Analysis by CFD for Flow Past Circular and Square Cylinder

Vikram C. K.

Ph.D. Student
Visveswaraya Technological University
Dept. of Mechanical Engineering
P.E.S.College of Engineering, Mandya, Karnatak state, India

Dr. Y. T. Krishne Gowda

Professor, Dept. of Mechanical Engineering
Maharaja Institute of Technology, Mysore, Karnataka state, India.

Dr. H. V. Ravindra

Professor
Dept. of Mechanical Engineering, P.E.S.College of Engineering, Mandya, Karnatak state, India

Abstract— Flow past circular and square body for is carried out for Reynolds number 100 and 200 numerically by using commercial CFD code fluent. 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 more 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. Strouhal number remains same even if magnitude of oscillations is increased while monitoring the velocity behind the cylinder. The results are presented in the form of streamlines, pressure distribution, monitored velocity, Lift and drag coefficient and Strouhal number.

Index Terms— Monitored velocity, Strouhal number, Square cylinder, vortex wavelength.

I. 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.

II. RESULTS AND DISCUSSIONS

A. Finite volume mesh and boundary conditions

The problem considered here is the flow past circular and square cylinder for Reynolds numbers 100 and 200. 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. The inflow, top and bottom boundaries have been located 6.5 cylinders with respect to the center of the cylinder. Similarly, in order to minimize the effects of the outflow boundary condition on the flow in the vicinity of the cylinder, the computational domain has been extended to 30 cylinders in the downstream of the cylinder.
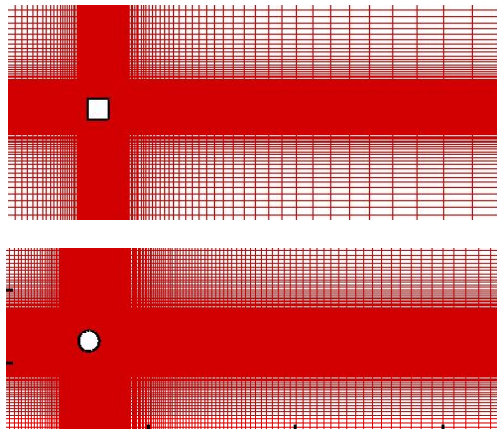

Fig . Finite volume mesh of square and circular cylinder.

In the present investigation, flow past circular and square cylinder has been computed by applying boundary conditions as follows.
(a) Inlet - Uniform flow (U = 1.0, V = 0.0)
(b) Cylinder surface -No slip (U = 0.0, V = 0.0)
(c) Top and Bottom Boundaries –symmetry boundary condition.
(d) Outlet Boundary -Continuative boundary condition can be expressed as (P = 0.0)

B. 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. As the flow forms a clockwise eddy, it rushes past the top of the cylinder somewhat faster than the flow across the bottom. When the clockwise eddy breaks away, the opposite pattern develops at the bottom. The eddies grow in size as they move away from the cylinder upto a certain length from the cylinder and then gradually die out and the flow becomes uniform as in the upstream. The tangential velocity of the Square cylinder is large and enlarges the separation area of square cylinder side face. When Reynolds number increased from 100 to 200 a similar flow pattern has been observed except the length of is vortex formation. For a square cylinder, the separation points are fixed, either at the leading edge or the trailing edge, depending on the Reynolds number. The square cylinder is a bluffer body as compared to the circular cylinder. Therefore, the vortex formation region is significantly broader and longer. This is presented in the form of
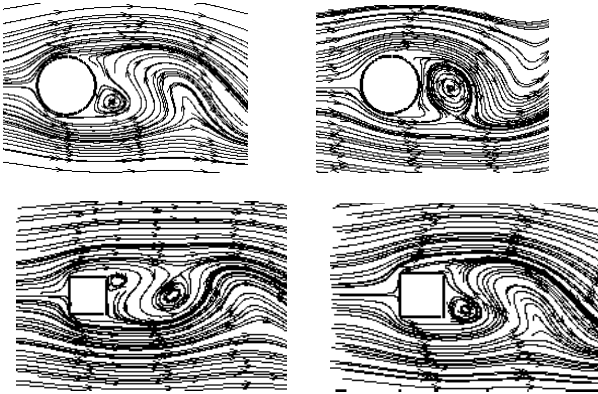
streamlines as shown in Fig.

Fig. Computational results of the instantaneous streamline plots of the circular cylinder and square cylinder for Re = 100 and 200.

C. Monitored velocity of circular and square cylinder

The temporal histories for the cross-stream component of velocity (v), along the axis of symmetry in the downstream region at two different points for Re=100 and Re=200 in circular cylinder and square cylinder by seeing the graph it can be said that circular cylinder is of higher magnitude compared with square cylinder. A typical plot of the monitored velocity is shown in Fig.
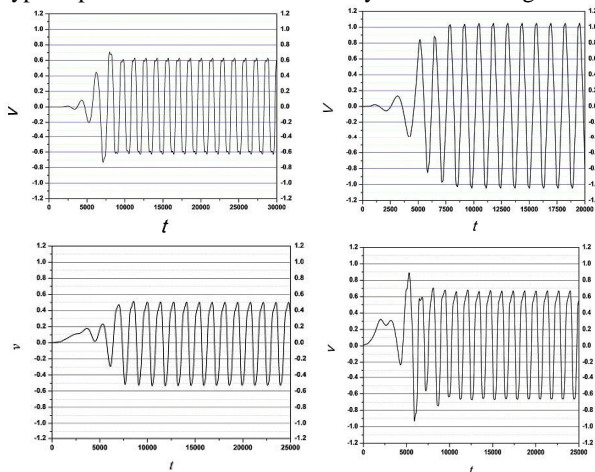

Fig. Monitored Velocity in the downstream of circular and square cylinder for Re=100 and Re=200

D. 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 4 shows a typical pressure distribution plot for the flow around circular and square cylinder for Re = 100 and 200. 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.
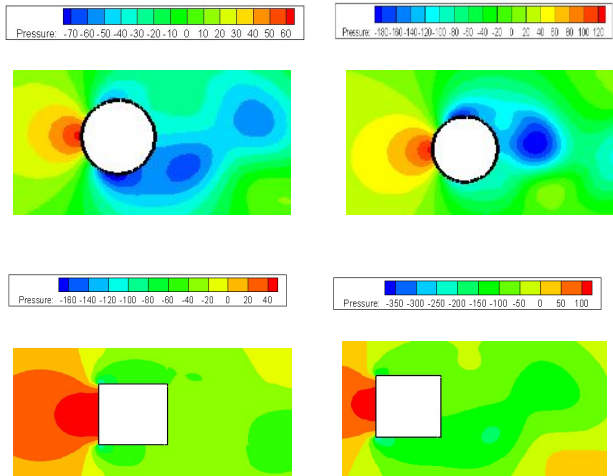

Fig. shows a typical pressure distribution plots of circular and square cylinder for Re = 100 and 200.

F. Lift and Drag coefficient

It can be seen from the graph that the Lift and drag coefficient gradually increases upto a certain time and becomes steady periodic for both circular and square cylinder. Lift coefficient is more in circular cylinder when compared with square cylinder. For a square or circular cross section, the Strouhal number increases with increase in the Reynolds number to a certain value beyond which further increase in Reynolds number results in decrease of Strouhal number reaching an asymptotic value. The easiest way, to compute the frequency f is to use the time between peaks of the y- velocity. The Strouhal number for square cylinder is less (0.14) when compared circular cylinder (0.19).
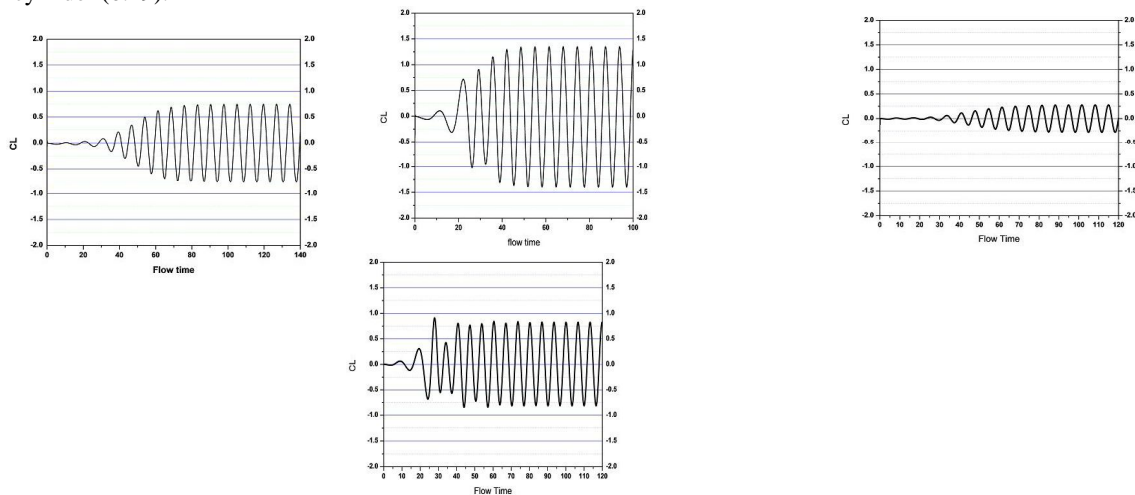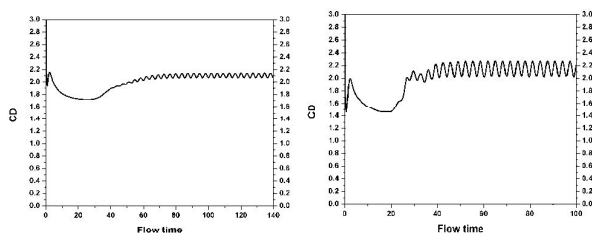

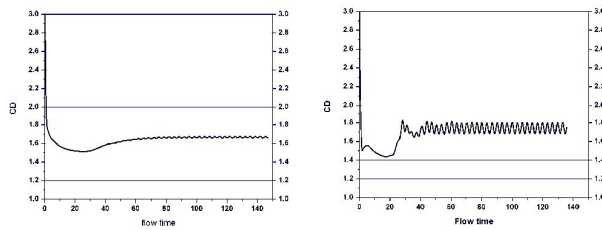Fig. Time History of Lift coefficient for circular and square cylinder for Re=100 and 200.

Fig. Time History of Drag coefficient for circular and square cylinder for Re=100 and 200.

III. 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 lift and drag coefficients of Square cylinder is less when compared with circular cylinder.

The increase in strouhal number, Results in decrease of vortex wavelength.

Strouhal number remains same even if magnitude of oscillations is increased while monitoring the velocity behind the cylinder.

The Strouhal number for square cylinder is less when compared to circular cylinder.

ACKNOWLEDGMENT

REFERENCES

[1] J. C. Hu, Y. Zhou and C. Dalton, "Effects of the corner radius on the near wake of a square prism," Exp Fluid, Vol. 40, pp. 106-118, 2006.

[2] Muammer Ozgoren, "Flow structure in the downstream of square and   circular cylinders," Flow Measurement and Instrumentation, Vol. 17, pp. 225–235, 2006.

[3] Breuer, M. Large eddy simulation of the subcritical flow past a circular   cylinder: numerical and modeling aspects. Int. J. Numer. Methods Fluids, Vol. 28, 1281–1302, 1998.

[4] M.M. Zdravkovich, Flow Around Circular Cylinders, Fundamentals, vol. 1, OUP, New York, 1997.

[5] 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.,Vol. 27,pp431–441, 2006.

[6] Dalton C, Zheng W, "Numerical solutions of a viscous uniform approach flow past square and diamond cylinders",. Journal of Fluids Structure, Vol. 18 pp. 455–565, 2003.

[7] Tamura T, Miyagi T, "The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes", J Wind Eng Indus Aerodyn, Vol. 83, pp.135–145, 1999.

[8] Zheng W, Dalton C, "Numerical prediction of force on rectangular cylinders in oscillating viscous flow". Journal of Fluids Struct. Vol.13, pp.225–249, 1999.

[9] Tamura T, Miyagi T, Kitagishi T, "Numerical prediction of unsteady pressures on a square cylinder with various corner shapes", J Wind Eng Indus Aerodyn. Vol. 74, pp.531–542, 1998.