A SIMPLE-BASED APPROACH TO SIMULATING INCOMPRESSIBLE FLOW OVER A CIRCULAR CYLINDER

Om Mahajan
Department of Mechanical Engineering
Indian Institute of Technology, Madras
Chennai, India, 600036
me22b174@smail.iitm.ac.in

ABSTRACT

This work presents a SIMPLE-based numerical approach for simulating steady incompressible flow over a circular cylinder using a finite volume method on a co-located grid. The Navier–Stokes equations are discretized using second-order schemes, and pressure–velocity coupling is handled via the SIMPLE algorithm with Rhie–Chow interpolation to prevent pressure oscillations. The methodology accurately captures flow separation and wake formation, demonstrating the effectiveness of SIMPLE on co-located grids for external bluff body flows.

NOMENCLATURE

$\boldsymbol{u}$	Velocity vector
u	x-velocity
v	y-velocity
p	Scalar pressure
i	x-direction unit vector
j	y-direction unit vector
μ	Dynamic viscosity (diffusion constant)
ρ	Fluid density
$\vec{A}$	Area vector
u_b	Boundary velocity
a_{nb}	Neighbor coefficient
u_{nb}	Neighbor velocity
V	Control volume
u^*	Guessed velocity (from predictor step)

INTRODUCTION

SIMPLE algorithm is utilized to compute steady, incompressible flow over a circular cylinder located at the center of a square domain on a co-located grid. Discretized Navier–Stokes equations are solved using second-order schemes: central difference scheme (CDS) for low Peclet numbers and up-wind scheme for high Peclet numbers. Pressure–velocity coupling is dealt with using the SIMPLE algorithm. A no-slip boundary condition is applied on the wall of the cylinder, and neumann bound- ary conditions on pressure are defined at the domain boundaries.

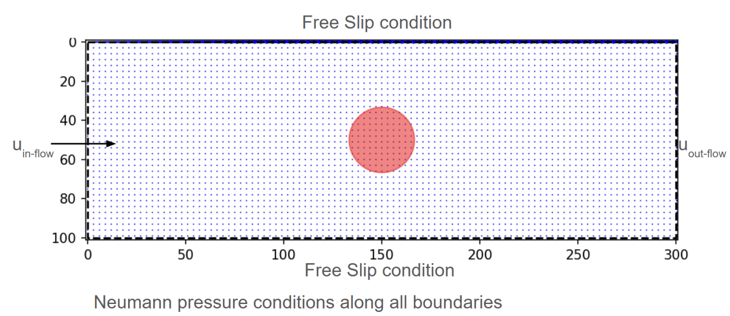

Figure 1: Computational domain and boundary conditions

GOVERNING EQUATIONS

1. Continuity Equation

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

2. Momentum Equations

x-momentum:

$$\frac{\partial}{\partial x}(\rho uu)+\frac{\partial}{\partial y}(\rho vu)=\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial u}{\partial y}\right)-\frac{\partial p}{\partial x}+S_u \tag{2}$$

y-momentum:

$$\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho vv)=\frac{\partial}{\partial x}\left(\mu\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right)-\frac{\partial p}{\partial y}+S_v \tag{3}$$

3. Discretization Approach: Hybrid Differencing Scheme

To discretize the convective terms in the momentum equations, we employ a **hybrid differencing scheme** that dynamically blends the Central Differencing Scheme (CDS) and the Upwind Differencing Scheme (UDS) based on the local cell Peclet number.

We define the convective fluxes and diffusive conductance as follows:

$$\begin{aligned}
F_e &= \rho\, u_e A_e = \rho\cdot\frac{u_{i,j}+u_{i,j+1}}{2}\cdot\Delta y\\
F_w &= \rho\, u_w A_w = \rho\cdot\frac{u_{i,j}+u_{i,j-1}}{2}\cdot\Delta y\\
F_n &= \rho\, v_n A_n = \rho\cdot\frac{v_{i-1,j}+v_{i,j}}{2}\cdot\Delta x\\
F_s &= \rho\, v_s A_s = \rho\cdot\frac{v_{i,j}+v_{i+1,j}}{2}\cdot\Delta x\\
D &= \frac{\mu}{\delta}\quad(\text{where } \delta=\Delta x \text{ or } \Delta y)
\end{aligned}$$

We then compute the local Peclet numbers:

$$\mathrm{Pe}_f=\frac{F_f}{D}\quad \text{for face } f\in\{e,w,n,s\}$$

The hybrid scheme is implemented using the following logic:

- If $|\mathrm{Pe}_f|<1$: the flow is diffusion-dominated $\rightarrow$ use **Central Differencing Scheme (CDS)**

- If $|\mathrm{Pe}_f|\geq 1$: the flow is convection-dominated $\rightarrow$ use **Upwind Differencing Scheme (UDS)** - To ensure stability and accuracy in all regimes, we use the `takeMax()` function as:

$$\begin{aligned}
a_E &= \max\left(-F_e, D-\frac{1}{2}F_e, 0\right)\\
a_W &= \max\left(F_w, D+\frac{1}{2}F_w, 0\right)\\
a_N &= \max\left(-F_n, D-\frac{1}{2}F_n, 0\right)\\
a_S &= \max\left(F_s, D+\frac{1}{2}F_s, 0\right)
\end{aligned}$$

This logic implicitly applies: - **CDS** when $D \gg F$ $\rightarrow$ diffusion-dominated, - **UDS** when $F \gg D$ $\rightarrow$ convection-dominated, - And blends both when $F\sim D$.

The final discretized momentum equation is:

$$a_P u^*_{i,j}=a_E u_{i,j+1}+a_W u_{i,j-1}+a_N u_{i-1,j}+a_S u_{i+1,j}+d_E(p^*_{i,j+1}-p^*_{i,j})$$

where $a_P=a_E+a_W+a_N+a_S+(F_e-F_w+F_n-F_s)$, and $d_E=-\Delta y/a_P$.

4. Discretized Momentum Equations

The discretized momentum equations are given by:

$$a_P u^*_{i,j}=\sum a_{nb}u_{nb}+(p^*_{i-1,j}-p^*_{i,j})A_x+b_u \tag{4}$$
$$a_P v^*_{i,j}=\sum a_{nb}v_{nb}+(p^*_{i,j-1}-p^*_{i,j})A_y+b_v \tag{5}$$

where a_P is the central coefficient

THE SIMPLE ALGORITHM

1. Pressure Correction Equation

We introduce pressure and velocity corrections:

$$p=p^*+p' \tag{6}$$
$$u=u^*+u' \tag{7}$$
$$v=v^*+v' \tag{8}$$

From the continuity equation (1), substituting the corrected velocities and discretizing, we obtain a pressure correction equation:

$$a_P p'_{i,j}=\sum a_{nb}p'_{nb}+b_p \tag{9}$$

The source term b_p is obtained from the divergence of the predicted velocity field: obtain a pressure correction equation:

$$b_p=\rho\left[\frac{u^*_{i+1,j}-u^*_{i,j}}{\Delta x}+\frac{v^*_{i,j+1}-v^*_{i,j}}{\Delta y}\right] \tag{10}$$

2. Velocity Correction Equations

Once the pressure correction p' is known, the velocity corrections are computed as:

$$u'_{i,j}=-\frac{d_u}{a_P}(p'_{i+1,j}-p'_{i,j}) \tag{11}$$
$$v'_{i,j}=-\frac{d_v}{a_P}(p'_{i,j+1}-p'_{i,j}) \tag{12}$$

Then the velocity field is updated:

$$u_{i,j}=u^*_{i,j}+u'_{i,j} \tag{13}$$
$$v_{i,j}=v^*_{i,j}+v'_{i,j} \tag{14}$$

3. Pressure Update

The pressure is updated with an under-relaxation factor α_p:

$$p_{i,j} = p^*_{i,j} + \alpha_p p'_{i,j} \tag{15}$$

α_p is taken to be 0.7 for these experiments.

4. Rhie-Chow Interpolation

To prevent checkerboard pressure fields on a co-located grid, face velocities must be computed using Rhie–Chow interpolation:

$$u_f = \bar{u}_f - \frac{1}{a_f}(p'_P - p'_N) \tag{16}$$

Here, $\bar{u}_f$ is the interpolated velocity from neighboring cells and a_f is the face momentum coefficient.

5. Iterative Procedure

Repeat the following steps until convergence:

1. Guess pressure field p^*
2. Solve momentum equations to get u^*, v^*
3. Solve pressure correction equation to get p'
4. Correct velocity and pressure fields
5. Check for convergence (e.g., residuals or variable change)

RESULTS

1. U-velocity field comparison

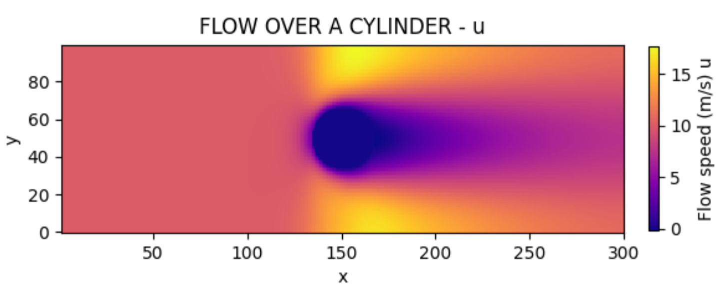

FIGURE 1: u field for inlet velocity $u = 10$

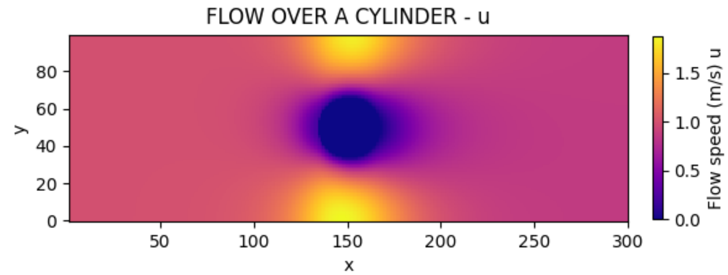

FIGURE 2: u field for inlet velocity $u = 1$

2. V-velocity field comparison

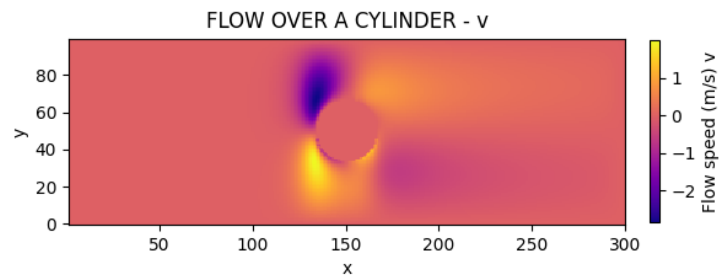

FIGURE 3: v field for inlet velocity $u = 10$

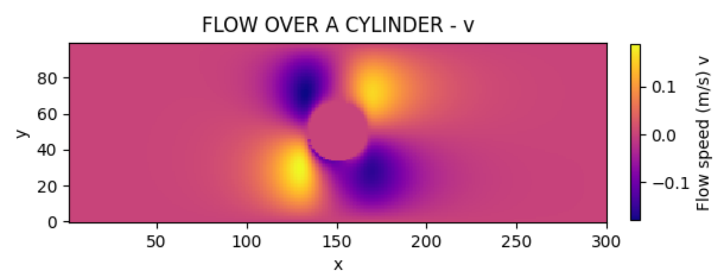

FIGURE 4: v field for inlet velocity $u = 1$

3. Centerline velocity for $u_{inlet} = 10$

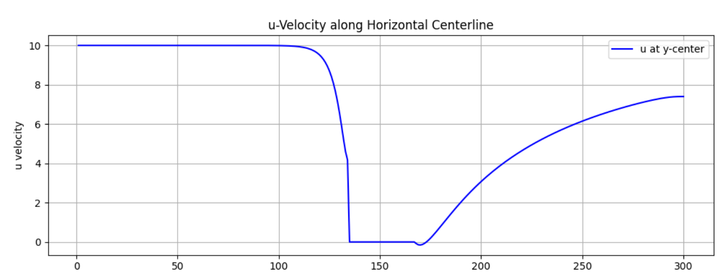

FIGURE 5: Centerline velocity u

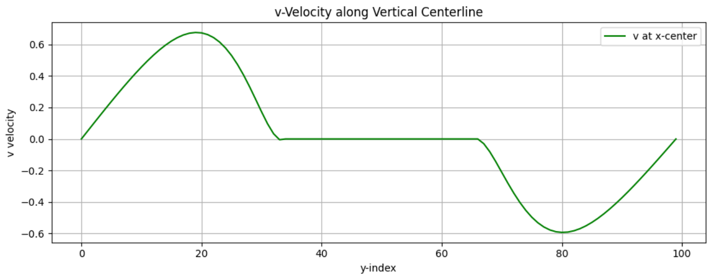

FIGURE 6: Centerline velocity v

4. Pressure field for $u_{inlet} = 1$

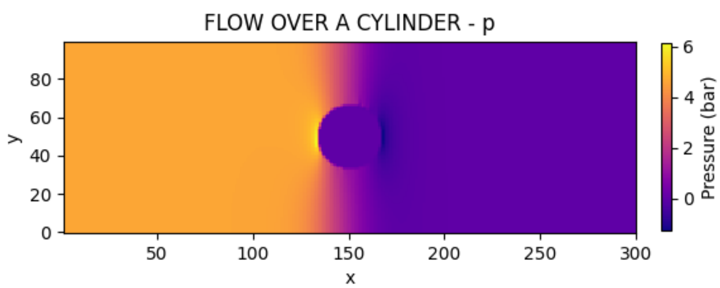

FIGURE 7: p field for inlet velocity $u_{inlet} = 1$

4. Streamlines for $u_{inlet} = 1$

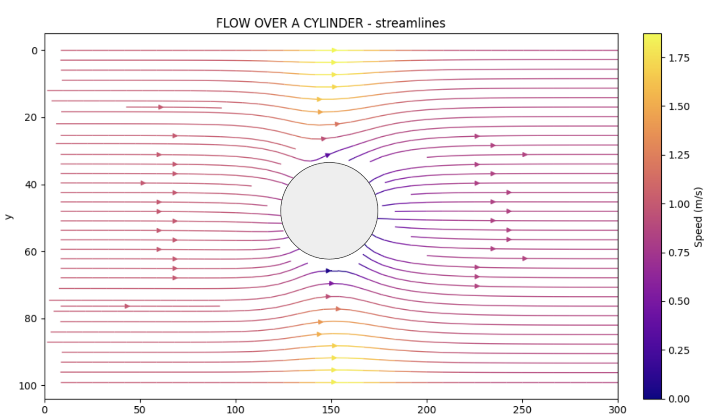

FIGURE 8: Streamlines for inlet velocity $u_{inlet} = 1$

4. Error vs Iterations $u_{inlet} = 1$

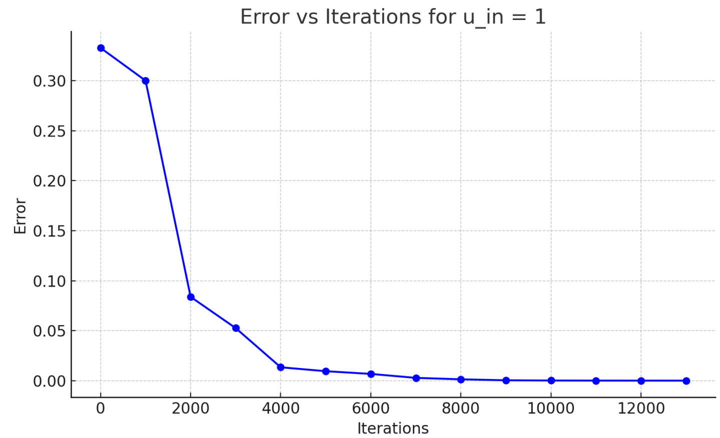

FIGURE 9: Error vs Iterations for inlet velocity $u_{inlet} = 1$

CONCLUSION

This project implemented a finite volume method on a structured, co-located grid to simulate incompressible flow over a circular cylinder. The co-located grid simplifies data storage and interpolation, while the structured layout aids discretization and indexing.

The convection-diffusion terms were discretized using the second-order Central Difference Scheme (CDS) for interior nodes:

$$\left(\frac{\partial \phi}{\partial x}\right)_e \approx \frac{\phi_E - \phi_P}{\Delta x}$$

However, at high Peclet numbers ($Pe = \frac{u\Delta x}{\Gamma} \gg 1$), convective transport dominates diffusion, leading to numerical instabilities with CDS. Hence, **upwind differencing** was used near boundaries and in regions of steep gradients:

$$\left(\frac{\partial \phi}{\partial x}\right)_e \approx \frac{\phi_P - \phi_W}{\Delta x}$$

The SIMPLE algorithm was employed to couple velocity and pressure fields. Rhie–Chow interpolation was applied to face velocities to eliminate pressure oscillations.

Pressure boundary conditions were set as **Neumann (zero-gradient)** on all the boundaries to match with the incompressible flow assumption.

This formulation successfully predicted major features of flow over a cylinder, i.e., wake creation and symmetry.

ACKNOWLEDGMENT

Thanks to Prof. Kameswararao Anupindi for his excellent dedication in teaching the course, and for teaching us

REFERENCES

[1] Versteeg, H. K., and Malalasekera, W., 2007, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, 2nd ed., Pearson Education.

[2] Patankar, S. V., 1980, *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corporation, Washington, D.C.