International Journal of Advanced Research in
Mechanical and Production Engineering and Development
Volume: 1 Issue: 2 08-Apr-2014,ISSN_NO: 2320-7256

CFD Analysis of 2-D Unsteady Flow Past a Square Cylinder at an Angle of Incidence

Kavya H.P[1], Banjara Kotresha[2], Kishan Naik[3]

Dept. of Studies in Mechanical Engineering, University BDT College of Engineering, Davangere, India [1,3]
Dept. of Mechanical Engineering, Govt Engineering College, Haveri, India[2]
kavyahp88@gmail.com[1], bkotresha@gmail.com[2], kishennaik@gmail.com[3]

ABSTRACT- *This paper presents a numerical simulation of two dimensional unsteady flow past a square cylinder at an angle to the incoming flow. The main objective of this study were to capture the features of flow past a square cylinder in a domain with the use of CFD for the Reynolds number (Re) considered in the range 50-200 so that flow is laminar. The shape and size of the re-circulation bubble downstream of the cylinder are strong function of orientation. Results are presented in terms of Strouhal number (st), Time-averaged velocity and Vorticity field. The Lift Co-efficient (cl) and velocity component in the wake region were monitored for calculation of Strouhal number. The variation of Strouhal number with Reynolds number was found from the analysis.*

Keywords__Square Cylinder, Strouhal number, Lift Co-efficient.

1. INTRODUCTION

The study of bluff body wakes is important for applications in aerodynamics, wind engineering. Examples of such cylindrical structure in engineering applications include sky scrapers, towering structures, long-spanned bridges and wires to name a few [6]. Since the vortices cause periodic force loading on the structure, leading to structural vibration and flow induced noise, many researchers have paid much attention to controlling the vortex shedding. It is also necessary to study flow around other bluff body shapes, such as sharp-edged rectangular cross sectional cylinders. Structures that typically have rectangular or near rectangular cross sections include architectural features on buildings, the buildings themselves beams, fences and occasionally stays and supports in internal and external flow geometries. When these structures are exposed to cross flow the separation takes places from the upper and lower portion of the body [3]. Due to instability the phenomenon of vortex shedding develops known as vonkarman vortex street.
Sushanta dutta., *et. al.*, [1] experimentally investigated using particle image velocimetry, hot wire anemometry, and flow visualization. The Reynolds number based on cylinder size and the average incoming velocity is equal to 410. Results are presented in terms of drag coefficient, Strouhal number, time-averaged velocity, stream traces, turbulence intensity,

power spectra and vorticity field. Vikram C.K.,*etal.*, [2] numerically investigated two dimensional unsteady flows past two square cylinders with in-line arrangements in a free stream. It has been found that the size of the eddy and the monitored velocity in between the square cylinders increases with increasing PPR. Gera.B .,*et al* ., [3] carried out a numerical simulation for flow past a square cylinder to see the wake behaviour for the Reynolds number (Re) considered in the range of 50-250 so that flow is laminar. The variation of Strouhal number with Reynolds number was found from the analysis for a zero angle of incidence.

2. GOVERNING EQUATIONS AND BOUNDARY CONDITIONS

Flow past a square cylinder has been simulated by solving numerically the unsteady Navier-Stokes equations for an incompressible fluid in a two dimensional geometry. The equations for continuity and momentum may be expressed in the dimensionless form as follows:

-Continuity

$$\frac{\partial \mathrm{u}}{\partial \mathrm{x}} + \frac{\partial \mathrm{v}}{\partial \mathrm{y}} = 0 \tag{1}$$

-X-momentum

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

-Y-momentum

$$\frac{\partial v}{\partial t} + \frac{\partial (uv)}{\partial x} + \frac{\partial (vv)}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{Re}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

In the above equations, the velocities are non-dimensionalised with average velocity U_{in} at the inlet and pressure at the outlet. Two dimensional pressure based unsteady finite volume code was used with a staggered grid arrangement. The unsteady formulation is first order implicit and an absolute velocity formulation is used. The PISO pressure-velocity coupling method is used for solving transient applications, with second order upwind momentum

3. NUMERICAL DETAILS

3.1 Computational Domain

The computational domain used for the analysis of flow past a square cylinder is shown in Fig 1.

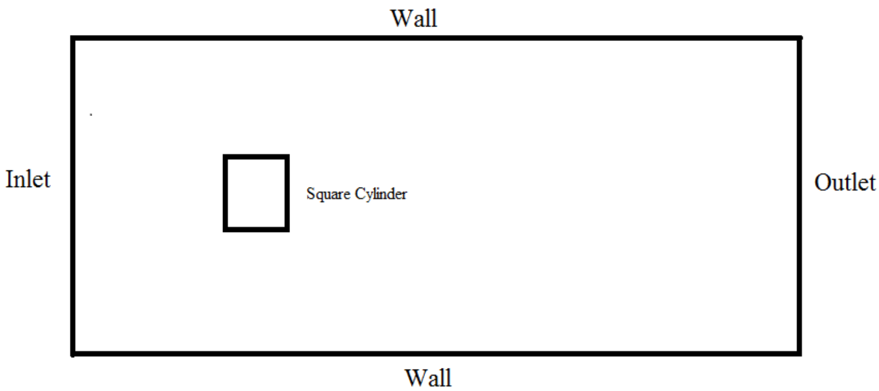


Figure 1. Computational domain for the flow past a square cylinder

The geometry and boundary conditions used for the study is same as used by other investigators. The inflow, top and bottom boundaries have been located 6.5 square cylinder with respect to the canter of the square cylinder, the computational domain has been extended to 30 square cylinder perimeters in the downstream of the cylinder [2].

3.2 Mesh Details

The computational domain shown in Fig 1 is meshed by using GAMBIT pre-processor and the mesh used for the simulation is shown in Fig 2.

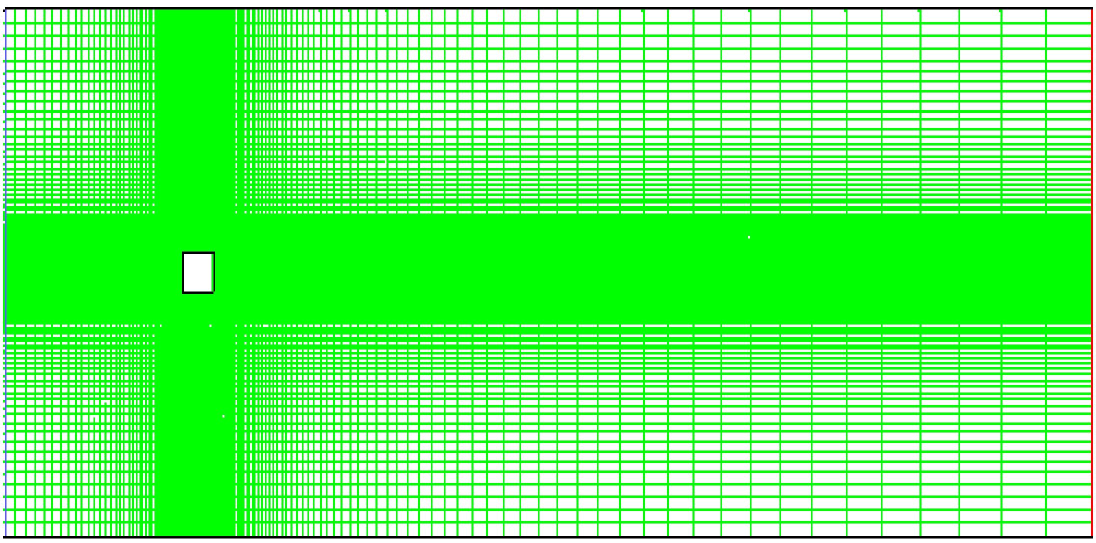


Figure 2: Finite volume mesh

In the present work, the grid size of 150×120 are considered for the computational domain. The time marching calculations were started with the fluid at rest. A constant time step $\Delta t = 0.02$ was used for all calculations. During the iterative sequence, convergence was assessed at the end of each iteration on the basis of the residual source criterion, which compares the sum of the absolute residual sources over all the residual sources overall the control volumes in the computational field, for each finite volume equation [3].

3.3 Boundary conditions

In the present study flow past square cylinder has been computed by applying boundary conditions as follows [2].

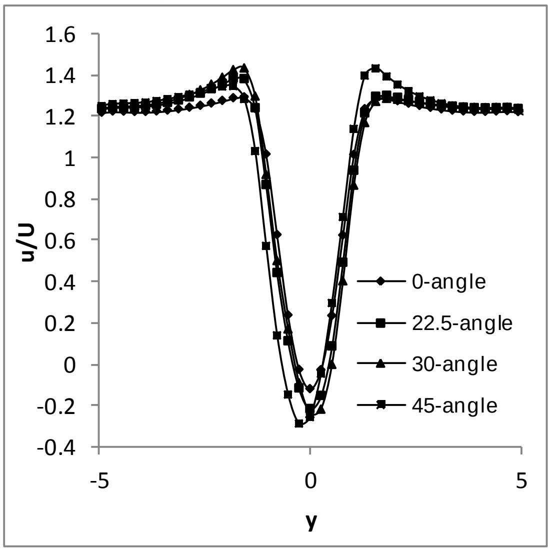
- Inlet – uniform flow (U=1.0, V=0.0)
- Cylinder surface – No slip (U=V=0.0)
- Top and Bottom boundaries (U=1.0 V=0.0)
- Outlet boundary – pressure outflow.

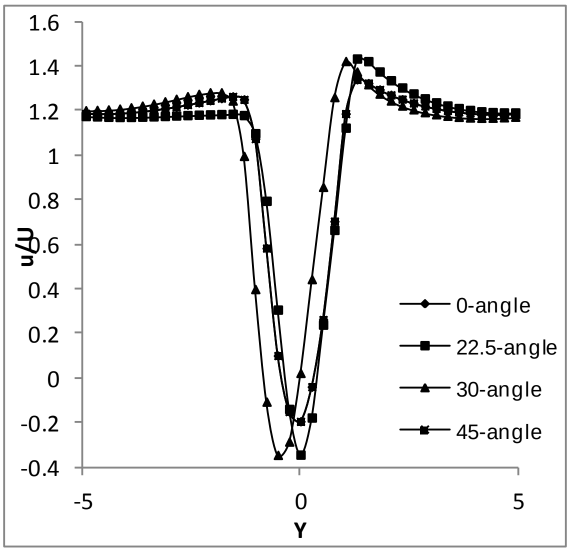
4. RESULTS AND DISCUSSION

Investigation was carried out for an angle of incidence of a square cylinder with limited range of Reynolds number from 50-200 recommended in literature. The axial velocity profiles, Lift Coefficient and Strouhal number is calculated from the numerical simulation.

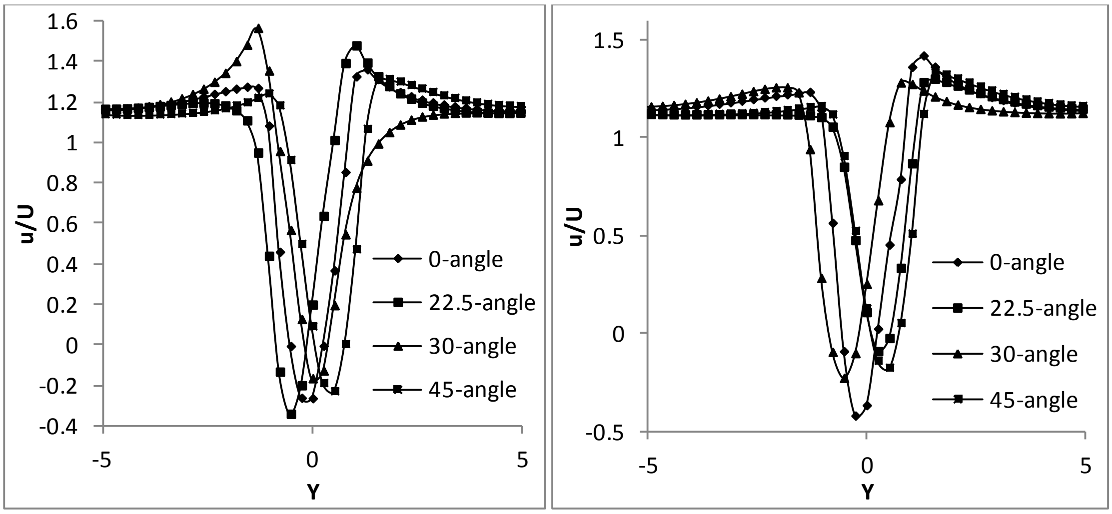
4.1 Velocity Profiles

Profiles of Time averaged axial velocity at x=2 from the square cylinder for four different angle of incidence (0, 22.5, 30, 45) at different Reynolds's numbers are plotted in section.





(a) **(b)**



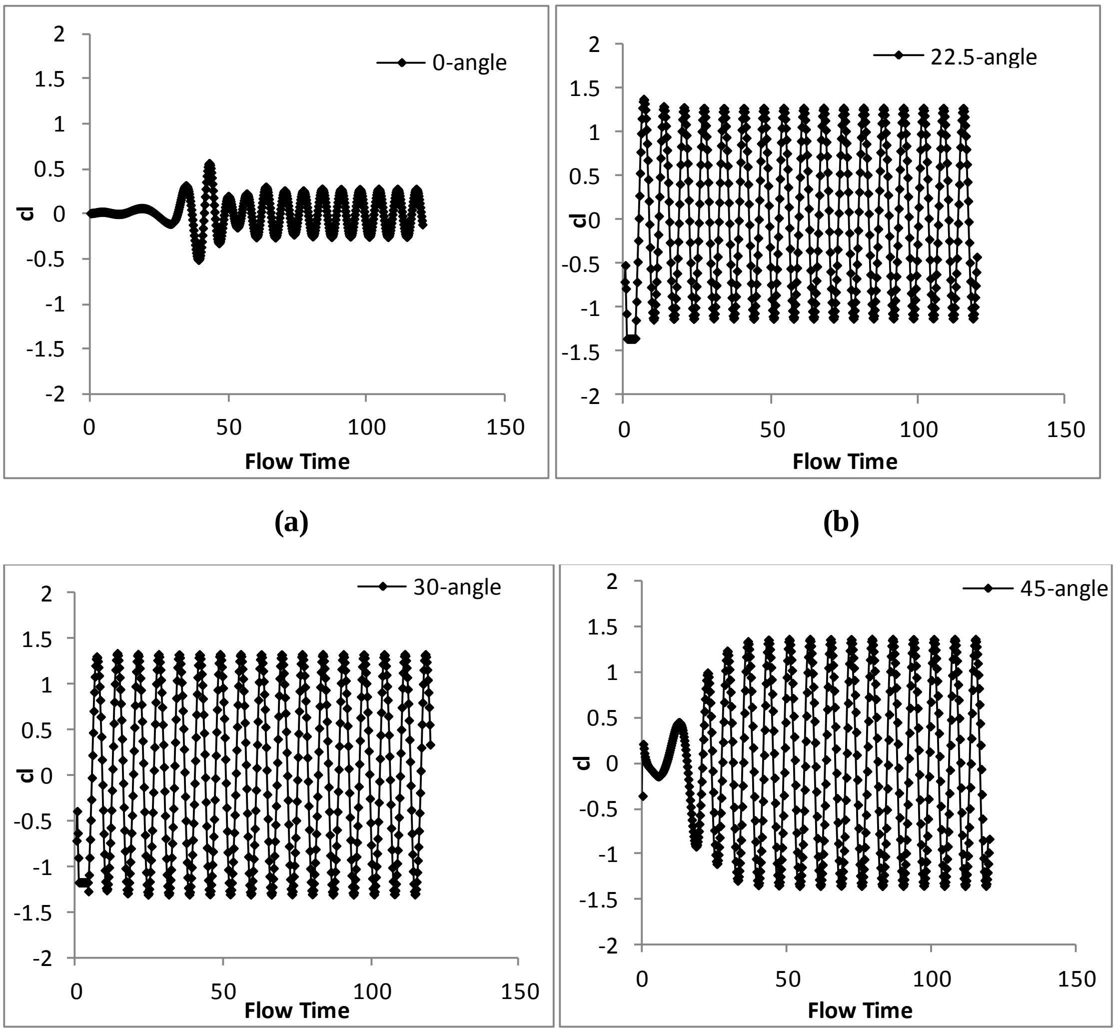
(c) **(d)**

Figure 3: Time Averaged Axial velocity at X=2 for different angle of incidence at (a) Re-50 (b) Re-100 (c) Re-150 (d) Re-200

Fig 3. Shows the Time averaged u velocity profiles for the different Reynolds number in the direction transverse to the main flow, for several orientation of the cylinder. The position of the velocity minimum is along the center line for symmetric configurations of Re 50 for all the angle of orientation and for other Reynolds number, the minimum shifts away from the mid plane (y=0) in the near wake (x=2) for all the orientation of a square cylinder.

4.2 Lift Coefficient

The time history of lift coefficient of square cylinder for different angle of incidence (0, 22.5, 30, 45) at Re-200 are plotted in this section.



(a) (b) (c) (d)

Figure 4: Lift Coefficient for different angle of incidence at Re-200 (a) 0^0 angle (b) 22.5^0 angle (c) 30^0 angle (d) 45^0 angle

Fig 4 shows the variation of Lift Coefficient with flow time for Re 200. The magnitude of Lift Coefficient is lower at Re 50 and for other cases the magnitude of Lift Coefficient is almost same. So that Strouhal number is initially low for 0^0angle compared to other cases.

4.3 Vorticity Contours

The following section gives the Vorticity line contours for Re-100.

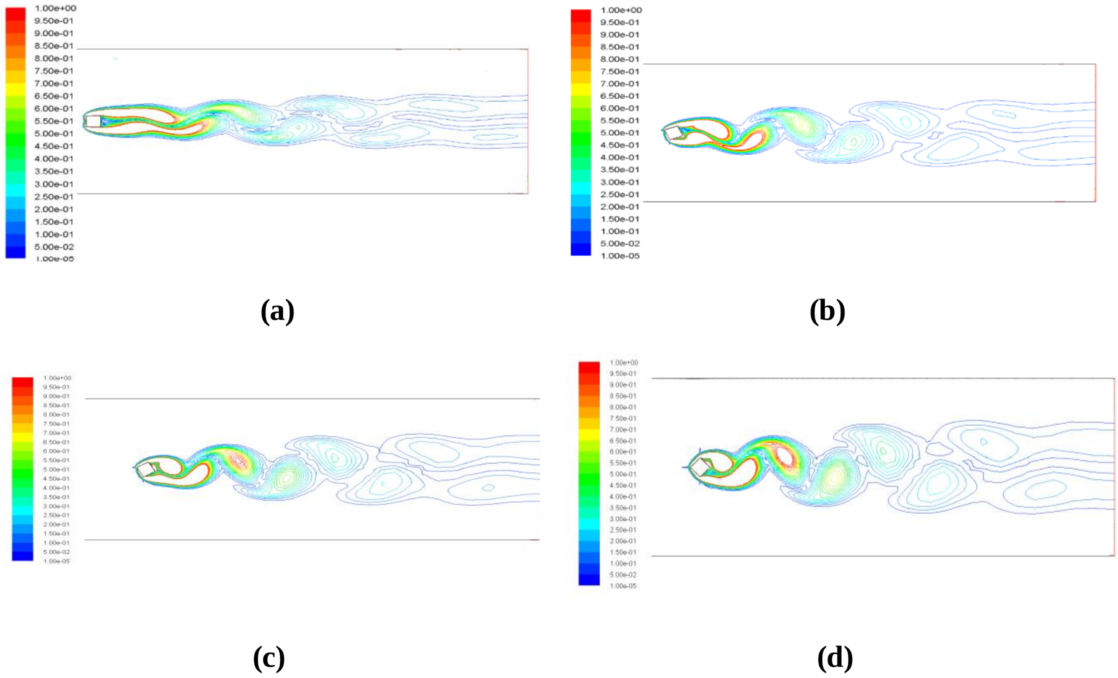


Figure 5: Vorticity Line contour for different angle of incidence at Re-100 (a) 0^0 Angle (b) 22.5^0 Angle (c) 30^0 Angle (d) 45^0 Angle

The change in angle of the cylinder affects the wake primarily due to the following two factors: (1) Change in the projected dimension normal to the flow and (2) Movement of the point of separation and hence the position of the dividing streamline. The study was carried out for a range of Reynolds number from 50-200 [3].

Table 1.Strouhal number obtained for different cases.

Re	0 degree	22.5 degree	30 degree	45 degree
50	-	0.1190	0.1190	0.1190
100	0.147	0.139	0.135	0.131
150	0.151	0.147	0.1428	0.1351
200	0.119	0.147	0.1428	0.1388

Table1. Shows the Strouhal number variation with cylinder orientation at different Reynolds number. Between Re 50-150; instability occurs and vortex shedding appears and flow becomes unsteady. Above the Re 200, the Strouhal number increases from the 0^0 to 22.5^0 incidence angle. The Strouhal number variation with incidence angle is related to an increase in the projected dimension of the cylinder with respect to the incoming flow. The vortex shedding frequency is influenced by the width between two free shear layers and the

free stream velocity. The increase in the incidence angle leads to an increase in the distance between the two free shear layers. Therefore, an increase in the incidence angle results in a reduced interaction between the two shear layers and a drop in the Strouhal number. Thus, a maximum seen at 22.5^0 incidence angle is due to competing effects of an increased projected dimension and a shorter vortex roll up distance [1].

5. CONCLUSION

CFD analysis was carried out for unsteady incompressible 2D flow past a square cylinder for different orientation at different Reynolds number ranging from 50 to 200 with respect to the incoming flow. Strouhal number (st), Lift coefficient (cl), Time averaged velocity and vorticity lines were determined from the numerical simulation. The Strouhal number remains almost uniform between Re 50 to 150 for different orientations of the square cylinder. At Re 200 the Strouhal number increases at 22.5^0angle of orientation and then decreases for other angles. The Time avaraged velocity profile in the direction of transverse to the main flow is minimum at Re 50. The magnitude of lift co-efficient is lower at Re 50 and for other cases the magnitude of lift coefficient is almost same. And the eddies size increases as the angle of incidence increases. Both Strouhal number and Lift Coefficient respond strongly to the orientation of the cylinder.

6. REFERENCES

[1]. SushantaDutta, P.K. Panigrahi and K. Muralidhar. "Experimental Investigation of Flow Past a Square Cylinder at an Angle of Incidence".

[2]. Vikram C.K., Dr. Y.T. KrishneGowda, Dr. H.V. Ravindraet al., "Numerical Simulation of two Dimensional Unsteady Flow Past Two Square Cylinders".

[3]. Gera. B, Pavan K. Sharma, Singh R. K. "CFD Analysis of 2D Unsteady Flow Around a Square Cylinder".

[4]. SushantaDutta, P.K. Panigrahi and K. Muralidhar. "Effect of Orientation on the Wake of a Square Cylinder at Low Reynolds numbers."

[5]. Obasaju, E. D. _1983_. "An investigation of the effect of incidence on theflow around a square section cylinder."*Aeronaut. Q.*, 34, 243–259.

[6]. Dong-Hyeog Yoon, Kyung-soo Yang, and Choon- Bum Choi. "Flow Past a Square Cylinder with an Angle of Incidence". *Physics of fluids22.*