The 2019 World Congress on
Advances in Structural Engineering and Mechanics (ASEM19)
Jeju Island, Korea, September 17 - 21, 2019

Effect of corner modification on flow characteristics of a 3D square cylinder

Ahmad Hammad[1)], **Emad Uddin[2)], *Zaib Ali[3)], Syed Atif Iqrar[4)] and Muhammad Yamin Younis[5)]

[1), 2), 3, 4)] *Department of Mechanical Engineering, SMME, National University of Sciences and Technology (NUST), H-12, Islamabad, 44000, Pakistan.*
[5)]*Department of Mechanical Engineering, Mirpur University of Sciences and Technology (MUST), Mirpur, AJK , Pakistan*
[2)] *emaduddin@smme.nust.edu.pk*

ABSTRACT

The flow characteristics around a square cylinder are prone to get effected by applying corner modifications. This research aims to study the turbulent wake region formed behind the cylinder and apply various modifications in order to reduce the wake, thus, decreasing the flow induced vibrations and improving structural integrity. The effect of applying corner modifications on the flow, mainly turbulent wake, around a wall mounted square cylinder is investigated using Improved Delayed Detached Eddy Simulation (IDDES) turbulence model in Fluent. Validation of flow characteristics was done against the results of wind tunnel experiments at a Reynolds number of 12,000 with similar geometric configuration. Cases with different corner radii and corner chamfer were studied at multiple velocities and their results are discussed in this paper. It was observed that after a certain increase in corner radii and chamfer the effects start to get adverse and result in increasing turbulent wake instead of improving flow behind the cylinder.

1. INTRODUCTION

Flow characteristics around a square cylinder with corner modifications have been an interesting topic for engineers and scientists because of its practical significance. Design of tall buildings, suspension bridges, flow past electronic components and cooling towers etc. are a few of the many applications that it holds. Fluid and structure interaction creates a complex flow field and understanding this interaction is of paramount importance in design. When a bluff body is immersed is a smooth and steady flow, it may bring oscillations which are damaging to the structure. The body experiences lift and drag forces and the variation of these forces cause Fluid Induced Vibrations. The vibrations generally have a negative impact on structures and have to be minimized. Due to its significance flow characteristics around square

** Corresponding author
* Presenting author
1, 4) Graduate Student
2, 3, 5) Assistant Professor

cylinders have been studied by many researchers. (Castro 1977) studied the physics of flow and wake characteristics around a wall mounted cube and found that the wake size is reduced in the context of the turbulent shear flow. It was also noted that the turbulence intensity also strongly influence the decay of velocity deficit behind the cube which was later confirmed by (Hunt 1978). The wake structure of finite circular cylinder with small width to height ratio was studied by (Sumner 2004) and it was found that the wake structure and power spectra for aspect ratio of 5 to 9 were quite similar but a variation was observed from that of aspect ratio 3. (Bourgeois 2011) conducted wind tunnel experiments and studied the large scale vortex structures in the turbulent wake of a finite surface mounted square cylinder at high Reynolds Number (Re) of 12,000 and aspect ratio of 4. The flow field was captured using Laser Doppler Velocimeter (LDV). These results serve as a benchmark for numerical solutions later on. It showed that due to the free end shear layer the alternating vortex shedding was deformed as it evolved downstream of the flow. Later on (El Hassan 2015) conducted a follow up study and observed the boundary layer thickness effects on the wake structures. (Saeedi 2016) used the same model as (Bourgeois 2011) and performed large-eddy simulations using numerical algorithm to investigate the turbulent wake behind the wall mounted square cylinder. The study was able to successfully reproduce the experimental results of (Bourgeois 2011) numerically. The results reflected that close to ground at low elevation, horse shoe vortex can be observed and it disappears and is replaced by two counter rotating vortices as the elevation rises. The effects of changing aspect ratios on the flow were studied by (Chen 2018). IDDES model was used in this study to simulate flow around a square cylinder using a commercially available CFD code. It was observed that by increasing the aspect ratio from 1 to 2 the recirculation of the flow increases and it maintained its size as it was further increased to 4. Studies have also been conducted for square cylinders having corner modifications. Corner cutoffs were introduced in a square cylinder by (Yamagishi 2009) and experimental investigation was conducted. It was found that the straight corner modification also knows as corner chamfer displayed the least drag coefficient. Numerical results for square and diamond cylinder with and without modification was presented by (Dalton 2003) at Re=250 and 1,000. A noticeable decrease in the drag and lift coefficients was noted by rounding the corners of the bluff body. Changing corner radius was introduced and its effect on the near wake of a square prism was observed by (Hu 2006) and it was observed that corner radius effects the flow structure greatly. (Monica 2019) conducted wind tunnel experiments to investigate effects of corner cutoffs on rectangular buildings. Various incidence angles were also introduced in the Computational Fluid Dynamics (CFD) study and Aerodynamic characteristics were noted. The studies discussed previously mainly focused on experimental and numerical solutions. A study using a CFD tool ANSYS, Fluent was conducted to note the effects of corner cutoffs by (Vikram 2012). Square cylinder cross-section with round and straight modification was studied at Re=100 and 200 and it was observed that by applying corner modification the velocity behind the cylinder becomes large. However, the lift coefficient decreases by applying corner modification. Few studies have been conducted in the past but it can be seen from above that effects of varying corner modification coupled with changing Reynolds Number is something that still needs to be explored. Therefore, the study conducted focused on finding the right corner modification and how changing the velocity i.e. the Reynolds Number will affect the flow field. The three dimensional vortex structure in the wake region and around the square cylinder for each case was also visualized. The model used for the study was same as used by (Bourgeois 2011) and (El Hassan 2015) in order to make validation of results easier. The initial model was validated against their experimental results and then modifications were applied at corner and its effect noted.

2. MODEL AND SETUP DETAILS

The study used the Improved Delayed Detached Eddy Simulation (IDDES) turbulent model with k-ω SST and the turbulent flow around a wall mounted cylinder was simulated. IDDES model was selected instead of RANS and LES because the particulars of turbulence quantities such as fluctuations in the velocity are not given by RANS model. Also simulating flow separation is a weakness of RANS model. It is known that LES predicts the flow behavior better than IDDES model but it requires a very small grid spacing and thus increasing the computational time according to (Chen 2018). Therefore an IDDES model was used to acquire adequate accuracy and maintain a better computational efficiency than the LES model.

2.1. Computational Domain and Setup

A wall mounted square cylinder was used in this study which is similar to the experimental work of (Bourgeois 2011) and (El Hassan 2015). The width of the cylinder was kept at d = 0.0127m and aspect ratio at 4 with Reynolds number (Re) = 12,000 and a freestream velocity (U_∞) = 15m/s, the dynamic viscosity of $1.855\text{x}10^{-5}$ Pa-s and turbulence intensity of 0.8%. At obstacle location the boundary layer thickness was approximately 0.18h. The square cylinder was placed downstream of the inlet at 4d and 23d upstream of the outlet. Each side wall was kept at 6.2d from the object in the span wise direction and total height was kept at 9d. Simulations were carried out at different velocity values of 5 m/s, 10 m/s and 15 m/s for various test model configurations. Their corresponding Reynolds number are 4,000, 8,000, and 12,000. A time step of 0.1 ms was taken to simulate the results and the domain was meshed with 2 million computational cells.

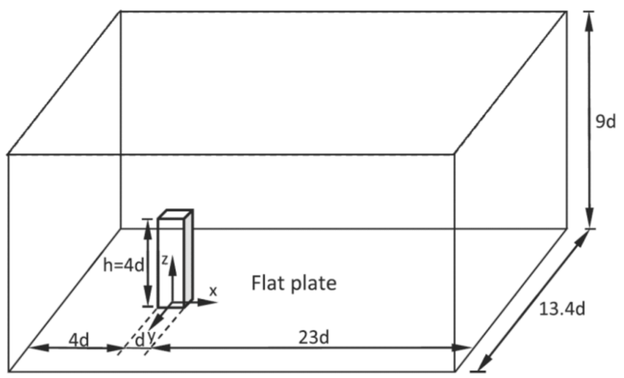

Fig. 1 Computational Domain for IDDES analysis

2.2. Test Model Details

The test model used for the study was a square cylinder with dimensions 0.0127m x 0.0127m x 0.0508m. Two types of corner modifications were then applied to the test model i.e. corner chamfer and corner radius. The corner ratio (C_R) was set at 0.2, 0.3 and 0.4 for both modifications. Where:

$$C_R = r/d \tag{1}$$

And '*r*' is radius for corner radius and chamfer in case of corner chamfer. The dimensions of corner modifications are given in table below:

Table 1 Corner Dimensions

Corner Ratio	Corner Chamfer	Corner Radius
0	0	0
0.2	0.00254m	0.00254m
0.3	0.00381m	0.00381m
0.4	0.00508m	0.00508m

The test models are shown in Fig. 2.

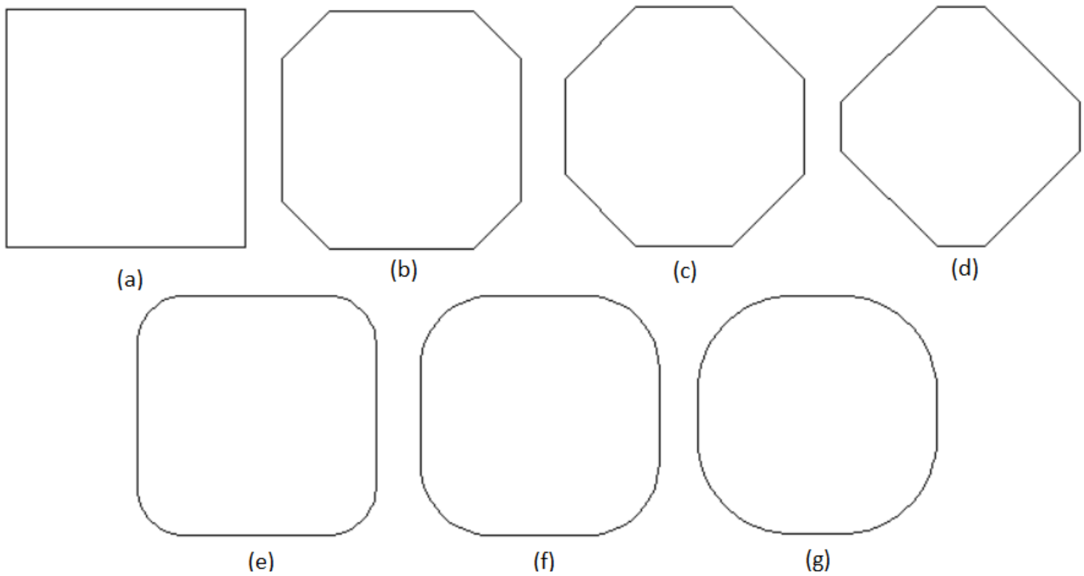

Fig. 2 Test models used for simulation with (a) without corner modification, (b) with 0.2d corner chamfer, (c) with 0.3d corner chamfer, (d) with 0.4d corner chamfer, (e) with 0.2d corner radius, (f) with 0.3d corner radius, (g) with 0.4d corner radius

3. SIMULATION RESULTS AND DISCUSSION

The simulations were carried out first for a simple model without any corner modification at Re=12,000. The results obtained were then compared with the experimental results of (Bourgeois 2011) and (El Hassan 2015). Velocity contour of x-z plane were compared for validation. The results are shown in Fig. 3.

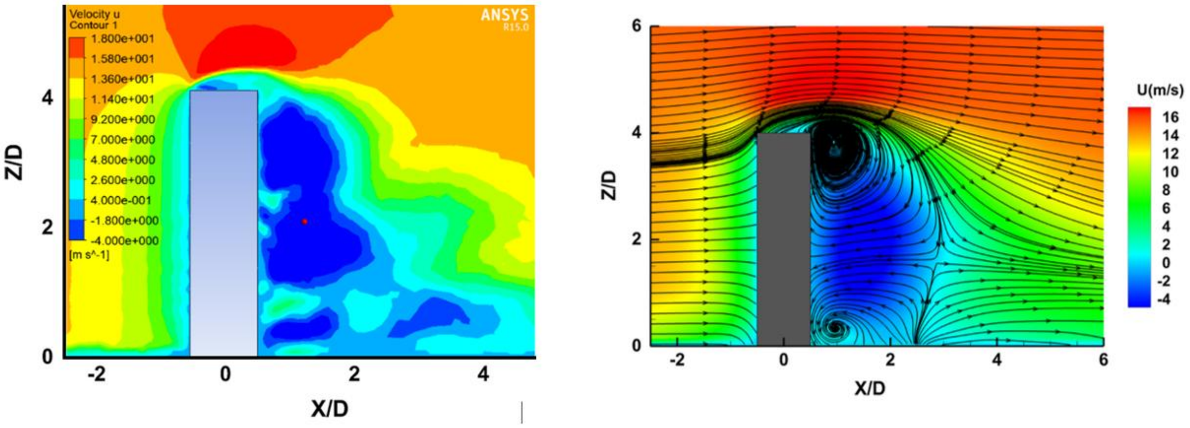

Fig. 3 Current simulation result (left) validated against the experimental result (right) by (El Hassan 2015) at the x-z plane at y = 0

The flow velocity contour on the x-z plane at y=0 can be seen in Fig.3. The result of current DES simulation is shown on the left side and previous wind tunnel result is shown on the right. It can be observed that in both cases the flow behind the cylinder moves downward at around x/d ≈ 2.

3.1. Effect on Velocity Contour with modification at Re=12,000

The time averaged normalized contours of velocity behind the object at central plane y/d = 0 gives an insight on flow pattern downstream of the test model. The large recirculating vortex formed behind the cylinder were observed as seen in Fig.3 for all the modified cases to observe the flow characteristics and its effect on the turbulent wake behind the cylinder. Fig. 4 shows the time-averaged velocity contour in central plane y/d=0 for cylinder with corner modification of 0.2d. The velocity was set at 15 m/s and Re=12,000. Velocity contours for corner chamfer are shown on the left while corner radius result is shown on the right side. The vortex region behind the two cylinders can be seen in the figure. It was noted that the vortex recirculation region for both cases was minimized. This is consistent with previous studies stating that by applying corner modification the maximum vorticity of the vortex attenuated (Hu 2006). The velocity behind the cylinder however, after applying modification, increased as compared to the results obtained for a cylinder without modifications.

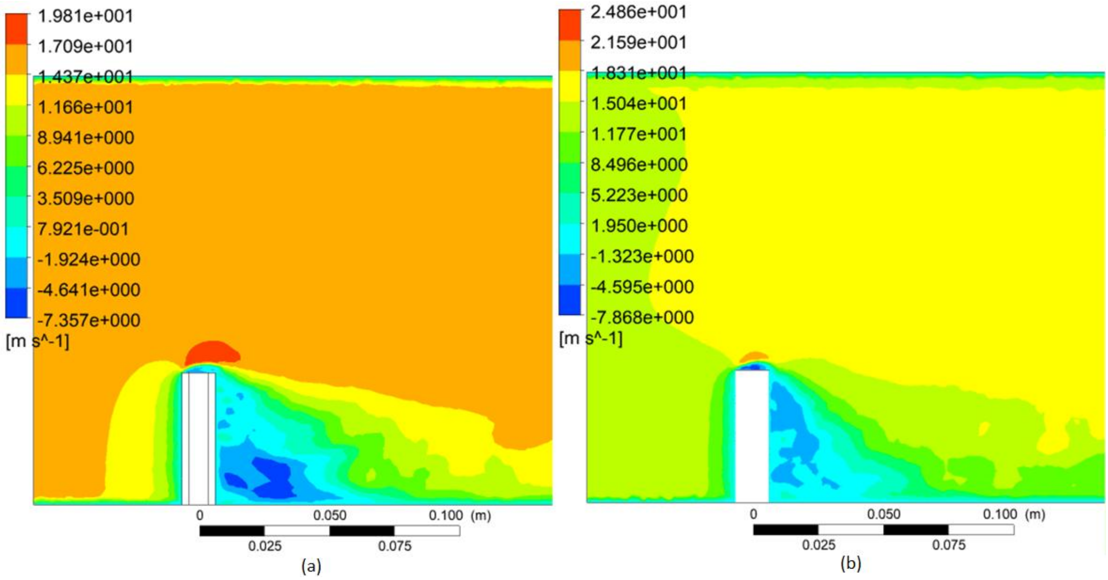

Fig. 4 Time-averaged velocity contour for Re=12,000 at central plane y/d=0 for (a) 0.2d Corner Chamfer & (b) 0.2d Corner Radius

Similar results were obtained for the case of 0.3d corner ratio and 0.4d corner ratio. By comparing the results of corner chamfer and corner radius it was seen that the vorticity region is smaller when corner radius is applied. This observation holds true since by applying corner radius the cylinder is shifting from square to circle thus a decrease in wake behind the cylinder. The time-averaged velocity contours for 0.3 corner ratio and 0.4 corner ratio can be seen in Fig. 5 and Fig. 6 respectively.

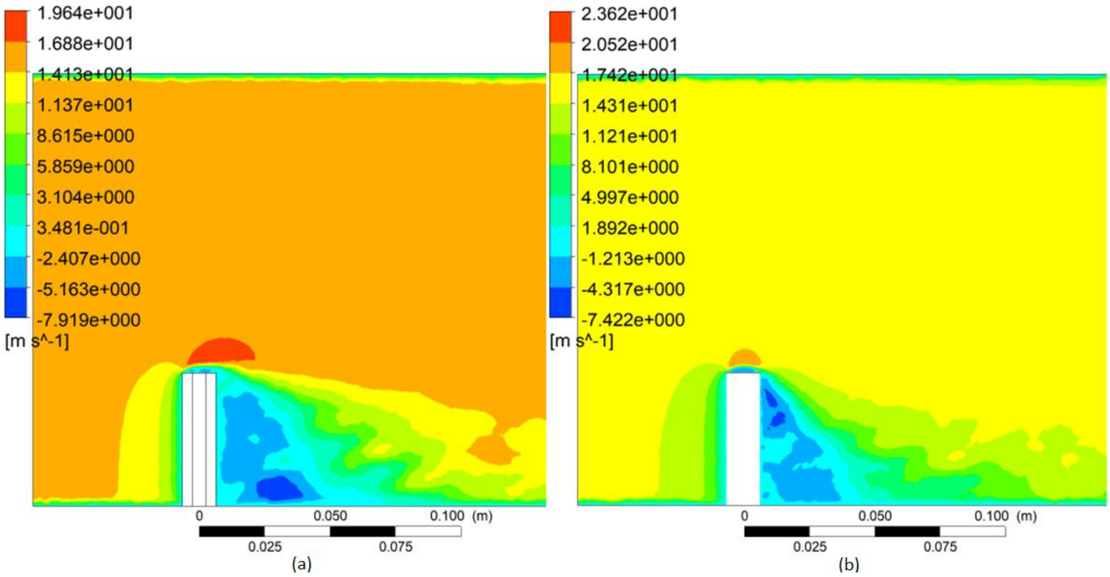

Fig. 5 Time-averaged velocity contour for Re=12,000 at central plane y/d=0 for (a) 0.3d Corner Chamfer & (b) 0.3d Corner Radius

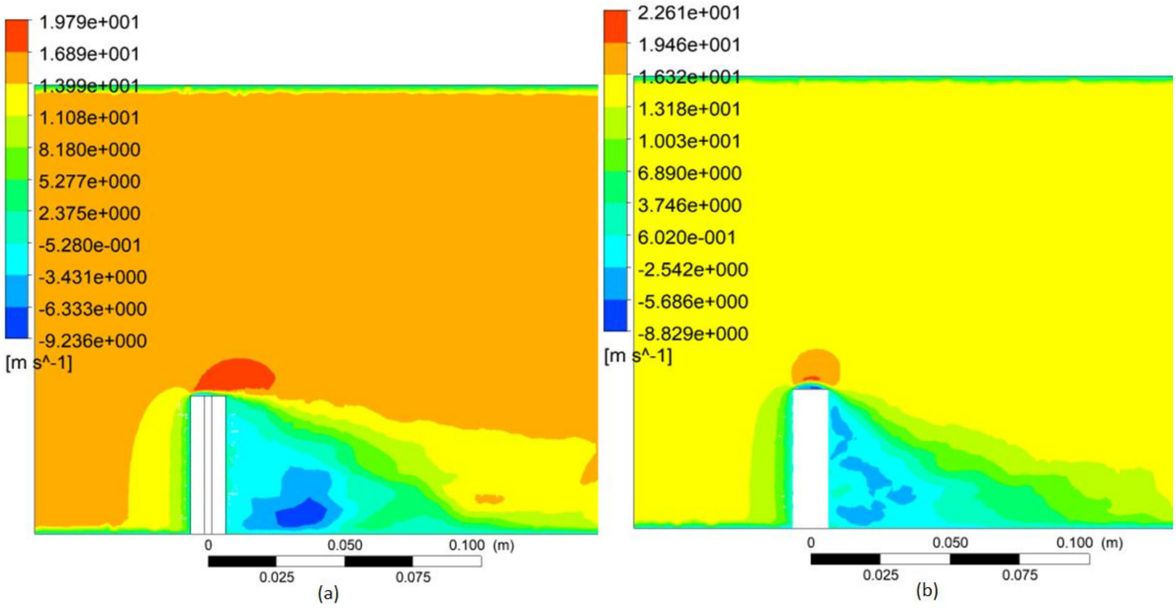

Fig. 6 Time-averaged velocity contour for Re=12,000 at central plane y/d=0 for (a) 0.4d Corner Chamfer & (b) 0.4d Corner Radius

Taking a look at Fig. 6 it can be observed that maximum velocity is at the top surface of the cylinder. The velocity increases when corner modification of 0.2d is applied and it can be seen clearly at the top of the cylinder surface and behind the cylinder. The increase in velocity will result in a decrease in lift coefficient as studied by (Vikram 2012) which additionally states that decrease in lift coefficient will increase the Strouhal number and as a result the vortex wavelength will decrease. The velocity then decreases as the corner modification ratio increases to 0.3. As the corner modification ratio reaches 0.4, the velocity for corner chamfer increases. This shows that after a certain increase in chamfer the effect start to reverse with increase in chamfer. For corner radius on the other hand, the velocity keeps on decreasing further.

3.2. Effect on Velocity Contour by changing Re

Time-averaged velocity contour at central plane y/d=0 were obtained for all the test cases at Reynolds number of 4,000, 8,000 and 12,000 respectively. The results for 0.3d corner chamfer and corner radius dimension are discussed here at all three Reynolds number with velocities 5 m/s, 10 m/s and 15 m/s respectively. Similar pattern was observed for 0.2d and 0.4d corner modification ratio. Time-averaged velocity contour can be seen in Fig. 7 for 0.3d Corner Chamfer on left and 0.3d Corner Radius on right at central plane y/d=0 at Re=4,000 (Fig. 7 (a) & (b)), Re=8,000 (Fig. 7 (c) & (d)) and Re=12,000 (Fig. 7 (e) & (f)). By decreasing the Reynolds number the vortices are minimized thus suggesting that decrease in Reynolds number will result in decrease in the turbulence of flow. This corresponds to the findings of (Raghavan 2011) which suggests that by increasing Reynolds number the vorticity is increased thus increasing the vibrational amplitude.

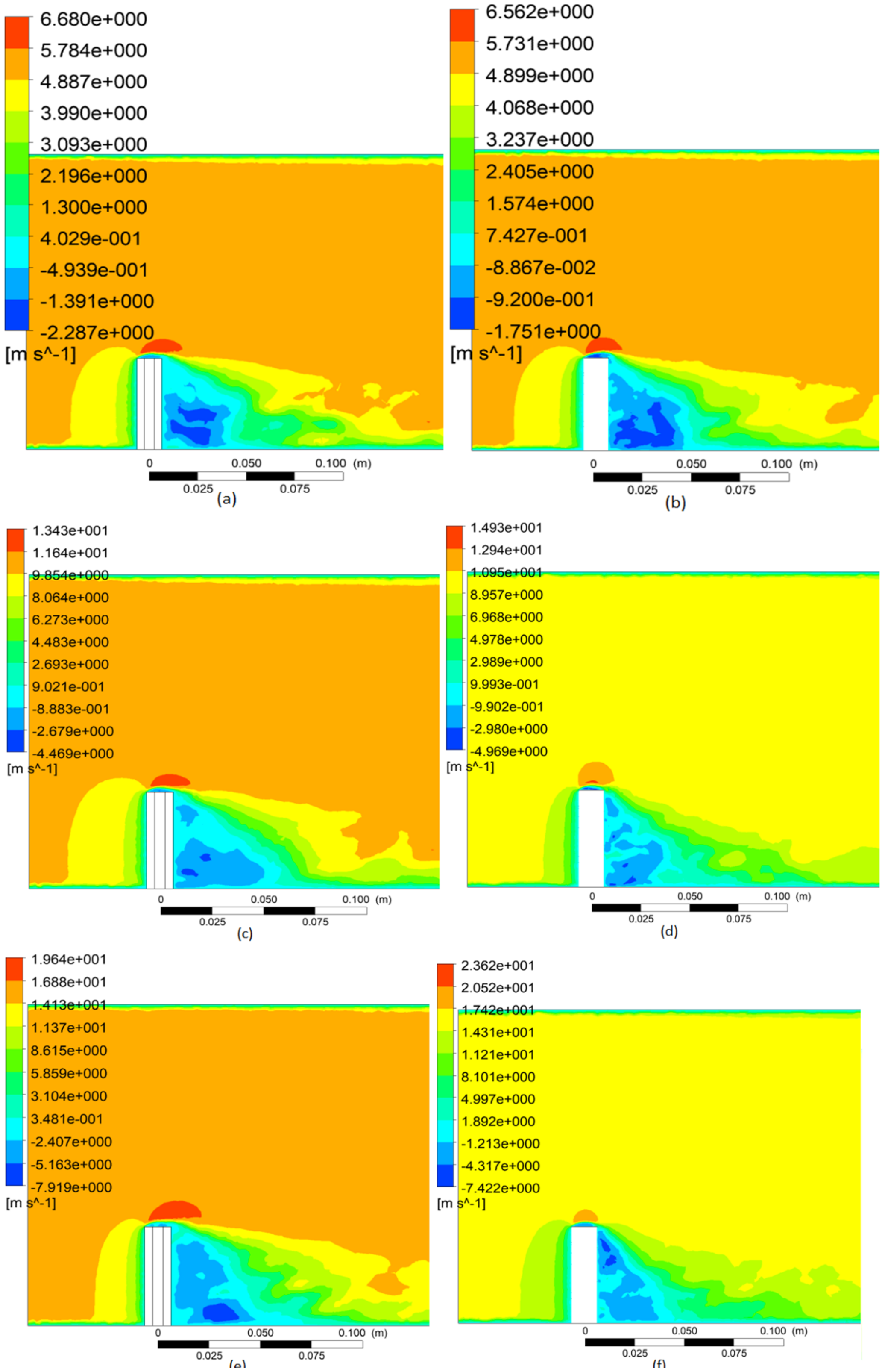

Fig. 7 Time-averaged velocity contour for 0.3d Corner Chamfer on left and 0.3d Corner Radius on right at Re=4,000 (a & b), Re=8,000 (c & d) and Re=12,000 (e & f)

3.3. Effect on Vortex structure by applying Corner Modification at Re = 12,000

The purpose of this study was to determine the best possible corner modification configuration and ratio in order to reduce the turbulent wake and vortices behind the structure. Two types of corner modifications were studied which were corner chamfer and corner radius. The two dimensional velocity contour at central place y/d=0 were discussed in the last section. The three dimensional vortex structure around the cylinder and downstream of the flow is discussed here which was obtained using low pressure iso-surfaces. The vortex shed from the cylinder without corner modification can be seen in Fig. 8. These vortices are developed around the cylinder and then approach ground at an oblique angle which corresponds to the structure obtained by (Saeedi 2016) using LES and serves as an additional validation of current IDDES results.

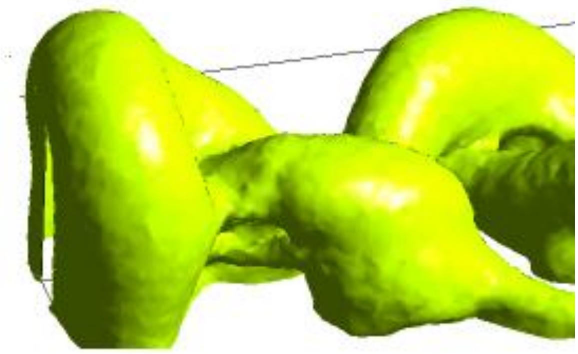

Fig. 8 Low pressure large alternating vortex tubes downstream of the cylinder

Corner modifications were applied to the test model under observation and the resulting vortex structures were obtained and are shown in Fig. 9. The figure shows low pressure large vortex tubes downstream of the cylinder at Re=12,000 for Corner Chamfer on the left and Corner Radius on the right with corner ratio of 0.2d (a) & (b), 0.3d (c) & (d) and 0.4d (e) & (f). It was observed that the vortex region reduces and moves downwards with increase in corner modification ratio. However, it was noted that with the ratio beyond 0.3d the trend starts to reverse and a rise in vortex structure was observed. Overall vortex strength for corner chamfer was observed to be greater than that of corner radius. Therefore, it can be claimed that the vorticity around a square cylinder by introducing corner radius is lower than that of corner chamfer. Similar pattern was observed for Re=8,000 and Re=4,000. Only the results for Re=12,000 are shown in Fig. 9.

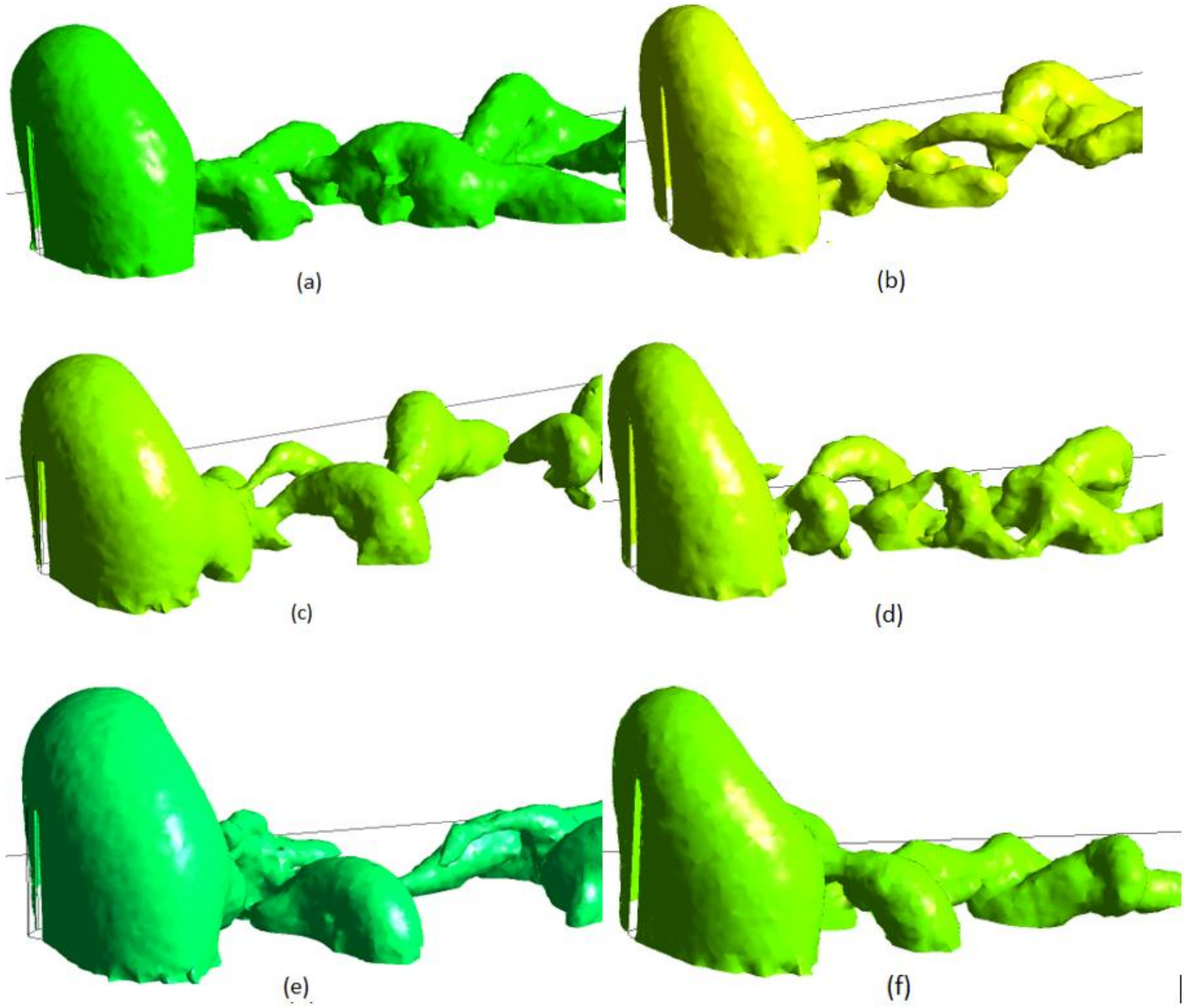

Fig. 9 Low pressure large alternating vortex tubes downstream of the cylinder at Re=12,000 for Corner Chamfer on the right and Corner Radius on the left with corner ratio of 0.2d (a & b), 0.3d (c & d) and 0.4d (e & f)

4. CONCLUSIONS

The three dimensional square cylinder was studied with and without corner modifications. The modifications applied were corner chamfer and corner radius of ratio 0.2d, 0.3d and 0.4d. As the aspect ratio of the cylinder is high, it results in an unsteady and complex flow around the cylinder. The simulation results lead to the conclusion that the square cylinder with corner medication show an increase in velocity at cylinder top and downstream of the cylinder compared to the cylinder with no modification. This increase in velocity will result a decrease in vortex wavelength. It can also be seen that for cylinder with corner radius of 0.2d, results indicate an increase in downstream velocity. Further increase in the corner ratio will result in decrease in velocity from that of 0.2d corner ratio. In case of corner chamfer, the velocity behaves the same as that of corner radius till 0.3d corner ratio. However, after that the velocity again starts to increase. A decrease in vortex formation behind the cylinder has been observed by decreasing the Reynolds number. It has been noted that the introduction of corner modification results in minimization of vortex recirculation region. With the introduction of corner chamfer the vortex region moves downward towards the bottom wall. The vortex region formed for corner radius has been noted to be lesser than that of corner chamfer.

REFERENCES

Bourgeois, J. A., Sattari, P., & Martinuzzi, R. J. (2011). “Alternating half-loop shedding in the turbulent wake of a finite surface-mounted square cylinder with a thin boundary layer”. Physics of Fluids, 23(9), 095101.

Castro, I. P., & Robins, A. G. (1977). “The flow around a surface-mounted cube in uniform and turbulent streams”. Journal of fluid Mechanics, 79(2), 307-335.

Dalton, C., & Zheng, W. (2003). Numerical solutions of a viscous uniform approach flow past square and diamond cylinders. Journal of Fluids and Structures, 18(3-4), 455-465.

El Hassan, M., Bourgeois, J., & Martinuzzi, R. (2015). “Boundary layer effect on the vortex shedding of wall-mounted rectangular cylinder”. *Experiments in Fluids*, *56*(2), 33.

Hunt, J. C. R., Abell, C. J., Peterka, J. A., & Woo, H. (1978). “Kinematical studies of the flows around free or surface-mounted obstacles; applying topology to flow visualization”. Journal of Fluid Mechanics, 86(1), 179-200.

Hu, J. C., Zhou, Y., & Dalton, C. (2006). “Effects of the corner radius on the near wake of a square prism”. Experiments in Fluids, 40(1), 106.

McClean, J. F., & Sumner, D. (2014). “An experimental investigation of aspect ratio and incidence angle effects for the flow around surface-mounted finite-height square prisms”. Journal of Fluids Engineering, 136(8), 081206.

Monica, A., Harikrishna, P., Nagamani, K., & Babu, G. R. (2019). “CFD and Wind Tunnel Investigations on Rectangular Building with Corner Cuts”. In Recent Advances in Structural Engineering, Volume 2 (pp. 53-68). Springer, Singapore.

Raghavan, K., & Bernitsas, M. M. (2011). “Experimental investigation of Reynolds number effect on vortex induced vibration of rigid circular cylinder on elastic supports”. Ocean Engineering, 38(5-6), 719-731.

Saeedi, M., & Wang, B. C. (2016). “Large-eddy simulation of turbulent flow around a finite-height wall-mounted square cylinder within a thin boundary layer”. Flow, Turbulence and Combustion, 97(2), 513-538.

Sumner, D., Heseltine, J. L., & Dansereau, O. J. P. (2004). “Wake structure of a finite circular cylinder of small aspect ratio”. Experiments in Fluids, 37(5), 720-730.

Chen, J. (2018). “Effect of Aspect Ratio on the Flow Structures behind a Square Cylinder” Electronic Theses and Dissertations. 7507.

Vikram, C. K., Krishne Gowda, Y. T., & Ravindra, H. V. (2012). “Influence of corner cutoffs on flow past square cylinder”. In Proceedings of International Conference on Challenges and Opportunities in Mechanical Engineering, Industrial Engineering, and Management Studies (ICCOMIM), Q (Vol. 33).

Yamagishi, Y., Kimura, S., Oki, M., & Hatayama, C. (2009, August). “Effect of corner cutoffs on flow characteristics around a square cylinder”. In 10th International Conference on Fluid Control, Measurements, and Visualization (FLUCOME), Moscow, Russia, Aug (pp. 17-21).