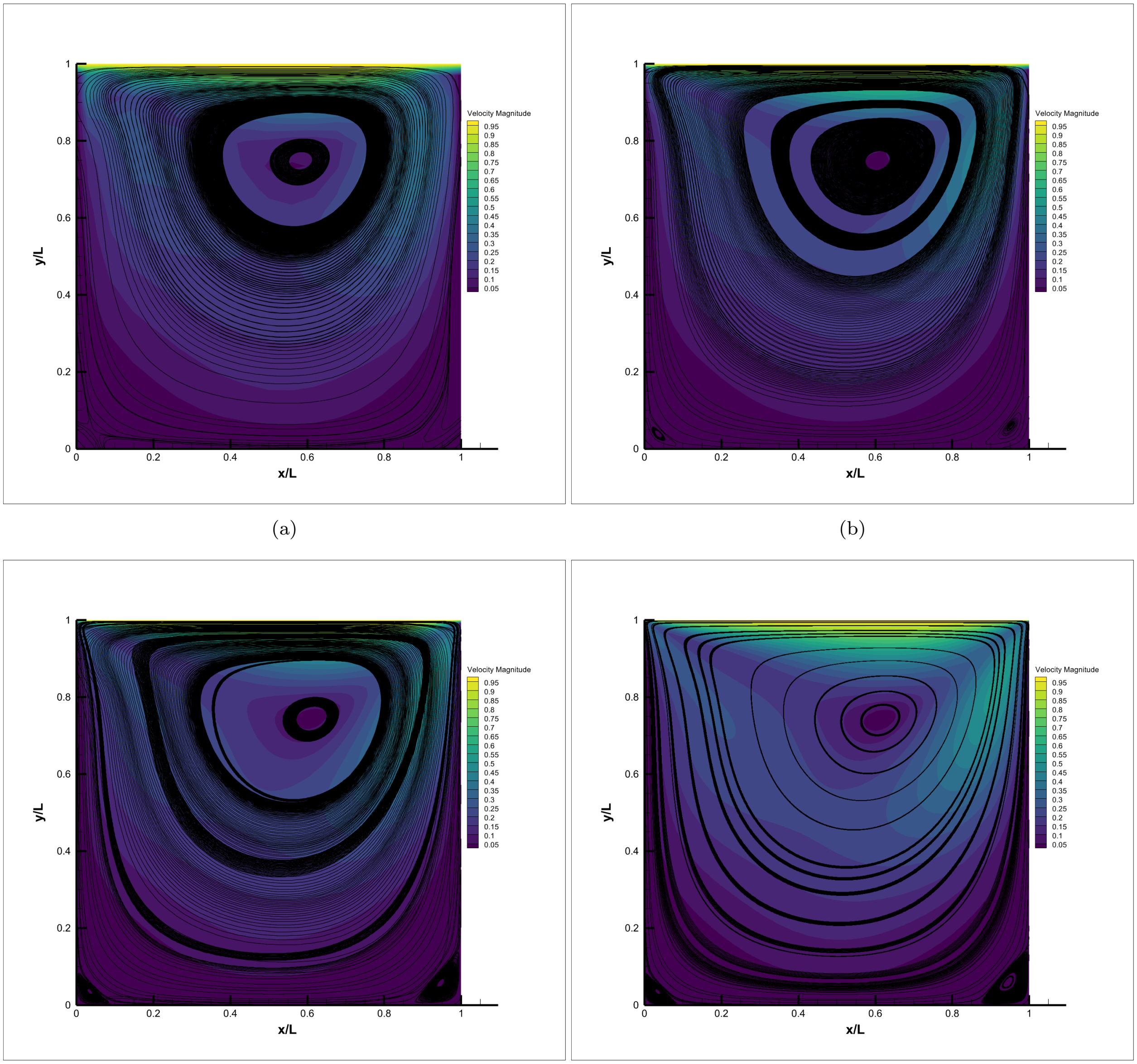


Figure 3: Streamlines by Ghia et al.



(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.

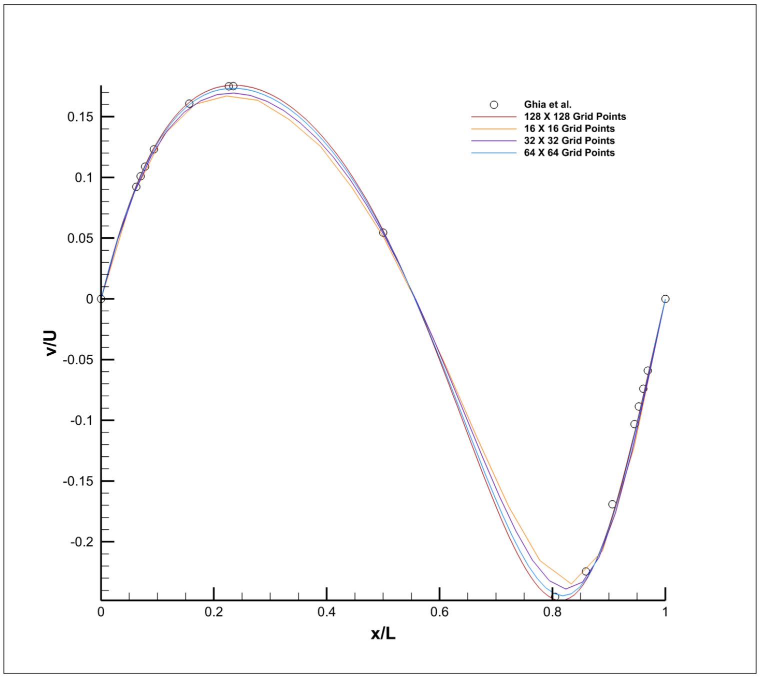


Figure 5: Variation of v/U along a horizontal line through the geometric center of the cavity

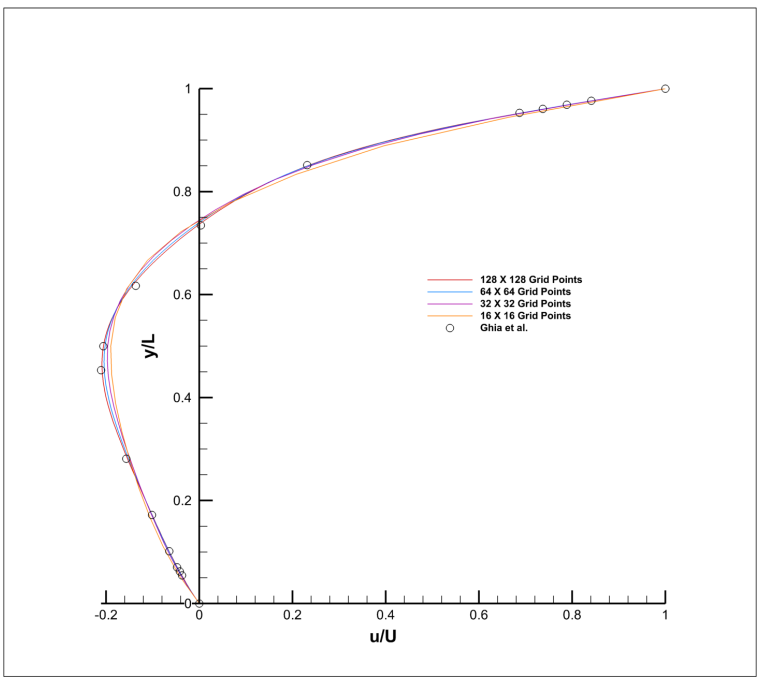
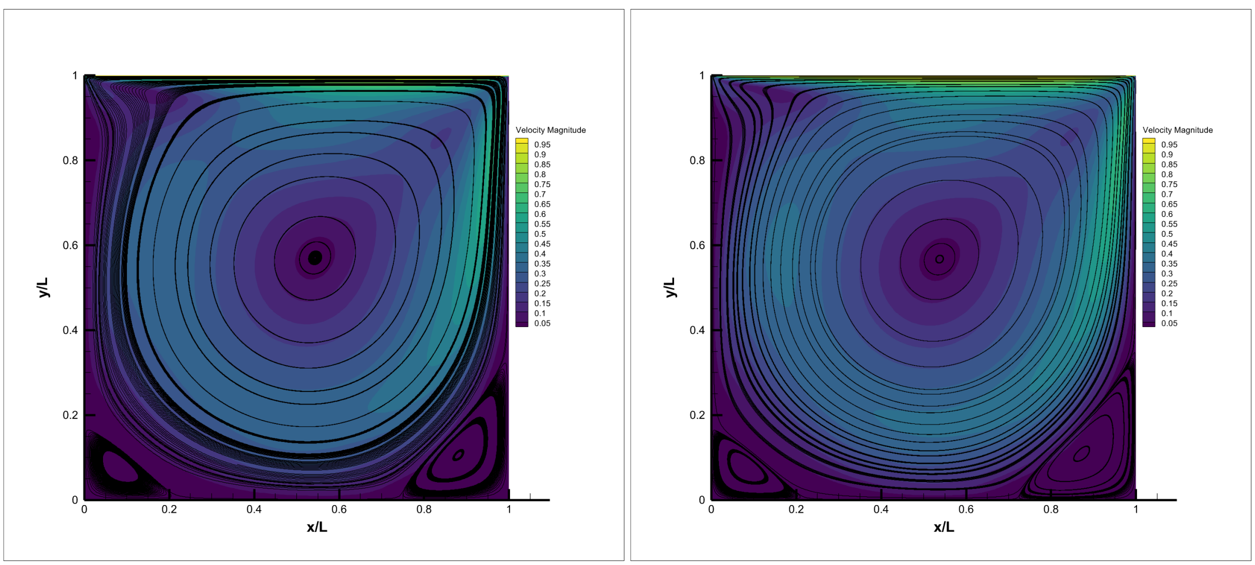


Figure 6: Variation of u/U along a vertical line through the geometric center of the cavity

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

Figure 7: Streamlines by Ghia et al.



(a)

(b)

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

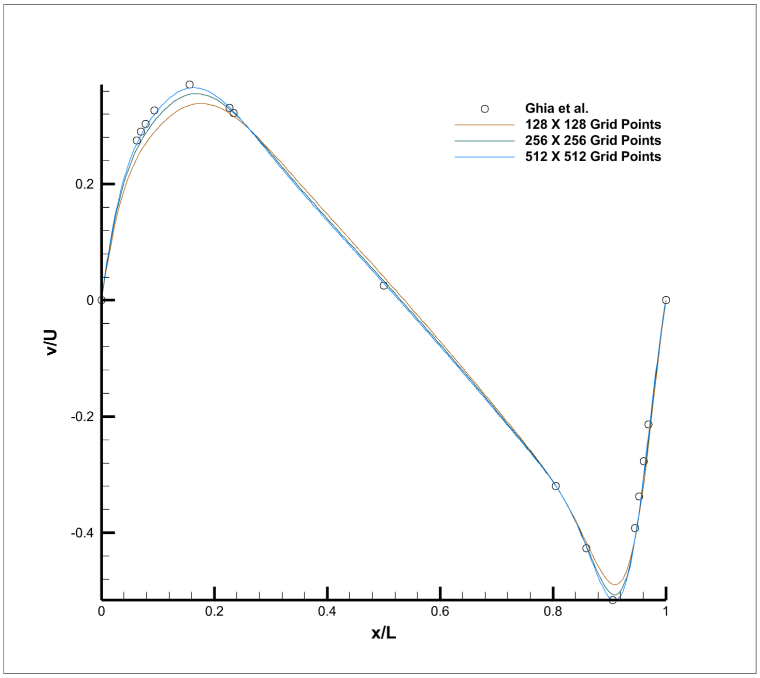


Figure 9: Variation of v along a horizontal line through the geometric center of the cavity

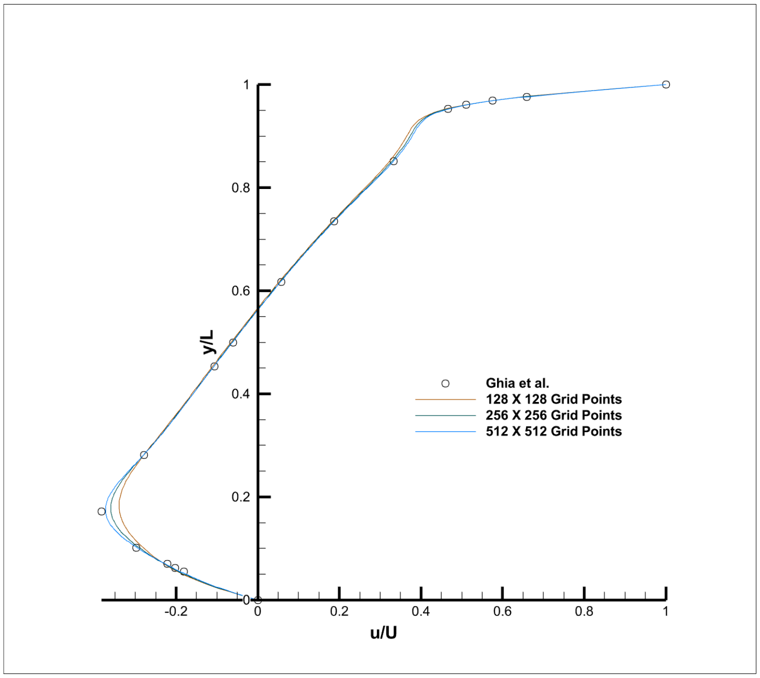


Figure 10: Variation of u along a vertical line through the geometric center of the cavity

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.