International Journal of Scientific & Engineering Research, Volume 5, Issue 10, October-2014 479
ISSN 2229-5518

Numerical investigation on flow past square cylinders with different corner shapes

Vikram C. K, Y. T. Krishne Gowda, H. V. Ravindra

Abstract— Flow past square cylinders with and without corner modification for Reynolds number 100 and 200 is carried out numerically in a tandem arrangement has been simulated by using commercial CFD code fluent. The flow is assumed to be two dimensional unsteady and incompressible. Results indicate, in case of chamfered and rounded corners in square cylinder, there is decrease in the wake width and thereby the lift and Drag coefficient values. The large velocity behind the cylinder decreases the lift and Drag coefficient. The lift coefficients of Square cylinder with corner modification decreases but strouhal number increases when compared with a square cylinder without corner modification. Strouhal number remains same even if magnitude of oscillations is increased while monitoring the velocity behind the cylinder. For the square cylinders of same perimeters with and without corner modification, the size of the eddy and the monitored velocity in between the square cylinders increases with increase in PPR. Frequency of vortex shedding decreases with the introduction of second cylinder either in the upstream or downstream of the first cylinder. The lift coefficient of square cylinder with corner modification decreases but Strouhal number increases when compared with a square cylinder without corner modification. The results are presented in the form of Streamlines, Monitored velocity, Pressure distribution. Drag coefficient, Lift coefficient and Strouhal number.

Index Terms— Drag coefficient, Lift coefficient, Pitch to perimeter ratio,Square cylinder, Strouhal number, Tandem, Vortex shedding, Wake.

1 Introduction

Flow passing square cylinder with and without corner modification has attracted a great deal of attention in the literature because of its practical significance in engineering e.g., Tall buildings, monuments, and towers 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. The highly asymmetric and periodic nature of flow in the downstream has attracted the attention of physicists and engineers alike. From the physics point of view, it is highly fascinating to understand how and why such a highly symmetric flow in the upstream becomes so highly asymmetric in the downstream. Vortex shedding observed in the wake of these bodies generates unsteady lift and drag forces also velocity fluctuations in the wake region. An initially smooth and steady flow across a square 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 square cylinder, the phenomenon of resonance is the obvious outcome. A lot of research has been carried out on flow past a square cylinder; however, there has been no complete investigation is carried in case, if corner of the square cylinder is chamfered or rounded then what will be the effect on the flow characteristics around such cylinders. Therefore, the simulation of unsteady flow past square cylinders with and without corner cutoffs has practical relevance. Hiroshi Hasebe et al. [2009] brought out the conclusions for flow field between square cylinders in tandem arrangement that surface pressure distributions vary considerably between the spacing and vortex shedding from the upstream cylinder has a great influence on the property of the turbulent flow structure between two square cylinders. The effects of the Reynolds number, spacing ratio and rotation angle of the downstream cylinder on flow characteristic modes, drag coefficients and vortex shedding properties were studied by Yen et al.[2008] for the case of two identical square cylinders were installed in tandem in a vertical water tank. Results show that the Strouhal number decreases as the Reynolds number increases in the viscosity- dominant flow field. But in the inertia- dominant flow field, the Strouhal number increases with the Reynolds numbers and approaches a constant for high Reynolds numbers. Lakshmana Gowda et al. [2009] studied the near wake flow field features of transversely oscillating square section cylinders with different corner radii. Results indicate that increasing the corner radius suppresses the possible instabilities of the cylinder. Bandyopadhyay [2004] has simulated flow past a square cylinder placed inside a channel with two different blockage ratios and for different Reynolds numbers. The average drag coefficient found to increase with an increase in blockage ratio for a given Reynolds number. Dalton and Zheng [2003] presented numerical results for a uniform approach flow past square and diamond cylinders, with and-without corner modifications at Re=250 and 1,000.They noted that rounding corners of the bluff bodies produced a noticeable decrease in the calculated drag and lift coefficients. The vortex shedding frequency is also found to increase with increase in blockage ratio. Tamura and Miyagi and [1999] Tamura et al. investigated numerically and experimentally the aerodynamic forces on square cylinders and observed a decrease in the wake width as well as Cd with the corner chamfered or rounded. Similar studies emphasizing corner effects were also conducted by Delany and Sorense [1953], Naudascher et

- *Vikram C.K is currently pursuing Ph.D in Visveswaraya Technological University, Dept. of Mechanical Engineering, P.E.S.College of Engineering, Mandya, Karnatak state, India,. E-mail:* ***Vikram_vilas@yahoo.co.in***
- *Dr. Y.T. Krishne gowda, Professor, Dept. of Mechanical Engineering, Maharaja Institute of Technology, Mysore, Karnataka state, India. E-mail: ytk_gowda@yahoo.com*
- *Dr. H.V. Ravindra, Professor, Dept. of Mechanical Engineering, P.E.S.College of Engineering, Mandya Karnatak state, India,E-mail: mailhovera@yahoo.com.*

ISSN 2229-5518

al.[1981], Kwok et al. [1988] and Okamoto and Uemura [1991].These investigations largely focused on the effect of corner radii On the aerodynamic or hydrodynamic characteristics, such as drag/lift forces and shedding frequency, of bluff bodies, how the corner variation may alter the near wake, however, yet to be sufficiently documented, particularly in the base region. Therefore, objective of the present work is to characterize quantitatively the corner effects on the near- wake flow structure.number.

2 Results and Discussions

2.1 Geometry and boundary conditions

FLUENT is one of the commercial packages available to solve the Navier-Stokes equations. It uses a standard finite volume method for the analysis of fluid flow and heat transfer problems. Its enhanced features such as complete mesh flexibility, solver capabilities, additional models to simulate the accompanying effects of turbulence, acoustics, heat transfer, species transport, reactions and combustion, dynamic mesh modeling, makes it an ideal choice to perform CFD simulations with.

The problem considered here is the flow past single square cylinder and two square cylinders of Pitch to perimeter ratios of 2, 4 and 6 for Reynolds number 100 and 200 with and without corner modification. it is important to locate the inflow and top and bottom boundaries at sufficient distance from the main cylinder such that the boundary conditions applied to these boundaries should not have any undesirable effects into the main region of interest i.e., around and behind the cylinder. The inflow, top and bottom boundaries have been located 6.5 square 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 square cylinders in the downstream.

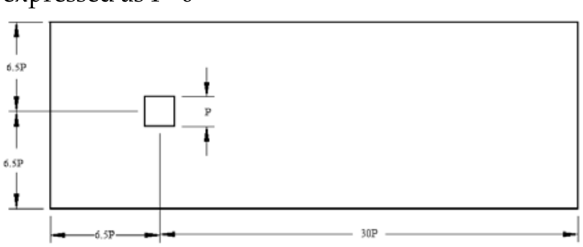
The following boundary conditions are used in the present investigation.

Inlet - Uniform flow (U = 1.0, V = 0.0)

Cylinder surface -No slip

Top and Bottom boundaries –symmetry.

Outlet boundary -continuative boundary condition can be expressed as P=0



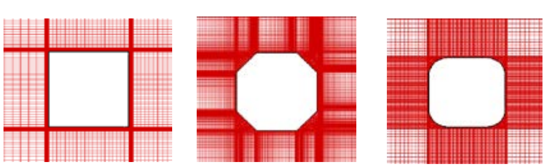


Fig.2.1 (a): Mesh of Enlarged view of Square cylinder and also corner modifications in square cylinder.

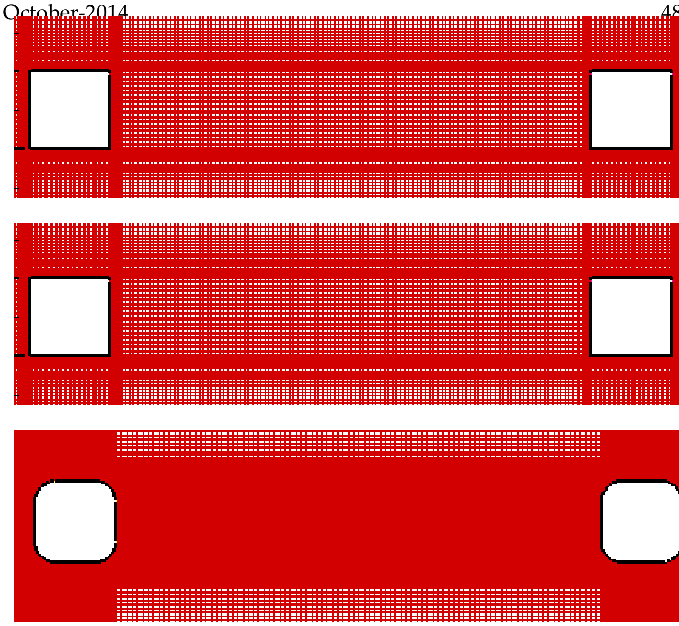


Fig.2.1 (b): Finite volume mesh of enlarged view of two square cylinders with and without corner modification when PPR=6.

2.2 Mesh sensitivity analysis

In the present work, a grid independence test is carried out using three different grid sizes 650x540,740x520, 890x765 for same computational domain size. The results obtained with 890x765 this grid gave a better result, hence decided to use for further computational work. Mesh density is thick in corner modified into chamfered and rounded in the square cylinders when compared with sharp square corners in order to resolve the gradients. The result in the form of Strouhal number is presented in the table 2.2 respectively. It can be seen readily that the results are independent of Mesh size. However 890x765 grid size has been selected to carry out the further computational work for square cylinder.

Grid size	Re=100	Re=200
650x540	0.15	0.16
740x520	0.15	0.16
890x765	0.15	0.16

2.3 Streamlines

In the case of flow over a square cylinder with and without corner modification for a square cylinder and two square cylinders when PPR is 2, 4 and 6 for Re=100, the flow is uniform and symmetrical in the upstream of the cylinder. The eddies are alternatively formed on either side of the square cylinder in the downstream for a square cylinder. As the flow forms a clockwise eddy, it rushes past the top of the square 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. In case of corner modification in square cylinder, a large eddy is formed behind the cylinder. The flow is uniform and symmetric in the upstream. The size of the eddy in between the cylinder is smaller when compared to the down-

ISSN 2229-5518

stream of the second cylinder for both with and without corner modification when PPR is 2. This is due to second cylinder is suppressing the eddy formation, also the formation of eddies in between the cylinders is less when compared to the downstream of the second cylinder because the distance between the two cylinders is very small. But in the case of PPR 4 and 6 the size of the eddy in between square cylinders and also corner modified in square cylinders is also get elongated. This is presented in the form of streamlines as shown in Figure. When Reynolds number increased from 100 to 200 a similar streamline pattern is observed except the length of its vortex formation.

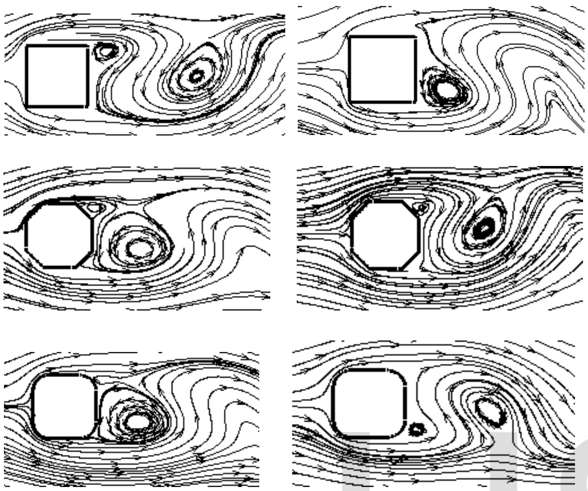


Fig.2.3 (a): Computational results in the form of streamlines around the square cylinder with and without corner modification i.e., chamfered and rounded for Re = 100 and 200.

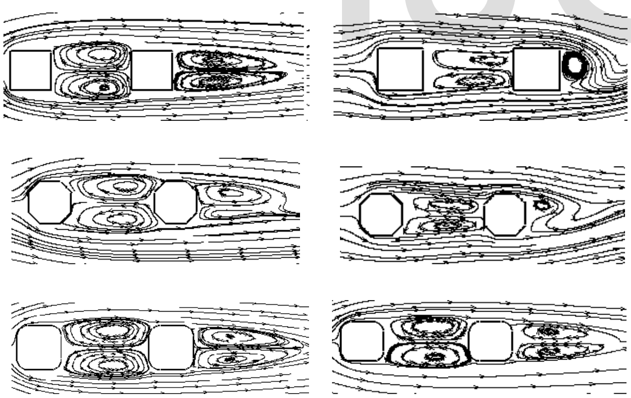
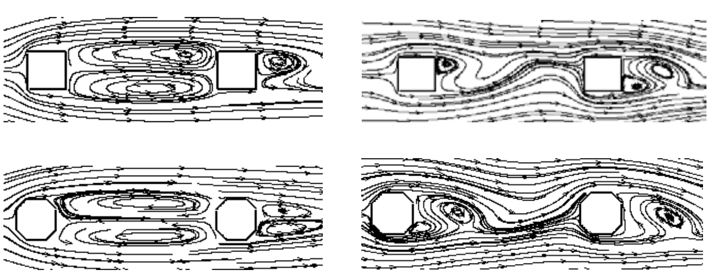


Fig.2.3 (b): Streamlines of square cylinder with and without corner modification when PPR=2 for Re=100 and 200.



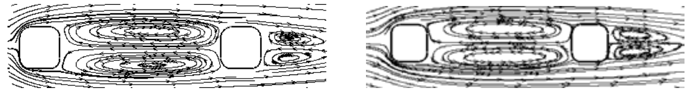


Fig.2.3 (c): Streamlines of square cylinder with and without corner modification when PPR=4 for Re=100 and 200.

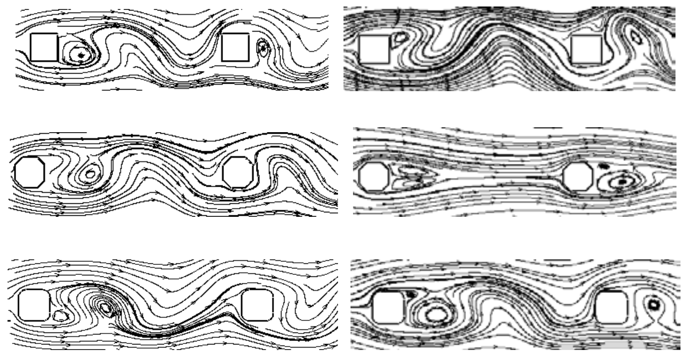


Fig.2.3 (d): Streamlines of square cylinder with and without corner modification when PPR=6 for Re=100 and 200.

2.4 Pressure distribution around a square cylinders

Pressure changes accordingly with the vortices motion in the vicinity of the bodies. Flow separates alternately around symmetrical bodies with and without 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 square 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. In case of square cylinder with and without corner modification it can be said that by seeing the numerical results, that pressure on the downstream side of the cylinder is smaller than that on the upstream side of the cylinder. It is clear that the pressure on the downstream side of the square cylinder with corner modification becomes greater than on the downstream side of the square cylinder without corner modification. Also for square cylinder without corner modification pressure is more at front end of upstream square cylinder.

In case of two cylinders, upstream cylinder experience maximum pressure than downstream cylinder. Without corner modification experiences highest pressure when compared with modification. Pressure on the downstream side of the cylinder is lower than that on the upstream side of the cylinder for all the three cases under investigation. Also front portion of the cylinder is experiencing maximum pressure compared to the second cylinder for all the three cases. It is due to fluid is brought to rest and hence the front cylinder experiences the absence of the kinetic energy and the domination of pressure energy.

ISSN 2229-5518

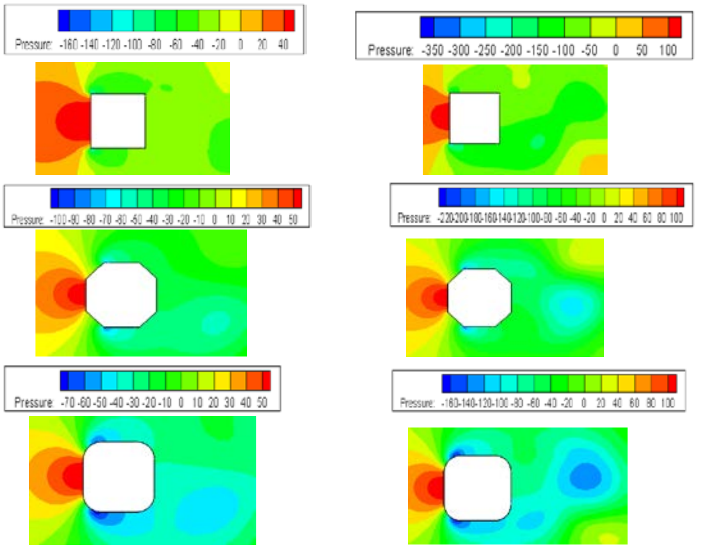


Fig.2.4 (a): Pressure distribution plot for the flow around a square cylinder with and without corner modification when Re = 100 and 200.

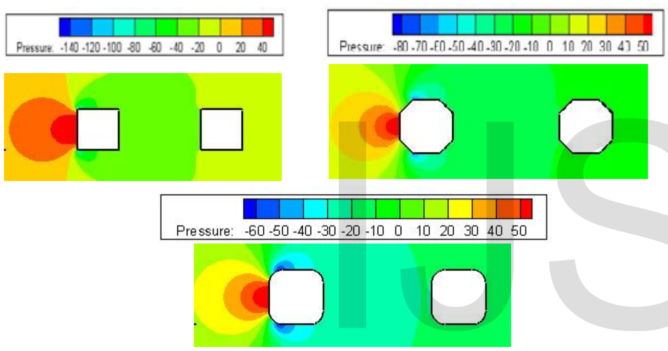


Fig.2.4 (b): Pressure distribution plot for PPR =2 with and without corner modification when Re=100.

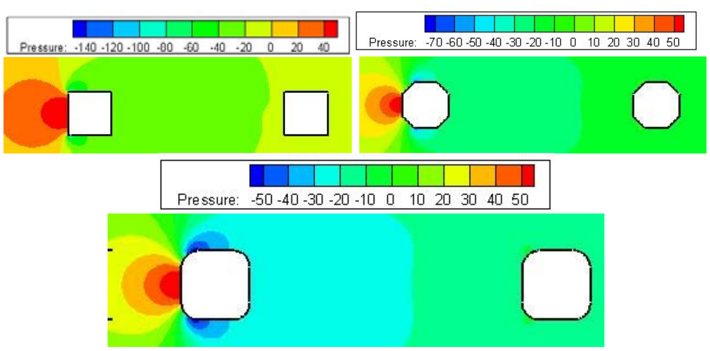
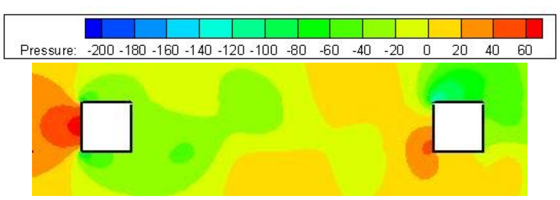


Fig.2.4 (c): Pressure distribution plot for PPR =4 when Re=100 with and without corner modification.



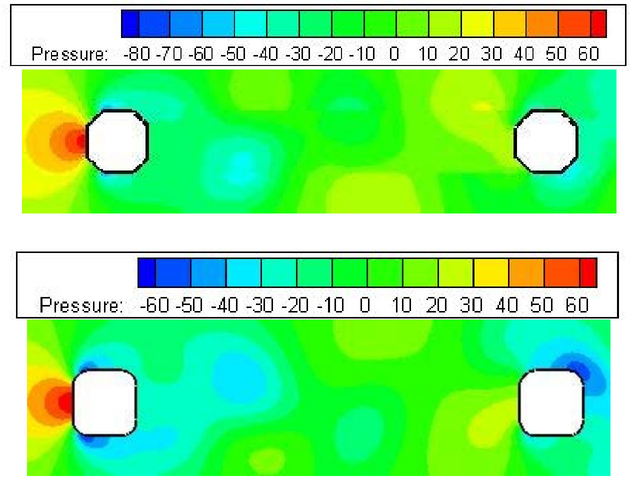


Fig.2.4 (d): Pressure distribution plot for PPR =6 when Re=100 with and without corner modification.

2.5 Lift and drag coefficient

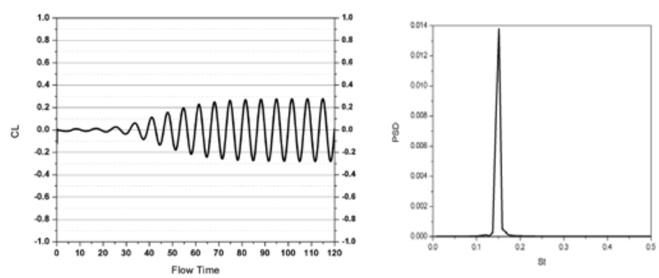
It can be seen from the graph that the Lift coefficient gradually increases upto a certain time and becomes steady periodic. Lift coefficient is more for Square cylinder with sharp corner when compared with chamfered and rounded corner. The strouhal number for square cylinder is less compared to chamfered and rounded corner cylinder which can be seen by Fourier Transform graph.

Strouhal (1878) proposed a dimensionless parameter, which Bénard (1926) called as 'Strouhal number' (St). The Strouhal number is a measure of the frequency of vortex shedding and is defined as

$St=fd/u$

Where f is frequency of vortex shedding from alternate sides of bluff body its value was extracted from the lift coefficient-time behavior plot, and u is the mean value of inlet velocity 1m/s and d is the diameter or perimeter of the cylinder 1m. To measure this quantity the values of the y-velocity at a point downstream of the cylinder at each time iteration were used. The easiest way, to compute the frequency f is to use the time between peaks of the y- velocity. An alternative way is to use a Fast Fourier Transform (FFT) of the y-velocity data over a given number of iterations, which easily can be done in Matlab.

The Strouhal number for square cylinder is less (0.15) when compared with chamfered (0.17) and rounded at the corner of square cylinder (0.185). Energy corresponding to the frequency is more for square cylinder then corner modification in square cylinders.



ISSN 2229-5518

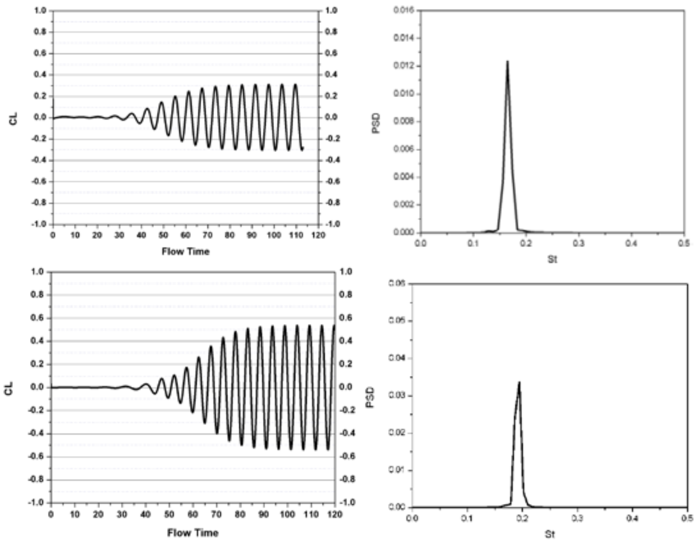


Fig.2.5 (a): Time History of Lift coefficient for square cylinder and chamfered and Rounded at the corner of square cylinder and also Strouhal number obtained by Fast Fourier transform for Re=100.

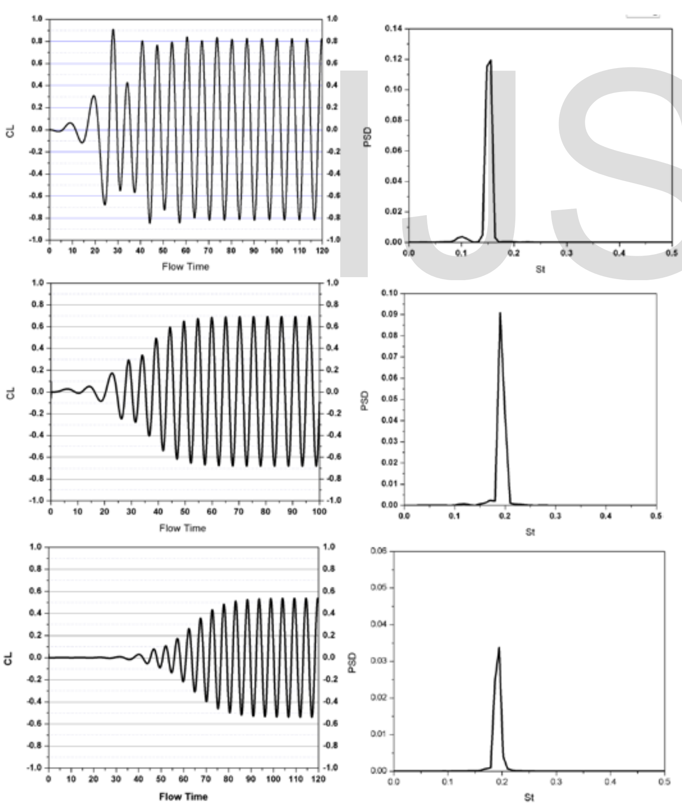


Fig.2.5 (b): Time History of Lift coefficient for square cylinder and chamfered and Rounded at the corner of square cylinder and also Strouhal number obtained by Fast Fourier transform for Re=200.

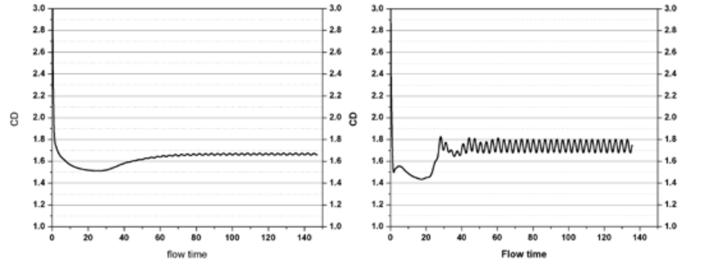


Fig.2.5 (c): Time history of Drag coefficient plot for square cylinder for Re=100 and 200.

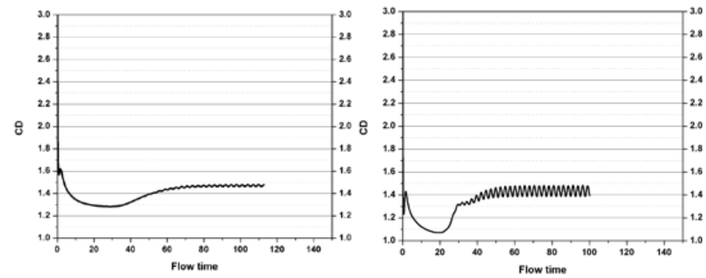


Fig.2.5 (d): Time history of Drag coefficient plot for chamfered at the corner of square cylinder when Re=100 and 200.

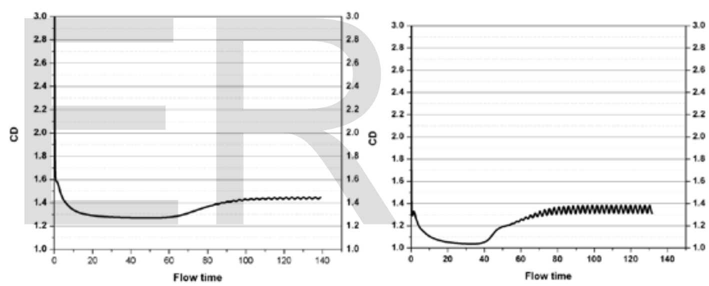


Fig.2.5 (e): Time history of Drag coefficient plot for rounded at the corner of square cylinder when Re=100 and 200.

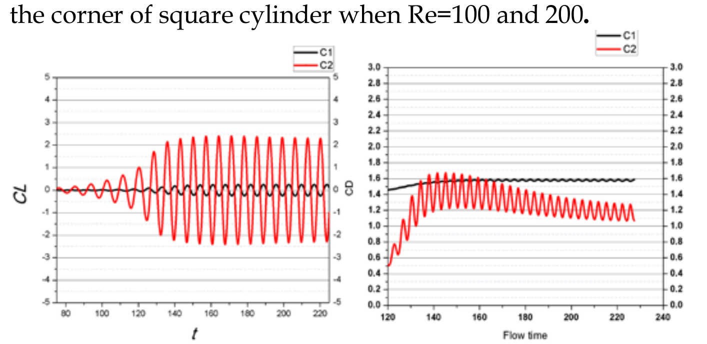


Fig.2.5 (f): Time history of lift coefficient and drag coefficient for upstream and downstream square cylinder for Re=100 when PPR=6.

ISSN 2229-5518

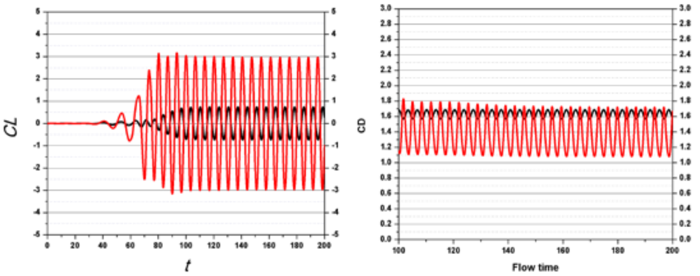


Fig.2.5 (g): Time history of lift coefficient and drag coefficient for upstream and downstream square cylinder for Re=200 when PPR=6.

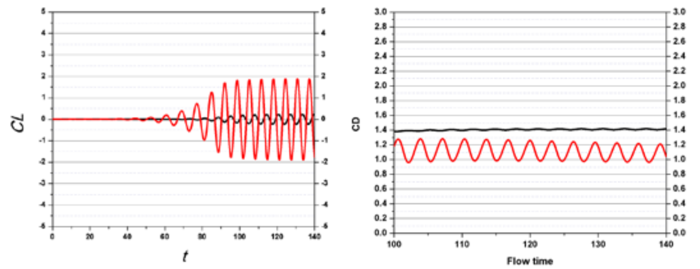


Fig.2.5 (h): Time history of lift coefficient for upstream and downstream of chamfered at the corner of square cylinder for Re=100 when PPR=6.

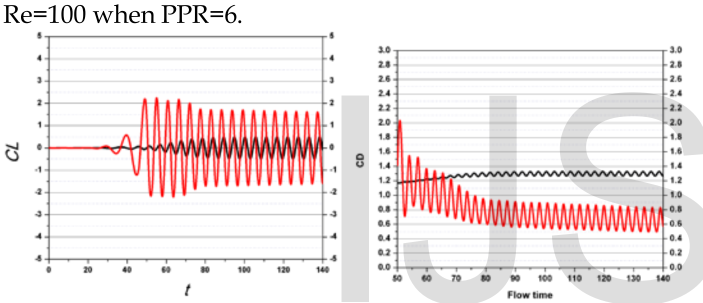


Fig.2.5 (i): Time history of lift coefficient for upstream and downstream of chamfered at the corner of square cylinder for Re=200 when PPR=6.

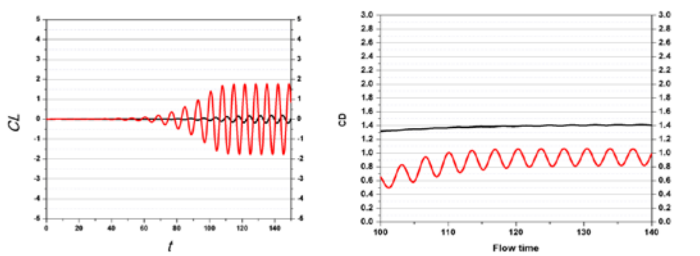


Fig.2.5 (j): Time history of lift coefficient for upstream and downstream of rounded at the corner of square cylinder for Re=100 when PPR=6.

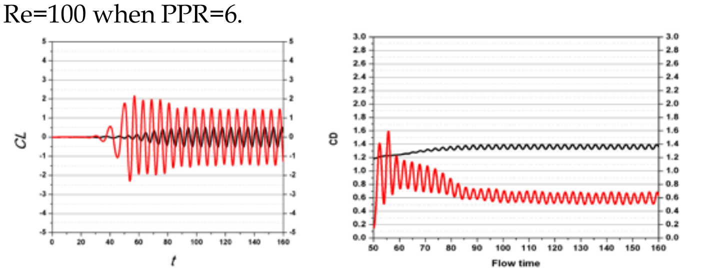


Fig.2.5 (k): Time history of lift coefficient for upstream and downstream cylinder for Rounded square cylinder for Re=200 when PPR=6.

2.6 Monitored velocity

The temporal histories for the cross-stream component of velocity (v), along the axis of symmetry (in the downstream region), at different nodal point for Re=100 and Re=200. It can be seen from the plots that Strouhal number remains same even if magnitude of oscillations is increased. A typical plot of the monitored velocity is shown in Fig. 2.6(a)–(i). In order to predict the frequency of vortex shedding velocities in the direction perpendicular to the flow, have been monitored for PPR 6. Velocities have been monitored in between the cylinders at different nodal point measured from the inlet section, which is the centre line of the geometry under investigation, also in the downstream. Monitored velocity in between the square cylinders and chamfered and rounded at the corner of square cylinders are less compared to the downstream of the second square cylinder. Frequency of vortex shedding can be decreased with the introduction a cylinder of with and without in the downstream of the second cylinder.

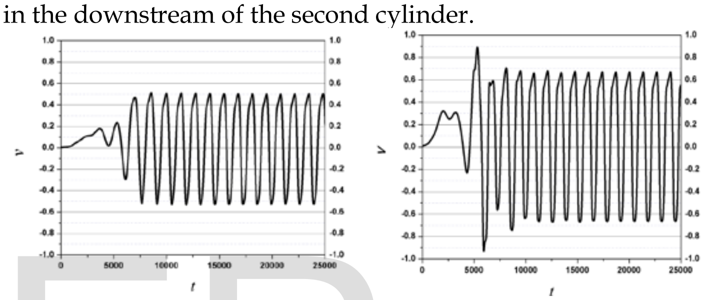


Fig.2.6 (a): Monitored Velocity in the downstream of square cylinder for Re=100 and Re=200

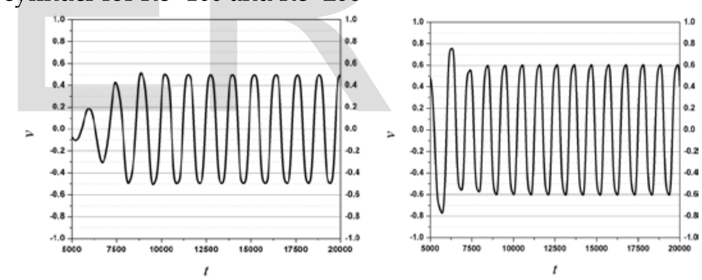


Fig. 2.6(b): Monitored Velocity in the downstream of square cylinder with corner chamfered for Re=100 and Re=200.

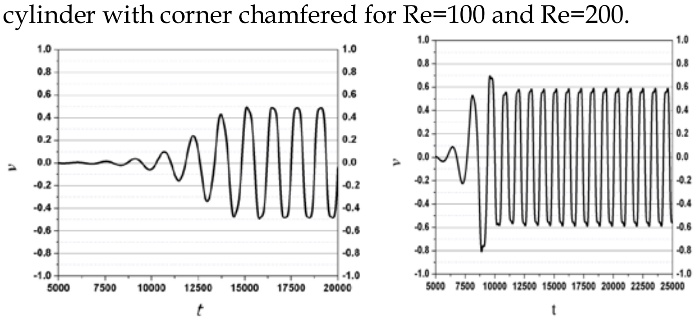


Fig. 2.6(c): Monitored Velocity in the downstream of square cylinder with corner rounded for Re=100 and Re=200.

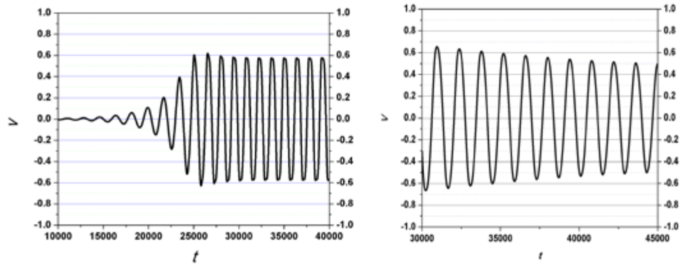


Fig. 2.6(d) Monitored velocity in between and downstream of square cylinders when PPR=6 for Re=100.

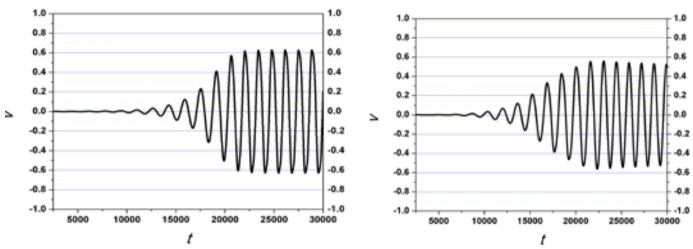


Fig. 2.6(e) Monitored velocity in between and downstream of chamfered at the corner of square cylinders when PPR=6 for Re=100.

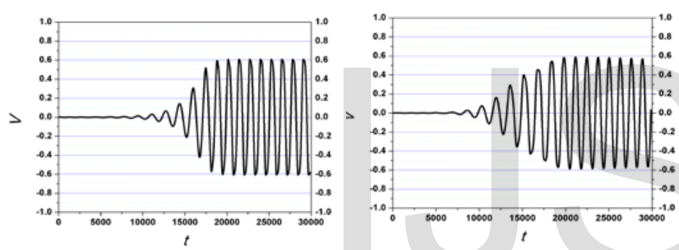


Fig. 2.6(f) Monitored velocity in between and downstream of rounded at the corner of square cylinders when PPR=6 for Re=100.

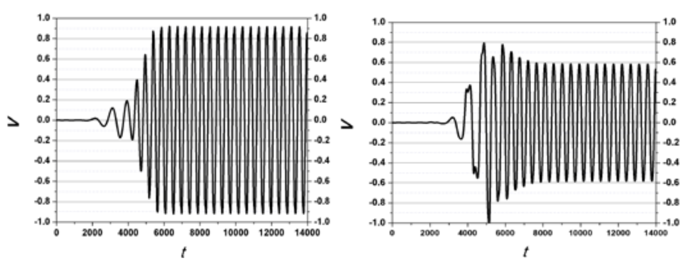


Fig. 2.6(g) Monitored velocity in between and downstream of square cylinders when PPR=6 for Re=100.

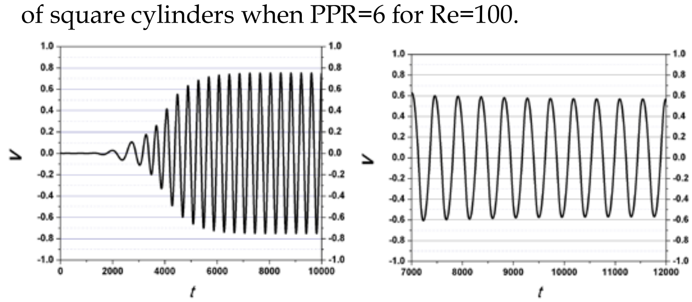


Fig. 2.6(h) Monitored velocity in between and downstream of chamfered at the corner of square cylinders when PPR=6 for Re=100.

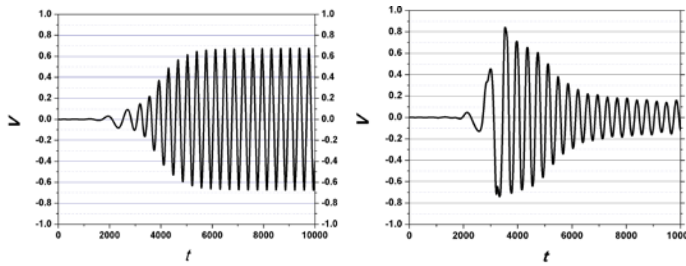


Fig. 2.6(i) Monitored velocity in between and downstream of rounded at the corner of square cylinders when PPR=6 for Re=100.

3 Conclusion

The results of the numerical analysis around square cylinders with and without corner modification lead to the following conclusions:

The tangential velocity of the Square cylinder is large when compared with corner Rounded and chamfered, and enlarges the separation area of square cylinder side face. Therefore, the width of the wake behind the Square cylinder with corner modification becomes small and thereby the lift and drag coefficient decreases.

The velocity behind the cylinder with corner modification becomes large. The large velocity behind the cylinder decreases the lift coefficient and drag coefficient.

The lift coefficients of Square cylinder with corner modification decreases but strouhal number increases when compared with a square cylinder without corner modification. Energy corresponding to the frequency is more for square cylinder then corner modification cylinders.

The increase in strouhal number decrease the vortex wavelength.Strouhal number remains same even if magnitude of oscillations is increased while monitoring the velocity behind the cylinder.

Monitored velocity in between the square cylinder, chamfered and rounded at the corner of the square cylinders is less compared to the downstream of the second cylinder. Frequency of vortex shedding is decreased with the introduction ofa cylinder in the downstream of the second cylinder.

For the square cylinders of same perimeters, the size of the eddy and the monitored velocity in between the square cylinders increases with increase in PPR.

The downstream cylinder is found to experience higher lift compared to the upstream cylinder.

The downstream cylinder is found to experience lower drag compared to the upstream cylinder.

Pressure on the downstream side of the cylinder is smaller than that on the upstream side of the cylinder for all the three cases. It is clear that the pressure on the downstream side of the square cylinder with corner modification becomes greater than on the downstream side of the square cylinder without corner modification. Also front portion of the cylinder is experiencing highest pressure compared to the second cylinder for all the three cases.

Acknowledgment

The authors wish to thank P.E.S college of Engineering Mandya, Karnataka, India for providing all the facility for our Research work in Mecanical Department.Manu, Research scholar in IISC, Banglore for his valuable suggestions.Manjesh, Vardaraju and Vilas C.K for supporting our work.

References

[1] Bearman PW, Graham JMR, Obasaju ED, Drossopoulos GM, "The influence of corner radius on the forces experienced by cylindrical bluff bodies in oscillatory flow",. Application Ocean Resarch, vol.6, pp. 83–89, 1984.

[2] 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.

[3] 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.

[4] 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.

[5] 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.

[6] Okamoto S, Uemura N, "Effect of rounding side-corners on aerodynamic forces and turbulent wake of a cube placed on a ground plane,"Exp Fluid. Vol.11, pp.58–64, 1991.

[7] Delany NK, Sorensen NE, "Low-speed drag of cylinder of various shapes," NACA Technical Notes 3038, 1953.

[8] Kwok KCS, Wilhelm PA, Wilkie BG, "Effect of edge configuration on wind-induced response of a tall building," Eng. Struct. Vol.10, pp.135–140, 1988.

[9] Andrea Mola, Giancarlo Bordonaro, and Muhammad Hajj, "Low-frequency variations of force coefficients on square cylinders with sharp and rounded corners," VI International Colloquium on Bluff Bodies Aerodynamics & Applications Milano, Italy, July, pp.20–24, 2008.

[10] Yoichi Yamagishi1, Shigeo Kimura, Makoto Oki et.al. "Effect of corner cutoffs on flow characteristics around a square cylinder,"Flucome-10th international conference on fluid control, Measurements, and visualization Aug. pp.17-21, Moscow, Russia, 2009.

[11] R. Ajith Kumar, Chang Hyun Sohn, and B. H. Lakshmana Gowda "Influence of corner radius on the near wake structure of a transversely oscillating square cylinder," Journal of Mechanical Science and Technology, Vol. 23. Pp.2390-2416, 2009.

[12] H. Kawai, "Effect of corner modifications on aeroelastic instabilities of tall buildings," J.Wind Eng.Ind.Aerodyn.Vol. 74-76 pp.719-729, 1998.

[13] 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.