International Journal of Advanced Engineering and Technology
ISSN: 2456-7655
Impact Factor: RJIF 5.54
www.newengineeringjournal.com
Volume 2; Issue 3; September 2018; Page No. 15-18

Analysis of circular and square cylinder at low Reynolds number using CFD

Rahul Kumar[1*], Kajal Sarmah[2], Kamlesh Kumari Sharma[3]

[1-3] Assistant Professor, Department of Mechanical Engineering, Modern Institute of Technology & Research Centre, Alwar, Rajasthan, India

Abstract

The Flow past over a circular and square body for are carried out for Reynolds number 100 and numerically by using CFD code fluid fluent. Direct numerical simulation of two-dimensional, unsteady and incompressible laminar flow is numerically investigated at Reynolds number of 100. In the present work Navier-Stokes equations are solved by using finite difference Marker and Cell (MAC) method. The accuracy of the Navier-Stokes solver is validated by the simulation of flow past a square cylinder without the splitter plate.

In the present analysis, the flow is assumed to be two dimensional. Results indicate that the In case of circular cylinder pressure in the upstream cylinder face has got large pressure than the front end face of square cylinder but in downstream of square cylinder has less pressure compared to circular cylinder.

The lift coefficients of Square cylinder are less when compared with circular cylinder. The increase in Strouhal number decreases the vortex wavelength. The results are presented in the form of streamlines pattern, pressure distribution over the different type of cylinder, velocity Counter, drag coefficient and Strouhal number.

Keywords: circular and square cylinder, Reynolds number, grid independence, drag coefficient, lift coefficient, Strouhal number

1. Introduction

In many mechanical engineering applications, separated flows often appear around any object. Fluid flow past a circular cylinder is a model problem of fundamental interest, as it impacts a number of practical engineering applications like Heat exchangers, boilers, condensers, economizers nuclear reactor fuel rods air conditioning coils etc. Vortices are formed and shed behind bluff bodies causing a sinuous wake in its downstream. Alternating eddies formed behind a bluff object gives rise to fluctuating lift and drag forces. The flow past bodies immersed in a fluid has been studied for a long time because of its importance in aero and hydrodynamic applications. A flow over a cylinder is considered as a model problem for a range of bluff body flow problems. A study of a flow past cylindrical bodies provides a general picture of the phenomenon of flow separation and bluff body wakes. The sharpness of cylinder corners in the experiments and the numerical treatment of these corners considerably influenced the shedding frequency. It is well known that the vortex wake information of a circular cylinder can be studied as a function of the Reynolds number. The first definition of flow regimes based on measurements of velocity fluctuations spectra and frequency given by Roshko (1954). He found a 'stable' (periodic) laminar vortex shedding regime for Re=40~150; a transition regime in the range Re=150~300, with an 'irregular' regime for Re=300~10000+, where velocity fluctuations shown distinct irregularities. One more bluff body square cylinder has its own application like Tall buildings, monuments, and towers bridges, skyscrapers, offshore structures, etc. are permanently exposed to wind. Similarly, piers, bridge pillars, and legs of offshore platforms are continuously subjected to the load produced by maritime or fluvial streams. These bodies usually create a large region of separated flow and a massive unsteady wake region in the downstream. An initially smooth and steady flow across a cylinder may bring about damaging oscillations, in cases where the natural frequency of the obstacle is close to the shedding frequency of the vortices. If the resulting excitation frequency synchronizes with the natural frequency of the cylinder, the phenomenon of resonance is the obvious outcome. Understanding the wake behavior and the associated dynamics of flow past square cylinder helps in the better design of the concerned or desired objectives, where the engineering parameters need to be designed with reasonable precision. A designer, therefore, is required to have a large database available in order to choose an optimal one among the different alternatives. To achieve this objective, extensive established correlations are required for different alternatives. Elaborate experiments were the order of the day. However with the advent of modern digital computers, numerical procedures are complementing with the experiments. This approach has substantially reduced monotony, time and higher labor costs involved with experimentation. In a numerical simulation changing the geometric parameters and fluid flow conditions can be easily accomplished by making suitable modifications in the input parameters. A lot of research has been carried out on flow past single circular and square cylinder; however, for the comparison study of these two which will give the difference in drag/lift forces and shedding frequency, monitored velocity pressure distribution, streamlines of bluff bodies.

2. Materials and Methods

2.1 Numerical Method

The numerical method which used in the present study is Marker and Cell (MAC) method. The MAC method first appeared in 1965. It was developed by Harlow and Welch [1] specifically for free surface flows as a variant of Particle-in-Cell (PIC) method. The original Particle-in-Cell (PIC) code [2] was developed in 1958, which used mass particles that carried material position, mass and species information. Despite the special capabilities of PIC for following discontinuities, it did not give accurate solutions in general because the transfer of information between the particles and the underlying grid resulted in numerical diffusion. For solving full Navier-Stokes equations, MAC method is used. In MAC method, there is an advection of marker particles with local fluid velocity and the distribution of marker particles determine the configuration of instantaneous fluid flow. If a cell contains a particle then it is also considered to contain fluid, therefore it provide flow visualization of the free surface. For the computation of velocity and pressure, Momentum and Poisson equations are used by this method respectively. It was developed to solve problems with free surfaces, but can be applied to any incompressible fluid flow problems. There have been many further developments and applications of the MAC method (Tome & Mckee; Chen, *et al.*) [3-4]. Its extension to three dimensions has been difficult and the coalescence and fragmentation of fluid regions meant enhanced complexity in the algorithm required if at all programmable. The computational domain is divided into Cartesian cells. Staggered grid arrangements are used in which velocity components are defined at the midpoints of the cell sides to which they are normal and the pressure is defined at the centre of the cell.

2.2 Governing Equations

The simulation for the present problem has been carried out by solving unsteady, conservative form of Navier-Stokes equations for an incompressible fluid in a two- dimensional geometry. The equations for continuity and momentum may be expressed in the dimensionless forms as follows:

Continuity equation

$$\frac{\partial u_i}{\partial x_i} = 0$$

Momentum equation

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_j u_i)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{R_e}\frac{\partial^2 u_i}{\partial {x_j}^2}$$

The above equations are the non-dimensional form of continuity and momentum equations. The non-dimensional parameters are given as $x_i = \frac{x_i}{D}, u_i = \frac{u_i}{D}, p = \frac{p}{\rho u^2}$ where D is the characteristic length scale, U is the average inlet velocity scale, p is the pressure, ρ is the density and Re is the Reynolds number given by $R_e = \frac{\rho U D}{\mu}$ (μ is the viscosity of fluid).

2.3 Boundary conditions

The boundary conditions for the present problem are:

Inlet B.C: A uniform velocity has been prescribed at the inlet of the 2- D channel i.e u(y) = 1.0 and v = 0.

Outlet B.C: The outlet boundary condition should have such specifications that it would not affect the flow in the upstream. Therefore, the convective boundary conditions which was proposed by Orlanski have been used:

$$\frac{\partial u_i}{\partial t} + u_c \frac{\partial u_i}{\partial x} = 0$$

where u_c, is the convective velocity in the above equation.

Confining boundaries: The confining boundaries (top and bottom boundaries) are modeled as the slip boundaries for example, at the transverse confining surfaces, the

$$y = \pm H/2, \frac{\partial u}{\partial y} = 0 \text{ and } v=0.$$

Obstacle: No slip (u = 0, v = 0) boundary conditions are used for the velocities on square cylinder as well as on the splitter plate surface.

2.4 Assumptions

The phenomenological near-wake model describes a cylindrical object which translates with constant speed U through ambient fluid and has lateral dimension D. This model is based on three major experimental and analytical findings related to the geometry of the vortex street, the energy associated with the stream wise motion.

2.5 Problem formulation

The present study is aimed at controlling the formation of vortex shedding and wake region behind the square cylinder by using detached splitter plate. The flow has been taken from left side to right side of the two-dimensional channel. Figure 1 represent the flow structure of the present problem.

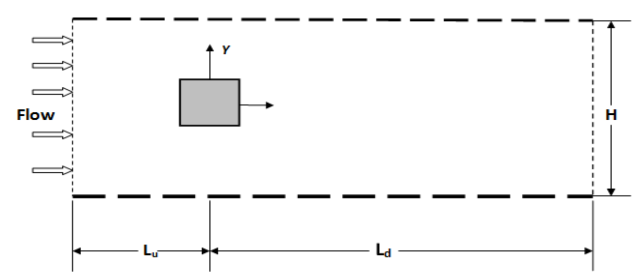

Fig 1: Flow domain for a square cylinder

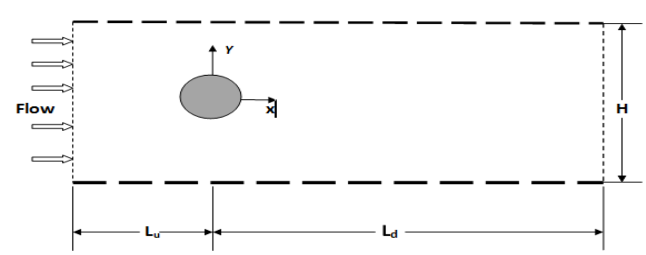

Fig 2: Flow domain for a circular cylinder

Some of the important parameters that are considered in the present problem are Reynolds number ($Re=\rho\ UD\ /\mu$), drag coefficient ($C_d=F_D\ /\ 0.5\rho U\ 2D$) and lift coefficient ($C_l=F_L\ /\ 0.5\rho U\ 2D$). Non-Uniform mesh is used for the present problem.

2.6 Finite volume mesh

The problem considered here is the flow past circular and square cylinder for Reynolds numbers 100. Choosing the numerical flow domain, which is neither too big nor too small, is still an art in computational fluid dynamics. Because only a finite computational domain can be employed for the numerical simulation, it is important to locate the inflow and far-field boundaries at sufficient distance from the main cylinder such that the boundary conditions applied at these boundaries do not introduce any undesirable effects into the main region of interest, around and behind the cylinder.

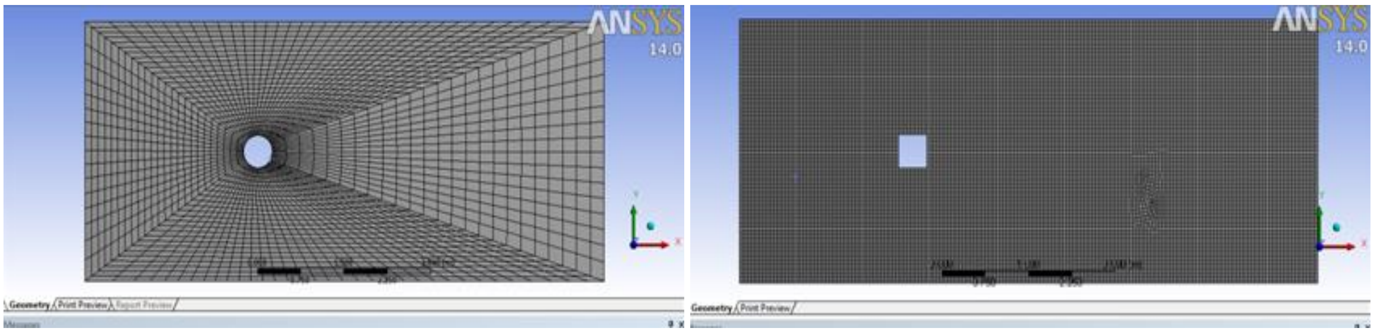

Fig 3: Mesh for a circular and square cylinder

3. Results

3.1 Streamlines

In the case of flow over a cylinder Re=100, the flow is uniform and symmetrical in the upstream of the cylinder. The eddies are alternatively formed on either side of the cylinder in the downstream. The tangential velocity of the Square cylinder is large and enlarges the separation area of square cylinder side face. The square cylinder is a bluffer body as compared to the circular cylinder. This is presented in the form of streamlines as shown in Fig. 4

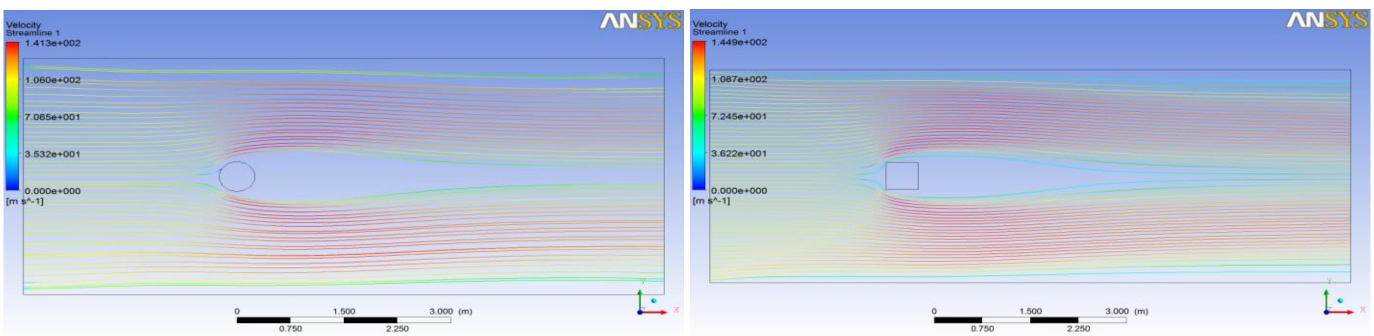

Fig 4: Computational results of the instantaneous streamline plots of the circular cylinder and square cylinder for Re = 100.

3.2 Pressure distribution around a circular and square cylinder

Pressure distribution is important in the study of flow around bluff bodies. Pressure changes accordingly with the vortices motion in the vicinity of the bodies. Flow separates alternately around symmetrical bodies with sharp corners such as the leading edge of a square section to form vortices around the cylinder. This usually introduces periodic forces on the body due to the pressure changes. This situation is particularly significant in flow involving fluid and structure interaction such as the flow around a tall building or suspension bridge. Although pressure induced force does not affect the simulation on a fixed cylinder very much. Vortex formation and progression induce forces on the bodies enveloped in the flow. A vortex creates a negative pressure suction area adjacent to the surface where it progresses. Thus the study of pressure distribution is important in the analysis of the aerodynamic forces around a structure. The pressure distribution near to the surface of the cylinder, flow momentum is quite low due to viscous effects and thus is sensitive to the changes of the pressure gradient. Figure 5 shows a typical pressure distribution plot for the flow around circular and square cylinder for Re = 100. In the case of flow past a bluff body with sharp corners i.e., square-cylinder the separation points are fixed. This is in contrast to the flow past an isolated circular cylinder, which does not have fixed points of separation. The location of these points of separation depends on the details of the attached boundary layer and the detaching shear layers. Fortunately, the issue of impingement is less severe than in the case of a square-cylinder, due to the relatively streamlined shape of the circular cylinder. In case of circular cylinder it can be said that pressure in the upstream cylinder face has got more pressure than the front end face of square cylinder but in downstream of square cylinder has less pressure compared to circular cylinder.

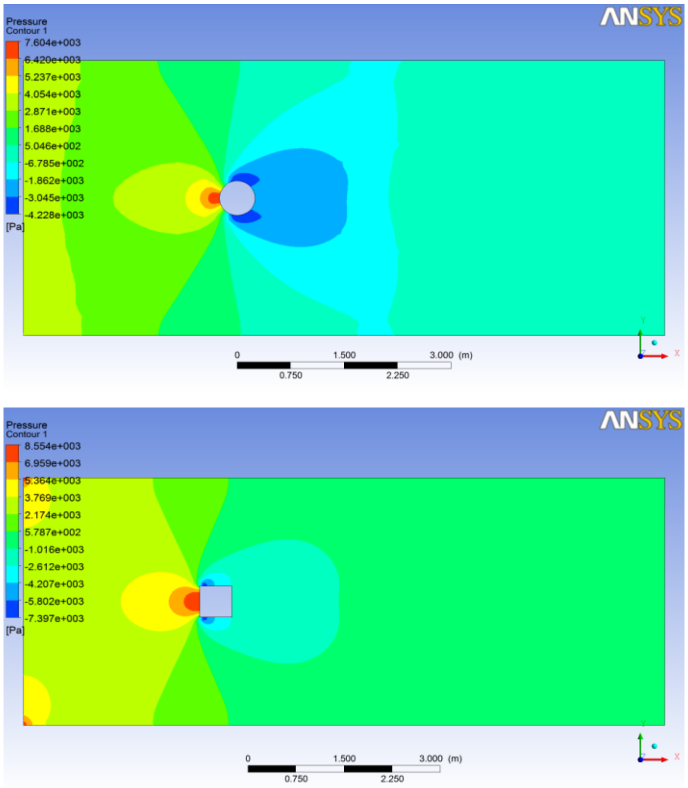

Fig 5: Pressure distribution around a circular and square cylinder for Re=100.

3.3 Counters of velocity magnitude

Velocity is often used interchangeably with the scalar quantity of speed, but the two terms have distinct differences. Speed measures the distance traveled per unit of time and ignores the direction traveled. Velocity, however, is a vector quantity that considers change in position over time (magnitude) and offers a direction of movement. The counters of velocity magnitude is shown in Figure 6.

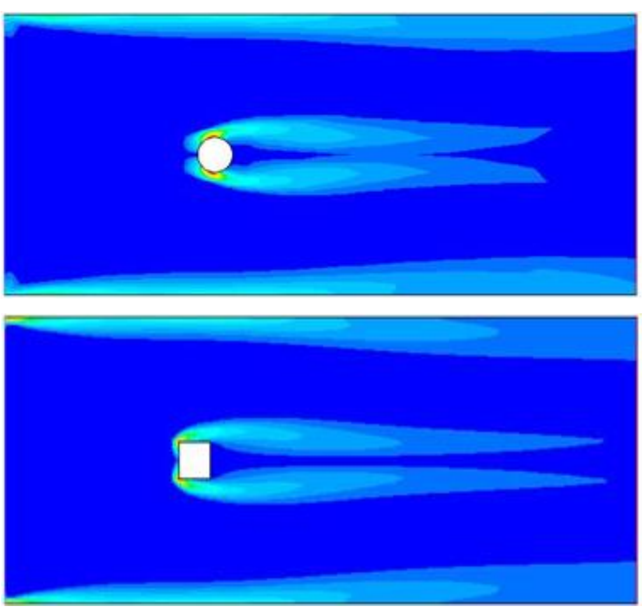

Fig 6: Counters of velocity magnitude a circular and square cylinder for Re=100.

4. Conclusion

The results of the numerical analysis around circular and square cylinder lead to the following conclusions:

- In case of circular cylinder pressure in the upstream cylinder face has got more pressure than the front end face of square cylinder but in downstream of square cylinder has less pressure compared to circular cylinder.
- The drag coefficients of Square cylinder is high when compared with circular cylinder.
- The Strouhal number for square cylinder is less when compared to circular cylinder.

5. References

1. Harlow FH, Welch JE. Numerical calculation of Time-Depentent Viscous Incompressible Flow, Phys. Fluids. 1965; 8:2182.
2. Evans MW, Harlow FH. The particle-in-cell method for hydrodynamic calculations, Los Alamos National Laboratory Report LA-2139, 1957.
3. Tome Murilo F, Mckee Sean. Gensmac: A computational Marker and Cell Method for free surface flows in General Domains J. Comput. Physc. 1994; 110:171-186.
4. Chen, *et al.* Velocity boundary conditions for the simulation of free surface fluid flow. J Comput. Phys. 1995; 116:262-267.
5. Hu JC, Zhou Y, Dalton C. Effects of the corner radius on the near wake of a square prism, Exp Fluid. 2006; 40:106-118.
6. Muammer Ozgoren. Flow structure in the downstream of square and circular cylinders. Flow Measurement and Instrumentation. 2006; 17:225-235.
7. Breuer M. Large eddy simulation of the subcritical flow past a circular cylinder: numerical and modeling aspects. Int. J. Numer. Methods Fluids. 1998; 28:1281-1302.
8. Zdravkovich MM. Flow Around Circular Cylinders, Fundamentals, OUP, New York, 1997, vol 1.
9. Erik Stalberg, Arnim Br¨uger, Per L¨otstedt, Arne V. Johansson, and Dan S. Henningson. High order accurate solution of flow past a circular cylinder. J Sci. Comput. 2006; 27:431-441.
10. Dalton C, Zheng W. Numerical solutions of a viscous uniform approach flow past square and diamond cylinders. Journal of Fluids Structure. 2003; 18:455-565.
11. Tamura T, Miyagi T. The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes. J Wind Eng Indus Aerodyn. 1999; 83:135-145.
12. Sohankar A, Norberg C, Davidson L. Simulation of three-dimensional flow around a square cylinder at moderate Reynolds numbers, Physics of Fluids. 1999; 11:288-306.