Shape Optimization of Corners Having Different Radius of High Rise Building

Shubham Jain[1], Imran Khan[2], MC Paliwal[3]

[1,2]*Student (ME), Department of Civil and Environmental Engineering, NITTTR, Bhopal (M.P.), India*
[3]*Assistant Professor,* Department of Civil and Environmental Engineering, NITTTR, Bhopal (M.P.), India

---***---

Abstract - *The shape of the building and orientation of the building is the main concern of an architect. While the aerodynamic loads is the concern of civil engineer or wind engineer. The aerodynamic forces can be reduce by the changing in shape orientation and shape optimization of structure as well as shape optimization of corners of structure. In this paper, we will see the results that changed by the optimization of shape of corners compare with original or non-modified shape. Aerodynamically shape optimization is the technique, in which shape modified by cutting the corners such as chamfered, roundness of corners or adding some material at the corners. This technique is applicable for high rise building as well as low rise building. But in the previous study, in high rise building, the shape optimization technique is more efficient while in low rise building it is expensive due to requirement of very high skilled person. The analysis is done in the Ansys Fluent which is worked on the principle of fluid dynamics, it is also called Computational Fluid Dynamics (CFD). Now these days, the CFD is highly used in this field, because it is most economical method than conventional methods i.e. wind tunnel test. An overview of the shape optimization aerodynamically namely major modification and minor modification, the minor modification is presented. It is expected that this research ignite the interest in the area of optimization of building.*

Key Words: **CFD, shape optimization, aerodynamic loads, drag force, lift force, vortex shedding.**

1. INTRODUCTION

A high rise building is a Multi Dwelling Unit (MDU) used as a residential, commercial, hotel etc. In these days the culture of construction of high rise building is more because of lack of land in metro cities and popular cities. Now these days high rise building is more popular because of luxurious facilities. In the field of civil engineering, the construction of high rise building is a challenge for structural and geotechnical engineers, especially if high rise building located in a seismically active region or if the soils have high compressibility then the risk factor is high, the serious challenge for the firefighters in high-rise structures during emergency condition. So many parameters are there which work against the high rise building such as wind load, seismic load etc. Wind load is most important parameter in the designing of high rise building, because wind speed increases with the height. At great height, it plays a vital role. Mostly buildings have sharp corners cause wind flow separation by which a strong induced force generates because of wind-structure interaction. But now these days, engineers and the architecture construct tall buildings with different geometries, such as Burj khalifa, Shanghai tower having 828m, 632m respectively. At the great height wind force is concern because it is the most powerful force which is impulse on the face of building.

Wind forces includes three forces drag force, lift force and torsional moment. The drag force is that force which acts in the direction of flow also called along-wind forces. The motion due to along-wind forces is the main concern results from pressure fluctuation on leeward face and windward face. The lift force is that force which acts in the perpendicular direction in the same plan also called cross-wind forces. The common source of cross-wind motion is associated with vortex shedding. Tall buildings are also called bluff bodies as opposed to streamlined that cause flow separation from the surface of the structure, rather than follow the body contour.

2. TECHNIQUES OR METHODS TO ANALYSE THE HIGH RISE BUILDING

2.1 Wind Tunnel Test

A wind tunnel is a mechanism used in aerodynamic research to study the effects of air/wind moving over the structure. A wind tunnel consists of a passage of tubular form with the object under test attached in the middle. Air is flow over the object by a powerful system having fan or other means. The test object, called a wind tunnel model, is instrumented with appropriate sensors to find aerodynamic forces, pressure distribution, or other aerodynamic-related characteristics.

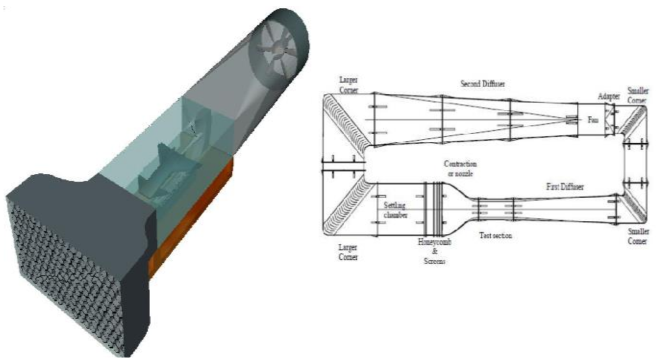

Fig -1: Wind Tunnel Test

2.2 Simulation using Computational Fluid Dynamics (CFD) tool in ANSYS

The advancement of computational fluid dynamics (CFD) modeling on high speed digital computers has shortened the demand of wind tunnel testing. However, CFD results are yet not fully reliable. That's why, we use wind tunnels test to verify CFD predictions. Analysis and design of tall buildings for lateral load such as seismic load and wind load are the major issues which are playing important role in recent decades. This paper represents a recent application of the computational fluid dynamics technique for wind analysis. Conventional wind analysis proposed wind tunnel experiment for appearing at the wind forces for a given structural form.

The modeling of tall building is done in Ansys Workbench and then the mesh file is import to the Ansys fluent for CFD simulation. After analysis in CFD, the wind forces are extracted from Ansys CFX-Post and imported to the structural model in SAP 2000 for the structural analysis.

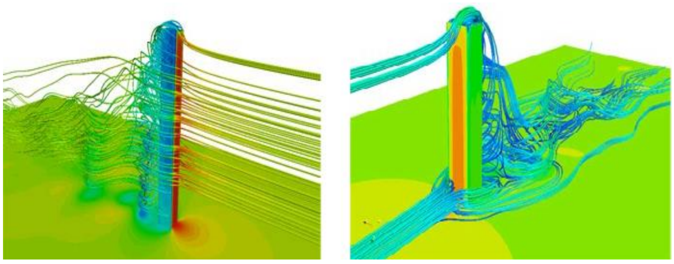

Fig -2: CFD model in Ansys Fluent

3. TYPES OF SHAPE MODIFICATION OF HIGH RISE BUILDING

The aerodynamic modifications have large effects on the structural systems. There are two approaches for reducing the wind loads on building. First approach is "Aerodynamic Mitigation" technique. This method effectively uses simple and new architectural feature to transform the aerodynamic shape of the buildings in order to decrease the wind loads. Aerodynamic modifications assist either by interrupt the formation of strong corner vortices or by splitting the coherent formation of vortices or by rerouting the flow in the separation zone above the roof edge or away from the weakened members. Second strategy to attain reductions in wind-induced forces on buildings is to use "Shape Optimization Aerodynamically" technique. Basically there are two types of modification are as follows;

3.1 Major Modification

In major modification, we modify the shape of building in which includes setbacks along the height, openings at the top, tapering effects, twist the building, etc. This modification has consequential effects on the structural design and architectural design of the building. For instance, varying the shapes of building are curvilinear form, Setback, Tapering, Adding opening and twisting respectively. This modification reduces the wind forces with large value on the building. In addition, we also use minor modification to reduce the vortex formation or vortex shedding.

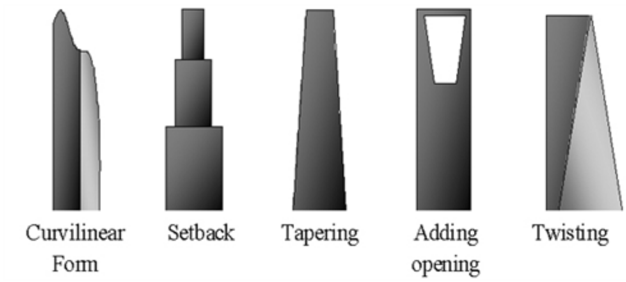

Fig -3: Various major modifications

3.2 Minor Modification

In minor modification, corner modifications are done such as fitting of vented fins, recessed corners (single recession, double recession, and triple recession), slotted corner, chamfered corners, and roundness of the corners and orientation of building is the most important factor to reduce the strong wind force. This modification has minor effects on the overall structural design and architectural design of the building. Most of the building shapes are square or rectangle which causes the strong vortex induced forces on the building experience. These forces (excitation forces) can be reduced via minor modifications to the aerodynamic shape optimization of the high rise building. For instance, modifications in the corners of the cross sectional shapes of the building like chamfered corners, slotted corners, recessed corner, roundness of corners and changing the orientation of the building according to the most frequent strong wind direction, the minor modification methods that can be done to improve the wind performance of high rise building. This method can be very effective in reducing the drag and fluctuating lift forces.

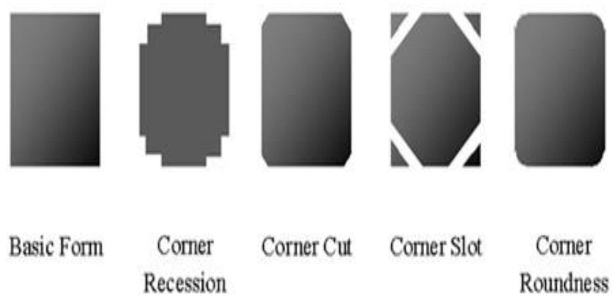

Fig -4: Different shapes of corners of building

In this study, roundness at the corner is taken with different radius such as R=0.00m to R=2.00m having 0.25m interval. There are nine geometries were taken but in this chapter the base model i.e. square shape with sharp corners or square corners and the optimized model i.e. square shape with modified corner having 2.00m radius at the corners were taken. In this chapter, we can see the velocity contour, velocity streamline in the figure. The data of the models were extracted from the software are following:

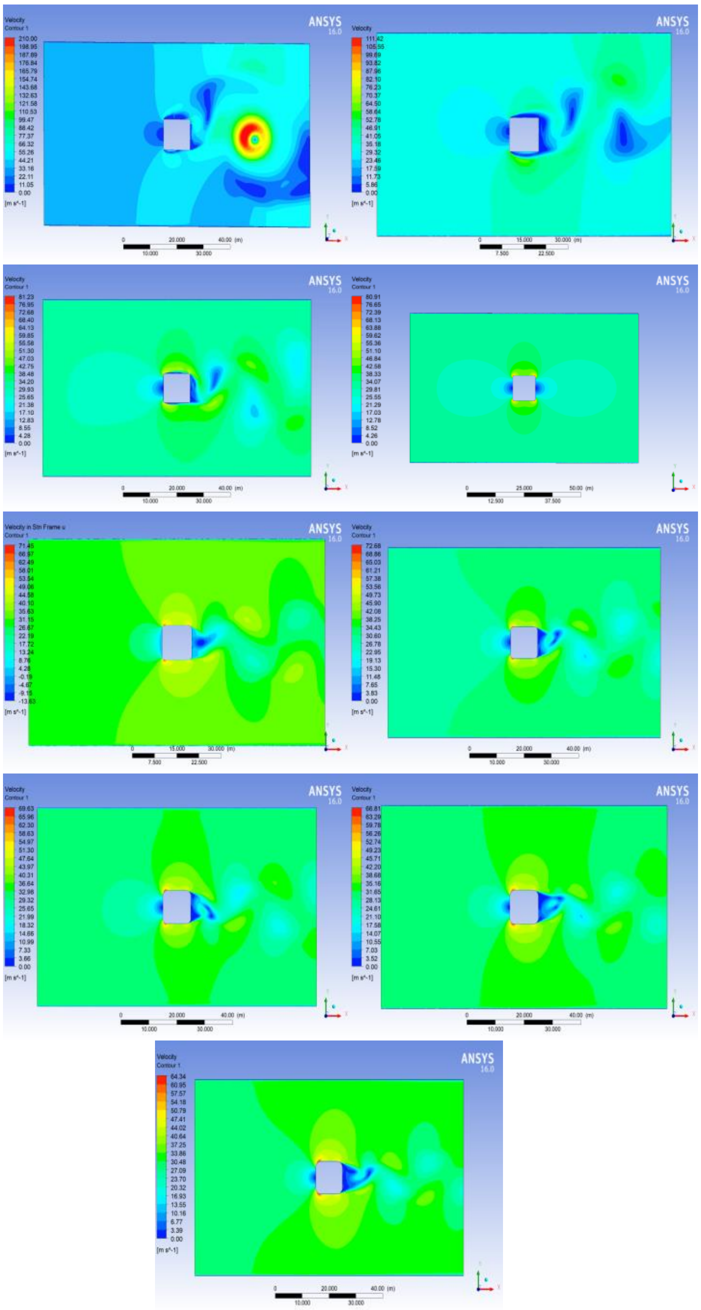

Fig -5: Flow field around different corner radius

The initial velocity of wind is 30 m/sec from velocity inlet. When wind is strike to the building then the flow pattern of the wind flow will change. The velocity of flow is going beyond 200 m/sec. While the velocity of Hurricane is 70 m/sec, it is considered in devastating category. But here, velocity goes beyond 200 m/sec. In high rise building, the structure experience 100 m/sec to 300 m/sec wind flow in any direction. In opposite face region vortex shedding occurs. The structure experience the drag force and lift force by which structure get unstable. The velocity of the wind get reduces due to reduction in area. The velocity at the co-ordinate (-5,5) is 64.34 m/sec while in square geometry having sharp corners has beyond 210 m/sec.

4. CONCLUSION

According to study, turbulence is created due to sharp corner and this is also the cause of flow separation. Vortex shedding is occurred at the opposite face of windward having the high intensity of flow. While in case of modified corners the intensity of vortex shedding is less due to the less decrement of in area of vortex shedding. Here, 8 modified models be simulated. According to study, when roundness at the corners is more the intensity of flow at the opposite face of windward face is less and the intensity of velocity of wind in vortex shedding area is also less, as well as drag force is also less. So, the stability is high of the structure. Here, we can see the results of drag forces of different geometries are following:

Table -1: Drag force against rounded corners having different radius

Roundness (R) in meter	Drag Force in Newton
0	22801.789
0.25	7544.404
0.50	7472.455
0.75	4288.341
1.00	3909.06
1.25	2844.363
1.50	2530.073
1.75	2214.362
2.00	2079.185

Table -2: Drag coefficient against roundness

Roundness (R) in meter	Drag Coefficient
0	41.363
0.25	13.685
0.50	13.555
0.75	7.779
1.00	7.091
1.25	5.159
1.50	4.694
1.75	4.016
2.00	3.771

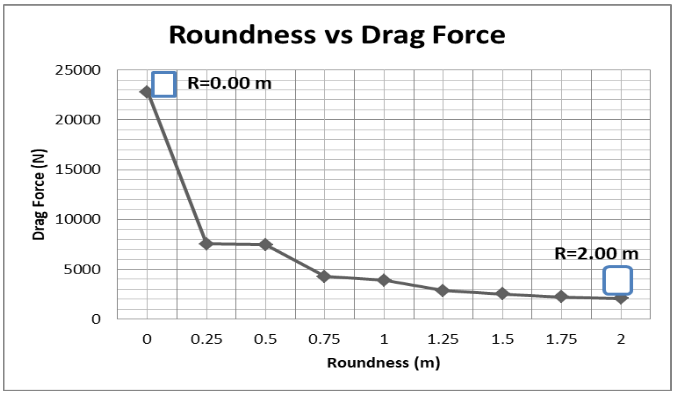

Chart -1: Roundness (m) vs Drag Force (N)

After the optimization, drag force is reduced from 22801.789N to 2079.185N as well as the decrement in drag coefficient.

REFERENCES

[1] A. Elshaer, G. Bitsuamlak, and A. El Damatty, "Vibration control of tall buildings using aerodynamic optimization," in 25th Canadian Congress of Applied Mechanics, 2015.

[2] A. Elshaer, G. Bitsuamlak, and A. El Damatty, "Wind Load Reductions due to Building Corner Modifications," in 22nd Annual Conference of the CFD Society of Canada, 2014.

[3] A. Kareem, S. Spence, E. Bernardini, S. Bobby, and D. Wei, "Using computational fluid dynamics to optimize tall building design," Counc. tall Build. Urban Habitat, 2013.

[4] A.Kareem, T.Kijewski, Y.Tamura, Mitigation of motions of tall buildings with specific examples of recent applications, Wind and Structures. 2(1999) 201–251.

[5] Bosch G, Rodi W, 1998. Simulation of vortex shedding past a square cylinder with different turbulence models. Int. J. Numer. Methods Fluids 616,601–616.

[6] Dutton R, Isyumov N., 1990. Reduction of tall building motion by aerodynamic treatments. J. WindEng. Ind. Aerodyn.36, 739–747.

[7] Ferziger, J H, Peric M, 1995. Computational Methods for Fluid Dynamics. Springer, Berlin, Germany.

[8] Hansen, G A, Douglass R W, Zardecki A, 2005. "Mesh Enhancement: Selected Elliptic Methods, Foundations and Applications," Imperial College Press, London, United Kingdom.

[9] HAYASHIDA H and IWASA Y. (1990), "Aerodynamic Shape Effects of Tall Building for Vortex Induced Vibration," Journal of Wind Engineering and Industrial Aerodynamics.

[10] P Irwin (2006), "Developing wind engineering techniques to optimize design and reduce risk". In 7th UK Conference on Wind Engineering, Glasgow, UK.

[11] J.A. Amina and A.K. Ahuja, "ASIAN JOURNAL OF CIVIL ENGINEERING" (BUILDING AND HOUSING) VOL. 11, NO. 4 (2010).

[12] K.T.Tse, P.A.Hitchcock, K.C.S.Kwok, S.Thepmongkorn, C.M.Chan, "J. Wind Eng. Ind. Aerodynamic", 97 (2009) 455–467.

[13] Slawomir Koziel and Leifur Leifsson, Reykjavik University, 101 Reykjavik, Iceland, AIAA JOURNAL, Vol. 51, No. 1, January 2013.

[14] Ms. Ameena, M Ansary1, Mr. M V Varkey2, Ms. Jiji Thomas, International Research Journal of Engineering and Technology (IRJET), Volume: 04 Issue: 06 | June - 2017.

[15] Peter A. Irwin, "Journal of wind Engineering and Indutrial Aerodynamics" 96 (2008) 761-712.

[16] P. Mendis, T. Ngo, N. Haritos, A. Hira, B. Samali, J. Cheung, "Wind Loading on Tall Buildings", EJSE Special Issue: Loading on Structures (2007)

[17] Shyam Baby, Jithin P N, Anna M Thomas, "A Study of Wind Pressure on Tall Buildings and Its Aerodynamic Modifications against Wind Excitation", 2015 IJEDR | Volume 3, Issue 4 | ISSN: 2321-9939.

[18] Tetsuro Tamura, Tetsuya Miyagi, "The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes", Journal of Wind Engineering and Industrial Aerodynamics 83 (1999) 135}145.

[19] Yi Li, Xiang Tian, Kong Fah Tee, Qiu-Sheng Li, Yong-Gui Li, "Aerodynamic treatments for reduction of wind loads on high-rise buildings", Journal of Wind Engineering & Industrial Aerodynamics 172 (2018) 107–115.