The Lid Driven Cavity Flow Analysis using Predictor Corrector Method on Staggered Grid Arrangement

Suraj Pawar*

Department of Mechanical Engineering,
Virginia Polytechnic Institute and State University
surajp92@vt.edu

   A finite volume flow solver is developed to solve unsteady two dimensional incompressible Navier- Stokes equation with primitive variables in non-dimensional form. The classical Lid Driven Cavity problem is studied using this flow solver for two Reynolds number. The grid size and the time step size is varied uniformly to study the vortices formed in the flow field. The Navier- Stokes equation is discretized using second order central difference scheme for spatial terms and with Adams- Bashforth method for unsteady term on staggered grid arrangement.

I. Nomenclature

u_{Lid}	=	Velocity of top moving Lid
i, j	=	Spatial index in x and y direction
ρ	=	Density of the fluid
μ	=	Dynamic viscosity of the fluid
u, v	=	Velocity in x and y direction
p	=	Pressure
L	=	Dimension of square cavity
ΔT	=	time step
D_i	=	Divergence Operator
G_i	=	Gradient Operator
Re	=	Reynolds Number
U, V	=	Non-dimensional velocities in X and Y direction
ρ^*	=	Non-dimensional density
μ^*	=	Non-dimensional dynamic viscosity

II. Introduction

   This work addresses the classical problem [1] of laminar flow inside a cavity in which the top lid moves with a constant velocity. This problem has been widely used for studying various numerical methods and for validation of codes to solve the Navier- Stokes equation [2–4].

   Several numerical methods have been developed to solve this problem. Ghia et al. [1] used vorticity stream function formulation with implicit multigrid method to study high Reynolds number flow on fine mesh. Hou et al. [5] used Lattice Boltzmann method to simulate the cavity flow. Barragy and Carey [6] implemented finite element method for the fully coupled stream function-vorticity formulation of the Navier-Stokes equations to study cavity flow.

   The scope of this work is to present numerical solution of unsteady incompressible Navier-Stokes equation with staggered grid arrangement for dependent variables like pressure and velocity. All the spatial terms are discretized using central difference schemes to increase the accuracy to second order. Adams- Bashforth method is used for unsteady term so that temporal error is also second order. All the calculation are done in double precision to reduce the machine round off error. Gauss Seidel method is used to solve the pressure correction equation. The steady state is assumed when L2 norm of the change in x- and y- directional velocity between two consecutive time step is less than 1.E-8

   Solution is presented for Reynolds number $Re = 100$ for three grid sizes and three time steps for each grid with uniform refinement. The code is validated by comparing the x- and y- directional velocities at the center line of the

---

cavity with results from Ghia et al. [1]. The calculation is done for $Re = 1000$ on fine grid and with largest possible time step.

III. Problem Description

Lid driven cavity flow consists of the flow in a square cavity with the top lid moving at certain velocity. The moving lid creates the strong vortex and the sequence of weaker vortices in the bottom two corners.

Since, the fluid velocity is very small as compared to the speed of sound the flow can assumed to be incompressible. The incompressible fluid flow is governed by continuity equation and momentum equation in each direction. There are four unknowns p, u, v, w and there are four equations. However, there is no dependent equation for pressure. This complicates the solution to Navier-Stokes equation because the pressure gradient contributes to the momentum equation. In compressible flow the continuity equation is used to determine the density and then pressure is calculated using equation of state. This approach can not be used for incompressible flow.

Usually for incompressible flows, the continuity and momentum equation are combined to obtain pressure equation. This pressure equation is elliptic in nature and has to be solved at every time step to obtain pressure field. The pressure calculated from this equation is used to correct velocity field such that the velocity field satisfies the continuity equation. These methods are referred to as the Predictor- Corrector method.

IV. Governing Equation and Boundary Conditions

For the Lid Driven Cavity problem, flow is assumed to be incompressible. Hence the density and viscosity of the fluid remains constant. We can obtain the velocity and pressure field by solving the Incompressible Navier- Stokes equation using Finite Volume Method.

A. Governing Equation

The assumption of Incompressible flow eliminates the need of solving energy equation. Dimensional form of continuity, x- momentum and y- momentum equation are given as

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \tag{1}$$

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho uu)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}(\mu\frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(\mu\frac{\partial u}{\partial y}) \tag{2}$$

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho vu)}{\partial x} + \frac{\partial(\rho vv)}{\partial y} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}(\mu\frac{\partial v}{\partial x}) + \frac{\partial}{\partial y}(\mu\frac{\partial v}{\partial y}) \tag{3}$$

Above equations can be non-dimensionalized using the lid velocity as the characteristic velocity scale and length of square cavity as the characteristic length scale. Below are the non-dimensional parameters

$$U = \frac{u}{u_{Lid}}; V = \frac{v}{u_{Lid}}; P = \frac{p}{\rho_{ref}u_{Lid}^2}; T = t\frac{u_{Lid}}{L}; X = \frac{x}{L}; Y = \frac{y}{L}; \rho^* = \frac{\rho}{\rho_{ref}}; \mu^* = \frac{\mu}{\mu_{ref}}; Re = \frac{\rho^* U_{Lid}L}{\mu^*}$$

Non-dimensional form of continuity equation is given as

$$\frac{\partial(\rho^*\rho_{ref}u_{Lid}U)}{\partial(XL)} + \frac{\partial(\rho^*\rho_{ref}u_{Lid}V)}{\partial(YL)} = 0$$

Since ρ_{ref}, L, U_{lid} are constants we can write non-dimensional continuity equation as below

$$\frac{\partial(\rho U)}{\partial X} + \frac{\partial(\rho V)}{\partial Y} = 0 \tag{4}$$

Non-dimensional form of X- momentum equation is given as

$$\frac{u_{Lid}^2}{L}\frac{\partial(\rho^*\rho_{ref}U)}{\partial T} + \frac{u_{Lid}^2}{L}\frac{\partial(\rho^*\rho_{ref}UU)}{\partial X} + \frac{u_{Lid}^2}{L}\frac{\partial(\rho^*\rho_{ref}UV)}{\partial Y} = -\frac{u_{Lid}^2\rho_{ref}}{L}\frac{\partial P}{\partial X} +$$
$$\frac{u_{Lid}}{L^2}\frac{\partial}{\partial X}(\mu^*\mu_{ref}\frac{\partial U}{\partial X}) + \frac{u_{Lid}}{L^2}\frac{\partial}{\partial Y}(\mu^*\mu_{ref}\frac{\partial U}{\partial Y})$$

We can multiply by $\frac{L}{U_{Lid}{}^2\rho_{ref}}$ throughout the above equation. We get

$$\frac{\partial(\rho^*U)}{\partial T} + \frac{\partial(\rho^*UU)}{\partial X} + \frac{\partial(\rho^*UV)}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{Re}\frac{\partial}{\partial X}(\mu^*\frac{\partial U}{\partial X}) + \frac{1}{Re}\frac{\partial}{\partial Y}(\mu^*\frac{\partial U}{\partial Y}) \tag{5}$$

Similarly, non-dimensional form of Y-momentum equation can be written as

$$\frac{\partial(\rho^*V)}{\partial T} + \frac{\partial(\rho^*VU)}{\partial X} + \frac{\partial(\rho^*VV)}{\partial Y} = -\frac{\partial P}{\partial Y} + \frac{1}{Re}\frac{\partial}{\partial X}(\mu^*\frac{\partial V}{\partial X}) + \frac{1}{Re}\frac{\partial}{\partial Y}(\mu^*\frac{\partial V}{\partial Y}) \tag{6}$$

Non-dimensional equation (4-6) are further used for obtaining discretized equations using staggered grid arrangement.

B. Boundary Condition

For all the walls no slip and no penetration boundary condition is used. No slip boundary condition assumes that relative velocity between the fluid and solid at the boundary is zero. Hence, for moving wall the fluid will have the same velocity as the lid velocity at the boundary. For non-moving walls the fluid velocity will be zero at the boundaries. For square cavity velocity boundary conditions cans be written as

$$U_{i,j} = U_{Lid}, \ V_{i,j} = 0 \quad \text{for } [i = 1 - i_{max}, \ j = j_{max}]$$
$$U_{i,j} = 0, \ V_{i,j} = 0 \quad \text{for } [i = 1 - i_{max}, \ j = 1]$$
$$U_{i,j} = 0, \ V_{i,j} = 0 \quad \text{for } [j = 1 - j_{max}, \ i = 1]$$
$$U_{i,j} = 0, \ V_{i,j} = 0 \quad \text{for } [j = 1 - j_{max}, \ i = i_{max}]$$

For pressure we use Neumann boundary condition at all walls. Hence, gradient of pressure normal to wall will be zero at all boundaries. Mathematically pressure boundary condition can be written as

$$\frac{\partial P}{\partial Y} = 0 \quad \text{for } [i = 1 - i_{max}, \ j = j_{max} \ and \ j = 1]$$
$$\frac{\partial P}{\partial X} = 0 \quad \text{for } [j = 1 - j_{max}, \ i = i_{max} \ and \ i = 1]$$

V. Numerical Discretization

In this section we discuss discretization of the Incompressible Navier-Stokes equation. Before we move to discretization we have to decide on the points at which to store unknown variables to be computed. There are two types of arrangement that are possible. The first type is a colocated grids in which all the variables (velocity and pressure) are stored at the nodes of control volume. The second type is a staggered grid arrangement in which the velocities are stored at the faces of pressure control volume. This type of arrangement is shown in Fig. 1. In this arrangement, several terms that requires interpolation with the colocated arrangement can be calculated without interpolation. The staggered grid has better coupling between the velocities and pressure. In the present study, staggered grid is used for discretization.
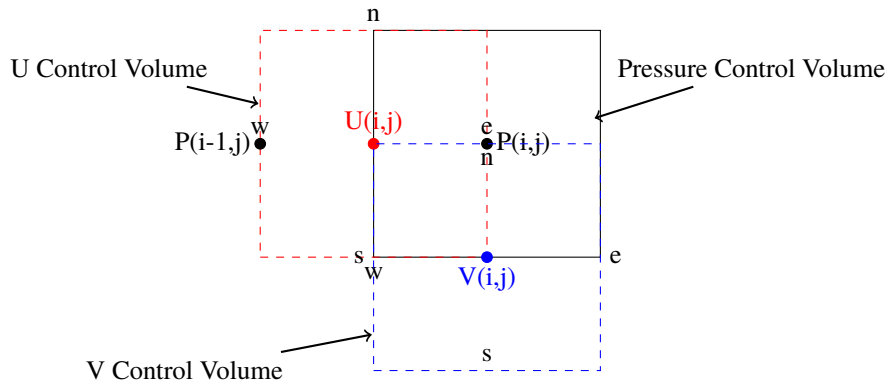

Fig. 1   Staggered Grid Arrangement

The momentum equation is discretized using finite volume method with central difference scheme for convection and diffusion terms. The discretization can be done as below:

$$\int_w^e \int_y \frac{\partial(\rho^* UU)}{\partial X}\,dy\,dx = ((\rho^* UU)_e - (\rho^* UU)_w)\Delta Y$$

$$\int_s^n \int_x \frac{\partial(\rho^* UV)}{\partial Y}\,dy\,dx = ((\rho^* UV)_n - (\rho^* UV)_s)\Delta X$$

$$\int_w^e \int_y \frac{\partial}{\partial X}(\mu^* \frac{\partial U}{\partial X})\,dy\,dx = ((\mu^* \frac{\partial U}{\partial X})_e - (\mu^* \frac{\partial U}{\partial X})_w)\Delta Y$$

$$\int_s^n \int_y \frac{\partial}{\partial Y}(\mu^* \frac{\partial U}{\partial Y})\,dy\,dx = ((\mu^* \frac{\partial U}{\partial Y})_n - (\mu^* \frac{\partial U}{\partial Y})_s)\Delta X$$

The pressure gradient term can be discretized using the pressure stored at pressure control volume nodes. Since we are using staggered grid arrangement pressure is directly available at nodes which is to be used in pressure gradient calculation. Using second order central difference

$$-\int_w^e \int_y \frac{\partial P}{\partial X}\,dy\,dx = -(P(i,j) - P(i-1,j))\Delta Y$$

The cell face flux can be calculated by interpolating the velocity stored the u and v velocity control volume nodes as

$$(\rho^* U)_e = \frac{\rho^* U(i+1,j) + \rho^* U(i,j)}{2}; \quad (\rho^* U)_w = \frac{\rho^* U(i-1,j) + \rho^* U(i,j)}{2}$$

$$(\rho^* V)_n = \frac{\rho^* V(i,j+1) + \rho^* V(i-1,j+1)}{2}; \quad (\rho^* V)_s = \frac{\rho^* V(i-1,j) + \rho^* V(i,j)}{2}$$

The velocity at the faces of control volume can be approximated using central difference scheme as below

$$U_e = \frac{U(i+1,j) + U(i,j)}{2}; \quad U_w = \frac{U(i-1,j) + U(i,j)}{2}$$

$$U_n = \frac{U(i,j+1) + U(i,j)}{2}; \quad U_s = \frac{U(i,j-1) + U(i,j)}{2}$$

The velocity gradient at the cell faces can be calculated using central difference scheme as follow

$$\frac{\partial U}{\partial X}_e = \frac{U(i+1,j) - U(i,j)}{\Delta X}; \quad \frac{\partial U}{\partial X}_w = \frac{U(i,j) - U(i-1,j)}{\Delta X}$$

$$\frac{\partial U}{\partial Y}_n = \frac{U(i,j+1) - U(i,j)}{\Delta Y}; \quad \frac{\partial U}{\partial Y}_s = \frac{U(i,j) - U(i,j-1)}{\Delta Y}$$

The time derivative of velocity is given as

$$\int_t^{t+\delta t} \frac{\partial \rho^* U}{\partial T} = \frac{\rho^* U^{n+1} - \rho^* U^n}{\Delta T}$$

Based on the value of other variables used in calculation of dependent variable at next time step the method can be either implicit, semi-implicit or explicit. In the present study, explicit discretization method is used. To increase the temporal accuracy Adams Bashforth method is used with the Euler method for first time step. The stability of explicit method is governed by Courant- Frederics- Levy condition and Von Neumann stability condition. In general performing stability analysis of complete Navier- Stokes equation is difficult because of its non linear nature. Hence, we can refer to linear Burger's equation for stability analysis for present problem. Instead if we use implicit method we can use larger time step because implicit method is not restricted by stability condition. If the time step used in implicit method is very large then temporal accuracy of solution will be not good and hence use of smaller time step is recommended for implicit method.

VI. Algorithm and Solution Procedure

As discussed in previous section the discretized can be done either by explicit method implicit method. For explicit method, the time step is restricted by stability criteria. For implicit method the system matrix is very stiff and difficult to invert. For deriving the discretized equation at each node it will be convenient to write Navier-Stokes equation in Cartesian Tensor notation

$$D_i \cdot (\rho U_i) = 0 \tag{7}$$

$$\frac{\partial \rho^* U_i}{\partial T} = -D_j \cdot (\rho^* U_i U_j) - G_i P + D_j \cdot (\mu^* G_j U_i) \tag{8}$$

In the above equation both convection and diffusion terms are treated at previous time levels $n, n-1, n-2...$ The solution of pressure at each time step is important but the history of pressure field is not needed for solving Navier-Stokes equation. The pressure at each time step should be such that the velocity field satisfies the continuity equation. If we neglect the pressure gradient term then the velocity calculated from momentum equation will not be a correct velocity. This velocity can be later corrected using the pressure correction which is calculated from continuity equation.
Let

$$H_i = -D_j \cdot (\rho^* U_i U_j) + D_j \cdot (\mu^* G_j U_i) \tag{9}$$

The above term can be calculated using Euler step. Euler explicit method is first order accurate in time. In the present study Adams-Bashforth time advancement method is used to get second order temporal accuracy. The intermediate velocity field is given by

$$\rho^* \tilde{U}_i = \rho^* U_i{}^n + \int_{T_n}^{T_n + \Delta T} (\frac{3}{2} H_i{}^n - \frac{1}{2} H_i{}^{n-1}) \tag{10}$$

The convection and diffusion terms in the above equation can be calculated using second order central difference scheme as discussed in previous section. The velocity has to be corrected using the pressure gradient term such that the corrected velocity satisfies the continuity equation. This can be done as follow

$$\rho^* U_i^{n+1} = \rho^* \tilde{U}_i - \Delta T G_i P^{n+1} \tag{11}$$

Substituting the above relation in continuity equation we get pressure correction equation as follow.

$$D_i \cdot (G_i P^{n+1}) = \frac{1}{\Delta T} D_i \cdot (\rho^* \tilde{U}_i^*) \tag{12}$$

The above equation is elliptic in nature and can be solved using an iterative procedure. There are various iterative methods such as Gauss Seidel method, Jacobi method, ADI method and Multigrid method can be used. Also, Krylov subspace methods such as Conjugate Gradient method can be used to speed up the convergence. In the present study Gauss Seidel method is used for solving pressure correction equation because of its simplicity. Special care has to be taken for pressure correction equation when dealing with the points adjacent to boundaries. Based on the location of grid, there are eight conditions for pressure correction boundary points.

All the calculations are done with an initial velocity and pressure field assigned to zero. It is assumed that the steady state is reached when L2 norm of the change in X-and-Y directional velocity between two consecutive time step goes below 1.E-8. Pressure correction equation is solved till the L2 norm of the residual goes below 1.E-5. The pressure field is normalized after every time step by subtracting the pressure at $(2, 2)$ location. If this is not done the truncation error will be large due to large value of pressure.

Since we are using explicit method for discretization the time step is limited by Courant–Friedrichs–Lewy condition and Von Neumann condition. Since Navier-Stokes equation is non linear it is difficult to find stability criteria for them. Hence we can used Burger's equation as reference for stability criteria since it has unsteady, convection and diffusion term. The maximum time step can be calculated using these condition as below

$$CFL = \frac{U \Delta T}{\Delta X} \leq 1; \ r = \mu^* \Delta T (\frac{1}{\Delta X^2} + \frac{1}{\Delta Y^2}) \leq \frac{1}{4}; \ Pe = \frac{CFL}{r} \leq \frac{4}{CFL} \tag{13}$$

The time step should be such that it satisfies all the above conditions. Using above conditions for $Re = 100$ and 32 x 32 grid the time step should be less than 0.0122 seconds for stability. For $Re = 1000$ and 128 x 128 grid the time step should be less than 0.0076 for stability and convergence.

Fig. 2 shows the algorithm of explicit method used for solving unsteady incompressible Navier-Stokes equation. The solver is developed using Fortran 95 as the programming language. The program is broken down into number of subroutines. The main program calls each of these subroutines in the correct order as per the algorithm. All the calculations are done in double precision using quad core Intel i7, 2GHz processor and 8 GB of RAM. The results are written in .dat file and later on plotted using Tecplot.
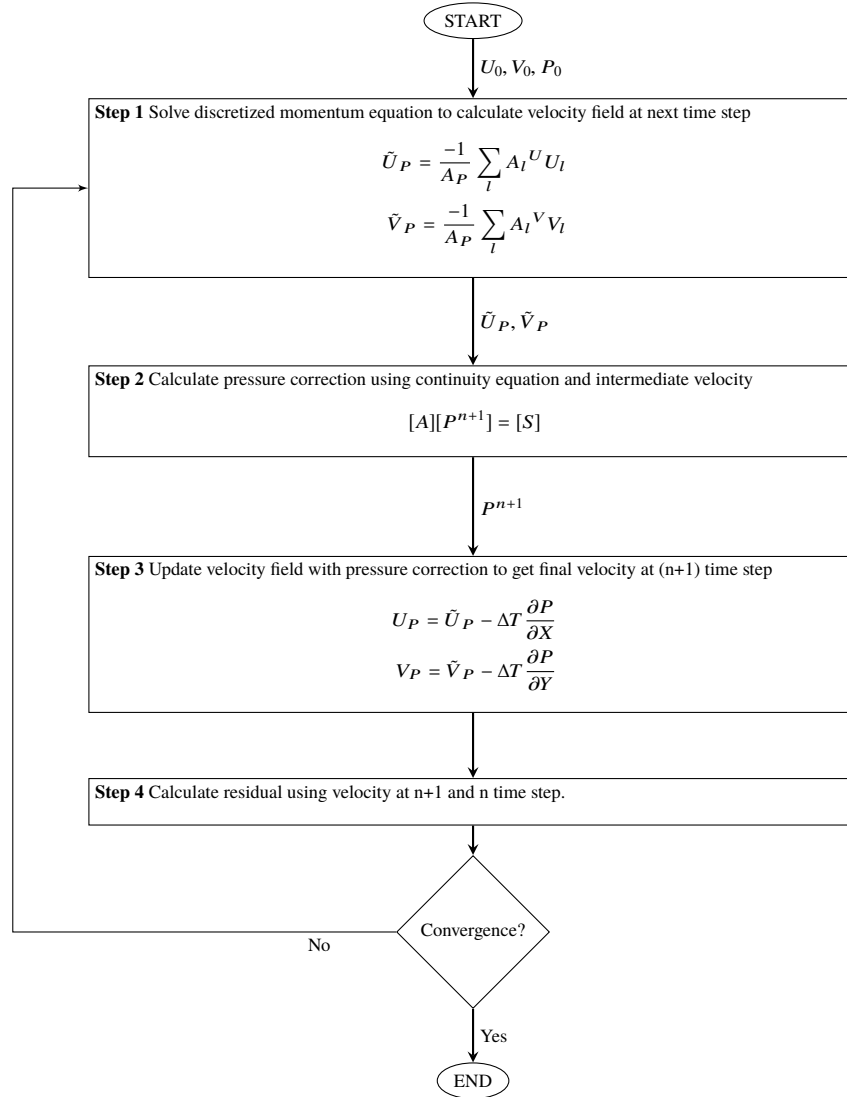
START

U_0, V_0, P_0

Step 1 Solve discretized momentum equation to calculate velocity field at next time step

$$\tilde{U}_P = \frac{-1}{A_P} \sum_l A_l{}^U U_l$$

$$\tilde{V}_P = \frac{-1}{A_P} \sum_l A_l{}^V V_l$$

$\tilde{U}_P, \tilde{V}_P$

Step 2 Calculate pressure correction using continuity equation and intermediate velocity

$$[A][P^{n+1}] = [S]$$

P^{n+1}

Step 3 Update velocity field with pressure correction to get final velocity at (n+1) time step

$$U_P = \tilde{U}_P - \Delta T \frac{\partial P}{\partial X}$$

$$V_P = \tilde{V}_P - \Delta T \frac{\partial P}{\partial Y}$$

Step 4 Calculate residual using velocity at n+1 and n time step.

Convergence?

No

Yes

END

Fig. 2   Algorithm for Explicit predictor-corrector method for Unsteady Incompressible Navier-Stokes Equation

VII. Results

A. *Re*= 100 Case

In this section we discuss the results for *Re*= 100 case.

1. Grid Convergence Study

Fig. 3 shows the grid convergence study done for the lid driven cavity problem. The grid was uniformly refined in each direction with the factor of two. At each grid level different time steps were used . In Fig. 3 the results are presented for smallest time step used on the corresponding grid. As the grid becomes finer the discretization error reduces and the results approaches benchmark results.[1].

(a) U- Velocity   (b) V- Velocity
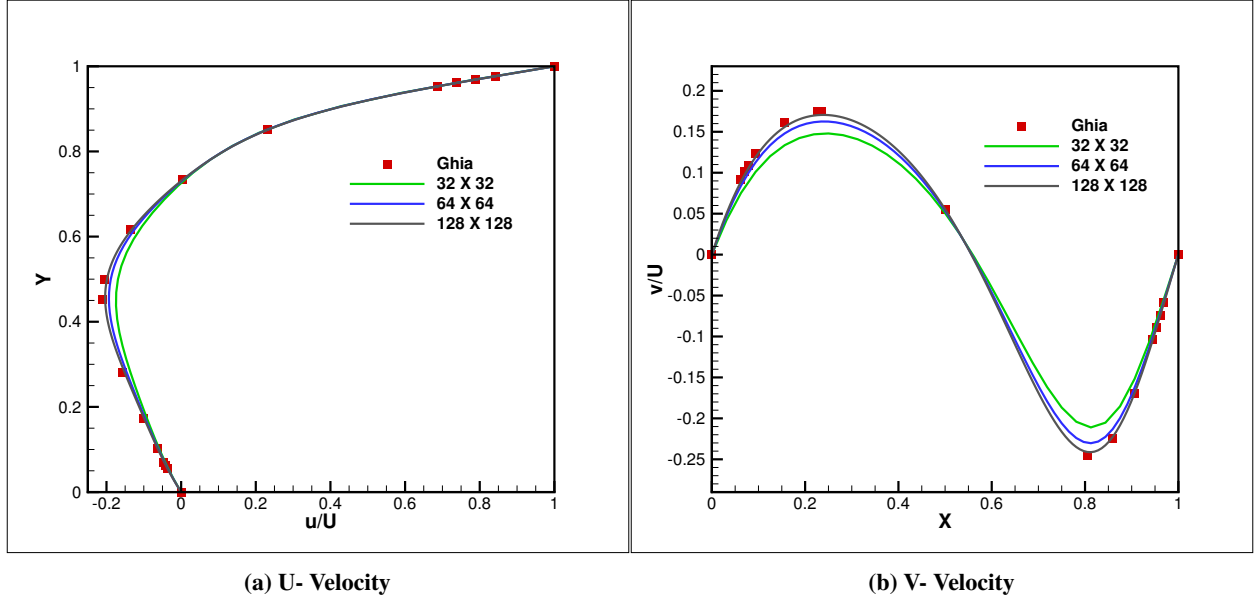
Fig. 3   Comparison of U and V Velocity profile through geometric center lines of square cavity for *Re* =100

2. Computational Time

The computational time required to reach steady state for different grids and different time step is given in Table 1. As the time step decreases the CPU time required to reach steady state increases. Also the time required for computation increases with the increase in grid density. For 32 X 32 grid and ΔT=0.02 the stability criteria given in equation 13 is not satisfied and hence the solution diverges.

Grid Size	Large (Seconds)	Medium (Seconds)	Small (Seconds)
32 X 32	-	1.4020	2.4102
64 X 64	11.5211	29.3786	48.0883
128 X 128	211.0041	356.8953	669.6274

Table 1   Computational time required to reach steady state for different grid sizes and different time step
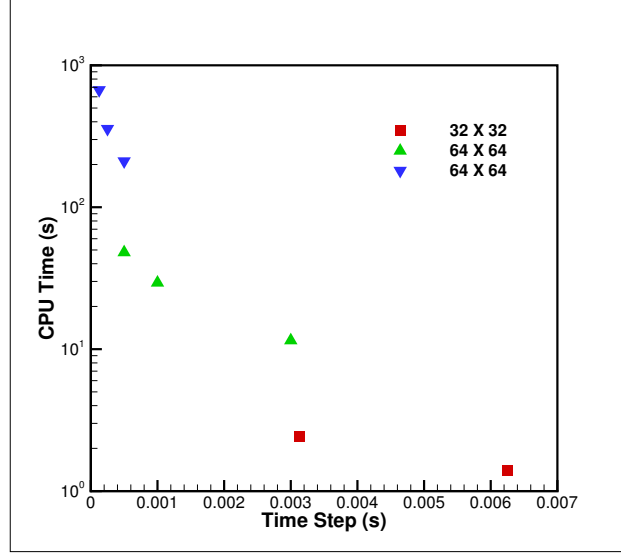
Fig. 4   CPU Time for all grid sizes and all time steps for *Re* = 100

3. Convergence to Steady State

Fig. 5 shows the residual history against non-dimensional time units for all grid sizes and time steps. The non-dimensional time unit is calculated by multiplying the time step and iteration number. The non-dimensional time units required to reach steady state decreases with the decrease in time step size. Also, the non-dimensional time units to attain steady state decreases with decrease in grid size.
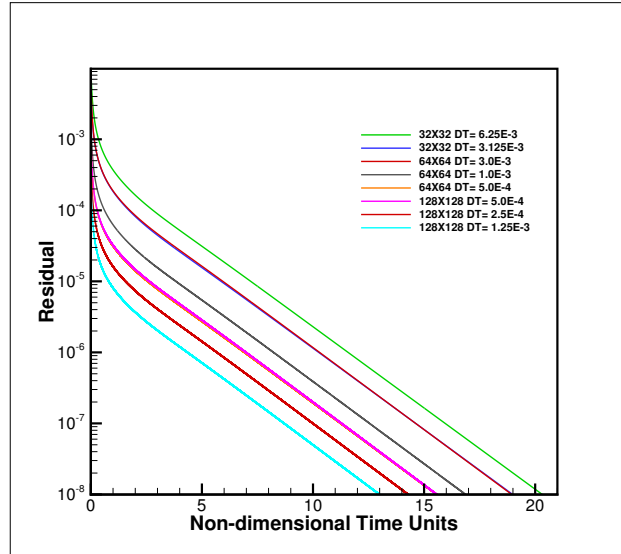

Fig. 5   Residual history against non dimensional time units for different grid sizes and time steps

4. Velocity and Pressure field

Fig. 6 shows the U-and-V velocity contours and pressure contour for *Re* =100 on fine grid and with the smallest time step $\Delta T = 0.000125s$. As can be seen from the Fig. 6a the strong vortex is formed in the square cavity along with two secondary vortices formed at bottom corners. The position and size of vortices are in good agreement with Ghia et al. [1]. From 6b we can see that the fluid has different direction of velocities in different part of the domain. From 6c we

8

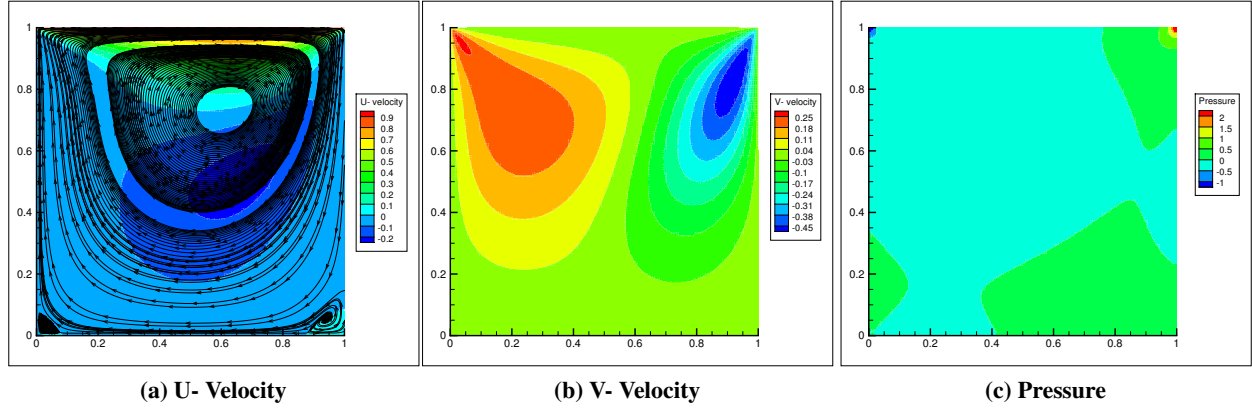can see the stagnation pressure at the top corners of the cavity.

(a) U- Velocity   (b) V- Velocity   (c) Pressure

Fig. 6   Velocity and Pressure contour in the square cavity for *Re* = 100, grid size 128 X 128 and ΔT= 1.25E-4

B. *Re*= 1000 Case

In this section we discuss the results for *Re* = 1000 case. The flow is simulated only for one grid size 128 X 128 with largest time step possible. Using equation 13 we get maximum time step to be 0.0076 that satisfies the stability condition. If we use ΔT =0.0076 the residuals reduces to 1.E-6 and then oscillates. The residuals converges below 1.E-8 when we use ΔT =0.0075. Fig. 7 shows the comparison of U and V velocity at geometric center lines with the results from Ghia et al. [1]. The accuracy of the results can be further increased by reducing the time step or by making the grid finer. The maximum and minimum V-velocity calculated in the present study is smaller than the velocity calculated by Ghia et al. [1]. Apart from that the results of the present study are in good agreement with the benchmark results.
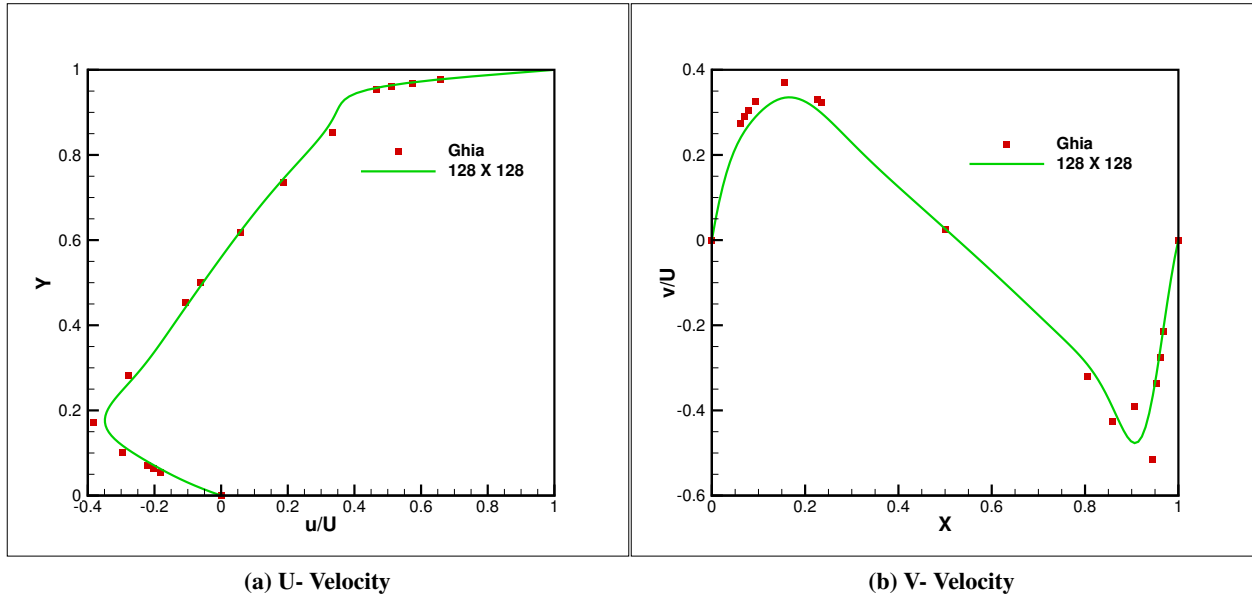

(a) U- Velocity   (b) V- Velocity

Fig. 7   Comparison of U and V velocity through geometric center lines of square cavity for *Re* = 1000 on 128 X 128 grid and ΔT =0.0075

Fig. 8 shows the comparison oh non-dimensional time units required to reach steady state for *Re* =100 and *Re* =1000 with ΔT =0.0005 and ΔT =0.0075 respectively. The non-dimensional time for convergence to steady state increases with increase in Reynold's number. Also the rate at which residuals reduces is not constant for higher Reynolds number

case. The rate of convergence decrease after 40 time units. Once the residual goes below a particular value the higher wave number errors are resolved. However there are still low wave number errors that needs to be resolved and fine grid takes more time to resolve these low wave number errors. If we instead use the coarse grid the convergence would be faster because these low wave number errors behave as high wave number errors on coarse grid.
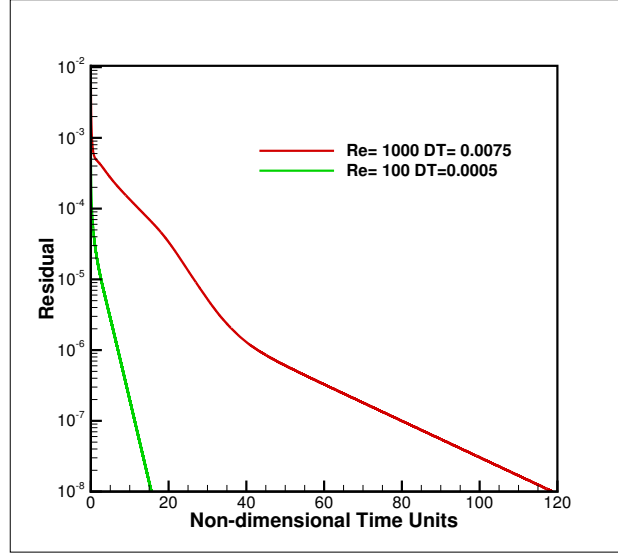

Fig. 8 Residual history against non dimensional time units for 128 X 128 grid and largest time step for two Reynolds Number

C. Improvement in Algorithm

In this section we discuss on what changes can be made in algorithm so that the problem reaches steady state faster. Below are some of the changes that can make the steady state convergence faster.

• The pressure correction equation is solved using Gauss Seidel method in the present code. Instead we can use faster methods such as Alternate Direct Implicit method, Krylov subspace methods like Conjugate Gradient method that solves the pressure correction equation faster than the Gauss Seidel method.

• Currently we are not using pressure gradient term when we calculate the intermediate velocity. The inclusion of pressure gradient term can make the steady state reach faster.

• The use of multigrid framework will also speed up the convergence to steady state. In multigrid framework, we solve the residuals equation for fine grid solution on coarse grid. The low wave number errors on fine grid are resolved faster on coarse grid because they act as high wave number error on coarse grid. This will accelerate the convergence to steady state.

• The current algorithm uses explicit method. This limits the maximum time step that we can use due to stability criteria. If we instead use semi-implicit or implicit method we can use larger time step and this will speed up the convergence.

VIII. Conclusion and Summary

Solution has been obtained for lid driven cavity flow using predictor-correction method for non-dimensional unsteady incompressible Navier-Stokes equation on staggered grid arrangement. The two parameters grid size and time step were studied for low Reynolds number flow. Various results such as computational time, non-dimensional time units, pressure and velocity field was studied for different grid sizes and time steps

The high Reynolds number flow was also analyzed on fine grid with largest time step possible. The results of present study are in good agreement with the benchmark results which validates the solver developed in present study. There are various ways in which the current solver can be improved and can me made more efficient and robust.

References

[1] Ghia, U., Ghia, K. N., and Shin, C., "High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method," *Journal of computational physics*, Vol. 48, No. 3, 1982, pp. 387–411.

[2] Marchi, C. H., Suero, R., and Araki, L. K., "The lid-driven square cavity flow: numerical solution with a 1024 x 1024 grid," *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, Vol. 31, No. 3, 2009, pp. 186–198.

[3] Botella, O., and Peyret, R., "Benchmark spectral results on the lid-driven cavity flow," *Computers & Fluids*, Vol. 27, No. 4, 1998, pp. 421–433.

[4] Bruneau, C.-H., and Saad, M., "The 2D lid-driven cavity problem revisited," *Computers & Fluids*, Vol. 35, No. 3, 2006, pp. 326–348.

[5] Hou, S., Zou, Q., Chen, S., Doolen, G., and Cogley, A. C., "Simulation of cavity flow by the lattice Boltzmann method," *Journal of computational physics*, Vol. 118, No. 2, 1995, pp. 329–347.

[6] Barragy, E., and Carey, G., "Stream function-vorticity driven cavity solution using p finite elements," *Computers & Fluids*, Vol. 26, No. 5, 1997, pp. 453–468.