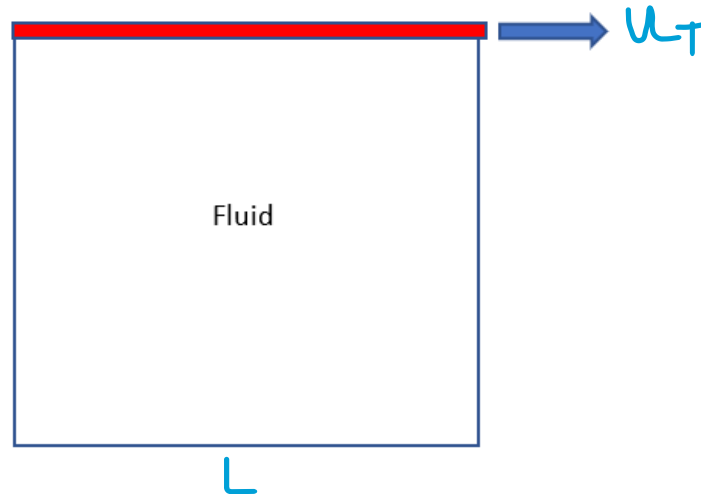


ME6434 Term Project report

Problem description

Consider a fluid inside a square cavity of dimension L bounded by four walls. All the walls except the top wall are fixed in space and time. The top wall is given a constant velocity u_T in the u direction, and we want to see the development of the u and v velocity profiles of the fluid in the whole domain, as it reaches steady state.



Governing equations and boundary conditions

For a two-dimensional fluid flow, the governing equations are the continuity and Navier Stokes equations. Assuming flow is incompressible, they are:

$$\left. \begin{aligned} \frac{\partial u^*}{\partial x^*} + \frac{\partial v^*}{\partial y^*} &= 0 \\ \frac{\partial u^*}{\partial t^*} + (u^* \cdot \nabla^*) u^* &= -\frac{1}{\rho^*} \nabla p^* + \nu^* \nabla^{*2} u^* \\ \frac{\partial v^*}{\partial t^*} + (v^* \cdot \nabla^*) v^* &= -\frac{1}{\rho^*} \nabla p^* + \nu^* \nabla^{*2} v^* \end{aligned} \right\} \text{---} \textcircled{1}$$

where $u^* = \text{velocity in the } x \text{ direction}$

$v^* = \text{velocity in the } y \text{ direction}$

$\rho^* = \text{density}$

$p^* = \text{pressure}$

$\nu^* = \text{kinematic viscosity}$

If we define a set of non-dimensional variables $x, y, t, u, v, p, \rho, \nu$ such that

$$(x, y) = \frac{(x^*, y^*)}{L}$$

$$t = u_T t^* / L$$

$$(u, v) = (u^*, v^*) / u_T$$

$$\rho = \rho^* / \rho_\infty$$

$$\nu = \nu^* / \nu_\infty$$

where L = length of the square cavity

u_T = lid velocity

ρ_∞ = undisturbed fluid density (Since fluid is assumed incompressible, $\rho^* = \rho_\infty$)

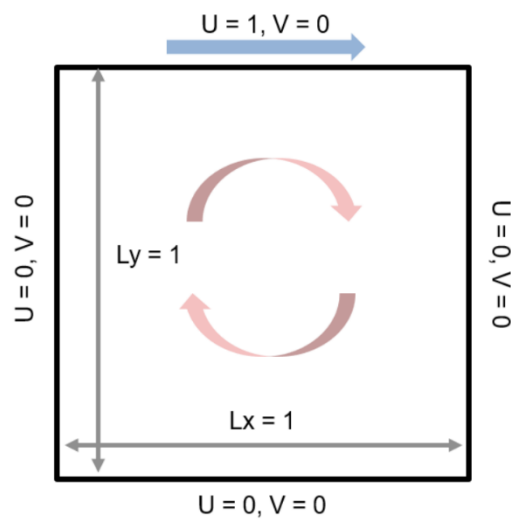
ν_∞ = undisturbed fluid kinematic viscosity

The non-dimensional Reynolds no. in this case would be

$$Re = \frac{u_T L}{\nu_\infty}$$

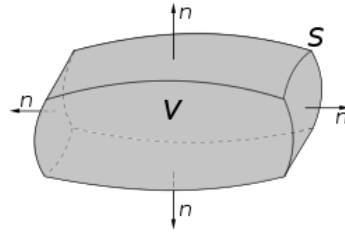
Substituting the variables in equation (1) in terms of their non-dimensional counterparts, we get

$$\left. \begin{aligned} \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0 \\ \frac{\partial u}{\partial t} + (u \cdot \nabla) u &= -\nabla p + \frac{1}{Re} \nabla^2 u \\ \frac{\partial v}{\partial t} + (v \cdot \nabla) v &= -\nabla p + \frac{1}{Re} \nabla^2 v \end{aligned} \right\} \text{--- (2)}$$



Discretization of governing equations

The above equations when integrated over a finite volume V look like below:



Continuity

$$\int_V \nabla \cdot \mathbf{u} dV = 0$$

$$\text{or, } \int_S \mathbf{u} \cdot \mathbf{n} dS = 0$$

X Momentum

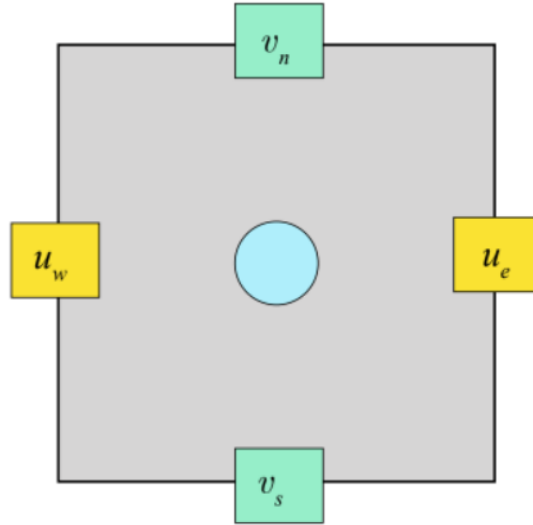
$$\frac{\partial}{\partial t} \int_V u dV = - \oint_S u(\mathbf{u} \cdot \mathbf{n} dS) - \oint_S p \cdot n_x dS + \frac{1}{Re} \oint_S \nabla u \cdot \mathbf{n} dS$$

Y Momentum

$$\frac{\partial}{\partial t} \int_V v dV = - \oint_S v(\mathbf{u} \cdot \mathbf{n} dS) - \oint_S p \cdot n_y dS + \frac{1}{Re} \oint_S \nabla v \cdot \mathbf{n} dS$$

where $\mathbf{u} = (u, v)$ is a vector and \mathbf{n} is the unit normal vector to the surface dS .

The staggered grid is used to discretize the above equations, which is explained below:



Continuity

$$\int_S \mathbf{u} \cdot \mathbf{n} dS = 0$$

$$\text{or, } \sum_{\text{faces}} (\mathbf{u} \cdot \mathbf{n})_{\text{each face}} dS_{\text{face}} = 0$$

$$u_e \delta y - u_w \delta y + v_n \delta x - v_s \delta x = 0$$

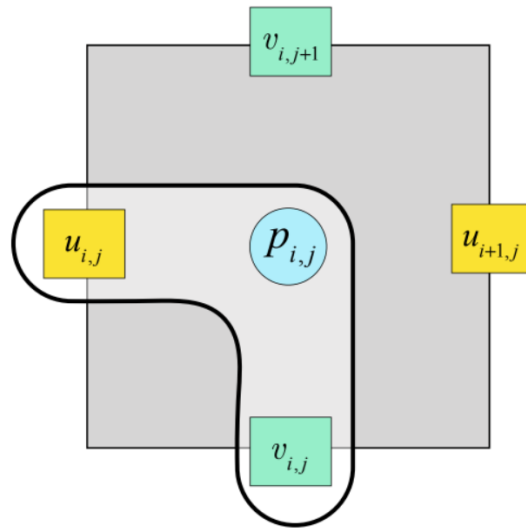
$$\text{or, } \frac{u_e - u_w}{\delta x} + \frac{v_n - v_s}{\delta y} = 0$$

If we want the new velocity at the next time step (n+1) to be divergence free, then,

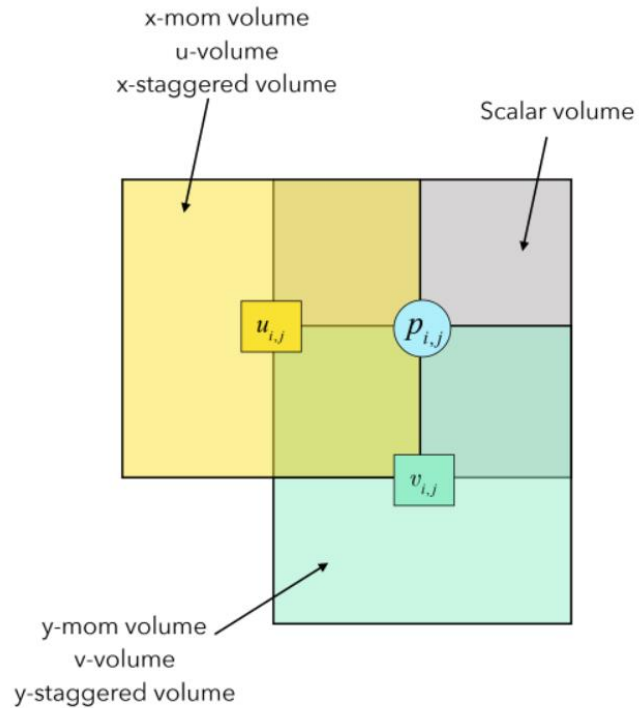
$$\frac{u_e^{n+1} - u_w^{n+1}}{\delta x} + \frac{v_n^{n+1} - v_s^{n+1}}{\delta y} = 0$$

Nothing has been said yet about the placement of variables. From the continuity equation, it seems that it is just natural to place the velocity field on the faces of the control volume. This leads us to the staggered grid arrangement.

There are several ways to stagger variables, however we will stagger the momentum and velocities to the negative faces of the control volume. Basically, the indexing of velocities on the minus side will be the same as the cell centered index.



Staggering implies different control volumes for different quantities. Scalars are placed at natural cell centers and are referred to as scalar volumes. Velocities (and momentum) have their own control volumes centered at the faces of the scalar volumes (for structured uniform grids).



The continuity equation the reads,

$$\frac{u_{i+1,j}^{n+1}-u_{i,j}^{n+1}}{\delta x}+\frac{v_{i,j+1}^{n+1}-v_{i,j}^{n+1}}{\delta y}=0$$

The diagram shows a central yellow square cell representing a control volume. The cell is bounded by a thick black line. The four corners of the cell are marked with teal squares containing the variables $v_{i-1,j+1}$ (top-left), $v_{i,j+1}$ (top-right), $v_{i-1,j}$ (bottom-left), and $v_{i,j}$ (bottom-right). The four edges of the cell are marked with yellow squares containing the variables $u_{i-1,j}$ (left), $u_{i,j}$ (center), $u_{i+1,j}$ (right), and $u_{i,j}$ (bottom). The edges are labeled with arrows and variables: the top edge is labeled $(vu)_n$ with an upward arrow; the bottom edge is labeled $(vu)_s$ with an upward arrow; the left edge is labeled $(uu)_w$ with a rightward arrow; and the right edge is labeled $(uu)_e$ with a rightward arrow.

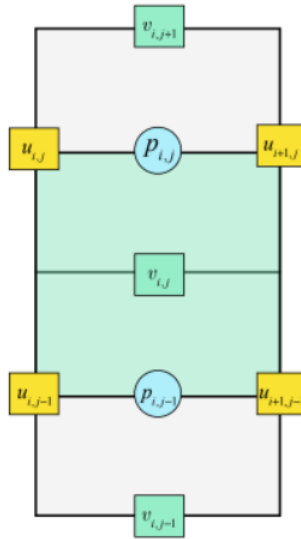
$$u_n = 0.5(u_{i,j} + u_{i,j+1})$$

$$u_s = 0.5(u_{i,j} + u_{i,j-1})$$

$$v_n = 0.5(v_{i-1,j+1} + v_{i,j+1})$$

$$v_s = 0.5(v_{i,j} + v_{i-1,j})$$

y-momentum convective flux

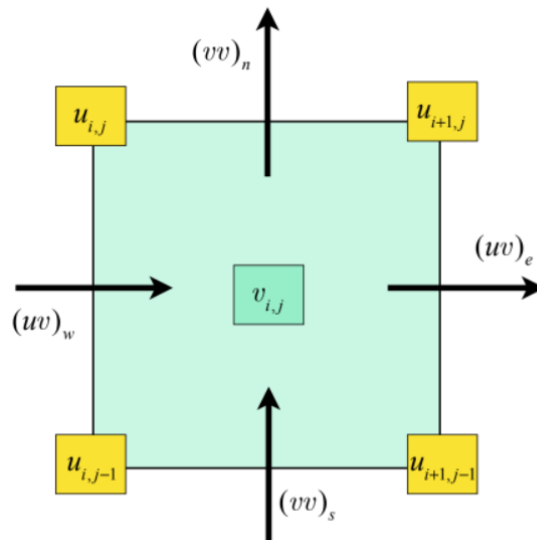


$$\oint_S v(\mathbf{u} \cdot \mathbf{n} dS) = (vu)_e \delta y$$

$$- (vu)_w \delta y$$

$$+ (vv)_n \delta x$$

$$- (vv)_s \delta x$$



$$u_e = 0.5(u_{i+1,j} + u_{i+1,j-1})$$

$$u_w = 0.5(u_{i,j-1} + u_{i,j})$$

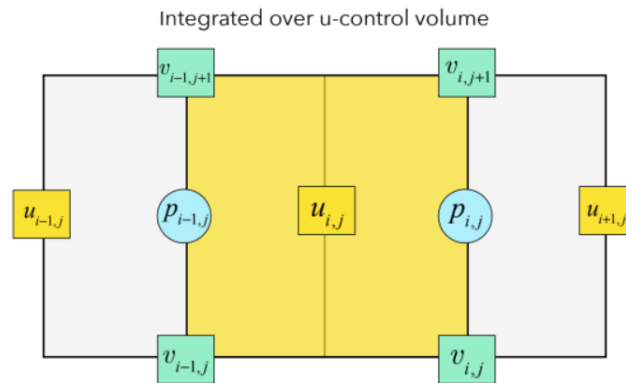
$$v_e = 0.5(v_{i,j} + v_{i+1,j})$$

$$v_w = 0.5(v_{i,j} + v_{i-1,j})$$

$$v_n = 0.5(v_{i,j} + v_{i,j+1})$$

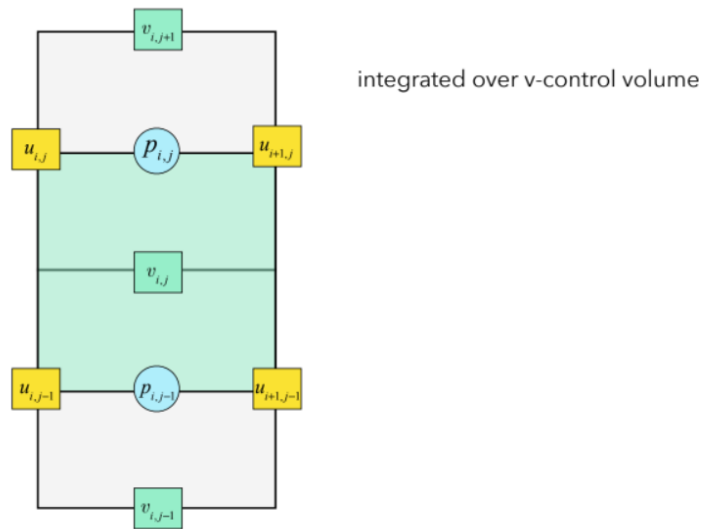
$$v_s = 0.5(v_{i,j} + v_{i,j-1})$$

x-direction pressure gradient



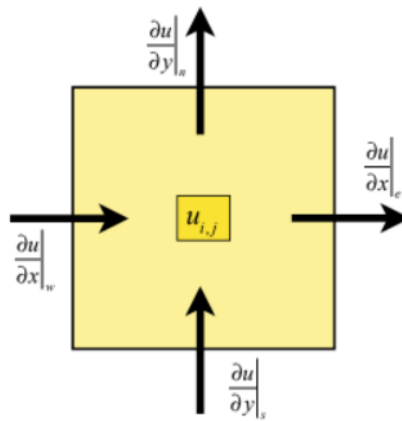
$$\oint_S p \cdot n_x = p_e \delta y - p_w \delta y = (p_{i,j} - p_{i-1,j}) \delta y$$

y-direction pressure gradient



$$\oint_S p \cdot n_y = p_n \delta x - p_s \delta x = (p_{i,j} - p_{i,j-1}) \delta x$$

x-direction diffusive flux



$$\oint_S \nabla u \cdot \mathbf{n} dS = \frac{\partial u}{\partial x_e} \delta y - \frac{\partial u}{\partial x_w} \delta y + \frac{\partial u}{\partial y_n} \delta x - \frac{\partial u}{\partial y_s} \delta x$$

Now,

$$\frac{\partial u}{\partial x_e} = \frac{u_{i+1,j} - u_{i,j}}{\delta x}$$

$$\frac{\partial u}{\partial x_w} = \frac{u_{i,j} - u_{i-1,j}}{\delta x}$$

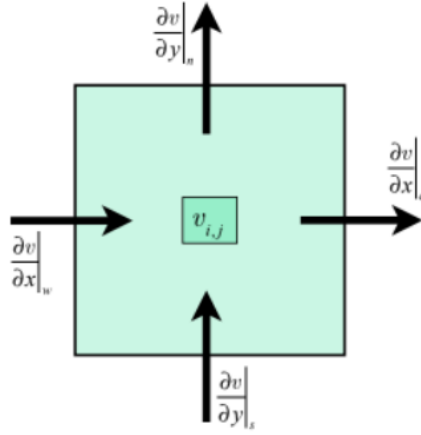
$$\frac{\partial u}{\partial y_n} = \frac{u_{i,j+1} - u_{i,j}}{\delta y}$$

$$\frac{\partial u}{\partial y_s} = \frac{u_{i,j} - u_{i,j-1}}{\delta y}$$

Thus,

$$\oint_S \nabla u \cdot \mathbf{n} dS = \frac{u_{i+1,j} - 2u_{i,j} + u_{i-1,j}}{\delta x} \delta y + \frac{u_{i,j+1} - 2u_{i,j} + u_{i,j-1}}{\delta y} \delta x$$

y-direction diffusive flux



$$\oint_S \nabla v \cdot \mathbf{n} dS = \frac{\partial v}{\partial x_e} \delta y - \frac{\partial v}{\partial x_w} \delta y + \frac{\partial v}{\partial y_n} \delta x - \frac{\partial v}{\partial y_s} \delta x$$

Now,

$$\frac{\partial v}{\partial x_e} = \frac{v_{i+1,j} - v_{i,j}}{\delta x}$$

$$\frac{\partial v}{\partial x_w} = \frac{v_{i,j} - v_{i-1,j}}{\delta x}$$

$$\frac{\partial v}{\partial y_n} = \frac{v_{i,j+1} - v_{i,j}}{\delta y}$$

$$\frac{\partial v}{\partial y_s} = \frac{v_{i,j} - v_{i,j-1}}{\delta y}$$

Thus,

$$\oint_S \nabla v \cdot \mathbf{n} dS = \frac{v_{i+1,j} - 2v_{i,j} + v_{i-1,j}}{\delta x} \delta y + \frac{v_{i,j+1} - 2v_{i,j} + v_{i,j-1}}{\delta y} \delta x$$

Solution method

The fractional step, or time-splitting, method solves the unsteady Navier-Stokes equations in a segregated manner. At each time step, an incomplete form of momentum equations is integrated to obtain an approximate velocity field, which is, in general, not divergence-free, then the velocity field is projected into the divergence-free field without changing vorticity. This projection step is achieved by solving the Poisson equation for pressure.

The incompressible Navier-Stokes equations in primitive variables are given by the equation

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u}$$

$$\nabla \cdot \mathbf{u} = 0$$

If we apply 2nd order central difference scheme for space discretization and 2nd order Adams Bashforth for time discretization, the above equation can be represented as

$$\frac{u^{n+1} - u^n}{\delta t} = -Gp + \frac{1}{2} \left\{ 3 \left(-Nu^n + \frac{1}{Re} Lu^n \right) - \left(-Nu^n + \frac{1}{Re} Lu^n \right) \right\}$$

$$Du^{n+1} = 0$$

In matrix form,

$$\begin{bmatrix} I & G \\ D & 0 \end{bmatrix} \begin{bmatrix} u^{n+1} \\ p' \end{bmatrix} = \begin{bmatrix} r^n \\ 0 \end{bmatrix}$$

where I is the identity matrix, N is the convective operator, G is the gradient operator, D is the divergence operator, r^n is the explicit right-hand side of the momentum equation, $p' = dt \cdot p$ and

$$r^n = u^n + \frac{\delta t}{2} \left\{ 3 \left(-Nu^n + \frac{1}{Re} Lu^n \right) - \left(-Nu^n + \frac{1}{Re} Lu^n \right) \right\}$$

LU representation of the fractional method

The fractional step method is related to the block LU factorization (Perot 1993).

$$\begin{bmatrix} I & 0 \\ D & DG \end{bmatrix} \begin{bmatrix} I & -G \\ 0 & I \end{bmatrix} \begin{bmatrix} u^{n+1} \\ p' \end{bmatrix} = \begin{bmatrix} r^n \\ 0 \end{bmatrix}$$

If we denote

$$\begin{bmatrix} I & -G \\ 0 & I \end{bmatrix} \begin{bmatrix} u^{n+1} \\ p' \end{bmatrix} = \begin{bmatrix} u^* \\ p' \end{bmatrix}$$

with the intermediate velocity u^* we obtain,

$$\begin{bmatrix} I & 0 \\ D & DG \end{bmatrix} \begin{bmatrix} u^* \\ p' \end{bmatrix} = \begin{bmatrix} r^n \\ 0 \end{bmatrix}$$

Now u^* is decoupled with the pressure term and calculations at each time step reduce to the following sequence of operations,

$$u^* = r^n$$

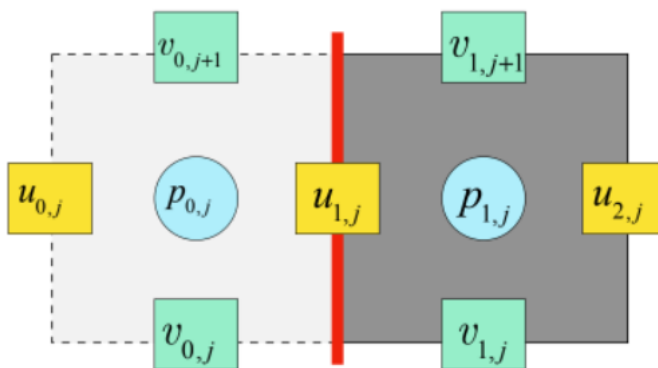
$$DGp' = Du^*$$

$$u^{n+1} = u^* - Gp'$$

Boundary conditions

Walls: Velocity

Example: take a wall on the x-minus (left) side



$$u_{1,j} = 0; \quad \forall j$$

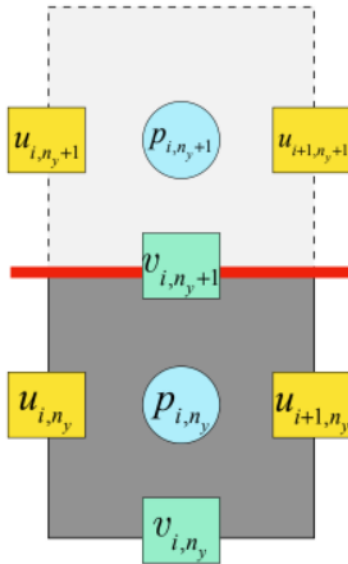
$$\frac{v_{1,j} + v_{0,j}}{2} = v_{\text{wall}}; \quad \forall j$$

$$u = 0$$

Can be directly specified on the boundary!

$$v = v_{\text{wall}} \quad \text{Use ghost cell specification:} \quad \frac{v_{\text{interior}} + v_{\text{ghost}}}{2} = v_{\text{wall}}$$

Example: take a wall on the y-plus (top) side



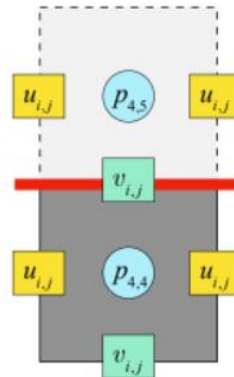
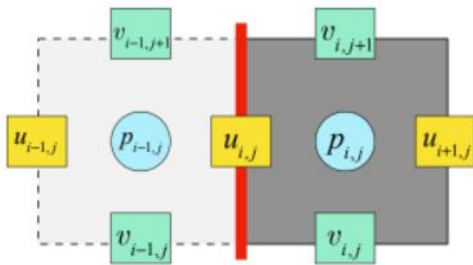
$$v = 0$$

$$u = u_{\text{wall}} \quad \text{Use ghost cell specification:}$$

$$\frac{u_{\text{interior}} + u_{\text{ghost}}}{2} = u_{\text{wall}}$$

$$v_{i,n_y+1} = 0; \quad \forall i$$

$$\frac{u_{i,n_y+1} + u_{i,n_y}}{2} = u_{\text{wall}}; \quad \forall i$$



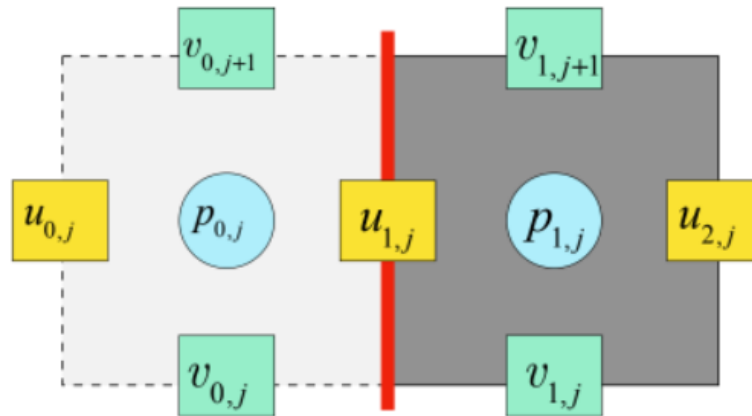
$$\mathbf{u} \cdot \mathbf{n} = 0 \quad \text{Can be directly specified on the boundary}$$

$$\mathbf{u} \cdot \mathbf{t} = v_{\text{wall}} \quad \text{Use ghost cell specification:}$$

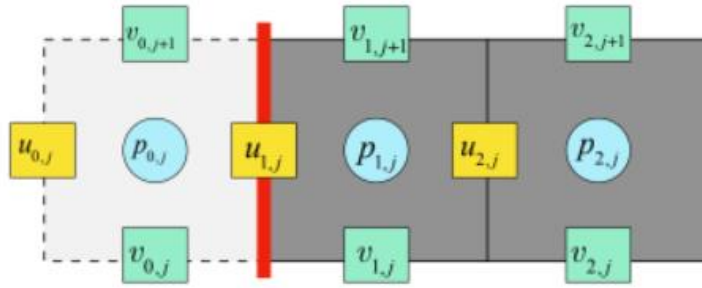
$$\frac{u_{\text{interior}} + u_{\text{ghost}}}{2} = u_{\text{wall}}$$

Walls: Pressure

Take an x-minus side for example



Apply continuity equation:
$$\frac{u_{2,j}^{n+1} - u_{1,j}^{n+1}}{\Delta x} + \frac{v_{1,j+1}^{n+1} - v_{1,j}^{n+1}}{\Delta y} = 0$$



Apply continuity equation:
$$\frac{u_{2,j}^{n+1} - u_{1,j}^{n+1}}{\Delta x} + \frac{v_{1,j+1}^{n+1} - v_{1,j}^{n+1}}{\Delta y} = 0$$

But, $u_{1,j}^{n+1} = 0; \quad \forall j \Rightarrow \frac{u_{2,j}^{n+1}}{\Delta x} + \frac{v_{1,j+1}^{n+1} - v_{1,j}^{n+1}}{\Delta y} = 0$

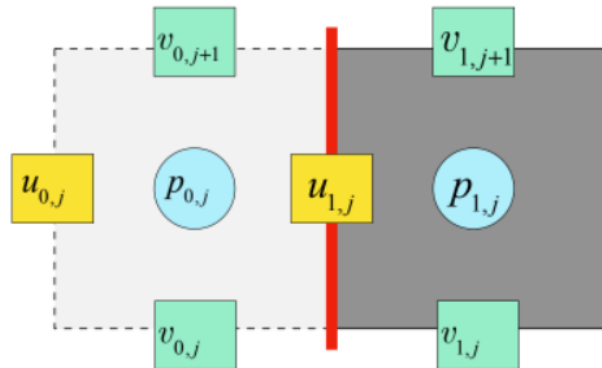
Now, use the fact that: $u_{i,j}^{n+1} = \tilde{u}_{i,j} - \Delta t \left. \frac{\partial p}{\partial x} \right|_{u_{i,j}}$

$$\frac{\tilde{u}_{2,j} - \Delta t \frac{p_{2,j} - p_{1,j}}{\Delta x}}{\Delta x} + \frac{\tilde{v}_{1,j+1} - \Delta t \frac{p_{1,j+1} - p_{1,j}}{\Delta y} - \left(\tilde{v}_{1,j} - \Delta t \frac{p_{1,j} - p_{1,j-1}}{\Delta y} \right)}{\Delta y} = 0$$

$$\frac{p_{2,j} - p_{1,j}}{\Delta x^2} + \frac{p_{1,j+1} - 2p_{1,j} + p_{1,j-1}}{\Delta y^2} = \frac{1}{\Delta t} \left(\frac{\tilde{u}_{2,j}}{\Delta x} + \frac{\tilde{v}_{1,j+1} - \tilde{v}_{1,j}}{\Delta y} \right)$$

Inlet: Velocity

Take a wall on the x-minus (left) side



$$u = u_{\text{inlet}}$$

Can be directly specified on the boundary!

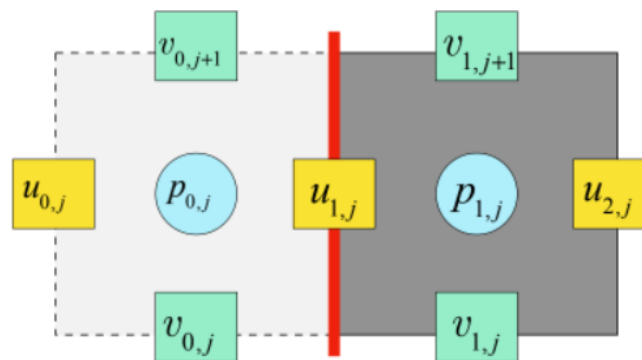
$$v = v_{\text{inlet}}$$

Use ghost cell specification:
$$\frac{v_{\text{interior}} + v_{\text{ghost}}}{2} = v_{\text{inlet}}$$

Inlet: Pressure

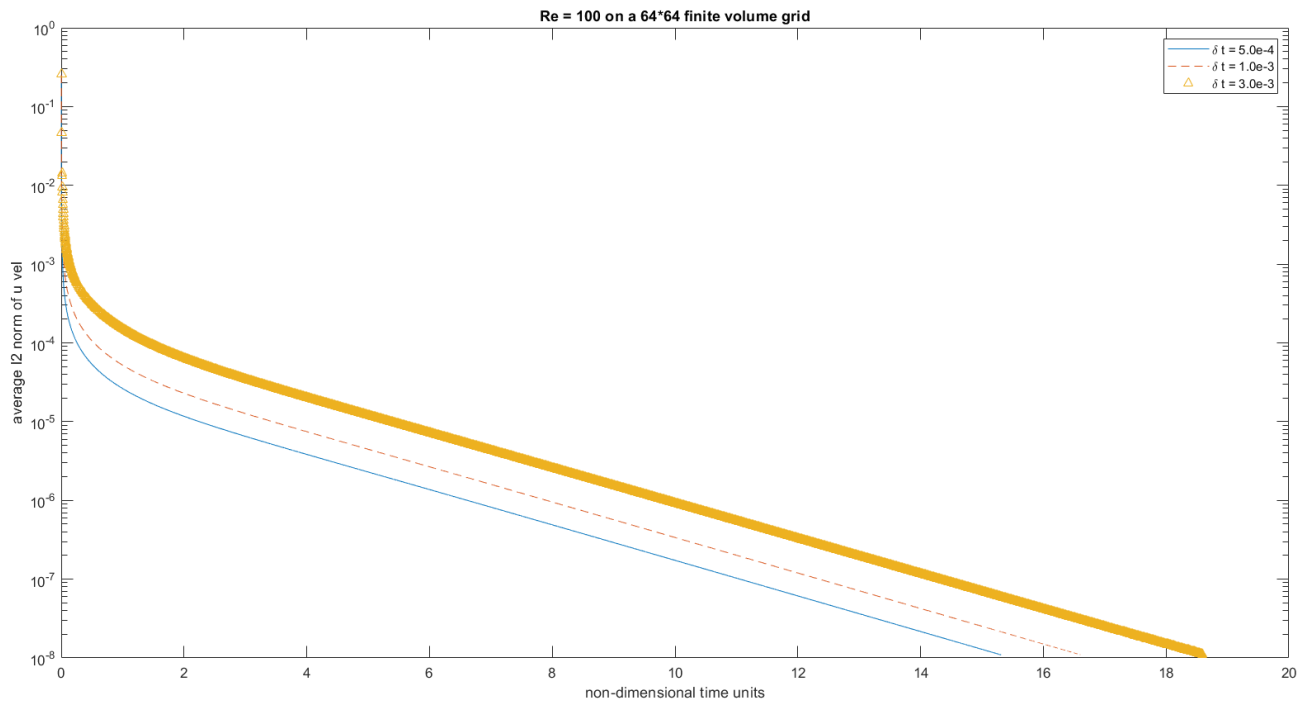
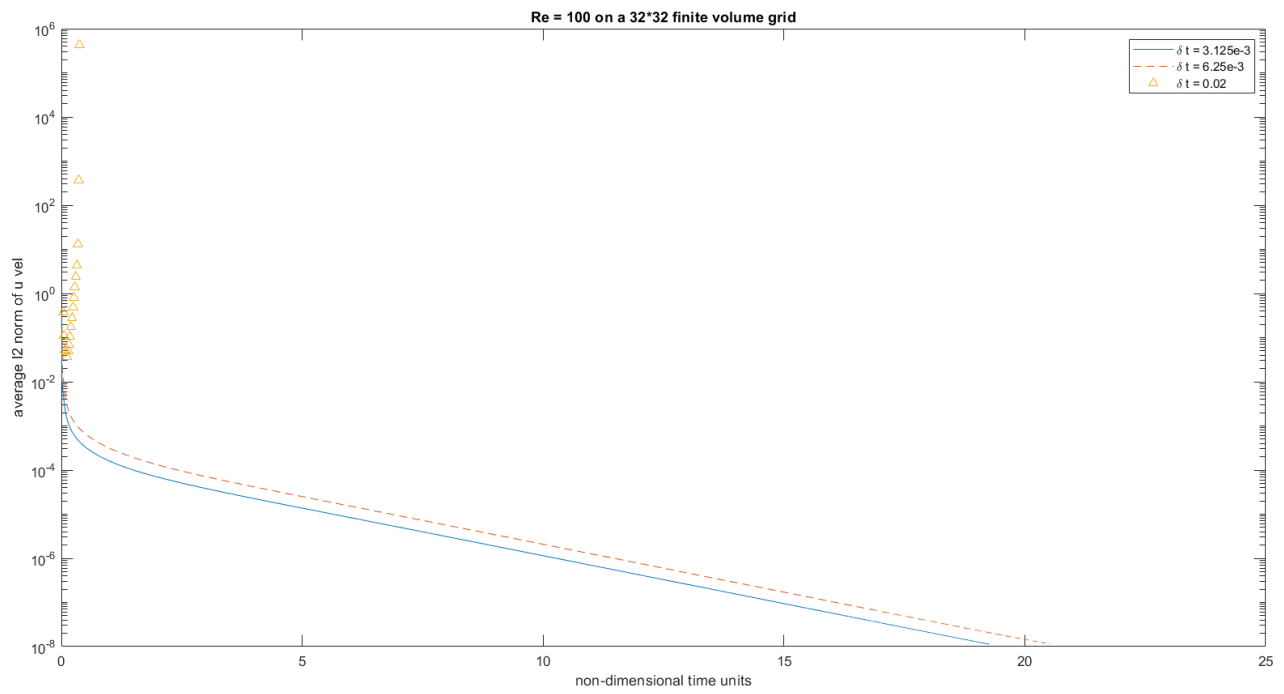
$$u = u_{\text{inlet}}$$

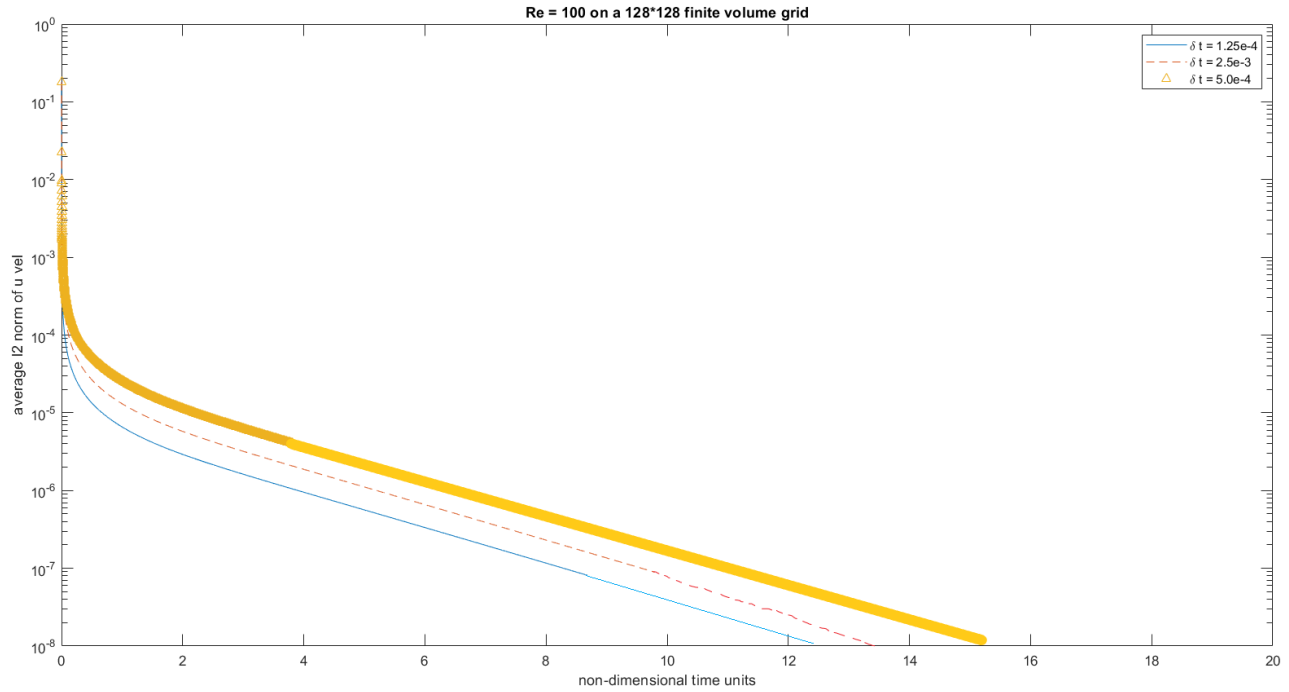
Apply continuity equation and eliminate corresponding pressure component



Results

Part A





We can see that for all the three grid resolutions, the number of non-dimensional time units required for convergence increases as the time step is increased. A very peculiar phenomenon occurs for the 32*32 grid at time step = 0.02 when the solver diverges. Peculiar because this value of time step is less than that dictated by the linear CFL condition and the 2d Neumann condition.

	Formula	Value of δt for CFL no. = 1
Linear CFL condition	$\delta t = CFL_{no} \delta x / u_T$	0.0313
2d Neumann condition	$\delta t = 0.25 \delta x^2 / \nu$	0.0244

So, why is it diverging? We investigated related literature and found that there are some more exotic conditions on stability of convection dominated flows, which are more stringent than the linear CFL criteria. Schneider et al. [1] have numerically proved that higher order methods such as Adams Bashforth and Runge-Kutta methods are restricted in stability by a non-linear CFL like condition which is given by:

$$\delta t \leq C \left(\frac{\delta x}{U} \right)^\alpha$$

where C is a value between 0 and 1 and α is dictated by the time integration scheme. Deriaz [2] did a more extensive theoretical von Neumann stability analysis of the 2D Burgers' equation as well as Euler equation and came up with the following stability conditions for various schemes:

Euler explicit:

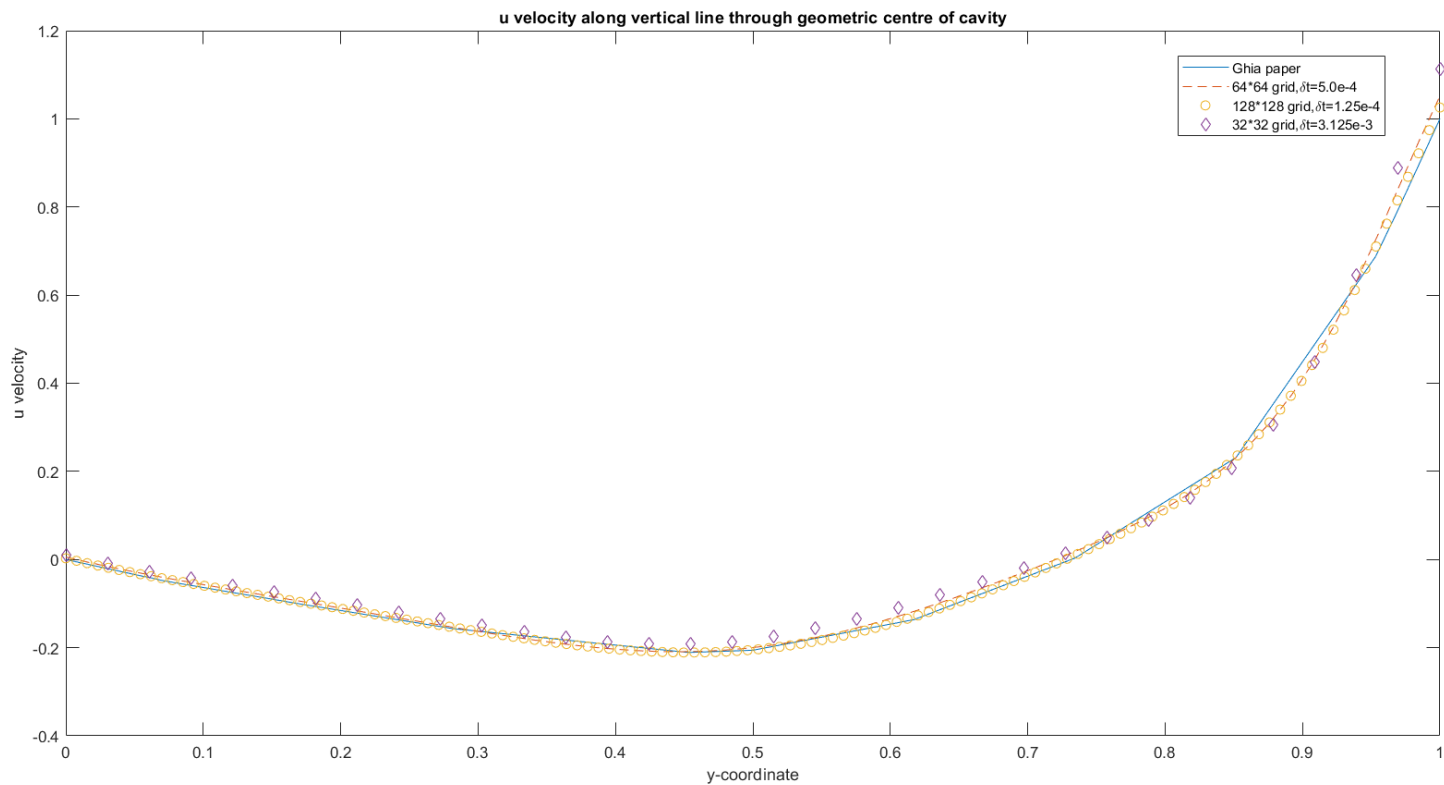
$$\delta t \leq 2C \left(\frac{\delta x}{U} \right)^2$$

2nd Order Adams Bashforth:

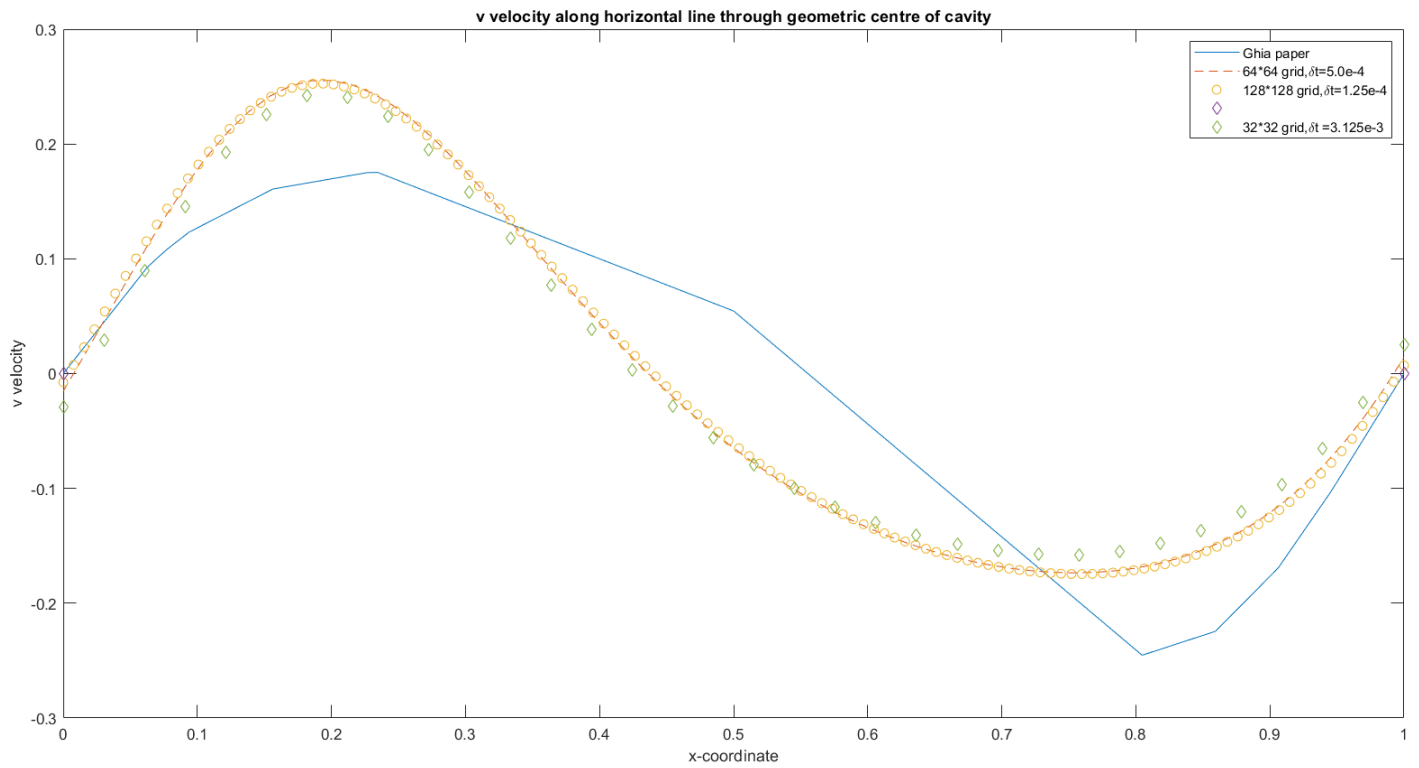
$$\delta t \leq 2^{\frac{2}{3}} C^{\frac{1}{3}} \left(\frac{\delta x}{U} \right)^{4/3}$$

For our 32*32 grid, taking a maximum value of C = 1 in the above condition for Adams Bashforth scheme gives $\delta t \leq 0.0156$. This maximum permissible value of the time step is less than 0.02, hence the solver diverges at $\delta t = 0.02$.

Comparison of our calculated velocities at Re = 100 with Ghia et al[3]

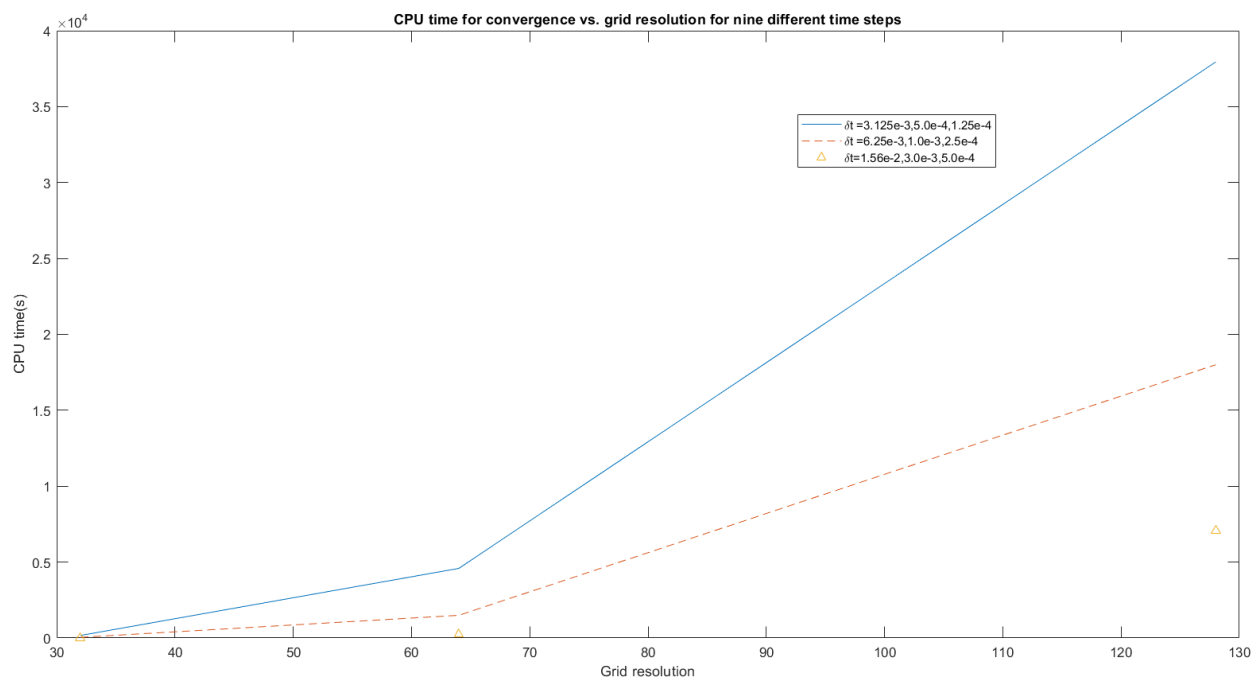


We can see that as far as the u velocity along a vertical line across the geometric center of cavity is concerned, there is very good agreement between our results and those of Ghia et al.



Our result for the v velocity has the same oscillatory trend as that of Ghia et al, although there isn't perfect match of values like the u velocity.

CPU time for convergence



We plotted the above graph in the following manner: the bottommost curve (yellow triangles) gives the CPU time for each grid resolution at the minimum of the three given time steps, the intermediate dotted red curve gives the CPU time at the intermediate time step for each grid resolution and the topmost solid blue line gives the CPU time at the maximum of the three time steps given for each grid resolution.

Approximate power law between CPU time and grid resolution

We did a basic curve fitting in MATLAB to find out the relationship between CPU time and the grid resolution. Indeed, a power law exists between the two which is given by:

$$y = ax^b$$

where y = CPU time to convergence

x = grid resolution (say 32 for 32*32, 64 for 64*64, etc.)

The values of a and b vary with the time step used, which is tabulated below

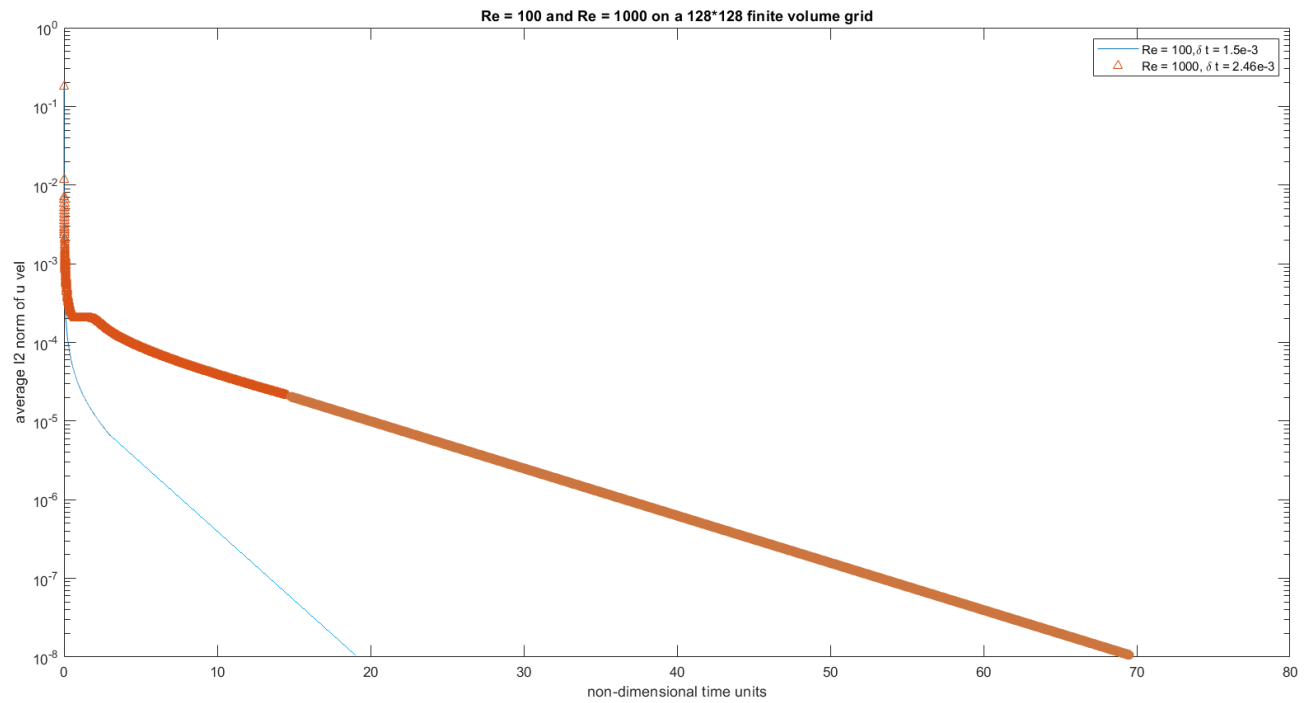
Time step	a	b
Minimum	0.01248	3.077
Intermediate	0.0004652	3.601
Maximum	7.391×10^{-7}	4.736

Largest permissible time step for $Re = 100$ vs. $Re = 1000$ on a 128*128 grid

Stability condition	Max. δt for $Re = 100$	Max. δt for $Re = 1000$
Linear CFL $\delta t = CFL_{no} \delta x / u_T$	0.0078	0.0078
2d Neumann $\delta t = 0.25 \delta x^2 / \nu$	0.0015	0.0153
Non-linear CFL $\delta t \leq 2^{\frac{2}{3}} C^{\frac{1}{3}} \left(\frac{\delta x}{U} \right)^{4/3}$	0.00246	0.00246

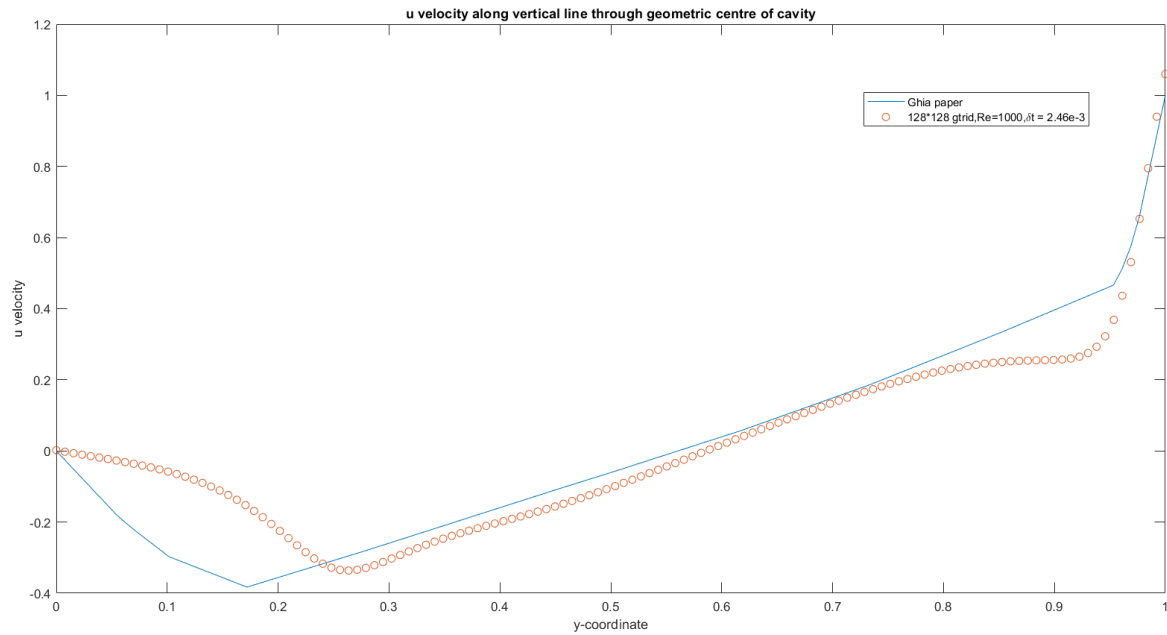
For $Re = 100$, the maximum permissible time step (0.0015) is dictated by the 2d Neumann condition, while for $Re = 1000$, the maximum permissible time step (0.00246) is given by a non-linear CFL like condition which is characteristic of convective flows. When Re is lower, viscous effects dominate, thus their dominating effect on stability. Whereas, when Re increases, convective forces start taking over more control and thus their dominating effect on stability.

Now we use the above maximum permissible time steps for $Re = 100$ and $Re = 1000$ to run our solver.

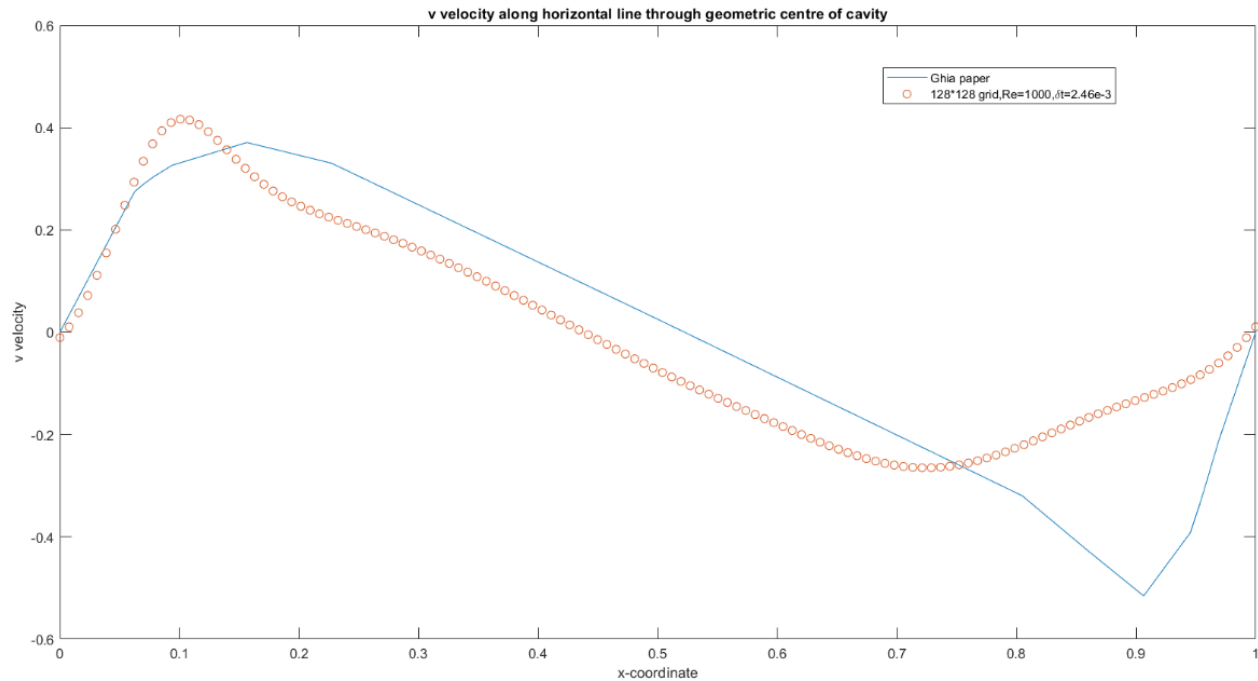


We can see from the above figure, that the no. of non-dimensional time units required for convergence is almost 3.5 times higher for Re = 1000 than for Re = 100. This is because the stabilizing effect of viscosity is very less at higher Reynolds numbers.

Comparison of our calculated velocities at $Re = 1000$ with Ghia et al[3]



We can see that there is a very good agreement between our results and those of Ghia et al as far the u velocity along a vertical line along the geometric center of cavity is concerned.



We draw similar conclusions for the v velocity along a horizontal line along the geometric center of cavity.

Summary and conclusion

From our exercise of trying to numerically solve the Navier Stokes equations, we realized that the solution is severely restricted by some exotic stability conditions, depending on the time discretization scheme, apart from the well known CFL or Neumann criteria. As a result, even taking a CFL no. quite less than 1 (say 0.3) can also result in divergence. Increasing the Reynolds no. magnifies this problem as viscosity is not strong enough to induce a stabilizing effect. As a result, the solver takes a longer time to converge for higher Reynolds numbers. This suggests that as CFD practitioners, we need to do an extensive von Neumann stability analysis of the problem at hand and not rely on the de-facto stability criteria. This of course is very difficult as we need to do stability analysis for each new problem we encounter. Life can be made a bit easier by keeping up to date with relevant literature where authors do the hard work for us.

Also, in our solver we were using the SOR iterative method to solve the pressure Poisson equation. We were using a relaxation parameter = 1.2 for all simulations. When we changed that to the optimum value of the relaxation parameter given by theory which is:

$$w_{opt} = \frac{2}{1 + \sin\left(\frac{\pi}{nx + 1}\right)}$$

where nx is the number of control volumes used to discretize the governing equations

we found that the solver indeed reaches convergence faster. Secondly, we also tried a 4 level V-cycle multigrid solver for the pressure Poisson equation and this time too, our solver converged faster than it did for $w = 1.2$.

References

1. Schneider, Kai & Kolomenskiy, Dmitry & Deriaz, Erwan. (2013). Is the CFL Condition Sufficient? Some Remarks. The Courant-Friedrichs-Lewy (CFL) Condition. 80 Years After Its Discovery. 10.1007/978-0-8176-8394-8_9.
2. Deriaz, Erwan. "Stability conditions for the numerical solution of convection-dominated problems with skew-symmetric discretizations." *SIAM Journal on Numerical Analysis* 50.3 (2012): 1058-1085.
3. Ghia, U. K. N. G., Kirti 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 (1982): 387-411.