IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 12, Issue 5 Ver. I (Sep. - Oct. 2015), PP 14-21
www.iosrjournals.org

Aerodynamic Studies of non – Circular High – rise buildings

Yechuri N V Mani Sandeep Kumar [1], V Dhana Raju[2], P Ravindra Kumar [3]
[1, 2, 3] *(Department of Mechanical Engineering, Lakireddy Balireddy College of Engineering, India)*

Abstract: *Wind induced load is an important issue for tall buildings, cable suspension bridges, electricity transmission towers, telecommunication towers and chimneys during natural disasters like strong winds of cyclone and earth quakes. One way to minimize wind-induced vibrations of tall buildings is to focus more on their shapes in the design stage. Important design aspect is that rather complicated sectional shapes are basically good with regard to aerodynamic properties for crosswind responses, which is a key issue in tall building wind-resistant design. Four models of non-circular high rise buildings were made and tested in a low speed wind tunnel. Triangular, taper square, square steps with sharp corners and square steps with filleted corners models are prepared. The study investigates the drags force and pressure distribution at the center of different models at different velocities with different orientations.*

Key words: *co- efficient of pressure, Drag force, high –rise buildings, Torsional Vibrations, Wind Tunnel.*

I. Introduction

A **tower block**, **high-rise building**, is a tall building or structure used as a residential and/or office building. In some areas they may be referred to as "MDU" standing for “Multi Dwelling Unit”. High–rise structures pose particular design challenges for structural and geotechnical engineers, particularly if situated in a seismically active region or if the underlying soils have geotechnical risk factors such as high compressibility or bay mud. Studies are often required to ensure that pedestrian wind comfort and wind danger concerns are addressed. In order to allow less wind exposure, to transmit more daylight to the ground and to appear more slender, many high-rises have a design with setbacks. Tall buildings have been traditionally designed to be symmetric rectangular, triangular or circular in plan, in order to avoid excessive seismic-induced torsional vibrations due to eccentricity, especially in seismic prone regions like Japan. However, recent tall building design has been released from the spell of compulsory symmetric shape design, and free-style design is increasing. Wind-induced load is an important and essential design issue for tall buildings, cable suspension bridges, electricity transmission towers, telecommunication towers and chimneys. For real structures, the flow field is very complex in nature; hence, experimental studies are mandatory. So far, some studies on the effect of increasing number of sides on aerodynamic characteristics have been conducted, but in the studies only mean drag force has been focused on by the numerical simulations under a uniform flow, no discussions were found on the characteristics of other wind force components and responses using a boundary layer flow.

Tamura et.al, 2010 [1], the present trend towards design of tall buildings is toward unconventional shapes such as square cross sections with different helical models as this provides good aerodynamic characteristics. Many researchers have tested wind pressures on irregular plan buildings Anim and Ahuja, 2008 et. al, [2], different rectangular cross sections tapered building models with taper ratios of 5% and 10%, and building models with set-back at mid-height Kim and Kanda, 2010 [3]etc., but, there have been very few studies on triangular cross-section tall buildings. The effects of building plan shape on aerodynamic forces and displacement response have been studied for super-high-rise buildings with square and triangular cross-sections with corner modifications Hayashida et.al. 1990 [4]. Aerodynamic modification of building shape, such as by changing the cross-section with height through tapering, alters the flow pattern around tall buildings, which can reduce wind-induced excitations of buildings with different rectangular cross-sections Lin et al., 2005 [5], tapered building models with taper ratios of 5% and 10%, and building models with set-back at mid-height Kim et.al., 2010 [6].

Besides changing building shapes, many current tall buildings have various polygon cross-sections, not being limited to the simple square cross-section. So far, there have been some studies that have investigated the aerodynamic characteristics of tall buildings with various polygon cross-sections. Chien et al. (2010) [7], examined mean drag forces, in terms of shape factors, for 8 kinds of polygon high mast structures using 2-dimensional CFD analyses. In their study, the mean drag force becomes constant for the polygon structures with more than 10-side.

Tang et al. (2013) [8], investigated the mean drag forces and wake flows on straight and twisted polygon tall buildings using numerical simulations. They examined the effect of increasing number of sides, rounded corners of square cross-section, and twisting angle of the square cross-section tall building. They used 10 straight tall buildings for various polygon cross-sections, 11 rounded square cross-sections with various fillet radii, and 9 twisted tall buildings with twisting angle up to 180º by the interval of 22.5º.The effectiveness of aerodynamic modification to reduce wind loads has been widely reported Miyashita et al. (1993) [9].

Six pressure models of high rise buildings were tested in a boundary layer wind tunnel. Five had triangular cross sections with configurations of Straight Triangle, Corner cut, 60^o Helical, 180^o Helical and 360^o Helical, and the other had a configuration of Clover. The results show that the helical models had more effect on aerodynamic characteristics Bandi et.al. [10]

II. Experimental Setup

The tests were conducted in low speed wind tunnel (shown in fig. 1). The size of the test section is 600 mm long and 300 mm side square cross- section. The high – rise building models used in this work are triangle prism, Taper Square, Square steps with sharp corners and Square steps with filleted corners (shown in fig 2 (a), (b), (c) & (d). The size of all the models is 100 mm, width of triangle side is 50mm, size of Taper Square is 60mm and for step models the size of sides is 80 mm. These models contain ports at its center at a distance of 5 mm each to measure the pressure acting on its surface. The experimentation is done for all the models at an angles of 0^0, 90^0, 180^0 and 270^0 to the flow direction and at different velocities 15 m/s, 20 m/s, 25 m/s. Values of drag, pitching moments and pressure on the surface of the models are measured using wind tunnel. The pressure distribution behind the models at X/D = 0, 1, 2 also measured using wake rake apparatus.

Fig 1. Low Speed Wind Tunnel

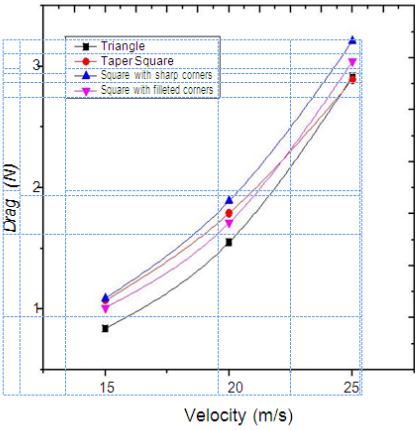
(a) **(b)**

(c)

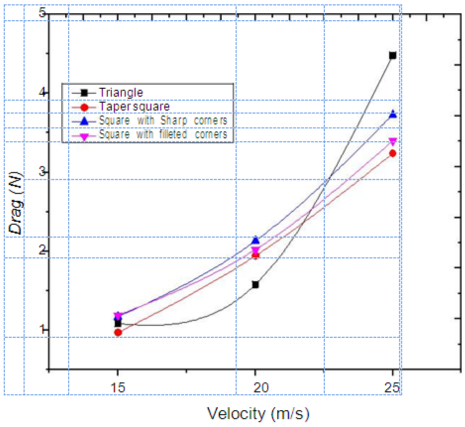
(d)

Fig 2. The above figures represents the fabricated models a) Triangular prism model, b) Taper Square model, c) Square steps with sharp corners, d) Square steps with filleted corners

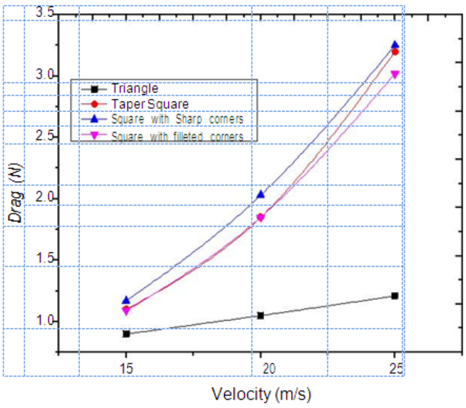
III. Results And Discussions



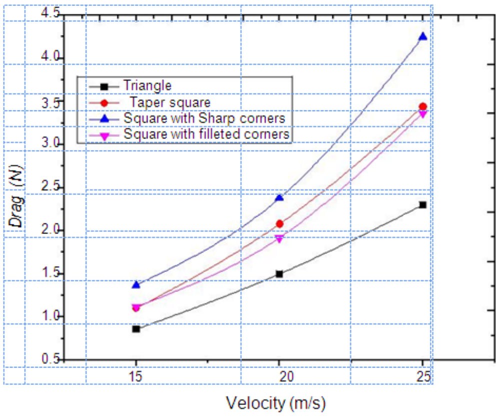
Variation of Drag at 90^0



b) Variation of Drag at 180^0



c) Variation of Drag at 180^0



d) Variation of Drag at 270^0

Fig 3. Variation of drag for different models at different angles with velocity

The plot (fig 3 (a), (b), (c), (d)) shows the variation of drag for different models at 0^0, 90^0, 180^0, 270^0 to the flow direction of wind at different velocities 15 m/s, 20 m/s, 25 m/s. In fig 3 (a) The plot shows that the drag is increasing for all the models with increase of velocity but for triangle model the drag is less at low speeds but it increases rapidly and reaches to the value of square models at high velocities of air. The drag for the tapered square is increases along with the remaining models up to the velocity 20 m/s and then the increment is deviating from the remaining shapes and reaches low value than triangle at high velocities. Fig 3 (b), The drag for corner filleted model is high at initial velocity increases with velocity but less than the sharp corner model. The filleted edges diverts the flow and decreases the force caused by flow. The drag for the taper square model increases with velocity but less than all the models at high velocities because the area exposed to the flow decreases with increase of height. Fig . 3 (c) The drag for the triangle at all velocities is less compared to the remaining models because of its shape the flow is divided by the sharp corner of the model and moves away from the model. Because of the same cross- sectional models the drag is almost changes equally for the remaining models, but because of the smooth corners the flow produces less drag compared to remaining models for the square steps with filleted corner model. At very high velocities this small change in shape plays main role. Fig 3 (d) The drag for steps with sharp corners is very much higher than remaining models because of sharp edges the flow is not smooth as the remaining models. The drag for taper and smooth edges square steps is nearly same at starting but for taper it is increases because of its shape.

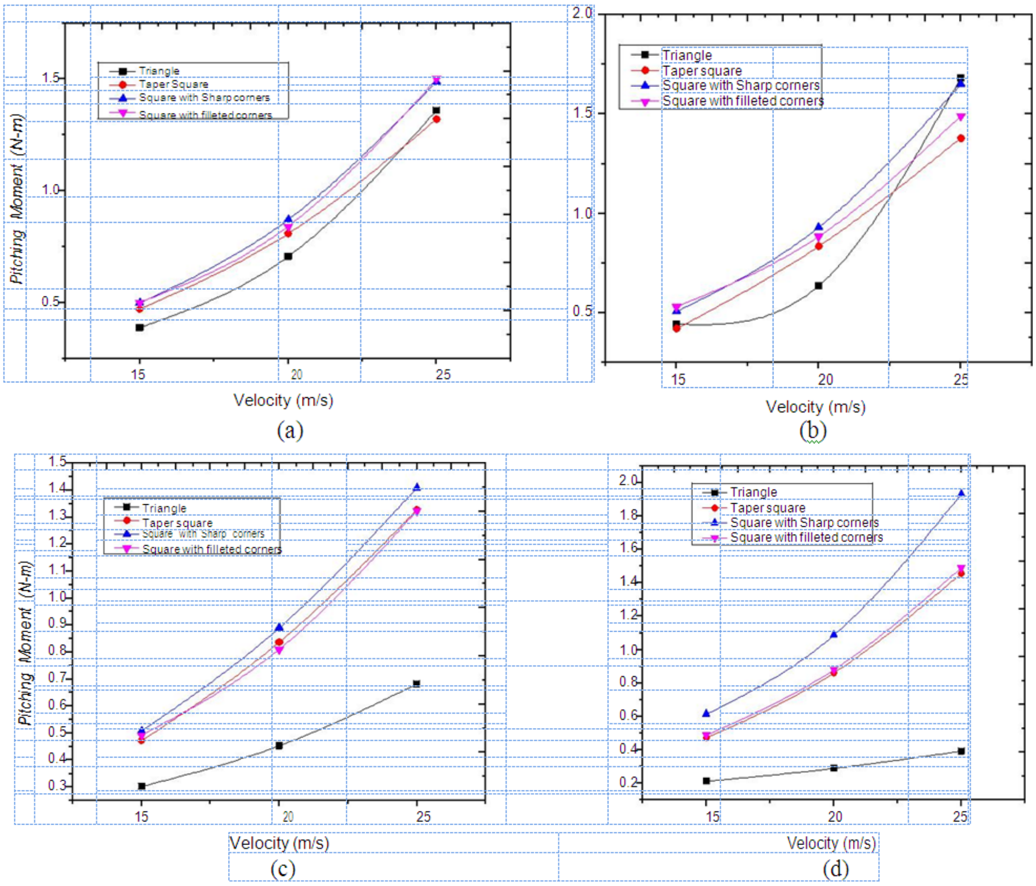
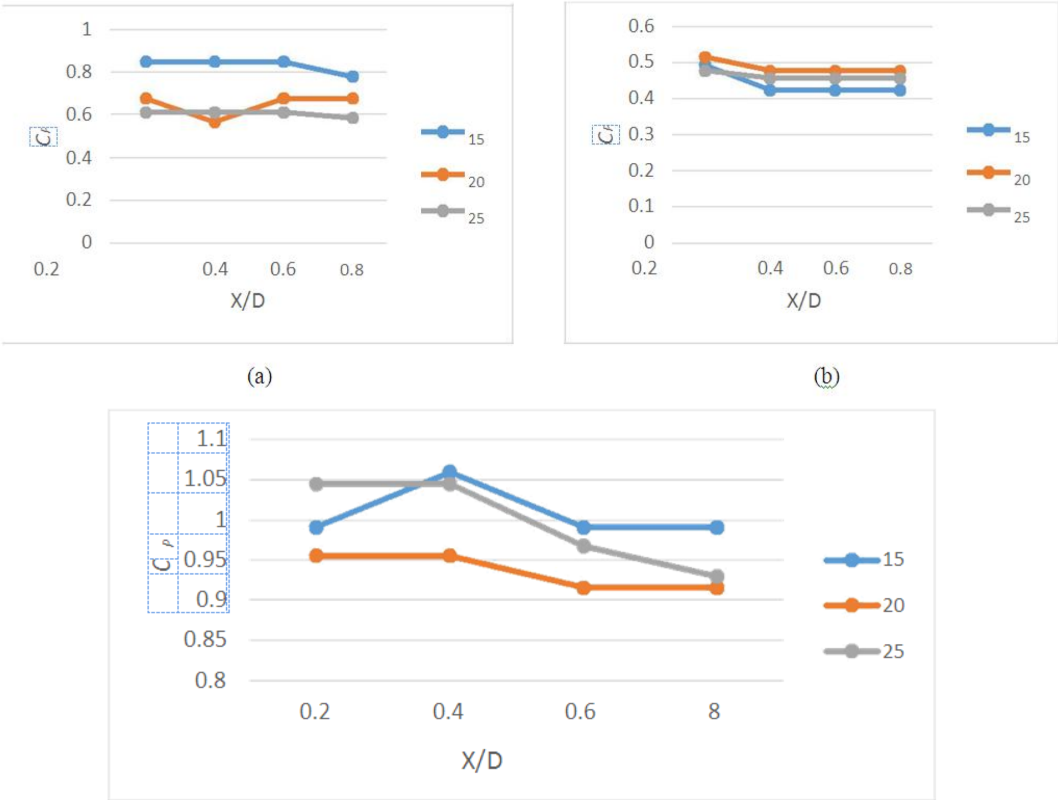


Fig 4. Variation of Pitching moment with respect to velocity at different angles to the flow

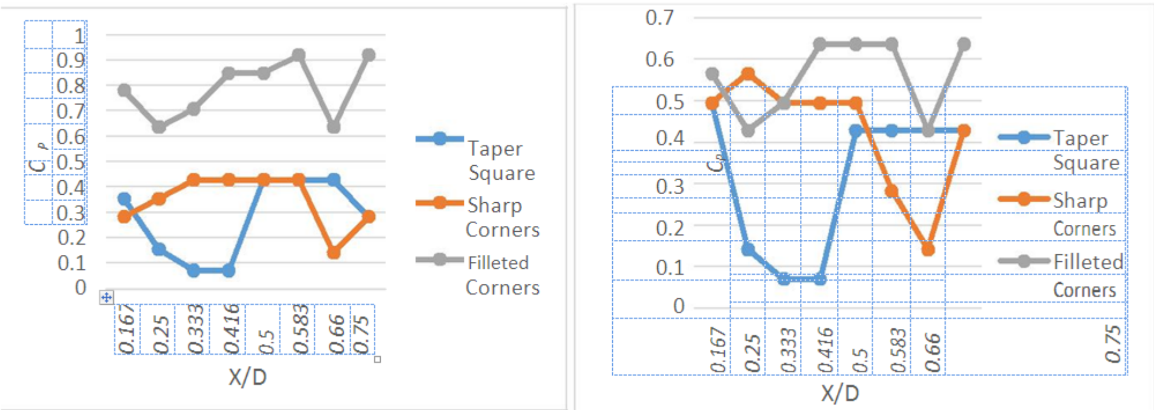
The plots shown in fig 4 (a), (b), (c), (d) shows the variation pitching moment for different models at 0^0 to the flow direction at different velocities i.e., 15 m/s, 20 m/s & 25 m/s. in fig 4 (a) The pitching moment is initially less for triangle model but increases with increase of velocity and reaches a value slightly more than taper square model. But for the models square steps with sharp corners and filleted corners has almost same values at all the velocities but filleted corner model has slight lesser value than the sharp corner model. For the taper model it is slightly higher than the triangle model but lesser than the remaining models but at height velocities it posses lesser pitching moment than the remaining models. In fig 4 (b) The triangle model has less pitching moment initially but increases drastically with increase of velocity and reaches a value higher than all the remaining models. Taper model has less pitching moment than all the models at all velocities. Square steps with filleted corner model posses less pitching moment than the model with sharp corners because it enables a smooth flow over the edges which causes less effect of flow at corners. In fig 4(c) The pitching moment for triangle model is less compared to the remaining models. Square with sharp corners posses more pitching moment than remaining models. Unlike as at 90^0 taper model posses more amount of pitching moment than filleted model. Among all the models of sqaure shape steps with filleted corners posses less pitching moment at all velocities of air. In fig 4 (d) Triangle has very less pitching moment than remaining models. Taper has less pitching moment over all the square models at all velocities in this orientation. After observing all the orientation results the triangle model has very less pitching moment than all the models. But among the square models the square taper model posses less pitching moment at almost all the orientations and all the velocities.



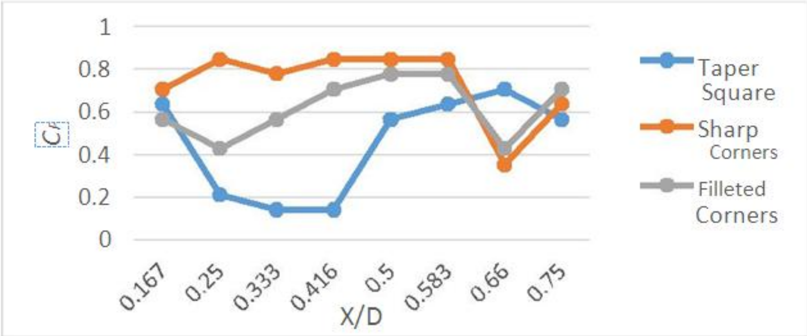
(c)

Fig 5. Variation of Co – Efficient of Pressure on the surface of the triangle model at different angles

The plots shown in fig 5. (a), (b), (c) 11 shows the variation of co-efficient of pressure on the surface of the wall of triangle model at different angles with respect to different velocities. The pressure is constant over the surface at 90^0 & 180^0, but at 270^0 the pressure varies because of the recirculation zone created behind the model. These pressure values of triangle model are higher than all the remaining models which has more tendency of lifting of walls which causes the damage of building. As velocity increases the air moves faster without proper contact to the wall hence as the velocity increases the pressure co-efficient decreases.



(a) (b)



(c)

Fig. 6 Variation of Co – Efficient of Pressure on wall surface of different models at different angles to the flow direction at 15 m/s velocity.

The plots shown in fig 6 (a), (b), (c) shows the variation of co-efficient pressure on the wall surface at an angle of 90^0, 180^0 & 270^0 to the flow direction at a velocity of 15 m/s. The pressure variation is following almost same scenario in all the direction for all the models but for taper square model it is almost same in all the direction. The recirculation zone is formed at 0.416 X/D location for Taper Square model hence the pressure drop is less at that location and then it increases and maintains constantly at 90^0 to the flow, but in the remaining cases after the recirculation zone also the pressure varies. For Square model with sharp corners has some higher pressure intensity on its wall surface at all directions. The recirculation zone forms at 0.66 X/D for this model and then the pressure increases. For the model square steps with filleted corners model the pressure intensity higher than all the remaining square models, recirculation zone is formed at 0.66 X/D and then the pressure increases.The maximum pressure co-efficient at 90^0 for the taper model at 15 m/s velocity is 0.429 it is same for sharp corner model but for filleted model it is very high up to a value of 0.920, at 180^0 angle for taper it is 0.429, for sharp corner model 0.566 and for filleted corner model it is 0.637, at 270^0 for taper 0.709, sharp corners model 0.779 and for filleted corner model it is 0.849 But among all the models taper square models has less pressure intensity on walls and square model with filleted corners has high pressure intensity. Hence at this velocity Taper model has less chance of damage in all the directions.

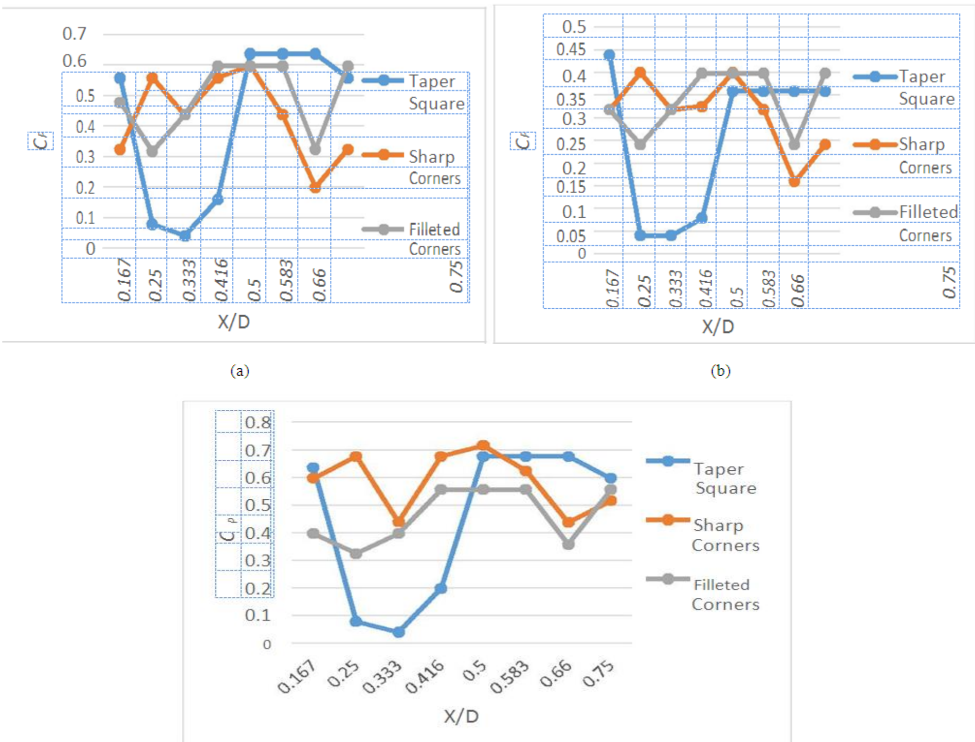


Fig. 7 Variation of Co – Efficient of Pressure on wall surface of different models at Different angles to the flow direction at 20 m/s velocity.

The plots shown in fig 7 (a), (b), (c) shows the variation of co-efficient pressure on the wall surface at an angle of 90^0, 180^0 & 270^0 to the flow direction at a velocity of 20 m/s. The pressure acting on taper square model is less compared to all the remaining models. The recirculation zone is formed at a distance of 0.33 X/D from the starting edge then the pressure raises and maintain almost constant in the all the directions. The pressure variation is almost following same scenario in all directions and for all models. The recirculation zone for square steps with sharp corners and filleted corners is formed at 0.66 X/D from the starting edge. Both of these models have almost equal values in 90^0, 180^0 but at 270^0 filleted model has less pressure co-efficient than the sharp corners model. For filleted model in all the direction the pressure co-efficient decreases at 0.33 X/D and then increases up to 0.416 X/D and maintain constant till 0.66 X/D here the pressure drop occurs due to the formation of vacuum by recirculation zone. Shape corner model has pressure fluctuations at all locations but major pressure drop is caused at 0.66 X/D because of recirculation zone. However the taper square model has less pressure intensity at 20 m/s velocity in the all directions.

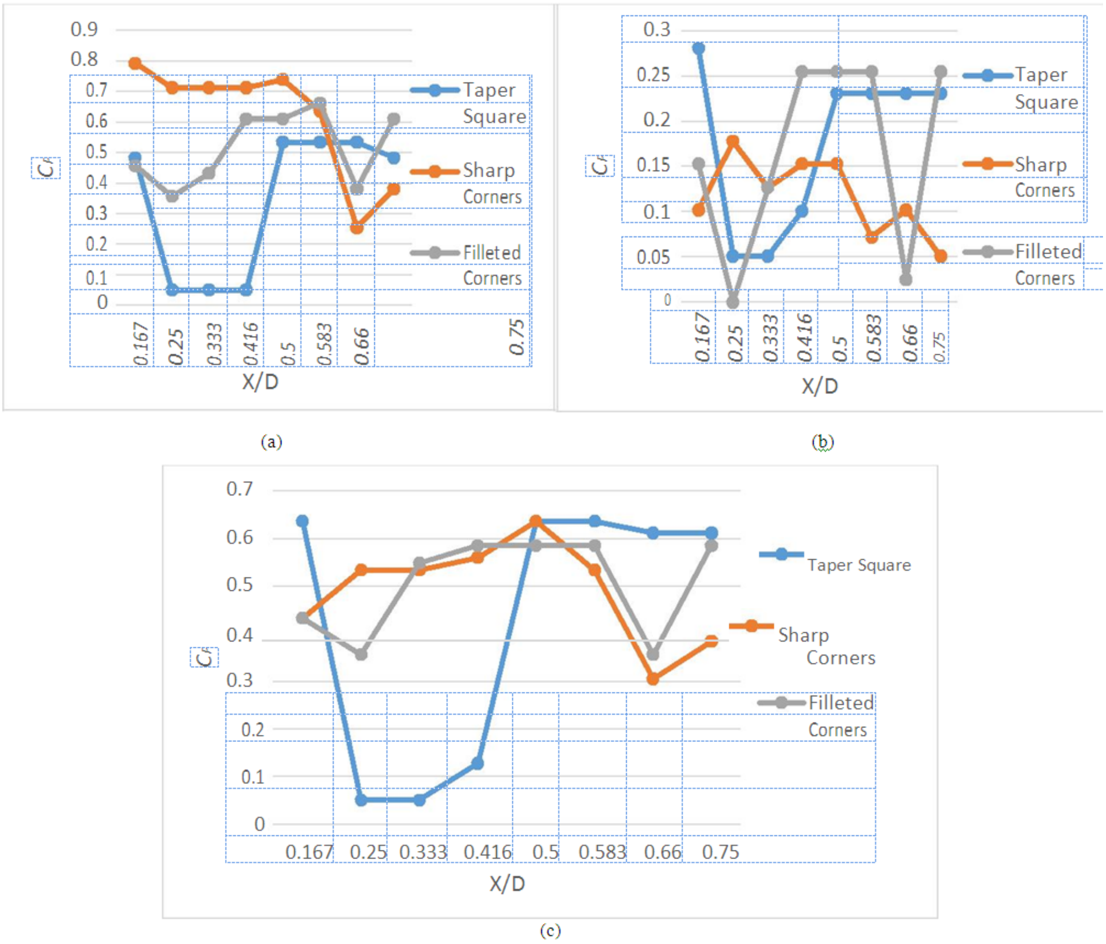


Fig. 8 Variation of Co – Efficient of Pressure on wall surface of different models at different angles to the flow direction at 25 m/s velocity.

The plots shown in fig 8 (a), (b), (c) shows the variation of co-efficient pressure on the wall surface at an angle of 90^0, 180^0 & 270^0 to the flow direction at a velocity of 25 m/s. The pressure acting on taper square model is less compared to all the remaining models. The taper square model follows same scenario at all the angles in all angles. Recirculation zone is formed in between 0.25 X/D to 0.33 X/D from the starting edge hence the pressure here decreases drastically and then increases and constant throughout the remaining section. For the remaining models the pressure does not following any scenario it changes with direction they both have recirculation zone at 0.66 X/D. At this velocity also in all the directions the taper square model has less pressure intensity on the wall surface. From the analysis of all models at different velocities at all angles the co-efficient of pressure is less for Taper Square model than all the remaining including triangle model also.

IV. Conclusion

In this work the experiments are conducted on four models Triangular, Taper Square, Square steps with sharp corners and Square steps with filleted corners to find the drag, pitching moment and co – efficient of pressures on the surface of the models at different angles to flow.

Drag is less for triangle model in the all directions expect at 90^0 to the flow direction. In this direction as the velocity increases the drag increase than the remaining models. The drag for all models is increases with respect to velocity.

Among the models of Square cross- section the drag is more for the model square steps with sharp corners. The remaining two models have less drag than this model and maintain just a small difference among these two at all velocities and in all directions.

The pitching moment is also less for triangle except in the direction 90^0 to the flow. Taper model has less pitching moment than all the square models in all directions and at almost all the velocities. Here the filleted corner model has more pitching moment than the taper model. The sharp corner model has very high pitching moment than all the remaining models.

The pressure co – efficient is high for triangle model than all the remaining models. Taper has less pressure co-efficient on the surface than the remaining two square models. Filleted corner model has high pressure co efficient than all the square models. Hence Taper Square model has less tendency of wall collapse during the heavy winds of a cyclone.

References

[1] Y.Tamura et al (2010), Aerodynamic characteristics of tall building models with various unconventional configurations, Proceedings of the 2010 Structures Congress.

[2]. J-A., Amin, A-.K.,Ahuja, 2008, Experimental study of wind pressures on irregular plan shape buildings, BBAA VI, 20-24.

[3]. Y.C., Kim, J., Kanda, 2010, Characteristics of aerodynamic force and pressures on square plan buildings with height variations, Journal of Wind Engineering and Industrial Aerodynamics 98, 449-465.

[4]. Hayashida, H.,Iwasa,Y.,1990.Aerodynamicshapeeffectsoftallbuildingforvortex induced vibration. Journal of Wind Engineering and Industrial Aerodynamics 33,237–242.

[5]. N., Lin, C., Letchford, Y., Tamura, B., Liang, O., Nakamura, 2005, Characteristics of wind forces acting on tall buildings, Journal of Wind Engineering and Industrial Aerodynamics 93, 217-242.

[6]. Y.C., Kim, J., Kanda, 2010, Characteristics of aerodynamic force and pressures on square plan buildings with height variations, Journal of Wind Engineering and Industrial Aerodynamics 98, 449-465.

[7]. Chien, C.W., Jang, J.J., Li, Y.C. (2010), "Wind-resistant design of high mast structures", Journal of the Chinese Institute of Engineers 33 (4), 597-615.

[8]. Tang, J.W., Xie, Y.M., Felicetti, P., Tu, J.Y., Li, J.D. (2013), "Numerical simulations of wind drags on straight and twisted polygonal buildings", The Structural Design of Tall and Special Buildings 22, 62-73.

[9]. Miyashita, K., Katagiri, J., Nakamura, O., Ohkuma, T., Tamura, Y., Itoh, M., Mimachi, T. (1993), "Wind-induced response of high-rise buildings", Journal of Wind Engineering and Industrial Aerodynamics, vol. 50, pp.319-328.

[10]. Experimental investigation on aerodynamic characteristics of various triangular section high-rise buildings Eswara Kumar Bandi, Yukio Tamura , Akihito Yoshida, Yong Chul Kim, Qingshan Yang.