# Final Computer Assignment
# Lid Driven Cavity Flow

Juan Carlos Cuevas

December 26, 2014

## 1 Results

The lid driven cavity flow was simulated using the splitting method and the results are presented in this report. To validate the code the velocity and pressure fields are plotted in the Figure 1.0.1. Then a contour plot of the streamwise velocity $u(x, y)$ is depicted in the Figure 1.0.2, and finally the results for the streamwise velocity (Figure 1.0.3) are compared with the results obtained by Ghia et. al [1] as a function of the wall normal position y. Additional results are shown for a Reynolds Number of $Re_L = 400$.

The numerical simulation considered an uniform square cavity of $50 \times 50$ grid, for two different Reynolds numbers $Re_L = 100$ and $Re_L = 400$ respectively. The simulation advance in time steps of $\Delta t = 0.01$ from the rest until the flow reaches a sort out of steady state with 10000 time steps ($t = 100$ sec). The spacing grid is $\Delta x = 0.02$. The number of time steps was limited by the capacity of the computer where it was run.

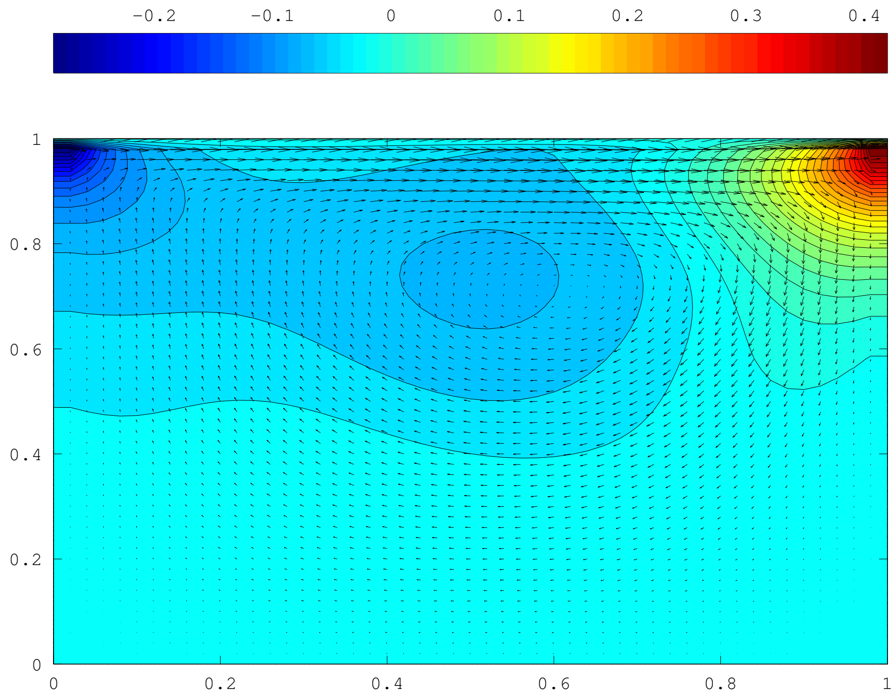


Figure 1.0.1: $Re_L = 100$. Velocity field and pressure contours for the lid driven cavity flow. The top color bar shows the intensity of the pressure contours ranging from a minimum value of -0.1 up to a maximum value of 0.4 normal units.

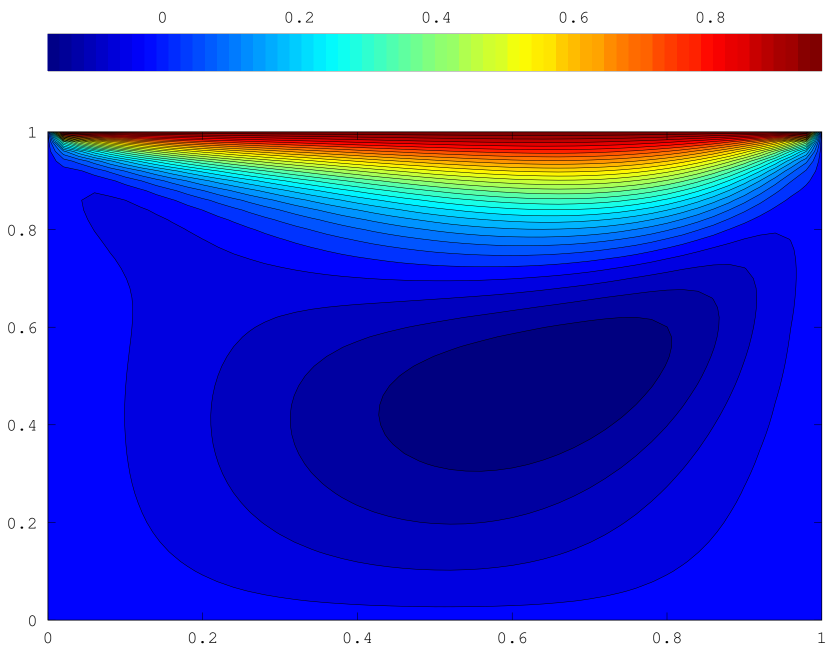


Figure 1.0.2: $Re_L = 100$. Streamwise velocity contours for the lid driven cavity flow. In the upper part can be seen like the speed is almost constant for u=1 normal units. The top color bar shows the intensity of the velocity contours ranging from a minimum value of -0.1 up to a maximum value of 1.0 normal units.

# References

[1] U Ghia, K.N Ghia, and C.T Shin. High-re solutions for incompressible flow using the navier-stokes equations and a multigrid method. *Journal of Computational Physics*, 48(3):387 – 411, 1982. ISSN 0021-9991. doi: http://dx.doi.org/10.1016/0021-9991(82)90058-4. URL `http://www.sciencedirect.com/science/article/pii/0021999182900584`.

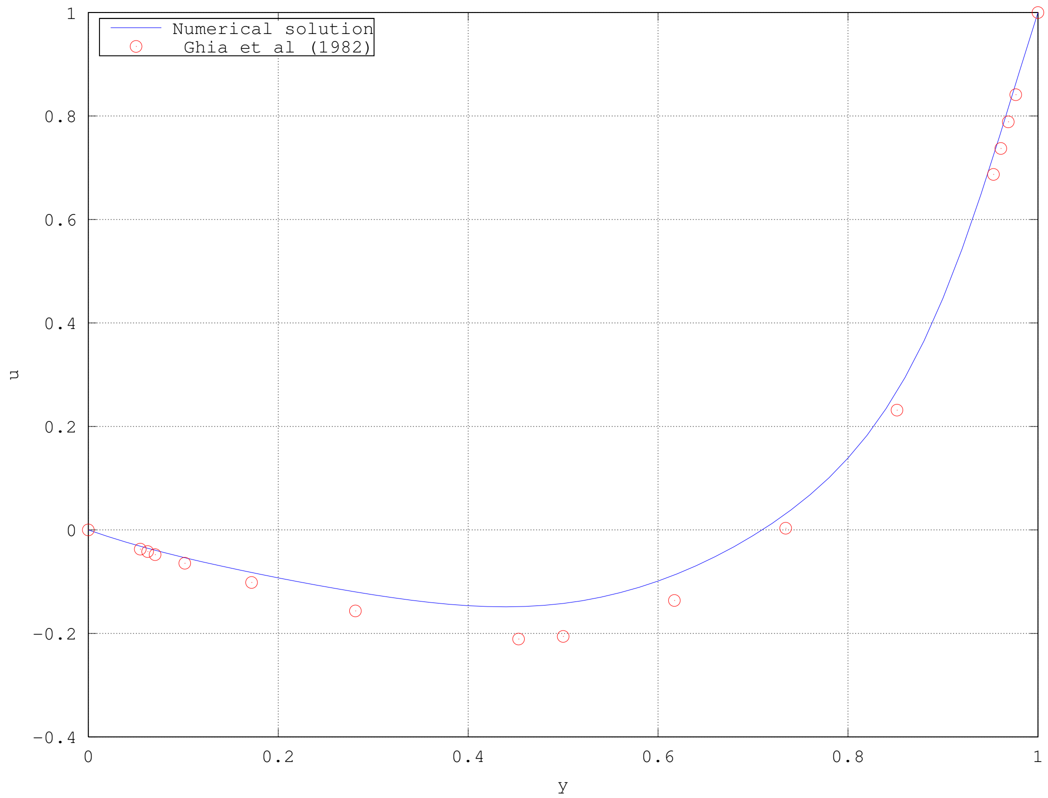


Figure 1.0.3: $Re_L = 100$. Variation of the streamwise velocity as a function of the wall normal position trough Geometric Center of the cavity $x/L = 0.5$. The results show a slight difference in the central region spanning from $y = 0.4$ to $y = 0.7$. Since the numerical method used is first order in the time. Major accuracy can be achieved with an implicit method in the time and smoother boundary conditions, thus improving convergence. In this case the number of iterations is limited by the technical specifications of the computer where the simulation was run. However the numerical results show an appropriate physical behavior.

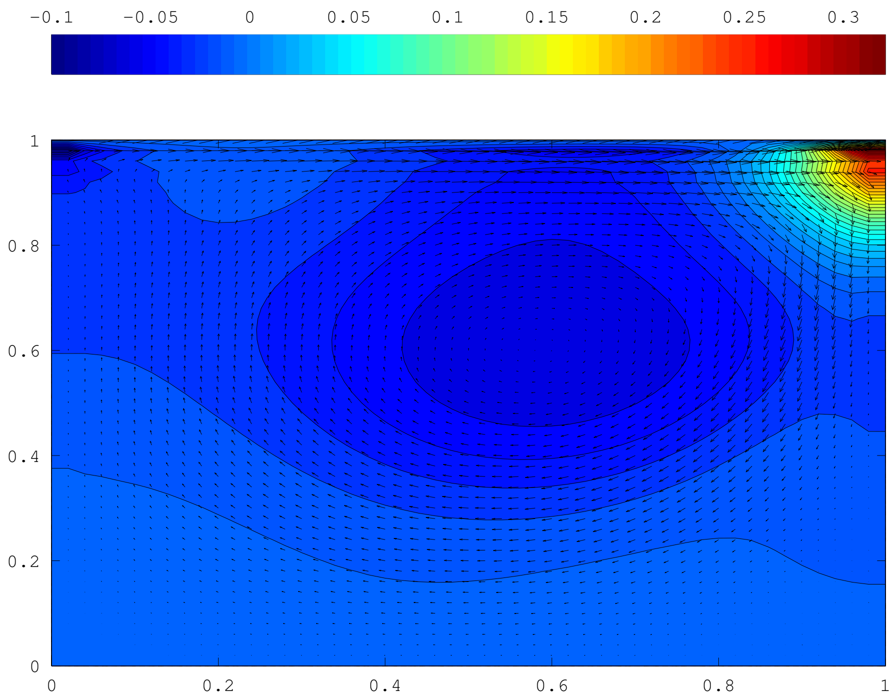


Figure 1.0.4: $Re_L = 400$. Velocity field and pressure contours for the lid driven cavity flow. The top color bar shows the intensity of the pressure contours ranging from a minimum value of -0.1 up to a maximum value of 0.3 normal units.

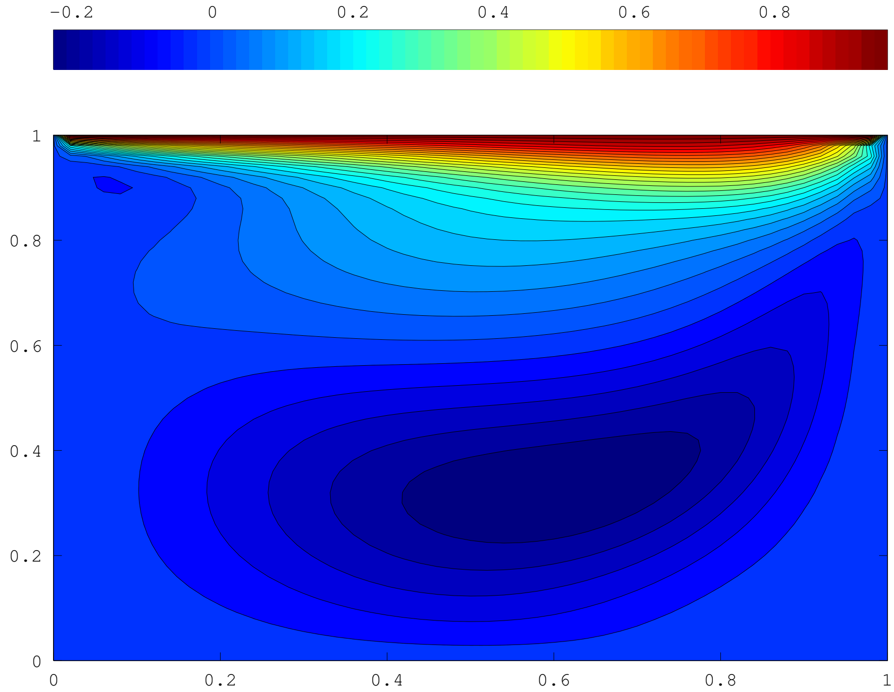


Figure 1.0.5: $Re_L = 400$ Streamwise velocity contours for the lid driven cavity flow. In the upper part can be seen like the speed is almost constant for u=1 normal units. The top color bar shows the intensity of the velocity contours ranging from a minimum value of -0.2 up to a maximum value of 1.0 normal units.

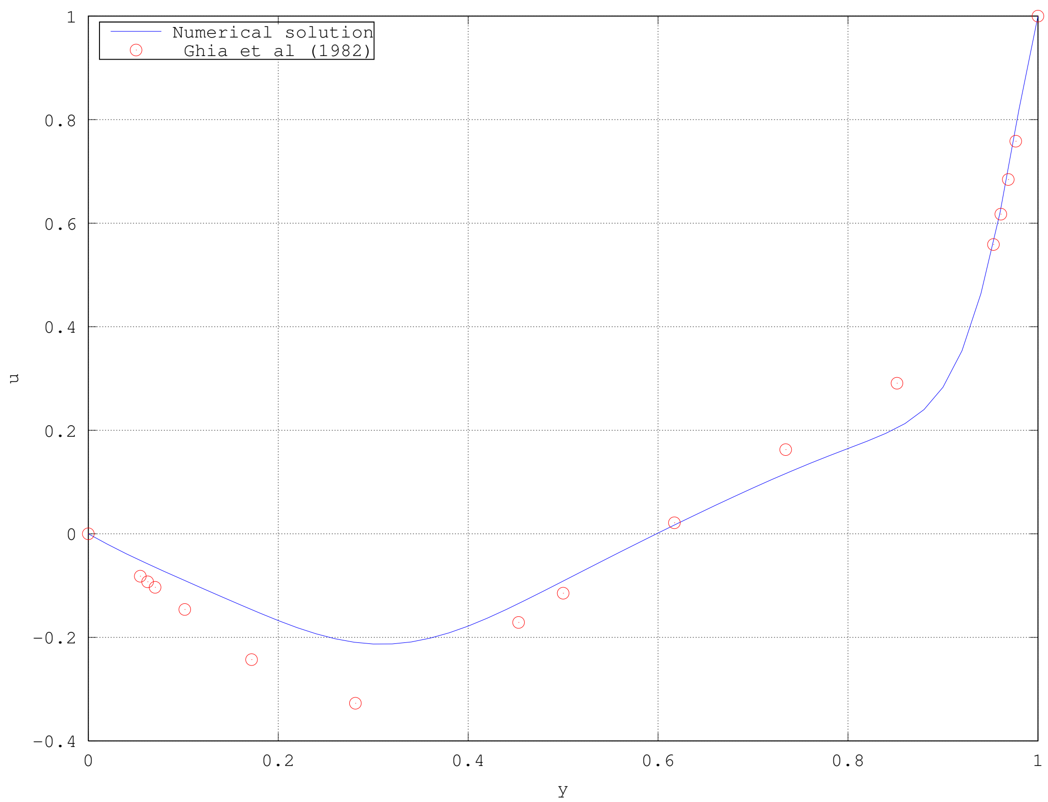


Figure 1.0.6: $Re_L = 400$. Variation of the streamwise velocity as a function of the wall normal position trough Geometric Center of the cavity $x/L = 0.5$. The results show that the numerical method decreases its accuracy for higher Reynolds Numbers. Other methods like the staggered grid method are necessary to simulate higher Reynolds numbers.