

CFD ANALYSIS OF 2D UNSTEADY FLOW AROUND A SQUARE CYLINDER

MD IMDAD UDDIN CHOWDHURY
ME17D023

ABSTRACT

Flows around bluff bodies is an area of great research for scientist for several years. Flows around buildings, chimneys are examples where the fluid is in motion. Atmospheric dispersion of pollutants around bluff bodies has intensified the need to understand wake behavior. In the present work a numerical simulation is carried out for flow past a square cylinder has been investigated for the Reynolds number considered in the range 5-250 so that flow is laminar. The main objective of this study is to capture the features of flow past a square cylinder in a domain with the use of CFD. Finite Volume Method is being used with staggered grid arrangement. The incompressible MAC algorithm is used for the velocity pressure coupling. Streamlines, velocity contour, velocity profile and integral parameters such as drag coefficients, recirculation length is investigated and validated with literature.

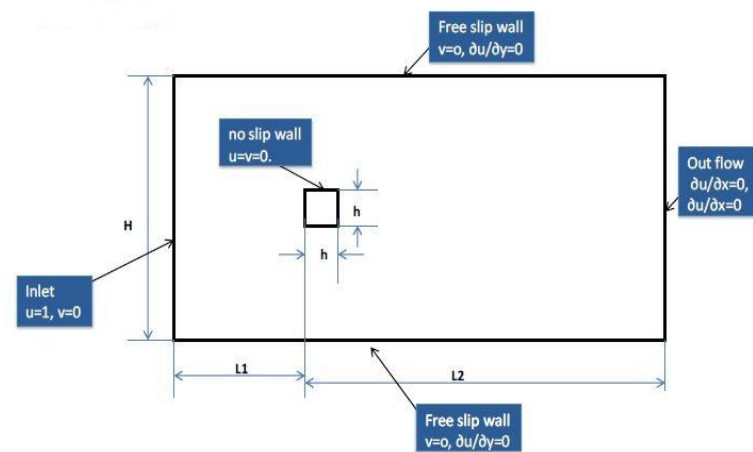
INTRODUCTION

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 a staggered mesh. In this project we compute the unsteady laminar flow over a square cylinder placed in a channel using MAC algorithm on a staggered mesh. It is observed in the literature that flow is steady in $0.5 \leq Re \leq 60$ and for $60 \leq Re \leq 300$ flow is unsteady laminar. Klekar and Patankar (1992) determined a critical value of $Re = 54$. When Re exceeds this value the free shear layer begins to roll up and form eddies. This phenomena is known as the Von Karman vortex street.

NOMENCLATURE

\mathbf{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.
 C_D Coefficient of drag.
 f unit quantity in transport equation.
 t Time
 F flux.
 A Area Vector.
 u_b boundary velocity.
 a_{nb} neighbor coefficient.
 u_{nb} neighbor velocity.
 L_r Recirculation length.
 V Volume
 u^* guess velocity.

PROBLEM DESCRIPTION



The domain is a rectangular duct with a square cylinder centered on the y-axis. Where $L1=5$, $L2=25$, $H=17$, $h=1$. Inlet is uniform velocity inlet having $u=1$, $v=0$. Outlet is an outflow which is upwinded using infinite Peclet number assumption and the walls of the square cylinder are all no slip wall boundaries. The walls of the domain are assumed as free slip wall. This case is run for various Reynolds numbers where Re is calculated using inlet velocity and length scale $h=1$. 400×250 staggered mesh is being used. Time step size between two iteration is taken as 0.001 sec.

GOVERNING EQUATIONS

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

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V) = 0 \quad (1)$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V u) = \nabla \cdot (\mu \nabla u) - \nabla p \cdot i + S_u \quad (2)$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V v) = \nabla \cdot (\mu \nabla v) - \nabla p \cdot j + S_v \quad (3)$$

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

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

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

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

$$\frac{\partial}{\partial t} (\rho \phi) + \nabla \cdot (\rho V \phi) = \nabla \cdot (\Gamma \nabla \phi) + S_\phi \quad (6)$$

Comparing the u-momentum equation 2 to general transport equation 6 we get $\phi = u$, $\Gamma = \mu$, $S_\phi = S_u - \frac{\partial p}{\partial x}$

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 $\nabla \cdot V = 0$ in eqs. 4 and 5. 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 MAC algorithm. Governing equations are being solved in non dimensional form. the velocities are non dimensionalized with the average velocity “ u ” at the inlet, all lengths with the obstacle height “ h ”, pressure by “ ρu^2 ”, and time by “ h/u ”. The non dimensional form of equations are

Continuity

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

x-momentum

$$\frac{\partial u}{\partial t} + \frac{\partial uu}{\partial x} + \frac{\partial uv}{\partial y} = -\frac{\partial P}{\partial x} + \frac{1}{Re} \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)$$

y-momentum

$$\frac{\partial v}{\partial t} + \frac{\partial uv}{\partial x} + \frac{\partial vv}{\partial y} = -\frac{\partial P}{\partial y} + \frac{1}{Re} \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)$$

DISCRETIZATION

Finite volume method is being used to discretize the equations. Entire solution domain has been sub-divided into a non overlapping cells (finite volumes). A staggered grid arrangements has been used where velocities are solved at face centre and pressure at cell centre. 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 discretization the corresponding coefficients will turn out to be different. Euler's explicit method is used for time integration. To

discretize convective and diffusive term, central difference scheme is being used.

x- momentum

$$(u_p^{n+1} - u_p^n)(\Delta x \Delta y)/\Delta t + \sum (F_f u_f)^n = - \sum (P_f S_{fx})^n + \left(\frac{1}{Re}\right) \sum (F_{duf})^n$$

y-momentum

$$(v_p^{n+1} - v_p^n)(\Delta x \Delta y)/\Delta t + \sum (F_f v_f)^n = - \sum (P_f S_{fy})^n + \left(\frac{1}{Re}\right) \sum (F_{dvf})^n$$

F_f denotes volume flux through a face f (i.e., normal component of velocity times area of the face). $F_{d\phi f}$ denotes diffusion flux of ϕ through the face (i.e., normal gradient of ϕ times area of the face).

SOLUTION METHOD

The solution method is based on MAC algorithm. The algorithm is as follows-

- 1) Guess pressure field P^*
- 2) Discretize and solve momentum equations with P^*

$$a_e^u u_e^* = \sum_{nb} a_{nb}^u u_{nb}^* + \frac{\Delta t}{\Delta x} (P_P^* - P_E^*)$$

$$a_n^v v_n^* = \sum_{nb} a_{nb}^v v_{nb}^* + \frac{\Delta t}{\Delta y} (P_P^* - P_N^*)$$

- 3) Calculate $(\nabla \cdot V)$
- 4) Calculate pressure correction term P'

$$P'_p = \frac{\nabla \cdot V}{-2\Delta t \left(\frac{1}{\Delta x^2} + \frac{1}{\Delta y^2} \right)}$$

- 5) Update velocity and pressure

$$a_e^u u'_e = \frac{\Delta t}{\Delta x} (P'_p - P'_E)$$

$$a_n^v v'_n = \frac{\Delta t}{\Delta y} (P'_p - P'_N)$$

$$u = u^* + u'$$

$$v = v^* + v'$$

$$P = P^* + P'$$

- 6) Check $\|\nabla \cdot V\| < \epsilon$

- 7) If yes then go to step 2 for time marching. If no then repeat step 3 to 6 until we get divergence free velocity field.

RESULTS

Investigation was carried out for a range of Reynolds number from 5-250 based on the parameters recommended in literature. Up to Reynolds Number 50, the flow is steady. Between Reynolds numbers 50 to 55, instability occurs and vortex shedding appears and flow becomes unsteady.

U velocity contour is being plotted for $Re=5$ and 55 respectively for $t=15$ sec at which the velocity field reaches steady state.

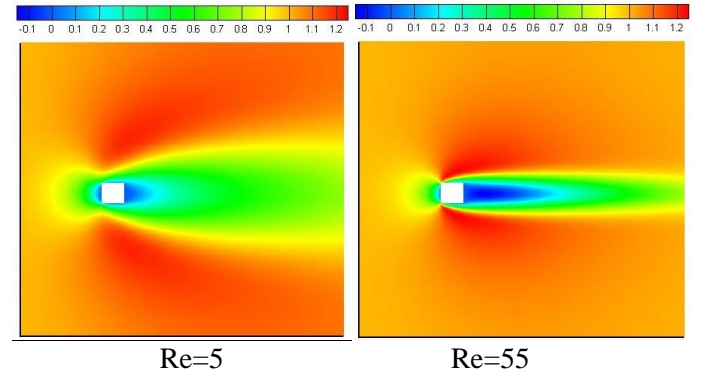


Fig : 'u' velocity contour at steady state

It is observed from the above figure that wake size is smaller for $Re=5$ as compared to $Re=55$ it is because of higher inertia of the fluid for later case. Uniform velocity $u=1$ is given at inlet and because of no slip boundary condition at cylinder walls we are getting a velocity gradient in both horizontal and vertical direction. Blue color of the velocity contour indicates lowest and red color is the highest velocity. In the recirculation zone as flow reverses so velocity magnitude is minimum at this zone as we can also see it in the above figure.

Coefficient of drag, C_D is also calculated for different Reynolds number and is being validated with literature. This C_D is taken after flow reaches steady state.

Re	C_D (Present)	C_D (Breuer[1])
5	5.38	5.5
30	1.92	2.0
55	1.55	1.6

To study how the contours are changing with time streamlines are being plotted for $Re=55$. Following are the plots to show how wakes are growing with time for $Re=55$.

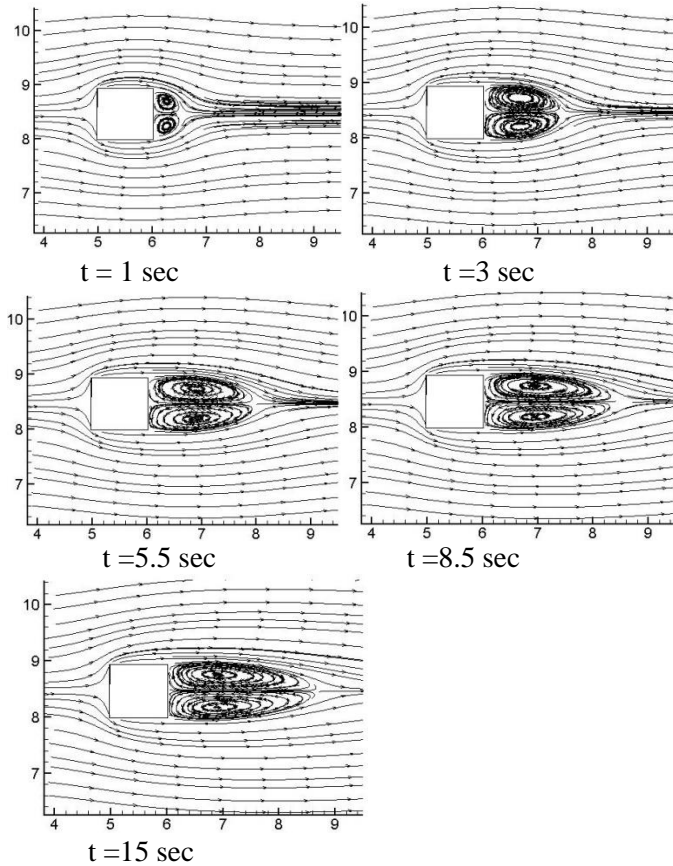


Fig: Streamlines for $Re=55$ at different time

It is observed from the plot

at $t=1$ sec, $L_r=0.4$

at $t=3$ sec, $L_r=1.4$

at $t=5.5$ sec, $L_r=1.8$

at $t=8.5$ sec, $L_r=2.3$

at $t=15$ sec, $L_r=2.8$

recirculation length increases with time and at $t=15$ sec it reaches steady state. This recirculation length is compared with Breuer (1999) and it is found that for $Re=55$, recirculation length is 2.9 which is very near to our numerical result.

To validate further with literature the code is being run for different Reynolds number 5,10,20,30,40 and 55. We restricted upto 55 because flow field reaches steady state upto this Reynolds number. Beyond 55 flow is unsteady and wakes start fluctuating. Recirculation length variation with Reynolds number is being plotted and validated with Breuer (1999). A good match is being

observed. This recirculation length is being taken after flow reaches steady state. On an average after almost 15 sec flow reaches steady state.

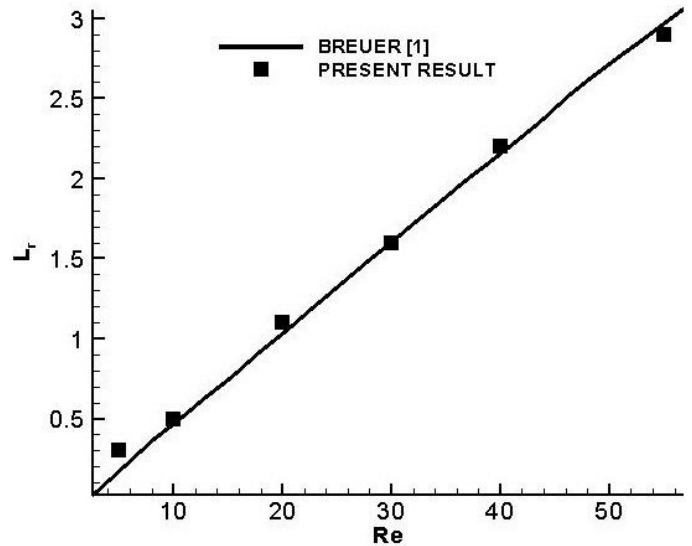


Fig: Variation of Recirculation length with Re

CONCLUSIONS:

CFD analysis is carried out using MAC algorithm for unsteady, incompressible 2D flow around a square cylinder at zero angle of incidence for Re ranging from 5 to 55. A study on recirculation length (L_r) of the wake region and Coefficient of drag (C_D) is being carried out and validated with literature. Beyond Re 55 flow becomes unsteady and an attempt is also made to capture the unsteady behavior of the flow field but it is observed in the literature that Lift force is dominant after 100 sec so due to time constraint unsteady analysis is being kept aside.

ACKNOWLEDGMENTS

Support from Dr. Kameswararao Anupindi.

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) Kelkar, K. M.; and Patankar, S. V., Numerical prediction of vortex shedding behind a square cylinder, International Journal Numerical Methods in Fluids vol (14), 1992, p.327

