2D SIMPLE SOLVER FOR LAMINAR FLOW OVER A SQUARE CYLINDER

Jeffrey R. Kennington
Department of Mechanical Engineering
Purdue University
West Lafayette, IN 47906
Email: jkenning@purdue.edu

Kameswararao Anupindi
Department of Mechanical Engineering
Purdue University
West Lafayette, IN 47906
Email: kanupind@purdue.edu

ABSTRACT

The solution to the Navier-Stokes equations. The SIMPLE solver is implemented using a staggered structured mesh. The solver is benchmarked against flow over a square cylinder over a range of Reynolds numbers.

NOMENCLATURE

u Velocity Vector.
u x velocity.
v y velocity.
p scalar pressure.
i x direction unit vector.
j y direction unit vector.
μ diffusion constant.
ρ density.
Re Reynolds Number.
∇ Divergence Operator.
ϕ unit quantity in transport equation.
Γ Diffusion coefficient.
J flux.
A Area Vector.
u_b boundary velocity.
a_{nb} neighbor coefficient.
u_{nb} neighbor velocity.
u_{up} upwind velocity.
V Volume
u^* guess velocity.
α_p pressure under-relaxation coefficient.
α_u momentum under-relaxation coefficient.

INTRODUCTION

Numerical simulation of incompressible Navier - Stokes equations using a sequential solution procedure poses two problems. The first one is that there is no governing equation for pressure (which is a path to solution issue), and the second one is pressure checkerboarding which leads to a wiggly solution (which is a discretization issue). Both these problems are overcome using SIMPLE algorithm on a staggered mesh. In this project we compute the steady laminar flow over a square cylinder placed in a channel using SIMPLE algorithm on a staggered mesh.

PROBLEM DESCRIPTION

The domain is a rectangular duct with a square cylinder centered on the y-axis. The geometry is made to match the domain used in [1] and is as shown if Figure 1. Where $D = 1$, $H = 8$, $L_{in} = 12.5$ and $L = 50$. The inlet is a constant velocity inlet using a parabolic profile with a maximum value of $u = 1$, the outlet is an outflow is upwinded using infinite Peclet number assumption and the walls are all no slip wall boundaries. The case was run using Re of 1, 10, 20, 30, 40 and 50. Where Re is calculated using the velocity inlet and the length scale D where $Re = D\mathbf{u}/\mu$.

GOVERNING EQUATIONS

The governing equations for this problem are the 2D steady, Navier - Stokes equations, which are nothing but the continuity equation and the momentum equations in two directions as given below:

$$\nabla.(\rho\mathbf{V}) = 0 \tag{1}$$

$$\nabla.(\rho\mathbf{V}u) = \nabla.(\mu\nabla u) - \nabla p.i + S_u \tag{2}$$

$$\nabla.(\rho\mathbf{V}v) = \nabla.(\mu\nabla v) - \nabla p.j + S_v \tag{3}$$

The source terms S_u, S_v appearing in above equations for a Newtonian fluid can be written as

$$S_u = f_u + \frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial x}\right) - \frac{2}{3}\frac{\partial}{\partial x}(\mu\nabla.\mathbf{V}) \tag{4}$$

$$S_v = f_v + \frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right) + \frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial y}\right) - \frac{2}{3}\frac{\partial}{\partial y}(\mu\nabla.\mathbf{V}) \tag{5}$$

But for the transport of any scalar ϕ we have a general transport equation which is given as follows:

$$\frac{\partial}{\partial t}(\rho\phi) + \nabla.(\rho\mathbf{V}\phi) = \nabla.(\Gamma\nabla\phi) + S \tag{6}$$

In a steady state case the general transport equation reduces to convection term balanced by diffusion and source terms as follows:

$$\nabla.(\rho\mathbf{V}\phi) = \nabla.(\Gamma\nabla\phi) + S \tag{7}$$

Comparing the u-momentum equation 2 to general transport equation 7 we get $\phi = u$, $\Gamma = \mu$, and $S = S_u - \frac{\partial p}{\partial x}$. Further in the present case viscosity (μ) and density (ρ) of the fluid are constant, and assuming that there are no body forces acting on the flow domain, the source terms S_u and S_v are zero. This can be shown by invoking the continuity equation for incompressible fluid $\nabla.\mathbf{V} = 0$ in eqs. 4, 5.

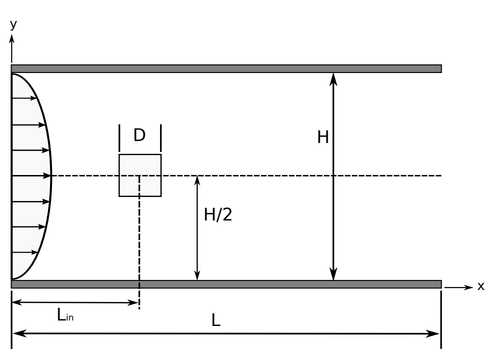


FIGURE 1: Problem Domain

The Navier - Stokes equations are nonlinear (because of the convection term) and they are coupled (because the u-momentum equation has v velocity in it and vice versa). In order to solve these equations we have to linearize the convective terms and the equations are decoupled using the current iterate values and are iterated till convergence using SIMPLE algorithm.

Discretization

In the finite volume method the first step is to integrate the governing equations on the control volume. In order to do this we recast the u-momentum equation 2 as

$$\nabla J = S \tag{8}$$

where, $J = \rho\mathbf{V}u - \mu\nabla\phi$, $S\delta = -\frac{\partial P}{\partial x}$

Upon integrating the equation 8 on a control volume and making use of divergence theorem we get:

$$\sum_{f=e,w,n,s} \mathbf{J_f}.\mathbf{A_f} = S.\Delta V \tag{9}$$

The reason behind using a staggered mesh is to avoid the checkerboarding of solution. Checkerboarding means the solution could converge to a value whose alternate cell values are equal but this kind of solution is not physically possible. Storing the velocity components on the cell faces instead of at the cell centers avoids checkerboarding and this is known as staggering. So the horizontal velocity u_e is stored on the east face of the control volume and the vertical velocity v_n is stored on the north face of the control volume. Pressure is stored at the cell centroid of the control volume. In this approach we will have three different control volumes, the pressure control volume which is the regular control volumes on the original mesh. The u velocity control volumes which are staggered from the pressure control volumes by $(\Delta x/2)$ in x-direction and the v velocity control volumes which are staggered from the pressure control volumes by $(\Delta y/2)$ in y-direction. Because the control volumes are different for u and v momentum equation discretizations the corresponding coefficients will turn out to be different.

The discretization of momentum equations results in equation 9 in which the fluxes have to evaluated on four faces of the control volume. The evaluation of flux term on the east face for u-momentum discretization is given below:

$$J_e.A_e = \rho\left(\frac{u_e + u_{ee}}{2}\right)^o u_{up}\Delta y - \mu\Delta y\left(\frac{u_{ee} - u_e}{\Delta x}\right) \tag{10}$$

In the above equation we have used upwinding for the velocity on the east face of the u-momentum control volume. The value of u_{up} assumes either u_e or u_{ee} depending on the mass flux direction which is multiplying it. And the mass flux on the east face is the linear interpolated value of the neighboring cells. The superscript o refers to the linearized convection terms in which the current iterate values are being used.

The evaluation of flux term on the north face is given below:

$$J_n.A_n = \rho \left(\frac{v_n + v_{ne}}{2} \right)^o u_{up} \Delta x - \mu \Delta x \left(\frac{u_{ne} - u_e}{\Delta y} \right) \tag{11}$$

In the above equation u_{up} assumes either u_{ne} or u_e depending on the mass flux direction. In a similar manner the face fluxes on the west and south faces can be evaluated. One the right hand side of the equation 9 we have the source term, for u-momentum equation this can be evaluated as below:

$$S\Delta\mathbf{V} = -\frac{\partial p}{\partial x}\Delta y \Delta x = (p_P - p_E)\Delta y \tag{12}$$

Application of Boundary Conditions

The boundaries are handled by using a fictitious cell across the physical boundary. As the solver sweeps across the physical domain the values across the boundaries are used in the solution and then updated after each iteration. The values for the u and velocity are assigned across the boundaries as shown in Figures 2 and 3. The opposite values are assigned to the outer cell resulting in a value of zero along the boundary.

Along the faces where the staggered cell center is on the boundary the set value is used. The pressure boundary outside values are set to the same as the cells along the boundary resulting in ∂p being equal to zero along all boundaries. The boundaries for the square cylinder are handled in the same manner as the outer boundaries for velocity and pressure.

Solution Method

The solution method is based on SIMPLE algorithm. The flow chart of SIMPLE algorithm is shown in Fig. 4. The discrete momentum equations [2] are given as follows:

$$a^u_e u_e = \sum_{nb} a^u_{nb} u_{nb} + \Delta y (P_P - P_E) \tag{13}$$

$$a^v_n v_n = \sum_{nb} a^v_{nb} v_{nb} + \Delta x (P_P - P_N) \tag{14}$$

In the SIMPLE algorithm, the discrete momentum equations 13 are substituted into discrete continuity equation. Then

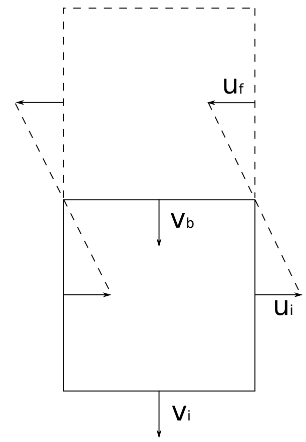


FIGURE 2: u-Velocity Boundary

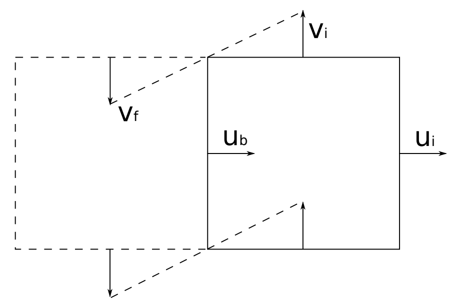


FIGURE 3: v-Velocity Boundary

by proposing corrections to velocity and pressure a pressure correction equation is derived. As shown in the flow chart, first we guess the velocity and pressure fields over the domain. Now using these guess values we solve the discrete momentum equations 13 and obtain u_e^*, v_n^*. Now these newly computed starred velocities satisfy momentum equations for the guessed pressure field but they do not satisfy the continuity equation since the guessed pressure is not the correct pressure field. So, corrections are proposed for velocity and pressure as follows:

$$u_e = u_e^* + u_e^{'} \tag{15}$$

$$v_n = v_n^* + v_n^{'} \tag{16}$$

$$p = p^* + p^{'} \tag{17}$$

SIMPLE algorithm demands that these corrected values should satisfy the discrete continuity equation. Substituting these proposed corrected velocities 15 into discrete continuity equation a discrete equation for pressure correction ($p^{'}$) can be derived which reads as follows:

$$a_P p_P^{'} = \sum_{nb} a_{nb} p_{nb}^{'} + (F_w^* - F_e^* + F_s^* - F_n^*) \quad (18)$$

$$a_E = \rho_e d_e \Delta y \quad (19)$$

$$a_W = \rho_w d_w \Delta y \quad (20)$$

$$a_N = \rho_n d_n \Delta x \quad (21)$$

$$a_S = \rho_s d_s \Delta x \quad (22)$$

$$a_P = \sum_{nb} a_{nb} \quad (23)$$

Where $d_e = \Delta y / a_e^u$, $d_w = \Delta y / a_w^u$, $d_n = \Delta x / a_n^v$, $d_s = \Delta x / a_s^v$ and $F_e = \rho u_e \Delta y$, $F_w = \rho u_w \Delta y$, $F_n = \rho v_n \Delta x$, $F_s = \rho v_s \Delta x$

Because of the nonlinear nature of the equations it is necessary to under-relax momentum equations. The under-relaxed discrete momentum equation is given below:

$$\frac{a_e^u u_e}{\alpha} = \sum_{nb} a_{nb}^u u_{nb} + \Delta y (p_P - p_E) + \frac{(1-\alpha)}{\alpha} a_e^u u_e^o \quad (24)$$

Similarly the pressure correction is under-relaxed when adding them to the starred values as follows:

$$p = p^* + \alpha_p p^{'} \quad (25)$$

In the present simulations we have taken $\alpha = 0.7$ and $\alpha_p = 0.3$.

RESULTS

The results of our cases were validated against published results [1]. As the Re increases the recircualtion length increase and the drag coefficient decreases due to the decrease in viscous forces. Only the viscosity was changed to produce the desired Re The recirculation length and drag coefficient were in good agreement as shown in Figures 5 and 6. Results for each case streamlines are compared in Figures 8.

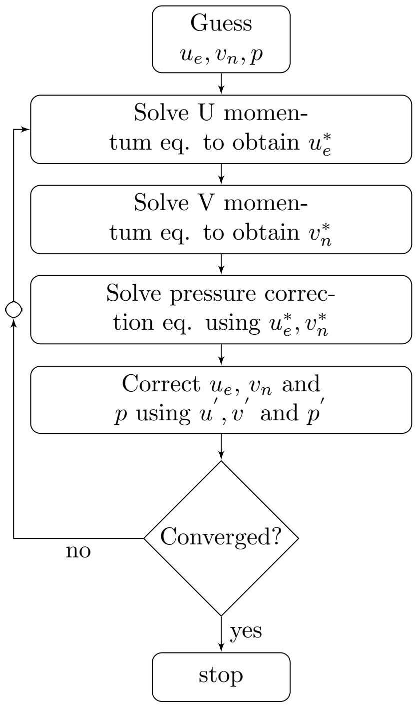


FIGURE 4: Flow chart of SIMPLE Algorithm

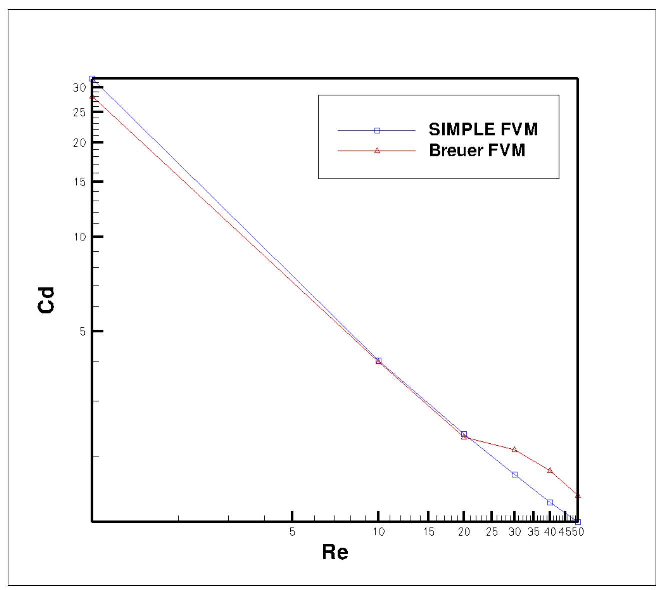


FIGURE 5: Drag Coefficient

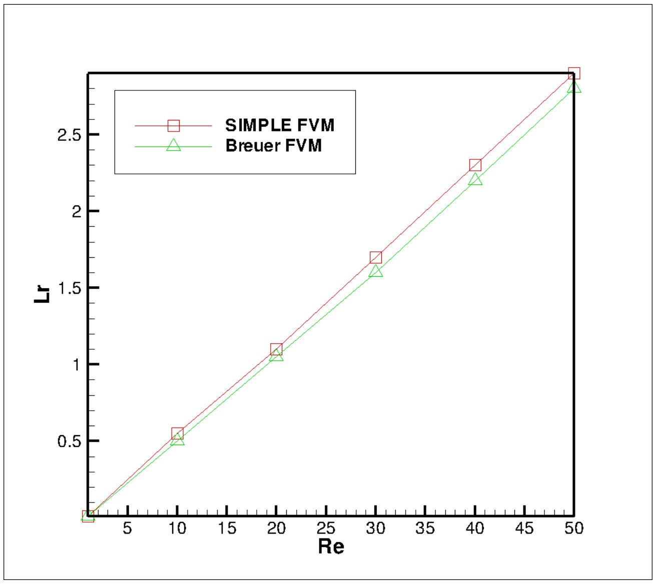
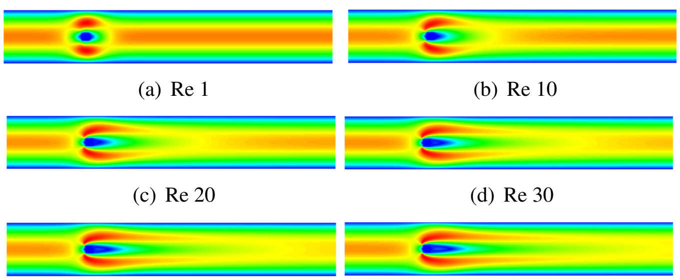
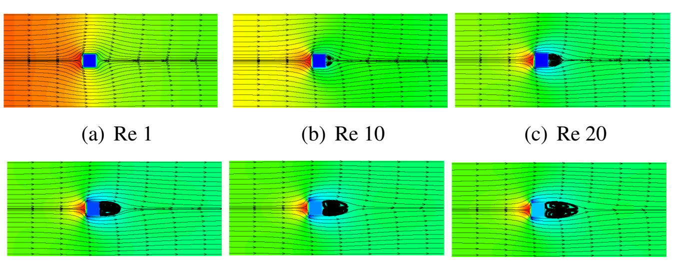


FIGURE 6: Recirculation Length vs Re



(a) Re 1 (b) Re 10 (c) Re 20 (d) Re 30 (e) Re 40 (f) Re 50

FIGURE 7: u-Velocity Profiles



(a) Re 1 (b) Re 10 (c) Re 20 (d) Re 30 (e) Re 40 (f) Re 50

FIGURE 8: Streamlines

CONCLUSION

The SIMPLE solver on a staggered mesh was successfully implemented to solve for 2D laminar flow over a square cylinder. The results agreed with published results and converged as expected. The staggered mesh worked to eleminate checkerboarding but is only practile on simple geometries and use of the collocated scheme is more robust in regards to mesh complexity. The SIMPLE algorithm, however, is robust and provides quality solutions to incompressible flows.

REFERENCES

[1] Breuer, M., Bernsdorf, J., and T. Zeiser, F. D., 1999. "Accurate computations of the laminar flow past a square cylinder based on two different methods: lattice-boltzmann and finite-volume".

[2] Murthy, J., 2010. Me 608 lecture notes.

ACKNOWLEDGMENT

Support from Prof. Murthy in our efforts.