International Journal of Advance Research in Science and Engineering
Volume No.06, Issue No. 11, November 2017
www.ijarse.com
IJARSE
ISSN: 2319-8354

Numerical investigation of unsteady incompressible viscous flow over flat plate having rectangular obstruction using vorticity-stream function approach

Rupam Deka[1], K. K. Das[2]

[1]*Mechanical Engineering Department,*
Dibrugarh University Institute of Engineering and Technology, Dibrugarh, (India)
[2]*Mechanical Engineering Department, Assam Engineering College, Guwahati, (India)*

ABSTRACT

In this work an attempt has been made to numerically simulate the effect of the flow past a flat plate having rectangular bluff body. A CFD code has been developed based on two-dimensional unsteady Navier-Stokes equations using Vorticity-Stream function approach. Results are in good agreement with the benchmark computation of flow over rectangular bluff body. The influences of the height of the bluff body and location of the bluff body have been investigated.

Keywords: Bluff Body, CFD, Stream Function, Simulation, Vorticity.

I INTRODUCTION

The study of wind flow over buildings has been an active area of research for several decades. Numerous experimental studies have been carried out to understand the flow behaviour of the wind when it interacts with a structure. The interaction of the wind with the building and building like structures affect the pedestrian comfort, pollution dispersion and also the ventilation within the building. In the present study, the physical problem of wind flow over a building has been simulated numerically.

In the present work a numerical code has been developed to study the flow around an obstruction. The two-dimensional Navier-Stokes equations have been solved using vorticity-stream function formulation.The numerical code developed has been used to predict the flow field and vorticity distribution. The effect of heights of the obstruction on the flow field is studied. Further, the variation of approach length to the obstruction on the flow is also studied.

The type of flow produced by the bluff bodies or building like structures in their wakes, particularly as regards the amount and level of organization of the vorticity field has the greatest importance both from the aero dynamical and from the design points of view. The numerical modelling of such type of flow is extremely difficult. However, experimental investigation is quite laborious, time consuming and costly.

Harlow & Welch (1965) [1] described a new technique for the numerical investigation of the time-dependent flow of an incompressible flow. The boundary of the flow is partially confined and partially free. The new technique is called the marker and cell method. Tamura &Kuwahara (1990) [2] investigated numerically the aerodynamic behavior of square cylinder in a uniform flow at high Reynolds numbers. They obtained the numerical solution for the unsteady flows by direct integration of the 2-D and 3-D incompressible Navier-Stokes equations in the generalized coordinate system. A systematic study has been carried out for the 3-D evaluation of wind flow around a building with emphasis on the boundary treatment by Stathopoulos &Bhaskaran (1990) [3]. Differential equations are discretized into difference form using the control volume method. Qasim et al.(1992) [4] performed 2-D computations to simulate the flow over the Field Test Facility Building located at Texas Tech University. Texas Tech Building is approximately 9.1m wide, 13.7m long and 4.0m high.Pearce et al. (1992) [5] investigated the periodic shedding of vortices from bluff body geometries with a 2D finite difference code.Zhang et al. (1993) [6] have investigated the effects of incident shear and turbulence on flow around building by a turbulent kinetic energy/dissipation k-ε model. Castro et al.(1997) [7] presented some aspects of computational work undertaken as part of a multipartner European Union projects on flow and dispersion around building.Bouris&Bergeles (1999) [8] have performed a two-dimensional large eddy simulation of the quasi-two-dimensional turbulent flow past a square cylinder with no-slip boundary conditions at the solid walls. Gao &Chow (2005) [9] studied air approach flow moving towards a cube. The RANSequation types of k-ε turbulence model are used. Flow separation at the corner above the top of the cube, level of separation and reattachment were investigated.

II GOVERNING FLOW EQUATIONS

In this work 2D unsteady Navier-Stokes equation has been solved using vorticity-stream function approach. The governing equations are vorticity transport equation and Poisson's equation.

$$\frac{\partial \xi}{\partial t} + u\frac{\partial \xi}{\partial x} + v\frac{\partial \xi}{\partial y} = \frac{1}{R_e}\left(\frac{\partial^2 \xi}{\partial x^2} + \frac{\partial^2 \xi}{\partial y^2}\right) \qquad (2.1)$$

$$\left.\begin{aligned} \xi &= \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \\ u &= \frac{\partial \psi}{\partial y} \\ v &= -\frac{\partial \psi}{\partial x} \end{aligned}\right\} \longrightarrow \frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = -\xi \qquad (2.2)$$

Where ξ is the vorticity, ψ is the stream function, u and v are x and y component of velocity.

III METHODOLOGY

The governing equations are converted into algebraic equations before being solved. The Finite Difference Method (FDM) has been used for the same and in this Forward Time Central Space (FTCS) scheme has been adopted.

The vorticity equation is parabolic, which is solved by time marching procedures whereas the stream function is elliptic and is solved by successive over relaxation methods (SOR). Thus, at every time step, both parabolic and elliptic equations have to be solved.

The number of grid points amounted to 150(x- direction) X 100(y- direction) = 15,000. The grid size is uniform throughout the zone. The computational domain has a downstream length and vertical height of 20 times the width of the obstruction and upstream length of 10 times of the width of the obstruction. The Reynolds number of the flow is 7000.

3.1 Boundary Conditions:

Impermeable no slip boundary condition has been applied on the solid boundary. The time marching procedure for solving the vorticity transport equation requires appropriate expressions for Ψ and ξ at the boundaries. The specifications of these boundary conditions are extremely important since it directly affects the stability and accuracy of the solution. Vorticity value at the inflow is taken as zero. Near the wall vorticity is given by $\xi = \left(\frac{\psi_{w+1}-\psi_{w}}{2}\right)+0.\Delta n$,where$\Delta n$ is the distance from (w + 1) to w, normal to the wall. At the outflow $\frac{\partial \xi}{\partial x}$ is taken as 0. Stream function is assumed to be equal to U.y and at the outflow $\frac{\partial \psi}{\partial x}$ = 0.On the solid surface it is having zero value. At inflow u = U and v = 0. At the outflow ∂u/∂x = ∂v/∂x = 0. At the far field u = U and v = 0. The velocities u = v = 0 at the walls due to no slip boundary condition.

IV RESULTS AND DISCUSSION

The flow characteristics over a rectangular building are shown in terms of streamline contour, vorticity contour and velocity profiles at various locations. The results are shown for Re = 7000.

Figure 4.1 shows the streamline plot and figure 4.2 shows the iso-vorticity plot. It is observed that the streamlines move downstream of the obstruction in curved paths. Behind the obstruction the streamlines have clockwise circular movements forming a rectangular zone characterized by flow reversal.

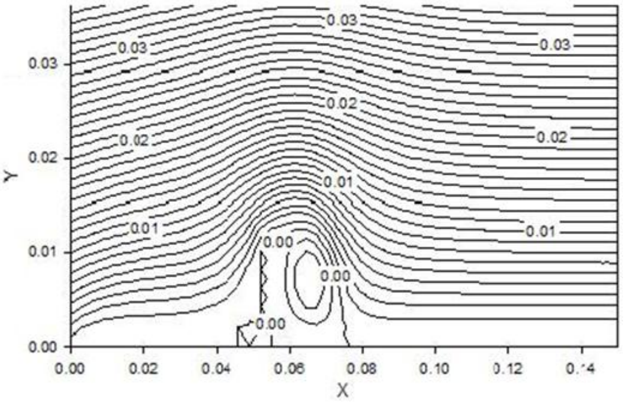


Fig-4.1 Streamline plot over building

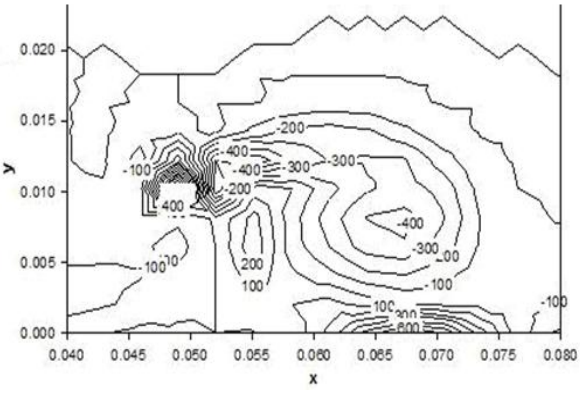
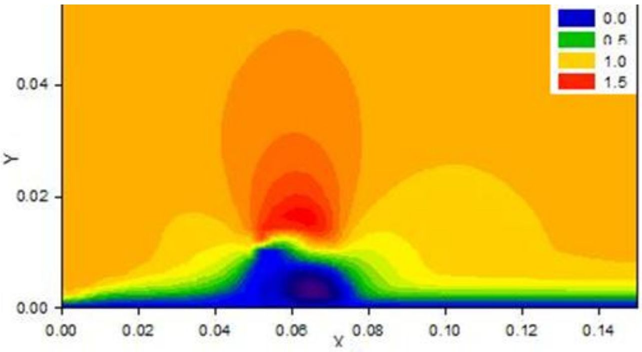


Fig-4.2Iso-vorticity plot

Both streamline pattern and iso-vorticity plots show that the flow field is fully turbulent and is characterized by strong stream line curvature, fluid rotation, flow separation etc.



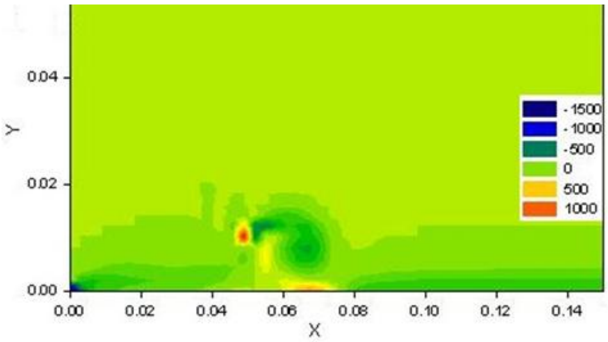
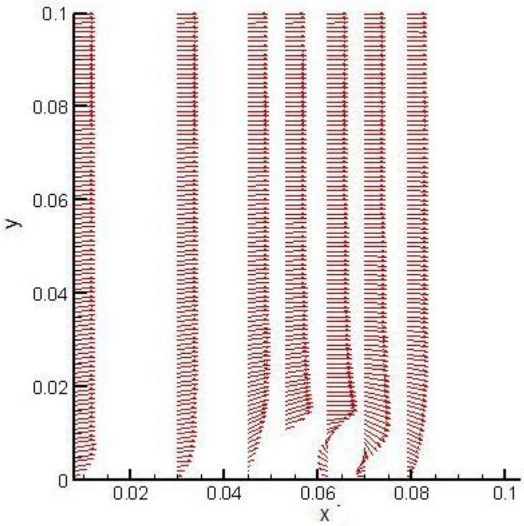


Fig-4.3 u Velocity contour for a normalised **Figure-4.4 vorticity contour for abuilding height**
(h=0.01)normalised building height (h=0.01)

Figure 4.3 and figure 4.4 show the u contour and vorticity contour for the normalised height (h=0.01) of the obstruction. Figure4.5 shows the velocity vector plot over a flat plate in the presence of a rectangular obstruction at different locations. The obstruction has been placed at a normalised distance 0.05 from the inflow. In the upstream of the obstruction the velocity profile is similar to that observed over a flat plate. As we move closer to the obstruction the velocity profile is modified. Reversal of flow is observed downstream of the obstruction. As we move further away this effect diminishes.



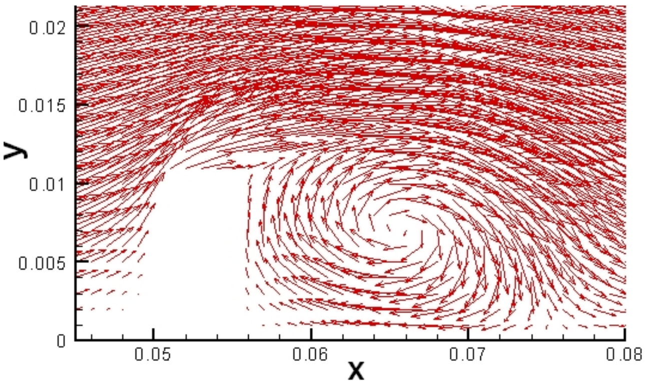


Fig- 4.5 Velocity profiles at different locations **Fig-4.6 Velocity vector plot**

In the figure-4.6 it has observed that stagnation occurs near the wall at inflow and windward side of the obstruction. In the downstream the velocity vectors show reversal of flow.

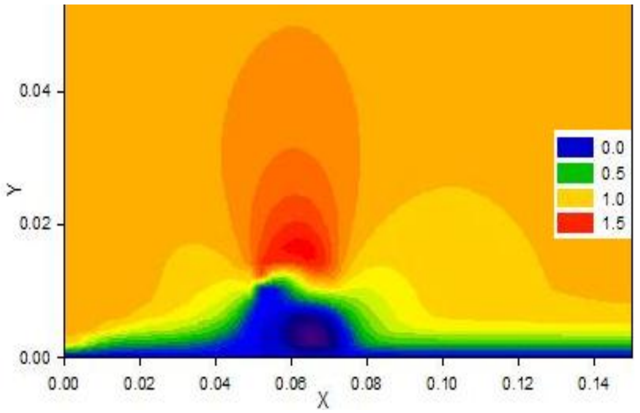


Fig-4.7 u Velocity contour for a normalised building height(h=0.008)

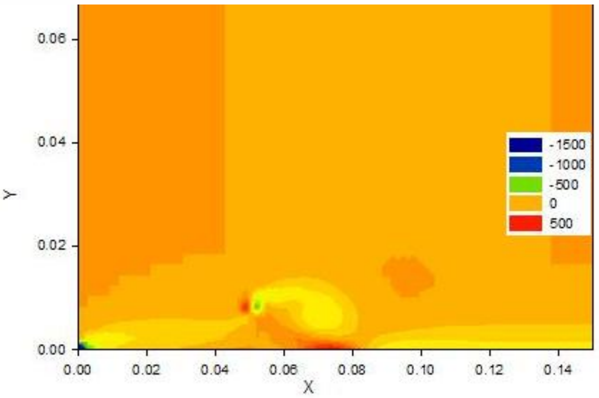


Fig-4.8 vorticity contour for normalised building height (h=0.008)

Figure 4.7 and figure 4.8 show the u contour and vorticity contour for the normalised height (h=0.008) of the obstruction.Figure 4.9 and figure 4.10 show the u contour and vorticity contour for the normalised height (h=0.012) of the obstruction. It is observed that with increase in height of the obstruction, the zone of disturbance in the downstream increases. The recirculation in the downstream is higher for greater height. The region above the building has higher disturbance for taller buildings.

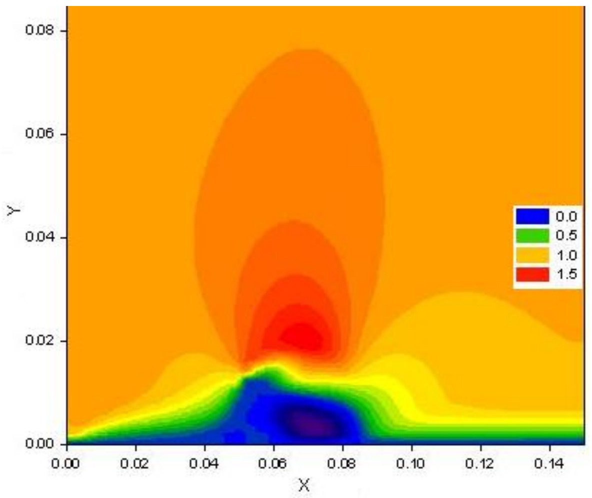


Fig- 4.9 u velocity contour for a normalised building height (h=0.012)

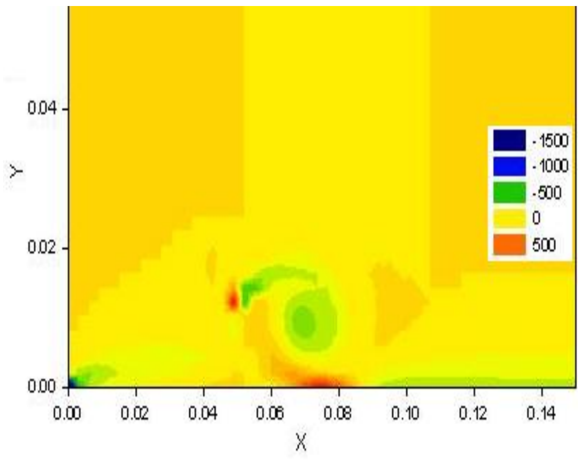


Fig-4.10 vorticity contour for normalised building height (h=0.012)

V.CONCLUSION

A numerical code based on 2D N-S equations has been developed using vorticity-stream function formulation. The code has been used to study the flow over a flat plate having a rectangular obstruction of different heights and at different distances from the leading edge. The results have been obtained in terms of streamline and iso-vorticity lines, velocity profile and velocity vectors at different locations. The results show that the height of the obstruction has significant effect on flow characteristics. With the increase in height of the obstruction the zone of disturbance in the downstream increases.The recirculation in the downstream is higher for greater height. In case of a taller obstruction a higher pressure drop occurs in the wake of the obstruction. Due to the variation in approach length the boundary layer formation in upstream of the obstruction has a prominent effect on the flow field around the obstruction.

REFERENCES

[1] Harlow F H & Welch J E (1965), " Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface", Phys. of Fluid, Vol.8, No.12, pp.2182-2189.

[2] Tamura T &Kuwahara K (1990), " Numerical Study Of Aerodynamic Behavior of Square Cylinder" , Jl. Of Wind Engineering & Industrial Aerodynamics, Vol. 33, pp. 161-170.

[3] Stathopoulos T &Bhaskaran A (1990) ," Boundary Treatment for the Computation of Three-Dimensional Wind Flow Conditions Around a Building", Jl. Of Wind Engineering & Industrial Aerodynamics, Vol. 35, pp. 177-200.

[4] Qasim A, Maxwell T T&Parameswaram S (1992) "Computational Prediction of Flow Over a 2-D Building" Jl.Of Wind Engineering & Industrial Aerodynamics, Vol.41-44, pp. 2839-2840.

[5] Pearce J. A, Qasim A, Maxwell T. T &Paramesawaran (1992) "A Computational Study of Coherent Wake Structure Behind 2-D Bluff Bodies", Jl. Of Wind Engg. & Industrial Aerodynamics, Vol.41-44, pp.2853-2861.

[6] Zhang Y Q , Huber A H , Arya S P S & Snyder W H (1993), " Numerical Solution to determine the effects of incident Wind Shear and Turbulence Level On the Flow Around a Building", Jl. Of Wind Engineering & Industrial Aerodynamics, Vol. 46 & 47, pp. 129-134.

[7] Cowan I R , Castro I P & Robins A G (1997), " Numerical consideration for simulation of flow & dispersion around buildings" , Jl. Of Wind Engineering & Industrial Aerodynamics, Vol. 67 & 68, pp. 535-545.

[8] BourisD &Bergeles G (1999) , " 2D LES of vortex shedding from a square cylinder" , Jl. Of Wind Engineering & Industrial Aerodynamics, Vol. 80, pp. 31-46.

[9] Gao Y & Chow W. K (2005), "Numerical studies on air flow around a cube", Jl. Of Wind Engg. & Ind. Aerodynamics, Vol. 93, pp. 115-135.