

Effect of Aerodynamic Modification on ‘V’ plan Shaped Tall Building Under Wind Excitation

Satadru Bhattacharya ^{1,*}, Sujit Kumar Dalui ²

¹ Research Scholar, Department of Civil Engineering, Indian Institute of Engineering Science Technology, Shibpur, Howrah, India,

² Assistant Professor, Department of Civil Engineering, Indian Institute of Engineering Science Technology, Shibpur, Howrah, India

Paper ID - 140215

Abstract

Advanced high-rise buildings with peculiar cross-sectional plan is the major challenging concern to all structural engineers as well as researchers due to variation in dynamic wind response. Irregular shaped tall buildings are vulnerable to wind force due to height and nonuniformity of the structure. In the present study, the principal objective is to investigate the significant role of the aerodynamic modification of the building model to reduce the aerodynamic effect of wind force on ‘V’ plan shaped high-rise building and to modify aerodynamic design criteria accordingly. Various cases have been made by considering different local modification like corner chamfered and corner rounded of ‘V’ plan shaped building model. The angle between the limbs is 90° which remain unchanged. The percentage is gradually increased from 5% to 20% of the total plan area with 5% regular increment for both chamfered and rounded corner. Wind incident angle is increased from 0° to 90° with 30° regular interval for each case. Computational Fluid Dynamics (CFD) is the basis of method to perform the numerical simulation of ‘V’ plan shaped building with local modifications such as corner chamfered, corner rounded which is similar wind environment as in urban terrain. Grid convergence study is performed to improve the accuracy of result by adopting very finer meshing of Computational Domain. Pressure coefficient of each face, force coefficient, velocity variation, pressure variation on each face is obtained by numerical analysis. Further, a comparison has been made with basic ‘V’ plan shaped tall building model without any modification to study the effectiveness of aerodynamic modification on wind-induced response of ‘V’ plan shaped building exposed to different wind incidence angle and observations have been made on the suitability of aerodynamic modification based on the numerical result.

Keywords: Aerodynamic Modification, Computational Fluid Dynamics (CFD), Irregular shaped tall buildings, local modification, wind-induced response.

1. Introduction

Advanced design of extremely tall and irregular structure which is exposed to extreme wind environment is the challenging concern of structural designers and researchers due to height and nonuniformity. Researchers are serious about the improvement of structural as well as architectural aspects by incorporating different types of shapes, sizes, aspect ratio etc and light in weight on the other side. Some essential global and local modification factors like aspect ratio, wind angle, taper, setback, chamfer, corner modification to reduce the force and pressure co-efficient of these high rise structures are considered by the Previous researchers. Shape modification of the building like corner modification, setback, tapering, opening etc. helps to reduce the wind load. Further, they incorporated wind tunnel test to find out the exact effect of wind load from different wind incident angle. Different types of geometrical shape like +, T, U, C, Y, H, E, L, O etc. are taken into consideration for study. Tapering of the sides of building and corner modification of the edges are very significant to reduce the aerodynamic forces using experimental setup in wind tunnel. **Gomes et al.** (2005) studied the result of the pressure

force and pressure co-efficient among all the modification. Among those unconventional shapes of building, V-shape building is studied by a very few researchers. In addition to that, IS 875 (part - III): 2016 has not mentioned any special criteria for this type of irregular shapes of building. So, for present study, V-plan shaped building with different corner modification is considered to be studied rigorously.

Previous researchers have explored response due to various global & local modification of irregular plan shaped building model exposed to wind force. **Jain and Mandal** (1992) have investigated the response of the dynamic Analysis of multi-storey V-shaped building with setback arrangement. **Stathopoulos and Zhou** (1994) observed the numerical prediction of turbulent wind pressure on the surfaces of L shaped building due to various wind direction using the k-ε turbulence model and cartesian rectangular grid generation. **Tamura and Miyagi** (1999) studied the effect of corner modification (sharp, rounded, chamfered) on

distribution of irregular plan shapes on wind tunnel test. Experimental models used in this test are L shaped, U shaped

*Corresponding author. Tel: +919046953650; E-mail address: satadru.civil2016@gmail.com

and cube-shaped model with 1:100 ratio. Further Computational Fluid Dynamics (CFD) is incorporated using a software PHOENICS to compare pressure variation in each face and wind flow pattern with wind tunnel test results considering various wind angle. **Kwok and Bailey (2006)** investigated the effect of various aerodynamic devices such as slotted corner, small fins on the wind induced response of high-rise building. **Kim et al. (2008)** have investigated the tapering effect on reducing the rms across-wind displacement responses of a high-rise building using aeroelastic model tested in wind tunnel. Three tapered model with 5% ,10% and 15% tapering ratio and one square model without any tapering were considered. **Gabbai and Simiu (2009)** have investigated a novel methodology for estimating the aerodynamic damping of an isolated rectangular plan shaped tall building. **Mara & Case (2010)** have studied the variation of overall building response with the corner modifications due to wind induced motion. They have considered corner cuts, chamfers, venting and fins as corner modification to reduce the wind force on building. **Tamura et al. (2010)** have explored the aerodynamic force and pressures acting on 31 tall building model with various configurations like rectangular, square, elliptical, with corner modification, with setbacks, helical, with opening and so on. **Deng et al., (2015)** investigated the effect of global and local strategies such as chamfered modification, opening slots on aerodynamic forces and wind induced responses by a series of wind tunnel tests which were conducted on tapered super tall building with square cross section by applying simultaneous pressure measurement technology. **Li and Li (2016)** have developed the optimal design model of an L shaped building using Kuhn - tucker conditions and validated the model with wind tunnel test results. **Elias, Matsagar and Datta (2017)** have studied multi-mode wind response control of chimneys with SSI consideration using distributed multiple tuned mass dampers (d-MTMDs) and compared with single tuned mass damper (STMDs) , arbitrarily installed distributed MTMDs (ad-MTMDs) to find out best effective mass damper and d-MTMDs gives best result among all the dampers. **Kumar and Dalui (2017)** have studied the variation of pressure distribution due to variation of internal angle between limbs of a regular cross plan shaped building for various wind direction (0° to 180°) using Computational Fluid Dynamics (CFD) with ANSYS CFX (k- ϵ turbulence model). Comparison has been made for numerical CFD simulation result and wind tunnel test results for the same. **Bhattacharyya and Dalui (2018)** explored wind flow pattern and mean pressure coefficient of E plan shaped building using wind tunnel test and CFD simulations with the help of ANSYS software. Wind incident angles are varied from 0° to 180° with 30° regular interval. **Elias and Matsagar (2018)** have investigated the effectiveness of multiple tuned mass dampers (MTMDs) to reduce the wind force of a tall building considering soil structure interaction (SSI) and compared the response with single tuned mass damper (STMDs) to find out suitable damper among both. **Baghaei Daemei et al. (2019)** have studied the effect of major (setback, helical, tapered) and minor (such as corner cut, chamfered, recessed, rounded) aerodynamic modifications for triangular building using CFD simulation.

2. Methodology of Work

2.1. Problem Formulation

In this study, a computational method is adopted to perform analysis of wind induced force acting on this type of irregular building. ANSYS CFX is used to create Model and perform fine meshing. Height of the V-shaped model is 180 m; both limbs of the model are 90m long with 30 m width. In the present study, 1:300 scale is considered to model (Fig. 1) the building. Various cases have been made by considering different local modification of corner cut like corner chamfered and corner rounded of ‘V’ plan shaped building model. The angle between the limbs is 90° which remain unchanged. Here corner modification increment is denoted by

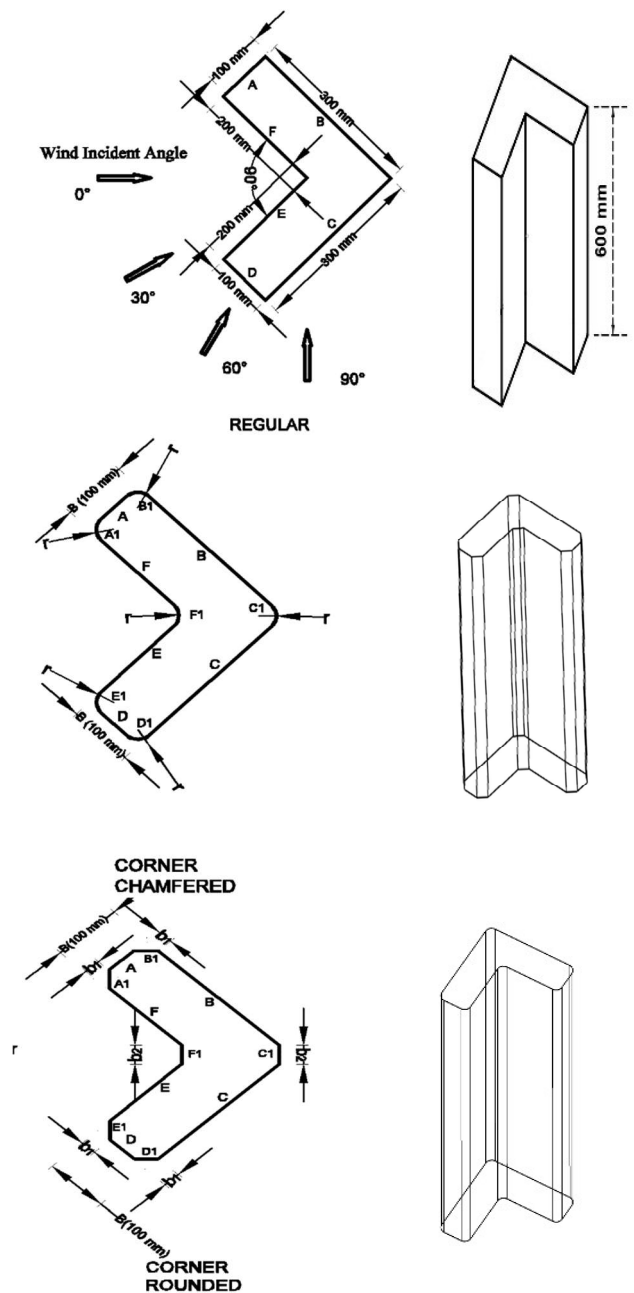


Fig. 1: Details of Corner modified V plan shaped model

Table 1: Input Parameter Details

Model name	Corner Cut length (b in mm) or radius (r in mm)	Corner cut Ratio $\mu_c = b/B$ ($2b1/B = r/B$)	Limb Angle (θ)	Wind Angle (β)
0% chamfer/rounded	0	0%	90°	0°, 30°, 60° & 90°
5% chamfer	5	5%	90°	
10% chamfer	10	10%	90°	
15% chamfer	15	15%	90°	
20% chamfer	20	20%	90°	
5% rounded	5	5%	90°	
10% rounded	10	10%	90°	
15% rounded	15	15%	90°	
20% rounded	20	20%	90°	

corner cut ratio (μ_c) which is the ratio of cutting width and total width of the limb. The percentage of corner cut ratio ($\mu_c = b/B$) is gradually increased from 5% to 20% with 5% regular increment for both chamfered and rounded corner. Wind incident angle is increased from 0° to 90° with 30° regular interval for each case. Modelling details with corner cut ratio is given in Table 1.

2.2 Solution Methodology

Computational Fluid Dynamics is the principle tool based on Navier-Stokes equation which is used in the simulation of the model. k-ε turbulence model which is based Reynolds Averaged Navier Stokes (RANS) equation is chosen among all the turbulence model to perform simulation in ANSYS CFX. Previously most of the researchers had performed analysis using wind tunnel test of the aero-elastic model but nowadays, the CFD method is being used widely by various researcher. Full-scale model is time and resource-consuming so length scale is reduced to 1:300 which is adopted for all the cases of corner modified V plan shaped building model. The wind velocity profile is generated by power law equation and basic wind speed is reduced to 1:5 scale. So, inlet velocity is considered as 10m/s.

Among all those RANS model, the standard k-ε model is widely used to solve computational fluid mechanics problems. Gradient diffusion hypothesis is adopted to relate the Reynolds stresses to the mean velocity gradients and the turbulent viscosity in k-ε turbulence models. The turbulent viscosity is product of turbulent length scale and turbulent velocity. The turbulent kinetic energy, k is defined as the variance of the fluctuation in velocity. ε is the turbulence eddy dissipation and has dimension per unit time.

According to the power law, the Wind velocity profile is generated by using the formula.

$$U_z = U_0 * \left(\frac{z}{z_0}\right)^\alpha \tag{1}$$

Where U_z is wind velocity at certain height Z, U_0 is boundary layer wind velocity, Z_0 is boundary layer depth and value of α is taken as 0.133 for open terrain with well obstruction. Inlet velocity is considered as 10m/s which is according to 1:5

scale (Basic Wind speed 50m/s) as the building model is considered in 1:300 length scale. Fig. 2 is mean wind velocity and turbulence intensity profile which shows very good agreement between the theoretical and analytical value.

3. Numerical Study

The meshing of the entire computational domain is required to maintain the accuracy and similarity with wind tunnel study. The computational Domain covers 5H in upstream, 5H from each side wall, 5H from top surface of the building model and 15H in downstream from the edges of the building model according to the recommendation of Franke et al. (2004) where H is the height of the building. Surfaces of the building are considered as no-slip condition and walls of the domain are considered as free slip condition. Outlet pressure is 0 pa.

The result depends on the fineness of the grid arrangement of the domain. The element size is arranged such that the element size is smaller near the building region and gradually increases to the wall of the domain to generate the inflation. Inflation is given to obtain more accurate response from the simulation.

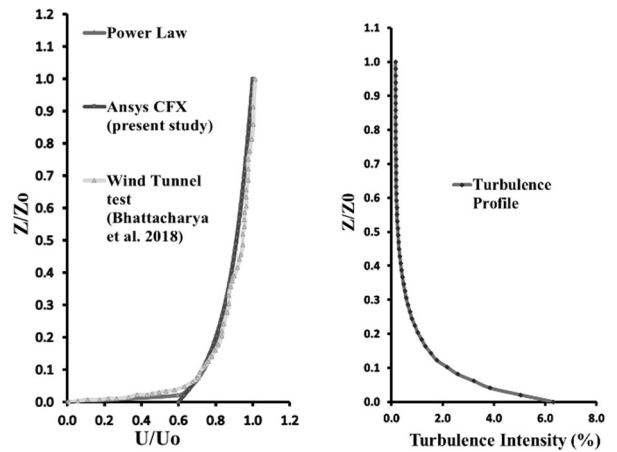


Fig.2 Mean Wind Velocity profile and Turbulence Intensity profile

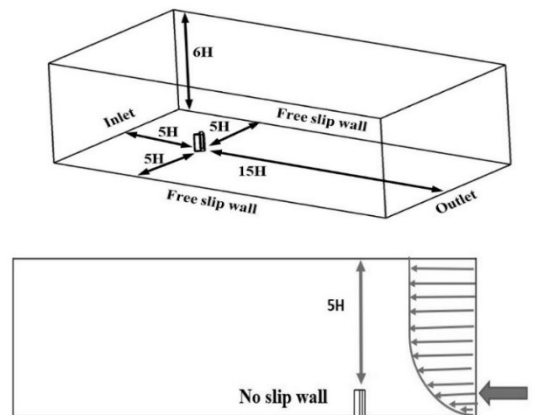


Figure. 3: Computational Domain of V shaped building model (a) Plan view (b) Elevation

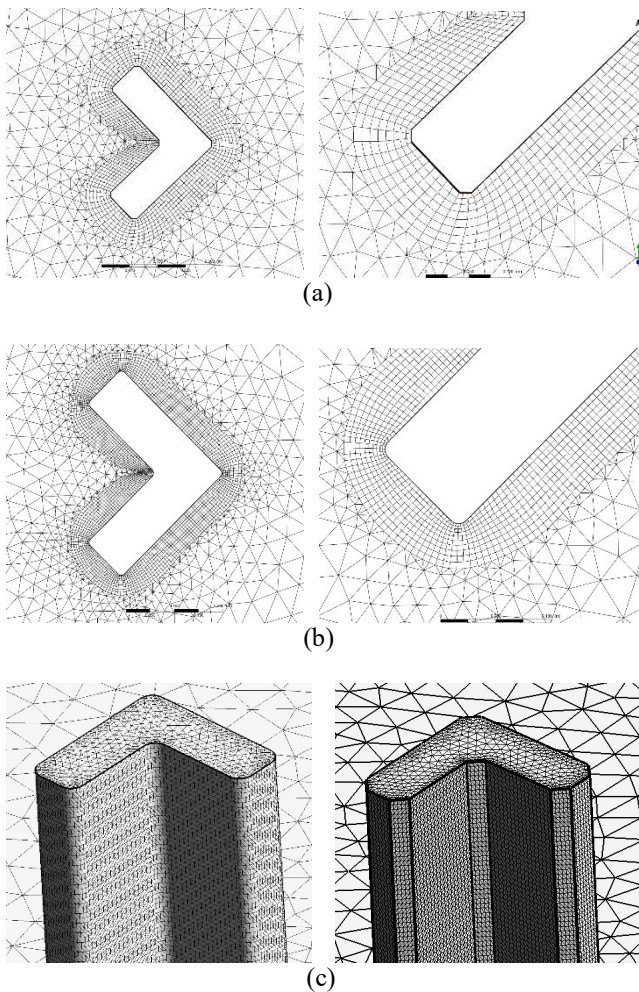


Fig. 4: Meshing arrangement of corner modified models (a) corner chamfered (b) corner rounded (c) Isometric view

3.1 Grid Sensitivity Study

A grid sensitivity study is an essential and major study to achieve more accuracy in the response obtained from the entire work. This study is based on finer grid arrangement of the computational domain until the outcome such that means pressure coefficient on each face, force coefficient etc converges with null error percentage. The result details of grid sensitivity study are given in the Table 2

Table 2: Grid Sensitivity Test Result

Model No	Model Name	Element number	Drag coefficient (C _D)	Error %
1	case 1	4599288	0.951	2.65
2	case 2	7827509	0.906	-2.12
3	case 3	10366750	0.959	3.50
4	case 4	12546014	0.961	3.72
5	case 5	15896489	0.958	3.40
6	case 6	17150303	0.952	2.76
7	case 7	20241094	0.941	1.59
8	case 8	22117942	0.932	0.64
9	case 9	27078552	0.926	----
10	case 10	33019015	0.926	----

It can be clearly observed from the study that, if meshing is finer then number of elements of meshing increases, then error percentage is found decreased so to reduce the error percentage in analysis this test should be performed. When the result of response is converged with the previous value, then tetrahedron (Cheng et al. 2011) and hexahedron size is considered as relevant which is used in further investigation of the corner modification.

4. Results and Discussion

4.1 Validation with previous work

Before proceeding to analysis of the present study, Validation of response obtained by previous researcher is performed with the experimental and numerical results obtained by Gomes et al. (2005). They explored the response of L shaped building with constant limb angle 90° in different wind incident angle from 0° to 180°. L shaped building acts as V shape when wind incident angle is 45° so in the present study, analysis of model maintaining same dimension as Gomes et al. (2005) has been performed using software package based on k-ε method to check the practicability of the software package. Inlet velocity in considered 10m/s. It is given in Fig. 5 which shows good agreement with numerical and experimental data.

4.2 Flow Pattern Variation Study

4.2.1 Wind Flow Pattern on The V Plan Shaped Tall Building with Various internal limb Angle

The flow separates at the edge of the corner portion of the model so velocity is high at this region due to flow separation. For corner chamfer or rounded faces, this velocity is extremely high due to side wash of flow. For rounded corner it is maximum and increases with corner cut ratio which can be observed in Fig. 6.

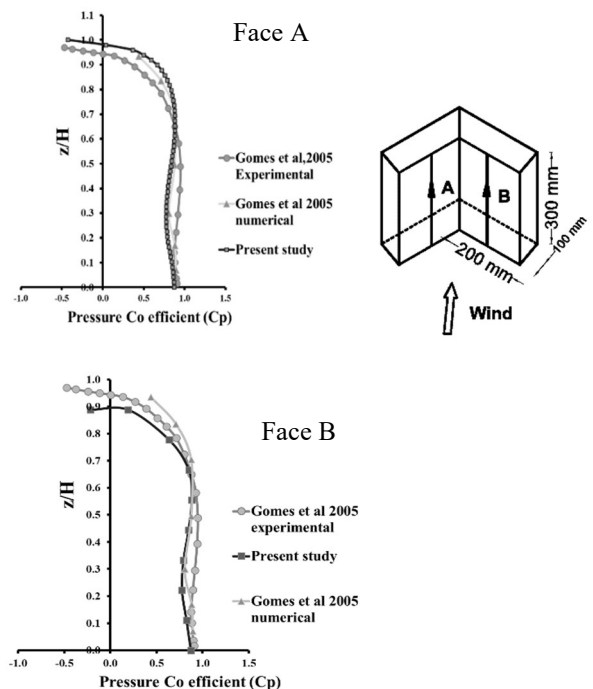


Fig. 5: Validation work of present study

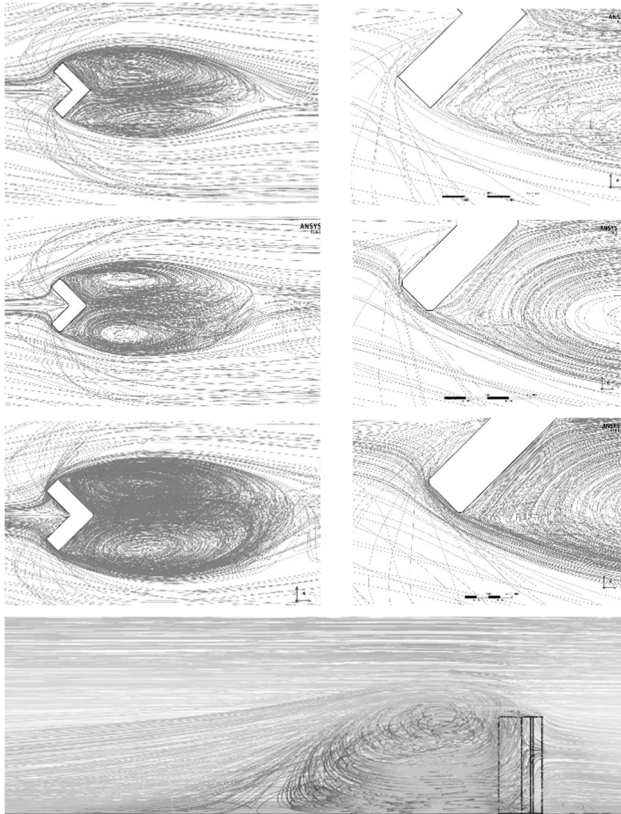


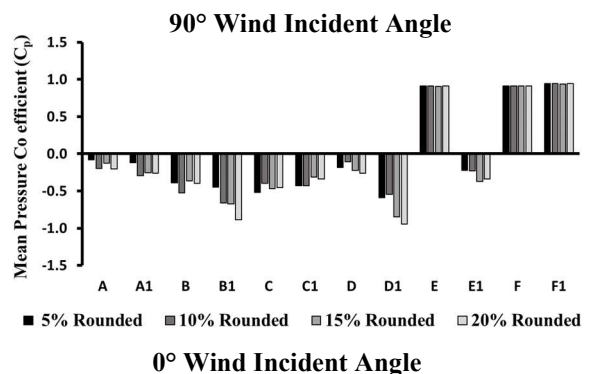
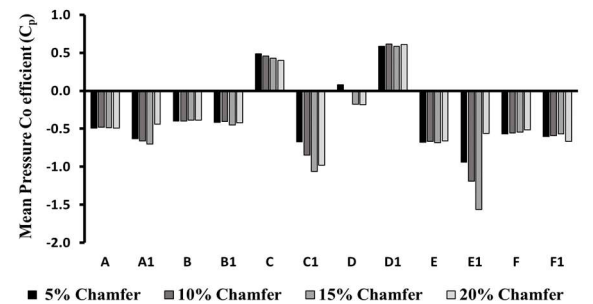
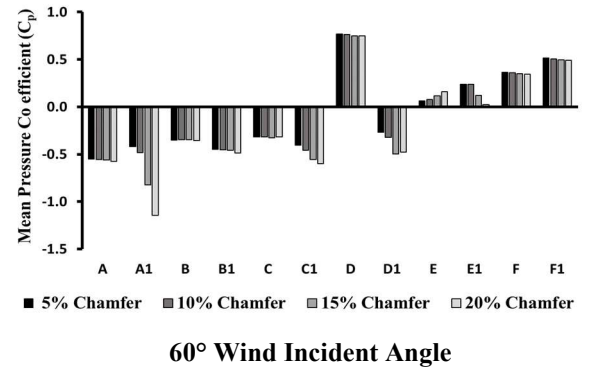
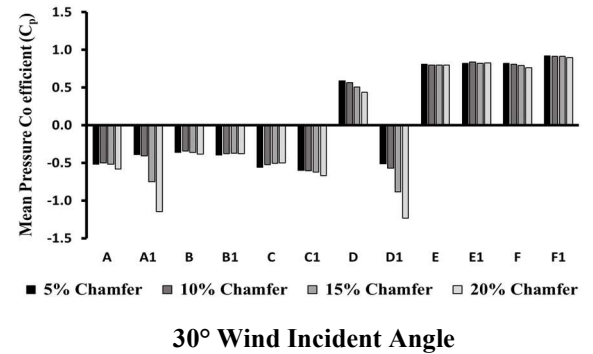
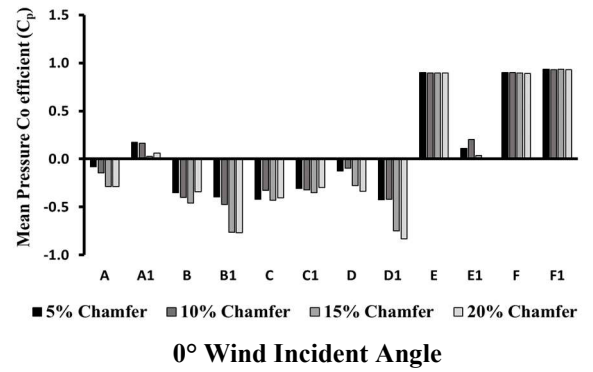
Fig. 6: Wind flow pattern of regular, corner chamfer and corner rounded V plan shaped model

At 0° wind incidence angle, vortices are almost symmetrical and even in the wake region of building model so pressure distribution is almost identical on the symmetrical faces. At 30°, 60°, 90° wind incidence angle, vortices are unsymmetrical and nonuniform in size in the wake region of building model so pressure distribution is not identical on the symmetrical faces.

4.3 Pressure Variation Study

4.3.1 Comparison of pressure coefficient of each faces in various corner cut ratio at different wind incident angle

Mean Pressure co efficient is obtained for each faces of V plan shaped building with different corner cut ratio with wind angle variation up to 90° which shows in Fig. 7. At 0°wind incident angle, for frontal identical faces, it is clearly seen that Mean Pressure co efficient is almost identical due to even eddies formation. With wind angle increment, it is no longer identical due to unsymmetrical vortex formation. Most of the faces experience pressure increment with the increment in corner cut ratio as it increases the local pressure at the region of flow separation. B1 and D1 face is critical for 0°wind incident angle. A1 and D1 face is critical for 30°wind incident angle. A1 face is critical for 60°wind incident angle & C1 and E1 face is critical for 90°wind incident angle For rounded corner cases, pressure increment is more with the increment in corner cut ratio as it increases the local pressure at the region of flow separation. Here also B1 and D1 face is critical for 0°wind incident angle. A1 and D1 face is critical for 30°wind incident angle. A1 face is critical for 60°wind incident angle & C1 and E1 face is critical for 90°wind incident angle



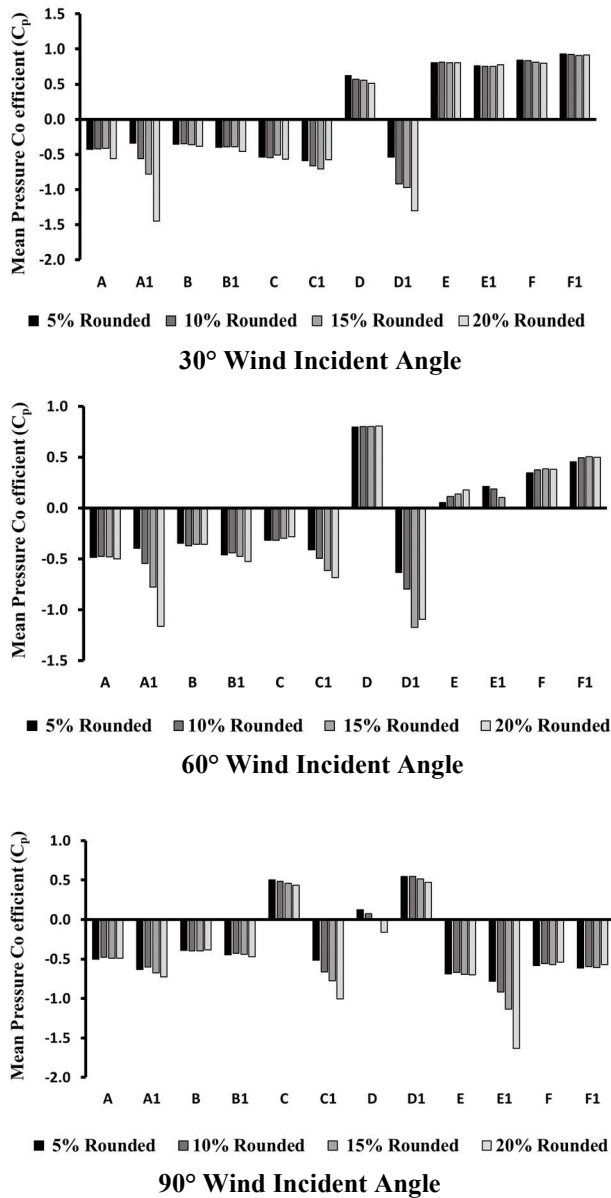


Fig. 7: Comparison of pressure coefficient of each faces in various corner cut ratio at different wind incident angle

4.3 Force Coefficient variation study

4.3.1 Comparison of Drag and Lift coefficient between various corner cut ratio at different wind incident angle

Drag coefficient and Lift Coefficient is obtained for each cases of V plan shaped building with different corner cut ratio with wind angle variation up to 90° which shows in Fig. 8.

For chamfered corner, at 0° wind angle, drag coefficient is maximum for corner cut ratio 0%. At 0°, 30°, 60° & 90° wind angle, drag coefficient is gradually decreasing with corner cut ratio increment. At 90° wind angle, also, lift coefficient is maximum for corner cut ratio 0%. At 0°, 30°, 60° & 90° wind angle, lift coefficient is also gradually decreasing with corner cut ratio increment.

For rounded corner, at 0° wind angle, drag coefficient is maximum for corner cut ratio 0%. At 0°, 30°, 60° & 90° wind angle, drag coefficient is gradually decreasing with corner cut ratio increment. At 90° wind

angle, also, lift coefficient is maximum for corner cut ratio 0%. At 0°, 30°, 60° & 90° wind angle, lift coefficient is also gradually decreasing with corner cut ratio increment. Force reduction is more for rounded corner than chamfered corner due to the size reduction of eddies in the wake region. For regular building eddies sizes are maximum.

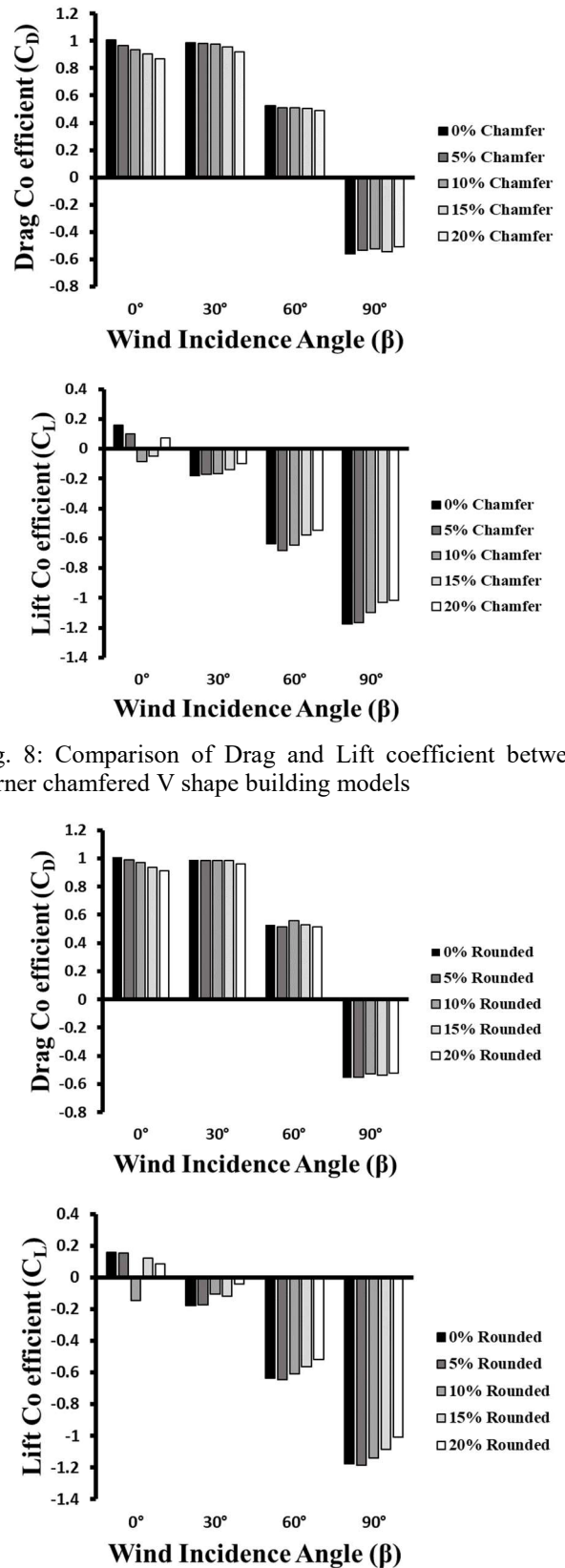


Fig. 8: Comparison of Drag and Lift coefficient between corner chamfered V shape building models

Fig. 9: Comparison of Drag and Lift coefficient between various corner rounded V shape building models

5. Conclusions

The wind induced response is obtained for V plan shaped building with different corner modification such as chamfered and rounded due to 0°, 30°, 60° & 90° wind incidence angle. Before proceeding to the analysis, a validation work from the previous study has been done to check the eligibility and practicability of the method. Based on the results obtained from the analysis of the wind-induced load on V plan shaped building different corner modification due to various wind incident angle, it is observed that, for corner chamfer or rounded faces, the flow velocity is extremely high at edges of corner region due to side wash of flow. For rounded corner it is maximum and increases with corner cut ratio. At 0° wind incidence angle pressure distribution is almost identical on the symmetrical faces due to symmetrical and uniform vortices formation. Irregularity in vortex formation occurs with the change of wind incidence angle and flow separation is affected by the wind directionality and corner modification. From pressure variation study, most of the faces experience pressure increment with the increment in corner cut ratio as it increases the local pressure at the region of flow separation. Similarly, for rounded corner cases, local pressure increases with the increment in corner cut ratio and more than corner chamfered cases as the velocity is more than chamfered cases at the edges of corner region.

It is observed from force coefficient variation study that, force coefficient reduces with the increment of corner cut

Deng, T., Yu, X. and Xie, Z. (2015) 'Aerodynamic measurements of across-wind loads and responses of tapered super high-rise buildings', 21(3), pp. 331–352.

Elias, S. and Matsagar, V. (2018) 'Wind response control of tall buildings with a tuned mass damper', *Journal of Building Engineering*, 15, pp. 51–60. doi: 10.1016/j.job.2017.11.005.

Elias, S., Matsagar, V. and Datta, T. K. (2017) 'Distributed Multiple Tuned Mass Dampers for Wind Response Control of Chimney with Flexible Foundation', *Procedia Engineering*. Elsevier B.V., 199, pp. 1641–1646. doi: 10.1016/j.proeng.2017.09.087.

Gabbai, R. D. and Simiu, E. (2009) 'Aerodynamic Damping in the Along-Wind Response of Tall Buildings', *Journal of Structural Engineering*, 136(1), pp. 117–119. doi: 10.1061/(asce)0733-9445(2010)136:1(117).

Gomes, A. et al. (2005) 'Experimental and numerical study of wind pressures on irregular-plan shapes', 93, pp. 741–756. doi: 10.1016/j.jweia.2005.08.008.

Hirsch, C. and Krüs, H. W. (2004) 'RECOMMENDATIONS ON THE USE OF CFD IN PREDICTING PEDESTRIAN WIND ENVIRONMENT', pp. 1–12.

Jain, B. S. K. and Mandal, U. K. (1992) 'Dynamics of buildings with v-shaped plan', 118(6), pp. 1093–1112.

Kim, Y., You, K. and Ko, N. (2008) 'Across-wind responses of an aeroelastic tapered tall building', 96, pp. 1307–1319. doi:

ratio for both type of corner modification. For regular building, eddies sizes are maximum so force coefficient is also maximum among all cases. Force reduction is more for rounded corner than chamfered corner due to the size reduction of eddies in the wake region and smooth separation of wind flow.

So, corner modification is beneficial specially rounded corner to reduce the aerodynamic force acting on a V plan shaped tall building but researchers and structural designers should give special attention for cladding design due to local pressure increment.

Disclosures

Free Access to this article is sponsored by SARL ALPHA CRISTO INDUSTRIAL.

References

Baghaei Daemei, A. et al. (2019) 'Study on wind aerodynamic and flow characteristics of triangular-shaped tall buildings and CFD simulation in order to assess drag coefficient', *Ain Shams Engineering Journal*. Ain Shams University, (xxxx). doi: 10.1016/j.asej.2018.08.008.

Bhattacharyya, B. and Dalui, S. K. (2018) 'Investigation of mean wind pressures on "E" plan shaped tall building', 26(2), pp. 99–114.

Cheng, C. K. C. et al. (2011) 'Wind-induced natural ventilation of re-entrant bays in a high-rise building', *Journal of Wind Engineering and Industrial Aerodynamics*. Elsevier, 99(2–3), pp. 79–90. doi:

10.1016/j.jweia.2008.02.038.

Kumar, D. and Dalui, S. K. (2017) 'Effect of internal angles between limbs of cross plan shaped tall building under wind load', 24(2), pp. 95–118.

Kwok, B. K. C. S. and Bailey, P. A. (2006) 'Aerodynamic devices for tall buildings and structures', 113(3), pp. 349–365.

Li, Y. and Li, Q. (2016) 'Across-wind dynamic loads on L-shaped tall buildings', 23(5), pp. 385–403.

Mara, T. G. and Case, P. C. (2010) 'The Effects of Incremental Corner Modifications on a 200m Tall Building', (2008), pp. 2949–2960. doi: 10.1061/41130(369)266.

Stathopoulos, T. and Zhou, Y. S. (1994) 'COMPUTATION OF WIND PRESSURES ON L-SHAPED BUILDINGS', *ASCE*, 119(8), pp. 1526–1541.

Tamura, T. and Miyagi, T. (1999) 'The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes', 83, pp. 135–145.

Tamura, Y. et al. (2010) 'Aerodynamic characteristics of tall building models with various unconventional configurations', *Structures Congress 2010*, pp. 3104–3113. doi: 10.1061/41130(369)278.

Tamura, Y., Tanaka, H. and Ohtake, K. (2010) 'Aerodynamic Characteristics of Tall Building Models with Various Unconventional Configurations', pp. 3104–3113.