Simulation of a lid-driven cavity flow using an incompressible Navier-Stokes solver based on the fractional step method

Tanmay C. Shidhore

West Lafayette, United States

1. Motivation

An incompressible Navier-Stokes solver developed using the fractional step method is validated for the case of a lid-driven cavity flow. Simulation data from Ghia et al. (1982) has been used as a benchmark. A comparison is presented for the case of a low and high Reynolds Number (Re=100 and 1000)

2. Problem Geometry

The geometry considered is a square cavity with side 1. The top wall of the cavity is moving with a constant velocity u_{wall} and the other walls are stationary.

3. Governing Equation

The physics of the problem is governed by the 2D incompressible Navier-Stokes equation-

$$\begin{gathered} \nabla^2 \vec{u} = 0 \\ \frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \vec{\nabla} \vec{u} = -\vec{\nabla} p + \nu \nabla^2 \vec{u} \end{gathered} \tag{1}$$

Here, $\vec{u}$ = u$\vec{i}$ + v$\vec{j}$, p = P/ρ = Normalised pressure and ν = Kinematic Viscosity

4. Numerical Scheme and meshing

Equation 1 has been discretized using 2^{nd} order spatial discretization. Time advancement is done explicitly using a 2^{nd} order Runge-Kutta method. Computations were carried out on a 48×48 stretched grid. The grid is constructed to be denser towards the boundaries and sparser in the centre.

5. Results

In order to compare numerical results with Ghia et al.(1982), the code was executed for a uniform wall velocity of +1. The Reynolds number was modified through the kinematic viscosity (0.01 for Re=100 and 0.001 for Re=1000). The following plots show the comparison for the centreline u and v velocities with y and x respectively.

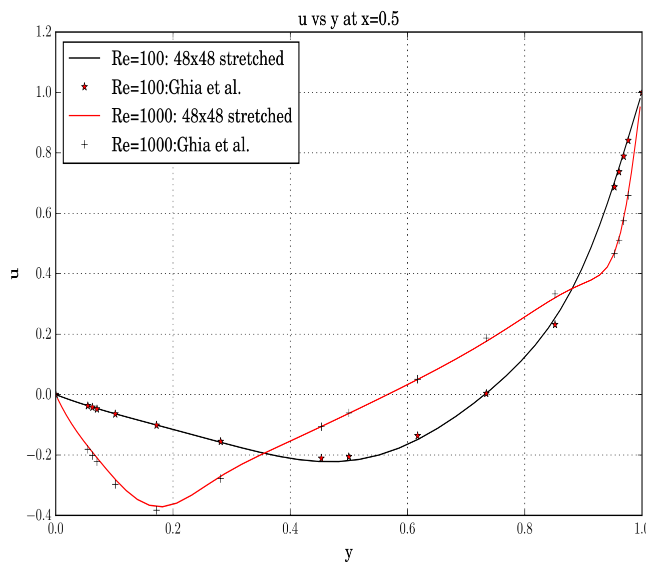

Figure 1: u vs y at x=0.5

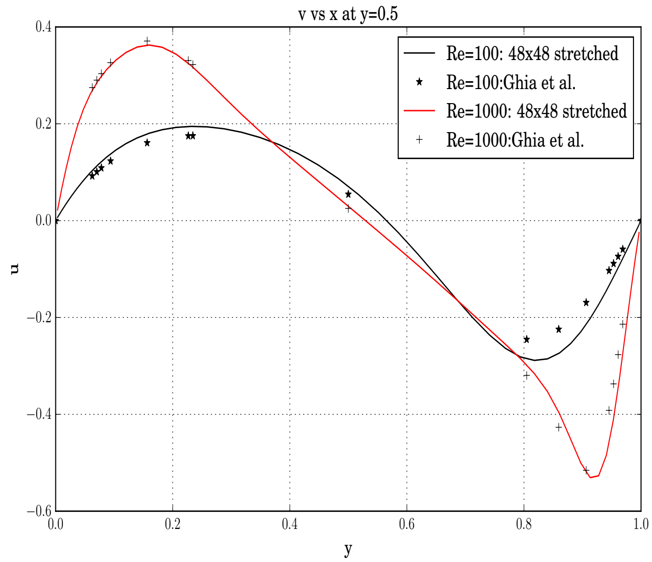

Figure 2: u vs y at x=0.5

As seen from the plot, the numerical predictions are in good agreement with the benchmark data. A slight mismatch can be observed towards the right wall at Re=100.

The following contour plots show the computed vorticity and streamfunction. The contour levels have been adjusted so as to match those in the paper.

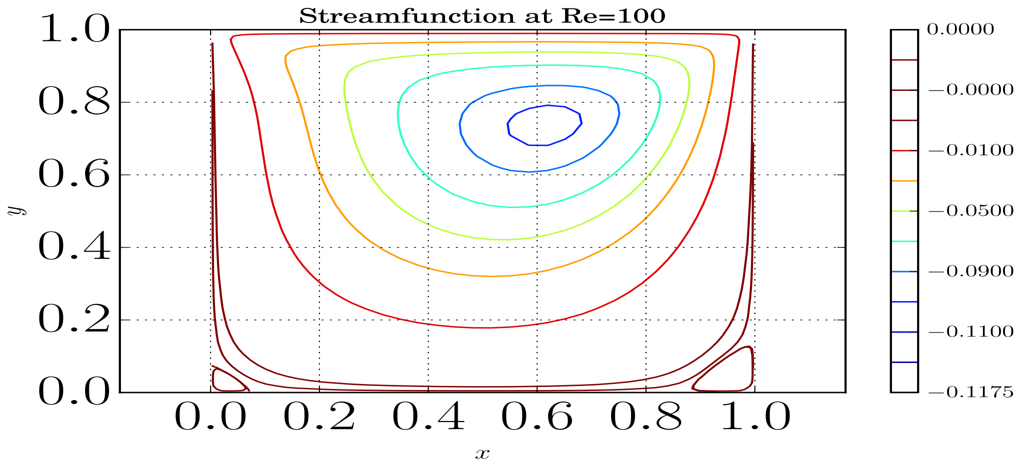

Figure 3: Contour plot of ψ for Re=100

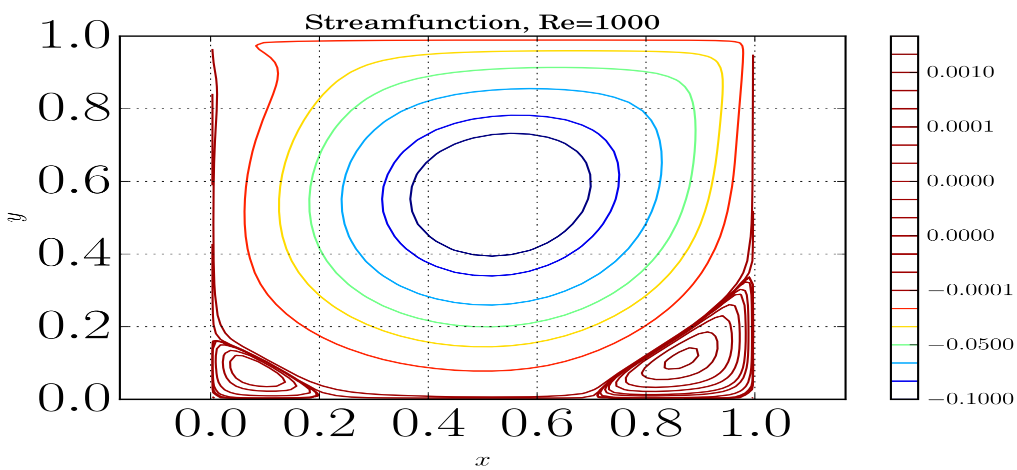

Figure 4: Contour plot of ψ for Re=1000

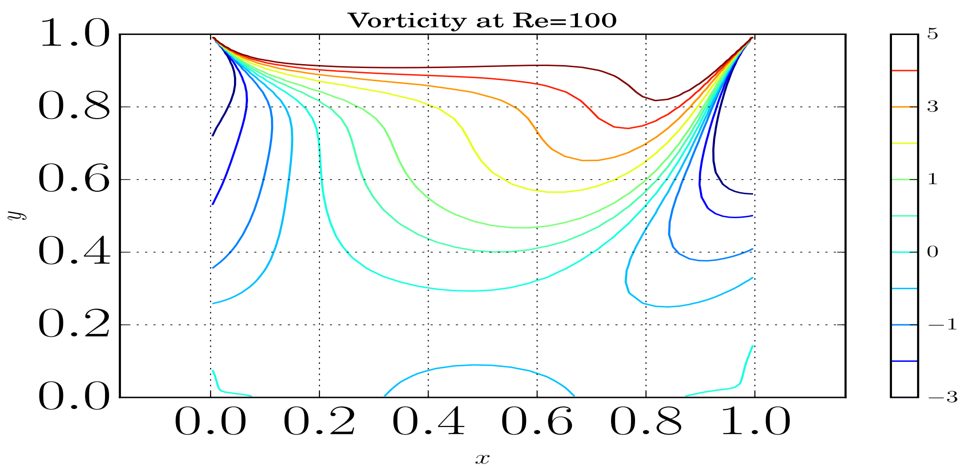

Figure 5: Contour plot of ω_z for Re=100

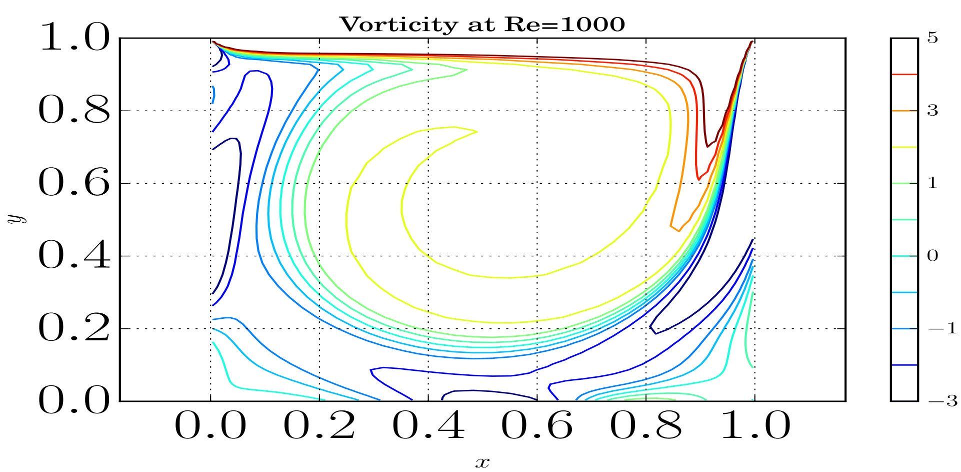

Figure 6: Contour plot of ω_z for Re=1000

A good agreement is observed with the contour plots from the paper for both Re values, in terms of the maximum and minimum values of ψ and ω. The corner vortices have been resolved adequately (Very low values of ψ appear to be blended due to the colour scheme of the contour plot). A mismatch is observed for vorticity at Re=1000 in the central region. This spurious behaviour can be attributed to the low resolution of stretched grid configuration in that area.

6. Bonus Part (HW3)

The following plots show the variation of the RMS error for the Taylor Green Vortex case at time t=1.0. Time advancement has been done explicitly using 1^{st}, 2^{nd}, 3^{rd} and 4^{th} order Runge-Kutta methods. A time step of 0.0001 has been used.

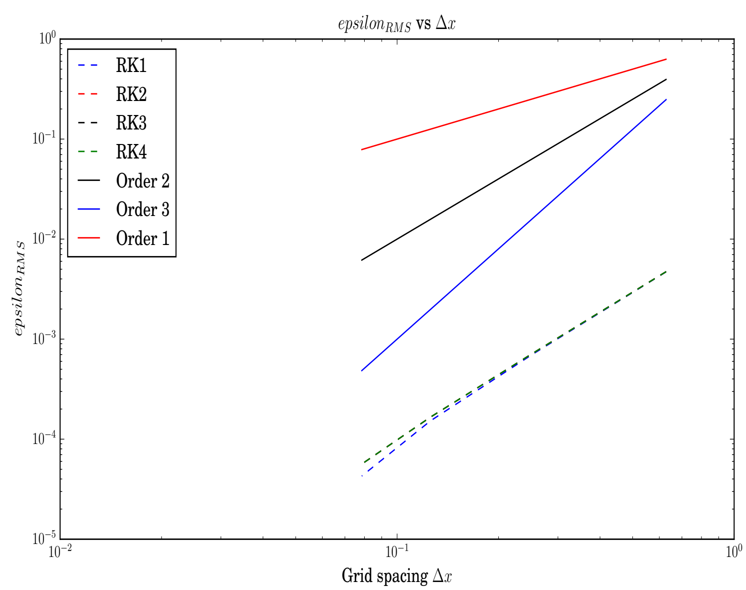

Figure 7: RMS error decay for u velocity

As seen from the plot, the error decay is second order, which is consistent with the spatial discretization being used.

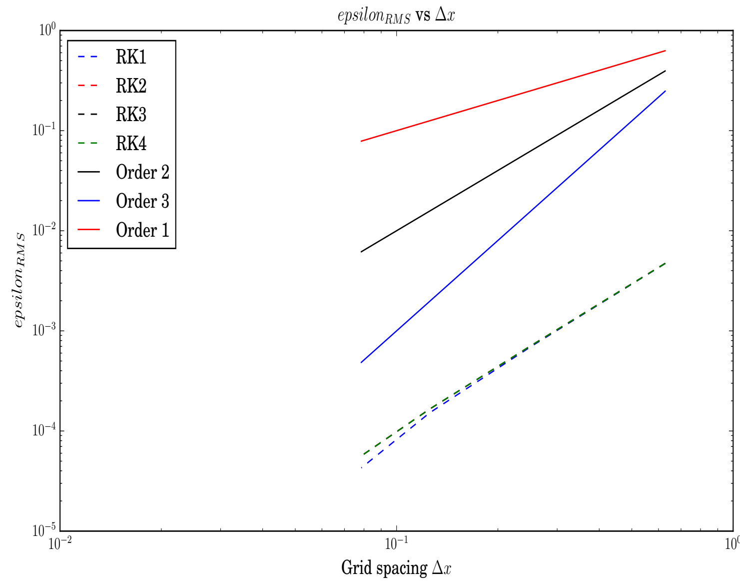

Figure 8: RMS error decay for v velocity

7. Conclusion

A finite-difference based Navier-Stokes solver was written in python. The solver was benchmarked against numerical results from Ghia et al. and showed reasonably good agreement with the published results. Furthermore, the Taylor Green Vortex problem was solved for in a periodic domain and the RMS error decay was found to be 2^{nd} order, which is consistent with the spatial discretization formulation being used.