INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

ISSN (ONLINE): 2321-3051

Vol.2 Issue.12,

December 2014.

Pgs: 32-41

INFLUENCE OF CORNER GEOMETRY ON THE FLOW STRUCTURE AND FLOW CHARACTERISTICS FOR FLOW PAST A SQUARE CYLINDER AT Re=150

Nidhul K

PG Student, Government College Of Engineering Thrissur, nidhul07@gmail.com

Abstract

Flow past a square cylinder with and without corner modifications is carried out numerically by using CFD fluent. The wake is generated by a uniform flow of Reynolds number (Re) 150 based on the characteristic length of the cylinder, D. 2D unsteady numerical simulation is done using FVM employing pressure based solver and PISO scheme. A computational grid independence study has been done to obtain a grid resolution which predicts the results without any discrepancies. The flow separation point for the square cylinder with and without corner modifications is obtained. The pressure distribution in the near wake region and around the square cylinder surface is also investigated for different corner geometries. The results are presented in the form of coefficient of pressure Vs Domain length/D, Coefficient of pressure along the cylinder wall and wall shear stress along the cylinder wall. The results indicate that the flow separation in case of square cylinder without sharp corners is delayed. The adverse pressure gradient along the surface of the cylinder and in the near wake region is smaller for round corners. The tangential velocity of square cylinder with sharp corners is large when compared with modified corners.

Keywords: Square Cylinder, Corner modifications, Reynolds number, wake, grid independence, flow separation point, Pressure distribution, Lift coefficient, Drag coefficient.

1. Introduction

Flow past bluff bodies 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. Bluff bodies are structures with shapes that significantly disturb the flow around them, as opposed to flow around a streamlined body. A bluff body is one in which the length in the flow direction is close to or equal to the length perpendicular to the flow direction. Examples of bluff bodies include circular cylinders, square cylinders and rectangular cylinders.

Flow around a bluff body creates a large region of separated flow and a massive unsteady wake region in the downstream when compared to a streamlined body. Due to the adverse pressure gradient in the flow direction, flow separates and flow reversal occurs which creates rotational lump of fluid masses known as eddies. Eddies formed are detached periodically from either side of structure, known as vortex shedding, generates velocity fluctuations in the wake region. Alternate shedding of vortices also induces a periodic oscillatory force in the transverse direction, known as the lift force. When the natural frequency of structure is close to the shedding frequency of the vortices, the phenomenon of resonance occurs. A lot of research has been carried out on flow past single square cylinder and passive methods to control vortex shedding. However, there has been no complete investigation on the effect of corner modification on the flow characteristics around a square cylinder. Therefore, the simulation of unsteady flow past square cylinder with corner modifications has practical relevance.

Sohankar et al. [1] constructed a non-uniform mesh near the wall of a square cylinder and, at 5 diameters away from the cylinder surface, a uniform mesh was applied. The size of the smallest cell was located at the leading edge of the square cylinder with cell size of h/D = 0.004. Inoue et.al [2] constructed a non-uniform mesh but divided the computational domain into three regions, each with a different grid ratio. The smallest cell was located along the edges of the square cylinder with the value of h/D = 0.01. Mohamed Sukri Mat Ali et.al [3] numerically investigated the sensitivity of the computed flow field to flow parameters for a flow with Reynolds number 150.They constructed computational meshes based on reasonable estimates of cell size and grid stretching ratios. Doolan [4] investigated the grid convergence for three different grid resolutions on DNS around a square cylinder and found that the solution converged when the smallest cell size along the square cylinder edge was h/D = 0.0167. Bearman et al. [5] investigated the corner radius effect on the hydrodynamic forces on cylindrical bluff bodies subjected to an oscillatory flow. They found that the drag coefficient, Cd, was sensitive to the corner radius in a steady flow and even more so in an oscillatory flow. Kawai [6] also has investigated the effects of corner modifications (such as corner cut, recession and rounding) on the flow characteristics and flow pattern past square prisms. He concluded that corner rounding is the best option to provide aerodynamic stability to prisms. Tamura et al. [7] and Tamura and Miyagi [8] investigated numerically and experimentally the aerodynamic forces on square cylinders and observed a decrease in the wake width as well as C_d with the round corners. Dalton and Zheng [9] 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. B. H. Lakshmana Gowda et al. [10] 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.

Over the years the investigations largely focused on the effect of corner radii on the aerodynamic characteristics, such as drag/lift forces and vortex shedding frequency. However, how the corner variation may alter the pressure variation in the near wake and along the cylinder surface has yet to be sufficiently documented. Hence one objective of this work is to characterize quantitatively the corner effects on the near-wake flow characteristics and flow structure, complementing the data in the literature.

2. Numerical Simulation Procedure

2.1 Description of model

A cylinder of square cross-section with a dimension of 4cm width and 4cm length is the model taken. R/D ratio for rounded corner is 0.125. For inclined corner, angle of inclination is 45^0. Rectangular corner is made by removing square portion of length 0.005m from each corner as shown in Figure 1.

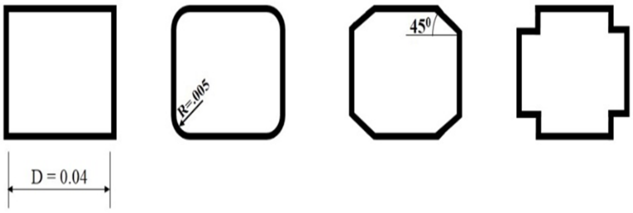


Figure 1: Cylinder with square cross-section and corner modifications

2.2 Flow field formulation

The governing equations on the flow field are the continuity and momentum equations (Navier–Stokes equations), which can be written as follows:

$$\frac{\partial \rho}{\partial t} + div(\rho V) = 0$$

$$\frac{\partial}{\partial t}(\rho u) + div(\rho V u) = -\frac{\partial p}{\partial x} + div(\mu\, grad u) + B_x$$

$$\frac{\partial}{\partial t}(\rho v) + div(\rho V v) = -\frac{\partial p}{\partial y} + div(\mu\, grad v) + B_y$$

Where ρ is the fluid density, μ is the fluid viscosity, V is the velocity vector of the flow field, p is the pressure, and u and v are the velocity components in the x- and y-directions, respectively.

B_x and B_y are also the body forces per unit volume, which are negligible in the present study. The fluid is assumed to be incompressible, and its properties has been taken as ρ=1.225 kg/m^3 and μ=1.7894×10^{-5} kg/m s.

A rectangular domain was used with a length of 38D and a width of 20D, where D=0.04 m is the cylinder diameter. The cylinder center has the coordinates x=10.5D and y=10.5D as shown in Figure 2.

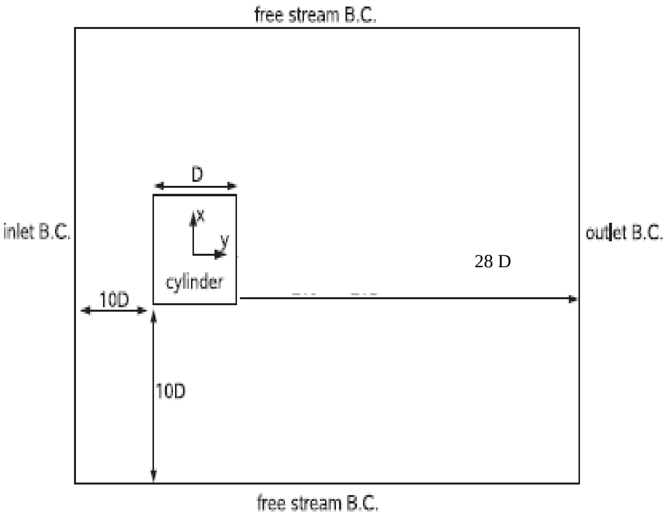


Figure 2: Computational flow domain for flow past a square cylinder

2.3 Discretization method

The governing equations have been discretized using the finite-volume method on a fixed Cartesian-staggered grid with non-uniform grid spacing. The grids in the region of the cylinder boundaries are sufficiently fine in order to achieve the reasonable accuracy. The temporal discretization has been done in conformity with the fully implicit scheme. For the spatial discretization, the hybrid scheme has been employed. PISO (Pressure-implicit with splitting of operators) procedure was applied to calculate the flow field variables.

3. Grid influence

Within the scope of predicting the flow field around a square cylinder using numerical analysis, many similar investigations have been made, but the results always show small discrepancies even though the overall global trends are similar. One of the reasons for these discrepancies is the difference in the construction of the mesh. So grid independence study has been done so that the grid chosen for simulation provides accurate results.

The mean axial velocity over the central plane of domain is shown in Figure 3. Six different 2D grids are used starting with 27200 cells upto 176094 cells.

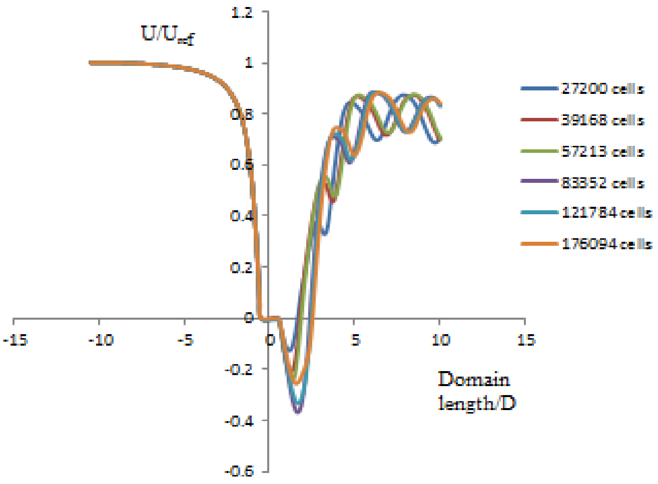


Figure 3: The mean axial velocity distribution on the center plane of the cylinder

The results of this case show a certain dependency on the grid resolution for the grids used. In Figure 3, it is observed that upstream all the runs predict identical results, whereas a faster recovery of mean axial velocity is predicted when the grid resolution is increased but the mean velocity of recirculation is approximately the same in all runs.

4. Results and Discussion

Preliminary simulations have been carried out in order to define adequate computational domains and to assure grid independence. Both uniform and non-uniform meshes have been tested, with different refinements. The values of C_D, C_L and Strouhal number for flow past a square cylinder obtained by simulation are compared with experimental and simulation work of other authors as shown in Table 1.The results obtained indicates that the numerical scheme, domain size, mesh resolution, and other parameters have been adequately chosen to capture the complex wake behaviour.

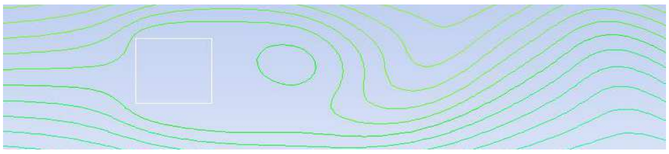
Then the results i.e. streamlines, vorticity magnitude, flow separation point, tangential velocity, pressure distribution in the near wake region and pressure distribution along the cylinder surface are demonstrated and analysed at Reynolds numbers of 150 for 4 cases i.e. for square cylinder with sharp corners, for square cylinder with round corners, for square cylinder with inclined corners and for square cylinder with rectangular corners.

Table 1: Comparison of results of case study with other authors works

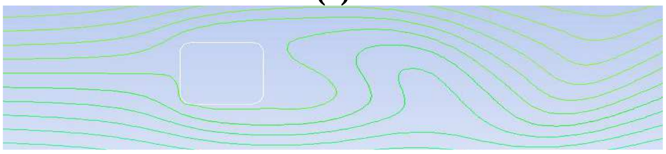
Study	St	C_D	C_L
Experiments(Okajima, 1982; Sohankar et al., 1999)	0.148-0.155	1.40	-
Sohankar et al. (1998)	0.165	1.44	0.230
Doolan (2009)	0.156	1.44	0.296
Inoue et al. (2006)	0.151	1.40	0.40
Ali et al. (2009)	0.160	1.47	0.285
Present study	0.149	1.60	0.44

4.1 Streamlines and Vorticity magnitude contours

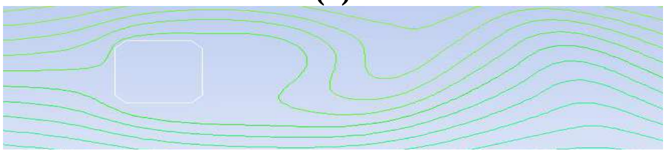
For flow over a square cylinder at Re=150, the flow is uniform and symmetrical in the upstream of the cylinder as shown in Figure 4(a). Due to adverse pressure gradient in the flow direction combined with low momentum near the wall surface, the flow streamline detach from the top corner of leading edge. Flow separation and flow reversal occurs. Hence rotating lump of fluid is created in the near wake region. As the flow forms a clockwise eddy, it rushes past the top of the square cylinder faster than the flow at the bottom.



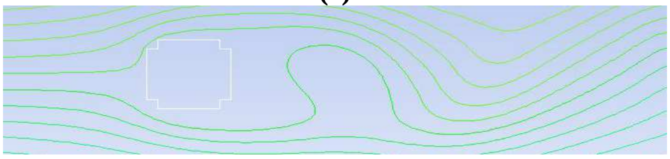
(a)



(b)



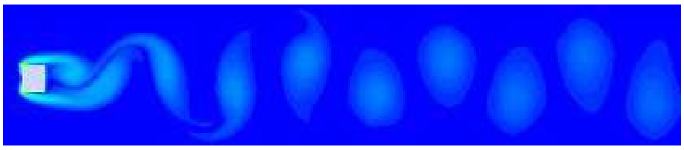
(c)



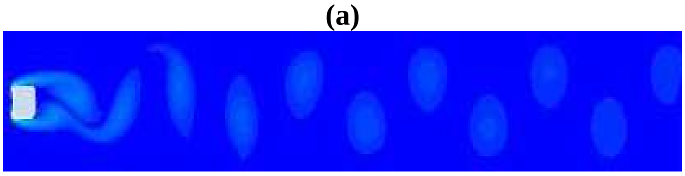
(d)

Figure 4: Streamlines flow around (a) square cylinder (b) square cylinder with round corners (c) Square cylinder with inclined corners and (d) Square cylinder with rectangular corners

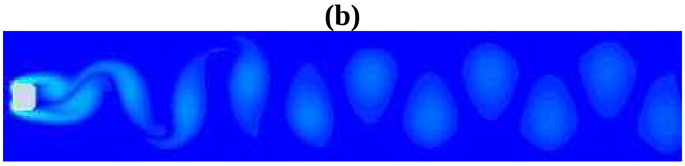
This eddy grows in size as they move away from the cylinder upto a certain length from the cylinder trailing edge dissipating energy to smaller eddies downstream. Vorticity magnitude contours for the different cases are shown in Figure 5.



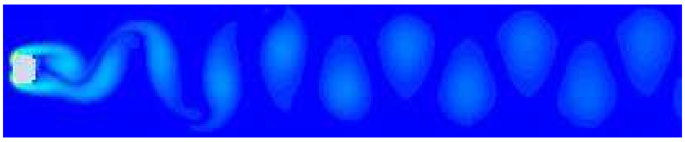
(a)



(b)



(c)



(d)

Figure 5: Vorticity magnitude contours for flow around (a) square cylinder (b) square cylinder with round corners (c) Square cylinder with inclined corners and (d) Square cylinder with rectangular corners

4.2 Tangential velocity and Wake width

Tangential velocity variation for flow over a square cylinder with and without corner modification is shown in Figure 6. It is observed that tangential velocity of square cylinder is large when compared to square cylinder with round, with inclined and with rectangular corners. Due to this separation area for square cylinder with sharp corners is enlarged. Therefore width of the wake behind the square cylinder with corner modification becomes large.

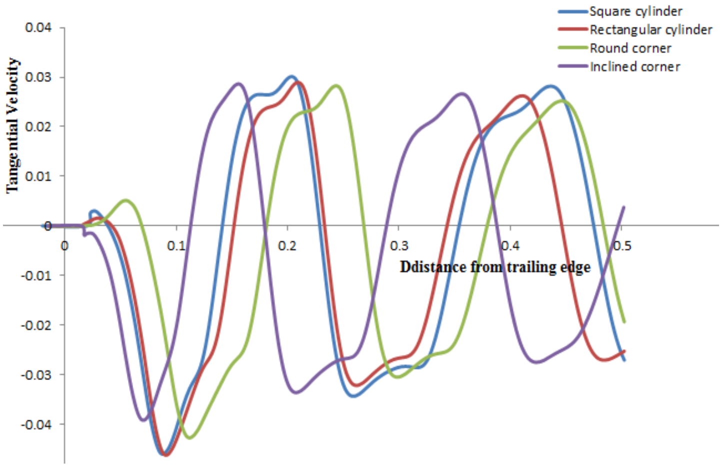
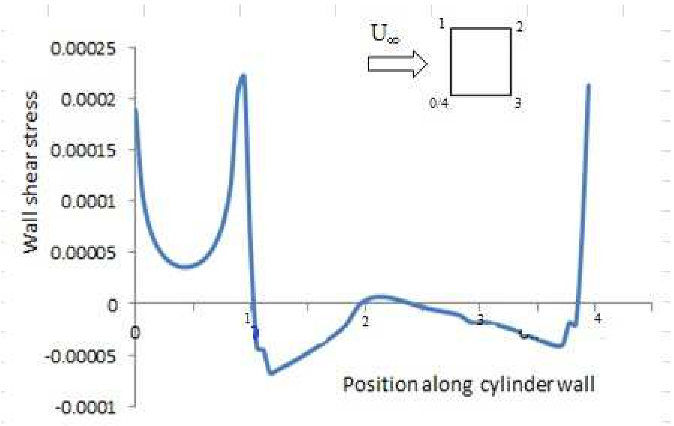


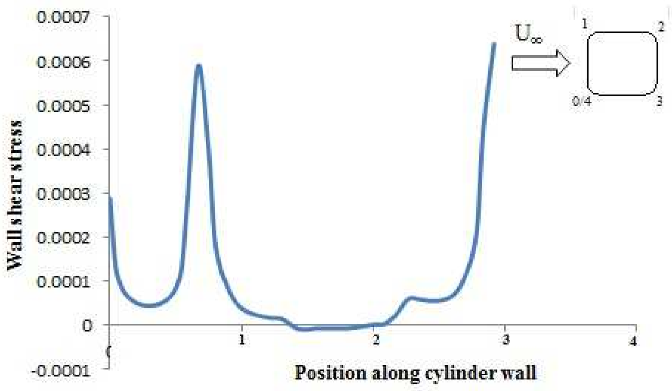
Figure 6: Tangential velocity variation in the near wake region

4.3 Flow Separation

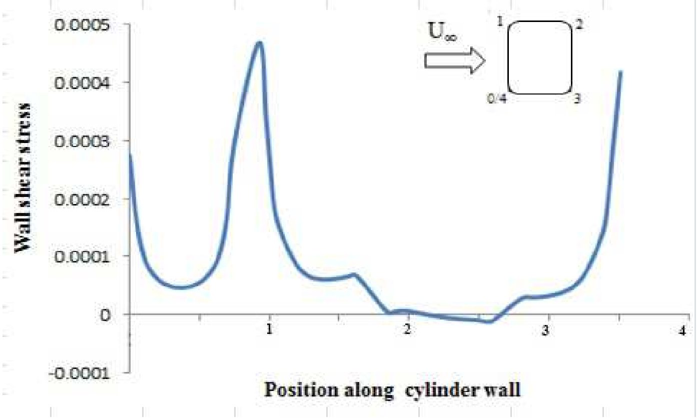
In fluid flow analysis over a bluff body, flow separation is an important term encountered. When the fluid flows over the top surface of square cylinder, flow momentum is quite low due to viscous effects and is thus sensitive to changes of pressure gradient. The flow has to move against adverse pressure gradient in addition to the viscous force. This leads to reduced velocity and wall shear stress. When the flow cannot overcome the adverse pressure gradient in the flow direction, boundary layer separates and flow reversal occurs causing the formation of lump of rotating fluid mass known as eddies.


Wall shear stress
0.00025
0.0002
0.00015
0.0001
0.00005
0
-0.00005
-0.0001
0
1
2
3
4
Position along cylinder wall
U∞
1
2
0/4
3

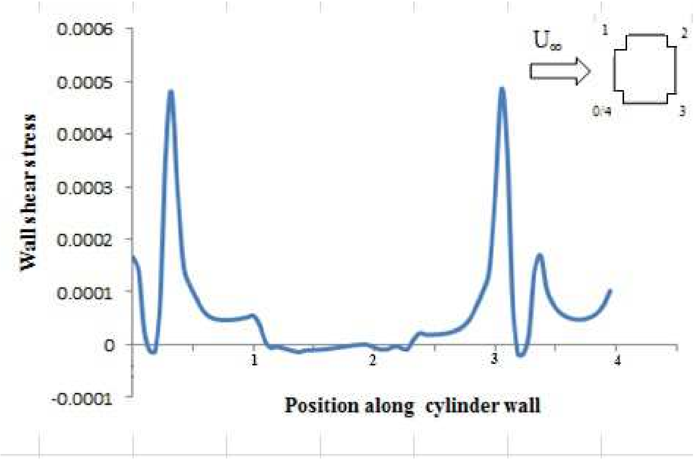
(a)


Wall shear stress
0.0007
0.0006
0.0005
0.0004
0.0003
0.0002
0.0001
0
-0.0001
1
2
3
4
Position along cylinder wall
U∞
1
2
0/4
3

(b)


Wall shear stress
0.0005
0.0004
0.0003
0.0002
0.0001
0
-0.0001
1
2
3
4
Position along cylinder wall
U∞
1
2
0/4
3

(c)



(d)

Figure 7: Flow separation points for flow past (a) Square cylinder with sharp corners (b) Square cylinder with round corners (c) Square cylinder with inclined corners and (d) Square cylinder with rectangular corners

At Re=150, flow separation occurs just after top left corner (1) of the square cylinder with sharp corners where wall shear stress is zero as shown in the Figure 7(a). Whereas with round corners, boundary layer remains attached even after top left corner and separates near the midway of top surface (1-2) of the cylinder as shown in Figure 7(b).For inclined corners, flow separation occurs near top right corner (2) and for rectangular corners, it occurs just after top left corner as shown in Figure 7(c) and Figure 7(d) respectively.

4.4 Pressure Distribution along the wall surface

Apart from velocity profile, pressure distribution is an important parameter in the study of flow around a cylinder. Pressure changes accordingly with the vortices motion in the vicinity of the body. At stagnation point, located at midpoint of face 0-1as shown in Figure 8, flow comes to rest (v=0) and pressure thus reaches maximum value (By Bernoulli’s theorem).

It is also noticed that the pressure shows symmetrical distribution; which suggest that problem has been simulated to satisfactory computational time to achieve a convergent solution. With different configurations analysed, adverse pressure gradient is larger in case of square cylinder with sharp corners and least for square cylinder with round corners as shown in Figure 8.

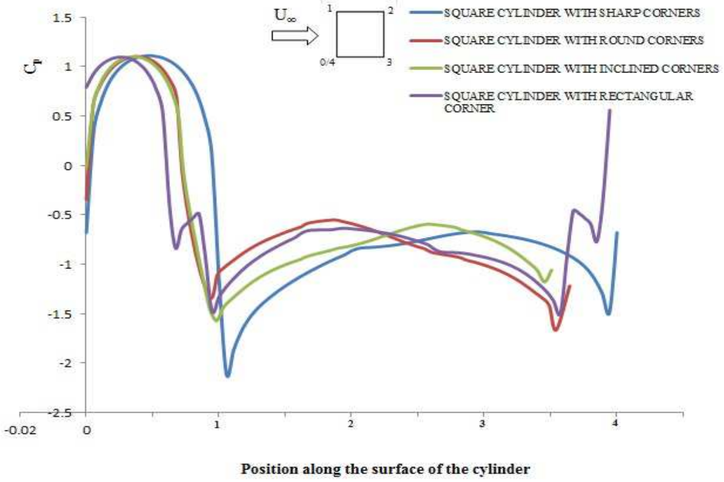


Figure 8: Coefficient of pressure variation along the cylinder surface

4.5 Pressure Variation in the near Wake region

In fluid flow analysis over bluff bodies, flow separates and creates eddies behind the object i.e. in the wake region. Eddies are alternatively shed from top and bottom of the body, inducing a transverse motion in the body. This periodic motion causes the body to vibrate and if the frequency of vibration matches with natural frequency of vibration of the body, resonance condition occurs and it may cause failure of the body or structure. Figure 9 presents the pressure distribution in the near wake region. Adverse pressure gradient is larger for square cylinder with sharp corners and least for square cylinder with round corners.

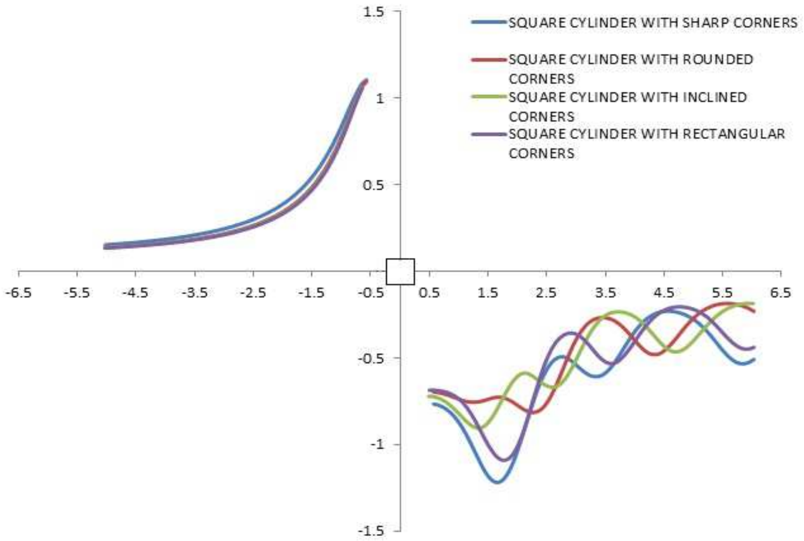


Figure 9: Variation of C_p in the near wake region

5. Conclusion

A numerical investigation of the flow around square cylinder with and without corner modifications at Reynolds number 150 has been presented. The simulation performed for flow past a square cylinder after grid refinement is in agreement with the theoretical results. 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.

- Flow separation is delayed when the sharp corners of square cylinder are modified to round corners. Also flow reattaches before reaching the top right corner.

- When the fluid flows over the top surface of square cylinder, flow momentum is quite low due to viscous effects and is thus sensitive to changes of pressure gradient. Pressure gradient on this top surface is smaller for square cylinder with round corners and larger for square cylinder with sharp corners.

- Similarly pressure variation in the near wake region is larger for square cylinder with sharp corners and least for square cylinder with round corners.

Hence in the structures with Square cross-section, the sharp edge corners can be rounded-off so that magnitude of adverse pressure gradient can be reduced and hence flow separation can be delayed without use of any external source or structure.

References

[1] Sohankar, A, Norberg, C, and Davidson, L (1998): ‘Low-Reynolds-number flow around a square cylinder at incidence: study of blockage, onset of vortex shedding and outlet boundary condition’, International Journal of Numerical Methods in Fluids, 26(1), 39-56.00.

[2] INOUE, IWAKAM, and HATAKEYAMA (2006). “Aeolian tones radiated from flow past two square cylinders in a side-by-side arrangement.” Physics of Fluids, 18(4), 046104.

[3] Ali, M.S.M., Doolan, C.J., Wheatley, V., (2009). Grid convergence study for a two dimensional simulation of flow around a square cylinder at a low Reynolds number. In: Seventh International Conference on CFD in The Minerals and Process Industries. CSIRO, Melbourne, Australia.

[4] Doolan, C.J (2009): ‘Flat-plate interaction with the near wake of a square cylinder’, AIAA Journal, 47,475-478.

[5] Bearman PW, Graham JMR, Obasaju ED, Drossopoulos GM (1984) The influence of corner radius on the forces experienced by cylindrical bluff bodies in oscillatory flow. Appl Ocean Res 6:83–89.

[6] H. Kawai (1998) Effect of corner modifications on aeroelastic instabilities of tall buildings, J.Wind Eng.Ind.Aerodyn. 74-76 719-729.

[7] Tamura T, Miyagi T, Kitagishi T (1998) Numerical prediction of unsteady pressures on a square cylinder with various corner shapes. J Wind Eng Indus Aerodyn 74:531–542.

[8] Tamura T, Miyagi T (1999) The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes. J Wind Eng Indus Aerodyn 83:135–145.

[9] Zheng W, Dalton C (1999) Numerical prediction of force on rectangular cylinders in oscillating viscous flow. J Fluids Struct13:225–249.

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

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

[12] Okajima, A., 1982, Strouhal numbers of rectangular cylinders. Journal of Fluid Mechanics 123, 379–398.

[13] Layukallo, T. and Y. Nakamura (2003), “Passive Separation Control on a Square Cylinder at Transonic Speed”, Transactions of the Japan Society for Aeronautical and Space Sciences, 45(150), 236-242.

[14] Strykowski P I and Sreenivasan K R (2002): “On the formation and suppression of vortex 'shedding' at low Reynolds numbers”, Journal of Fluid Mechanics 1990; 218: 71-107.