Journal of Engineering Research and Application
http://dx.doi.org/10.55953/JERA.2023.2102
Volume 2, June 2023
pp 12-21

Analysis of Wind Effects on Different Shapes of Tall Buildings

Asmita Sharma[1], Aryan Singh Verma[1] , Aaditya Tanwar[1], Aasheesh Sharma[1], S K Verma[1] and Pramit Kumar Choudhary[2*]
[1] Punjab Engineering College, Chandigarh, India
[2] National Buildings Construction Corporation Limited
* Corresponding Author choudhary.pramit@gmail.com

Abstract:
The rapid expansion of cities and increasing population means that land is limited and what land is available is shaped in an erratic manner, while its demand is increasing day by day. Therefore, to cope with this increasing demand and to achieve maximum utilization of resources, tall buildings are being built. However, these buildings are extremely vulnerable to lateral loads, particularly wind loads.To ensure the safety of these buildings, wind tunnel studies are used to examine the effects of wind loads. To combat the issue of shortage of wind tunnel facilities in the world, numerical analysis is used instead. Thus, this project aims to analyse the impact of wind effects on high-rise buildings of different shapes with the help of CFD, by making use of ANSYS. The wind effects have been assessed by obtaining the velocity profile and turbulent profile, pressure contours for different faces of the building, and velocity streamlines for the plan and elevation of the building.

1.Intoduction

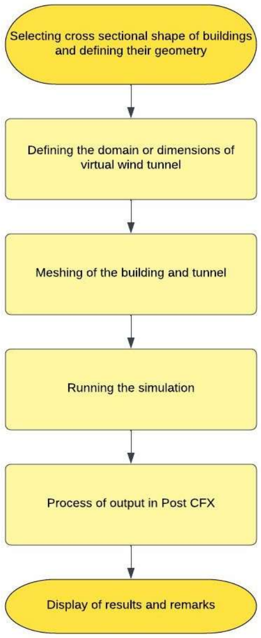
The term 'Wind Load' refers to the lateral load caused by winds on buildings or any pressure or force that the building or structure may experience due to wind. According to IS 875 Part 3 (2015), tall building may be defined as any multiple-level structure with a height greater than or qual to 50 metres. An ever-increasing population and increasing land prices due to limited resources have led to a need for buildings that are higher vertically instead of horizontally expanded ones. However, these high-rise buildings are very vulnerable to wind loads. As wind energy is focused at lower frequencies and thus can cause severe harm to structures with similar frequency ranges, such as high-rise or tall buildings. Therefore, with an exponential increase in the construction of various models of tall buildings across the world and particularly in
India, and as the effect of wind loads start to become increasingly crucial with a rise in height of the building, it is extremely necessary to study the effect of wind loads in depth to ensure the safety of these high-rise buildings that seem to have taken over the world.These wind effects are usually examined with help of wind tunnel studies in which scaled-down atmospheric wind loads are applied to scaled models of buildings under a controlled laboratory setup. The model's sensors can then be used to measure the relevant physical quantities such as pressure, velocity, shear, etc. Later, these obtained quantities can be converted into prototypes using model scale laws. One such software that can be used in high-functioning computing to deliver CFD

simulation results required to estimate the impact of wind loads more quickly is ANSYS CFX.However, these wind tunnel studies are arduous and expensive, therefore CFD (Computational Fluid Dynamics) software can be used to undertake virtually simulated studies instead of losing resources in performing actual wind tunnel studies. Also, the Indian Standard Code concerning wind loads, IS:875 (Part 3: Wind loads) does not extend to buildings with unconventional shapes or unique environmental conditions or odd locations. Thus, in these situations, specialized investigations are required to determine the wind loads and their impacts. Various international standards [1-5] provide information on the effects of wind load on popular plan shapes; Sun et al. [6] carried out a wind tunnel test on a model of a 1040-meter building; Irwin et al. [7] looked at the impact of corners. Bhattacharyya and Dalui [8] presented wind effects on an "E"-shaped building; Koliyabandara et al. [9] examined the wind loads on irregularly shaped buildings while the numerical simulation was carried out using the Computational Fluid Dynamics (CFD) module Midas NFX. Bandi et al. [10] conducted an experiment on six building models; Raj et al. [11] explored the reaction of the "+" shape; Nagar et al. [12] studied the impacts of wind action around the "H" and the square building model. Carassale et al. [13] studied corner influence on buildings. Pal et al. [14] approximated drag and lift force coefficients on interactinsquare and fish plan shape models that were interfering; Paul and Dalui [15] carried out studies for the "+" shape; Hajra and Dalui [16] carried out studies for the octagonal shape; Gaur and Raj [17] carried out studies for the square model; Sanyal and Dalui [18] carried out a study to gauge the effect of wind excited action by finding the effect of side ratio on "Y" shape model under a wind load. The main objective of this project is to study four different shapes of buildings for wind effects, i.e., N-Plan shape, V-Plan shape, L-Plan shape, and I-Plan shape building. This includes evaluating the wind flow pattern around the building models and finding out pressure contours, velocity streamlines, and subsequently pressure coefficients for these models with the help of ANSYS CFX.

2. MATHODOLOGY

The k-Ɛ turbulence model has been chosen to be used in the software to solve complicated fluid problems as it is known to provide efficient and authentic results for tall buildings which are in line with experimental studies. The study consists of two main parts:

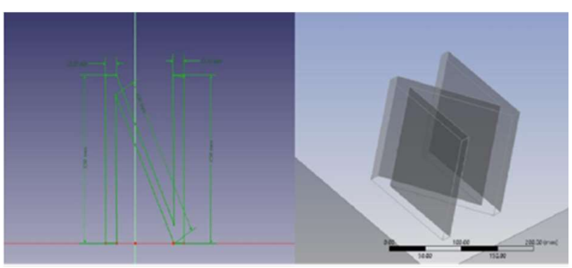
1. Mathematical modelling using Ansys: It consists of defining the geometry and dimensions of the shape, meshing of the building and wind tunnel & set up in the software
2. Setup and Results: First, the simulation is setup in the software. Then the pressure contours for all the faces of the building and pressure coefficients for windward and leeward faces are obtained using the software. Velocity Streamlines for plan (mid plane) and elevation (side plane) have also been obtained. The study has been performed in a systematic manner through the following steps:



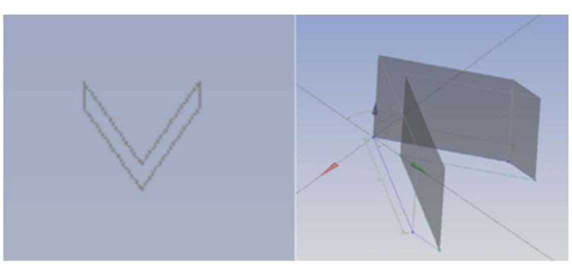
2.1 Selection of building shapes and defining their geometries:-

Tall building models with four different plan shapes-N, V, L, and I have been considered, with a geometric scale of 1:250 and designed such that their scaled-down cross-sectional area comes out to be 6000mm^2 and height is kept as 200 mm.

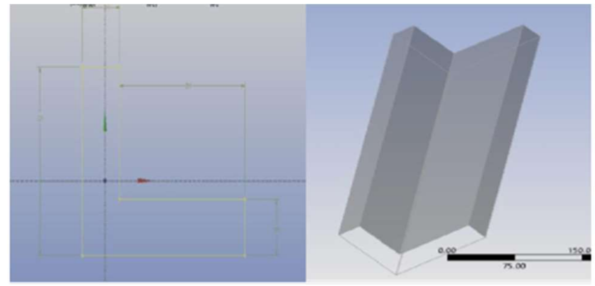
N-plan shaped building



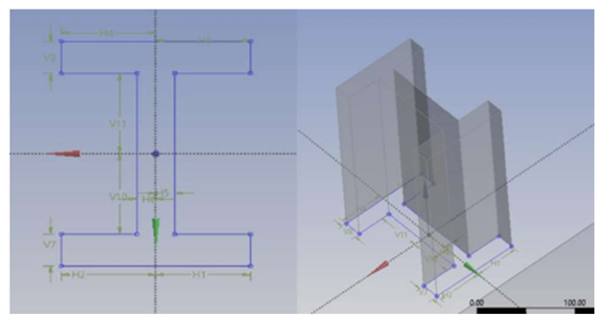
V-plan shaped building



L-plan shaped building

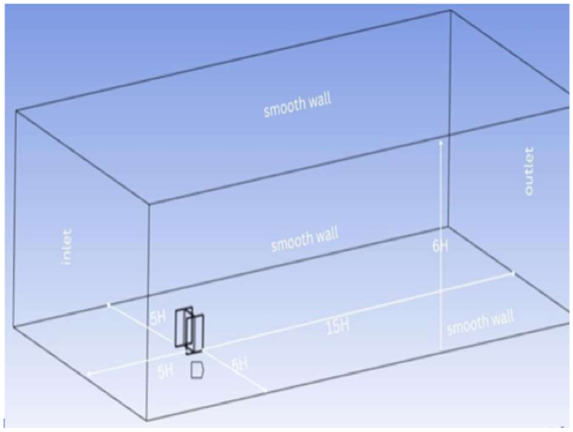


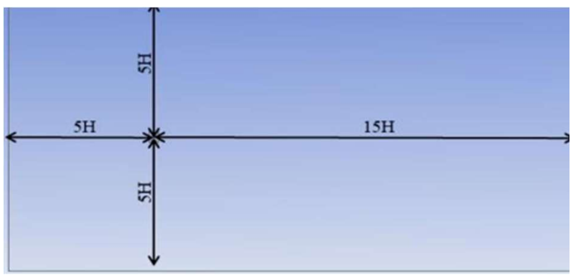
I-plan shaped Building



2.2 Defining Boundary Conditions or Virtual Wind Tunnel:

The study's domain layout was chosen such that an actual wind tunnel can be appropriately simulated. The building's windward side faces are kept 5h away from the inlet and the two side faces of the domain while the main building's leeward sides are 15h away from the domain's exit (where h is the height of the structure). The buildings' roof is also 5h away from the top of the domain, i.e., the total height of the wind tunnel is 6h.

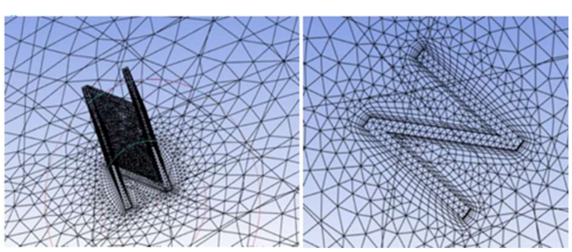




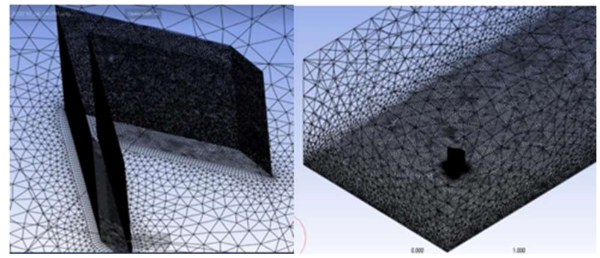
2.3 Meshing:

Since we are using Ansys CFD which uses Finite Element Method to generate the simulation, the geometry is divided into multiple smaller parts. This is done to increase the precision of the simulation-generated solution. Inflation is applied at places where there is a surface interchange for all the models in the CFD simulation to lessen the anomalous flow. For this project, tetrahedron meshing has been done with an element size of 0.0025m for the building structure and 0.025m for the virtual wind tunnel. Post that, an inflation of 5 layers has been applied to reduce the anomalous flow.

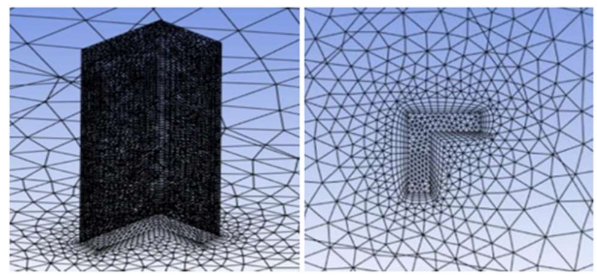
N-plan shaped building



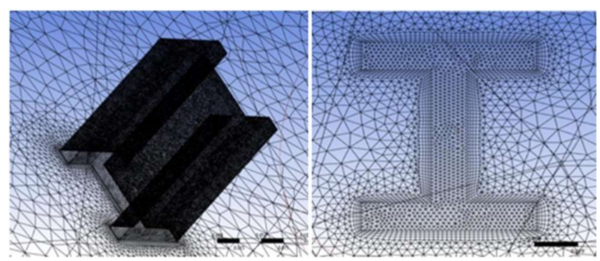
V-plan shaped building



L-plan shaped building



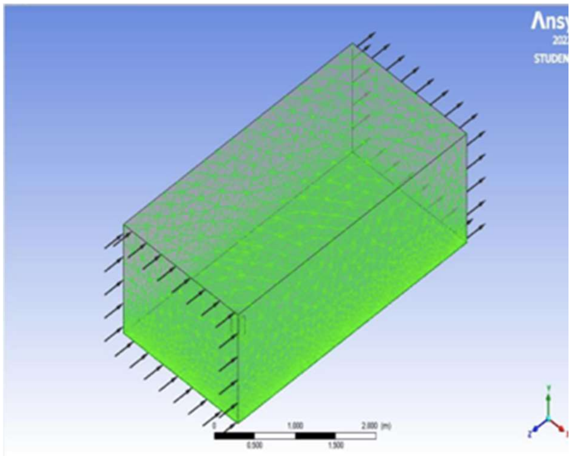
I-plan shaped building



3 Set-up and Results

3.1 Set-up of the simulation:

It is important to define the boundary details or the domain of the simulation before running it. The inlet, outlet and side walls of the wind tunnel are defined first. Then the faces, top of the building and the ground surface is defined. Analysis is done under steady state assumption.

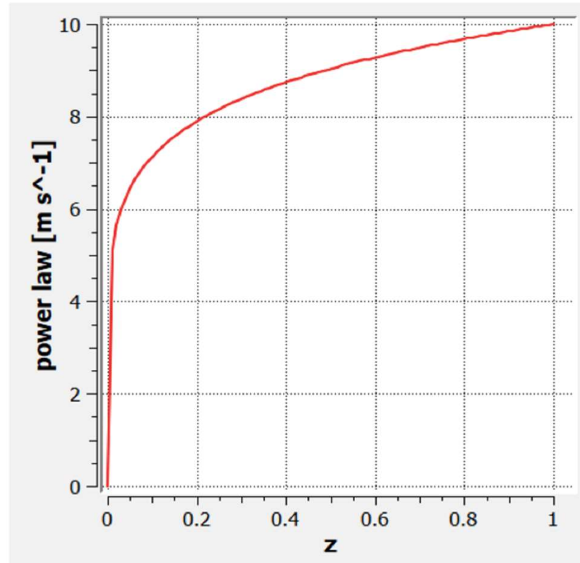


As per research studies, k-epsilon model has been giving the most accurate results for tall building therefore tit has been chosen as the turbulence model in the software. In order to approximate the wind profile, the power law express is defined as below:

u= uRef *(y/ yRef)^alpha

where u is the wind speed (in meters per second), at height y (in meters), uRef is the known wind speed (assumed = 10m/s here at a reference height yRef (assumed = 1m here).

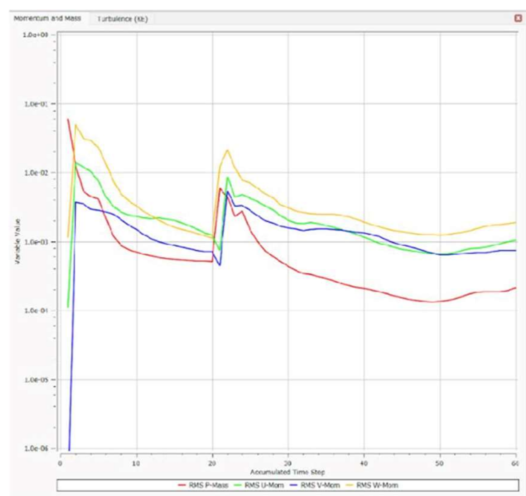
The exponent (α) is an empirically derived coefficient and varies according to terrain category, for Terrain 2 it is assumed to around 0.147. This expression has been defined such that the speed increases with height till it attains 10 m/s speed at 1m height after which it stays constant at 10 m/s.



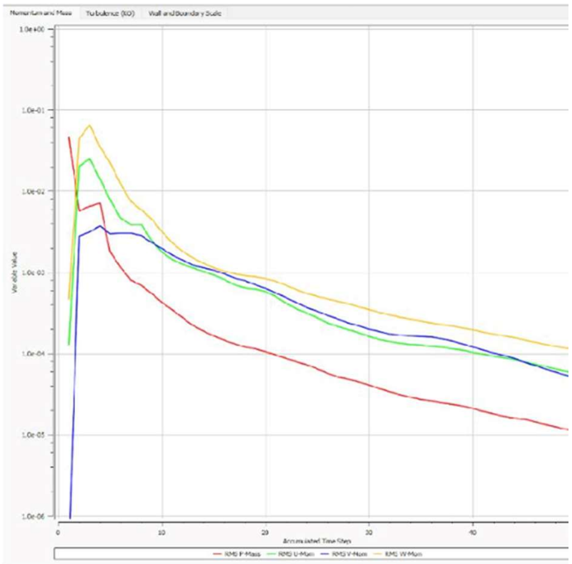
3.2 Running of Simulation:

After the set-up is done, the Ansys CFX solver is run and results in the form of graphs for u, v, w direction are obtained. Full Run Type is employed in large problem settings with double precision.

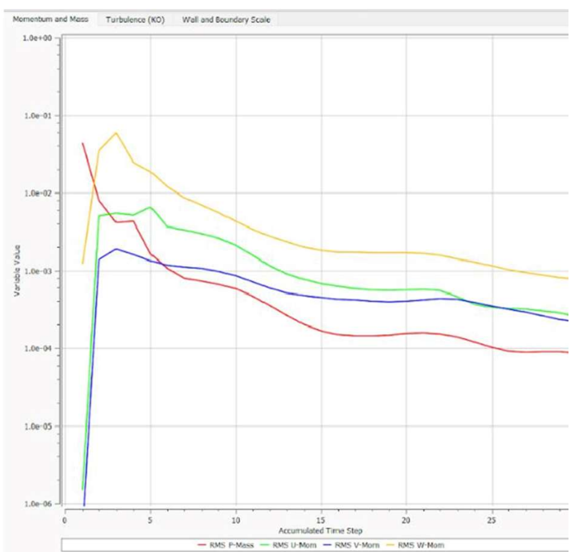
N-plan shaped building



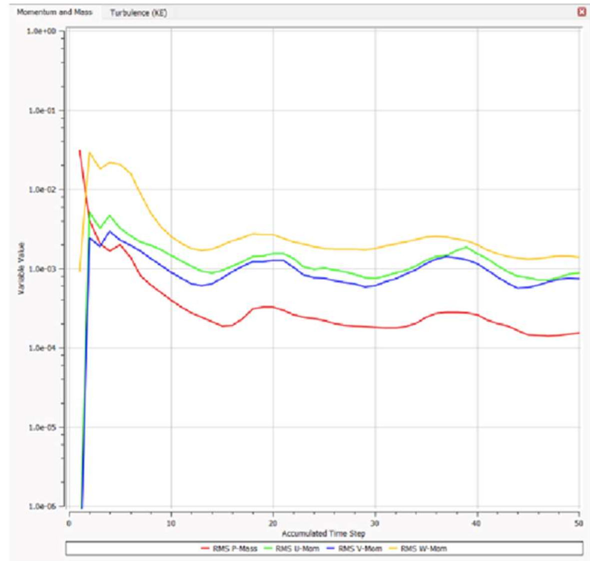
V-plan shaped building



L-plan shaped building



I-plan shaped building



4 Results:

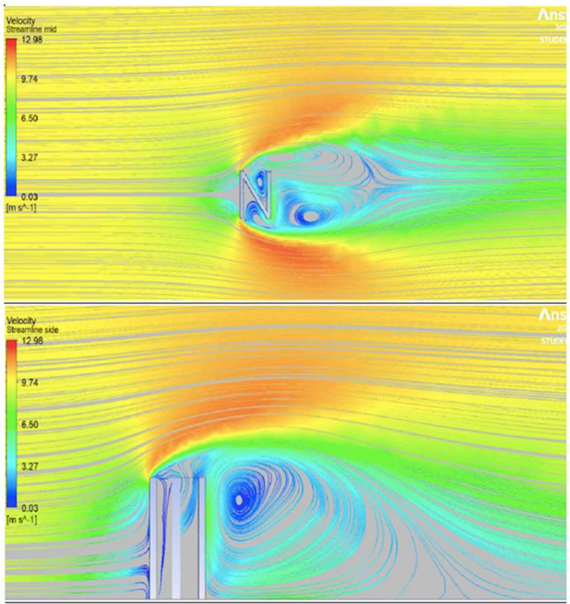
4.1 STREAMLINES

An imaginary line in a fluid called a streamline is one whose tangent at any point corresponds to the direction of the velocity of a fluid particle there. The streamline for 0°wind incidence angle for N-plan, V-plan, L-plan and I-plan shaped buildings in plan (plane passing through the idle of the building) and in elevation (side plane) have been shown below. On the downstream side, vortices are observed in all the shapes but unique flow patterns are found for all four models because of the variation in the plan shape of thebuildings

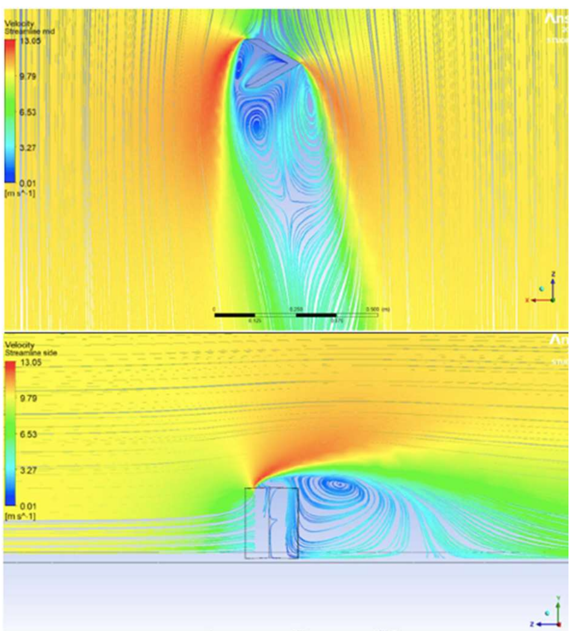
PRESSURE CONTOURS

The pressure contours are the lines that connect points that all have the same value of pressure. The contour lines have a colour and a value associated with them and are usuallylabelled with that value

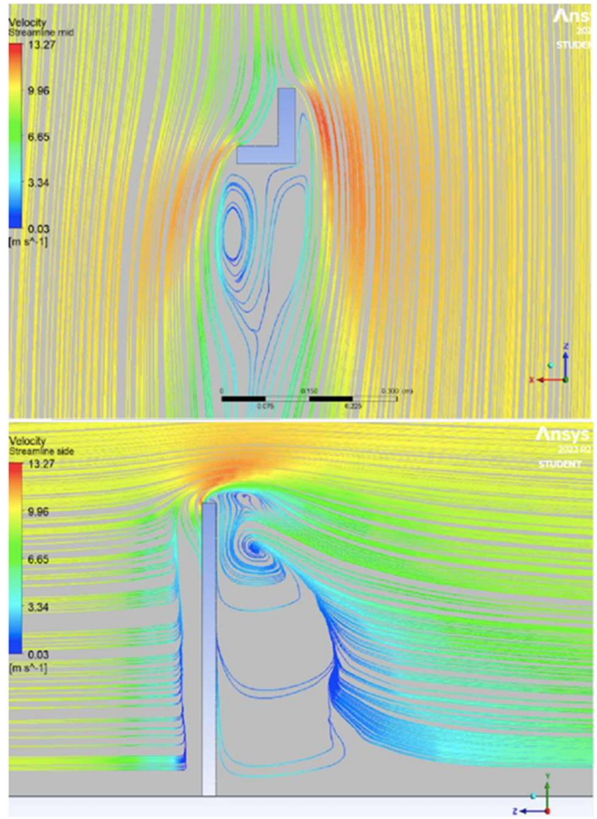
N-plan shaped building



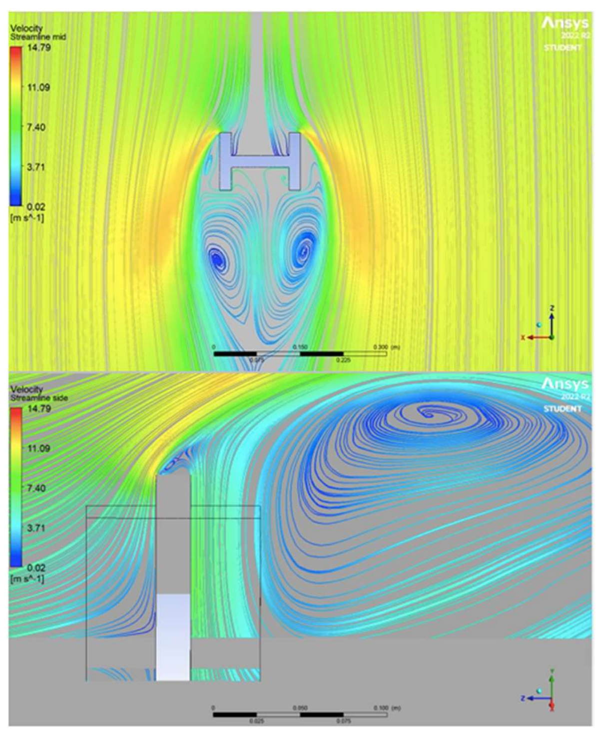
V-plan shaped building



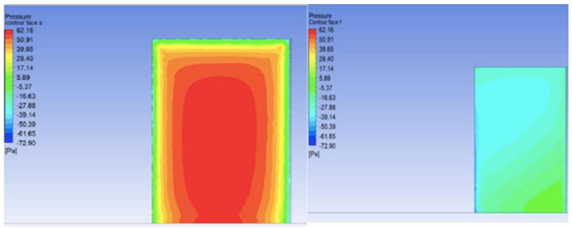
L-plan shaped building



I-plan shaped building

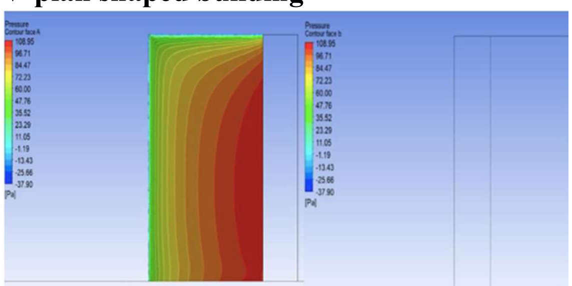


N-plan shaped building

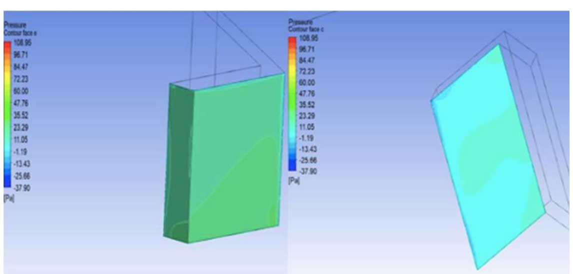


--Windward faces --Leeward Faces

V-plan shaped building

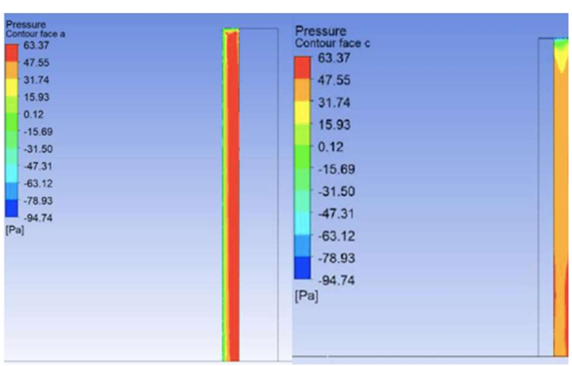


--Windward faces

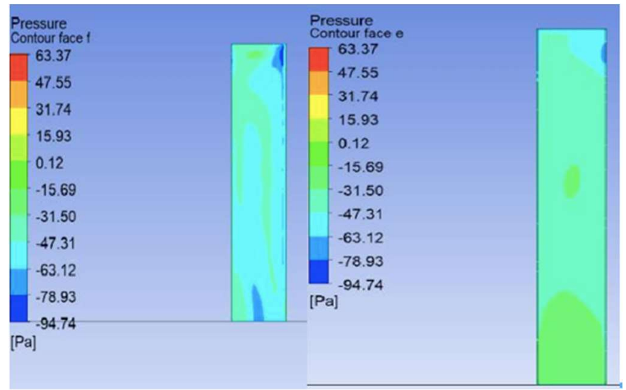


--Leeward faces

L-plan shaped building



--Windward faces



--Leeward faces

PRESSURE COEFFICIENTS

Wind Pressure is often defined in terms of Wind Pressure Coefficients. It is a dimensionless number which describes the relative pressure throughout a flow field in fluid dynamics. Pressure Coefficients of Windward and Leeward Faces of different plan shapes are tabulated below: These have been calculated using the formula:

$$Cp = P/ (0.6 Vr^2)$$

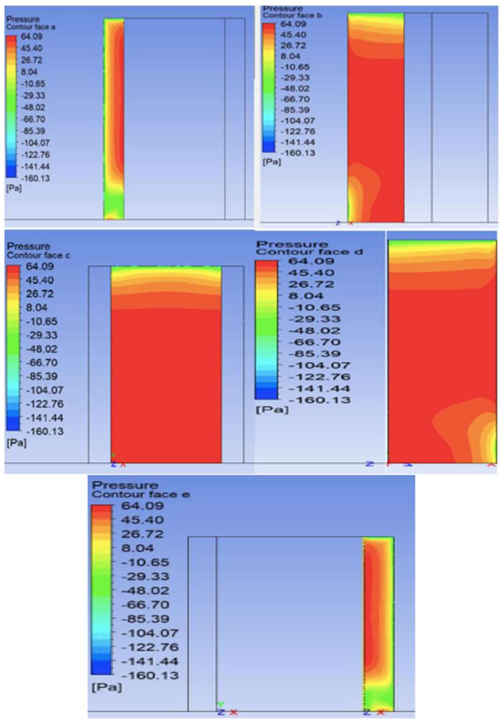
SHAPES	Windward Faces	Lee-Ward Faces
I	0.55	-0.46
V	0.58	-0.53
N	0.61	-0.59
L	0.53	-0.56

Conclusions-;

- For 0^0 wind incidence angle among I, V, N and L shape building models, the tall building model having a plan shape in the 'N' shape is having maximum wind pressure coefficient on windward face while model "L" is having minimum wind ward coefficient.
- In the case of 0^0 wind the wind pressure on the leeward side is having maximum pressure for N shape building model while the model having I plan shape is

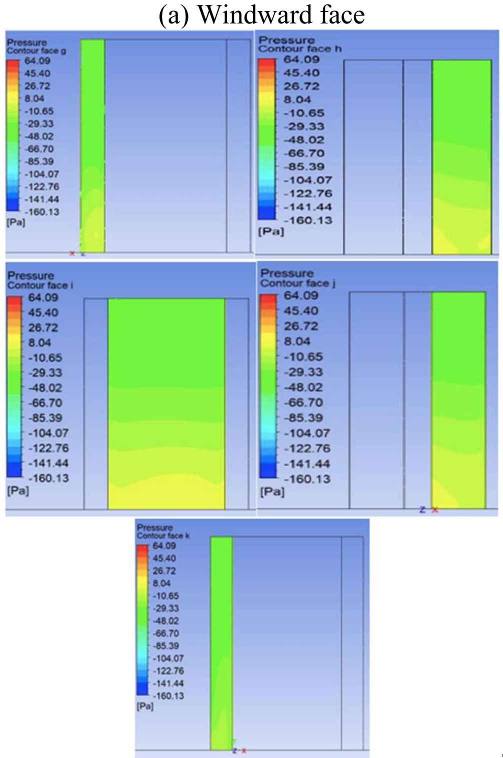
observed to have the least pressure coefficient.

I-plan shaped building



--

(a) Windward face



--

--Leeward face

- It is observed from the all-shape model that the wind ward face is always under the effect of positive pressure while the leeward face is always under the negative pressure zone which is minimum among all.

REFERENCES

1. IS-875 (Part-3), in Indian Standard Design Load (Other Than Earthquake) for Building and Structures-Code of Practice, BIS, Old Delhi, India, 2015.
2. AS/Nzs11702, Structural Design Actions - Part 2, Wind actions. Standards Australia/Standards New Zealand, Sydney, Australia, 2011.
3. Asce716, Minimum Design Loads and Associated Criteria for Buildings and Other Structures, Structural Engineering Institute of the American Society of Civil Engineering, Reston, 2017.
4. GB 50009-2001, National Standard of the Peoples' Republic of China, Ministry of Construction (MOC) and the General Administration of Quality Supervision, Inspection and Quarantine (GAQSIQ) of the People's Republic of China, Beijing, China, 2002.
5. H. Kong Building Department, Code of Practice on Wind Effects in Hong Kong 2019, Building Department Hong Kong, Hong Kong First Issue, 2019.
6. X. Sun, H. Liu, N. Su, and Y. Wu, "Investigation on wind tunnel tests of the Kilometre skyscraper," Engineering Structures, vol. 148, pp. 340–356, 2017.
7. P. Irwin, J. Kilpatrick, and A. Frisque, "Friend or foe, wind at height," CTBUH 2008, 8th World Congress. - Tall Green Typology a Sustain. Urban Future. Congress. Proc., pp. 336–342, 2008.
8. B. Bhattacharyya and S. K. Dalui,

"Experimental and numerical study of wind-pressure distribution on irregular-plan-shaped building," Journal of Structural Engineering, vol. 146, no. 7, Article ID 04020137, 2020.
9. H.M.A.D. Jayasundara, S.M.N.H. Koliyabandara and K.K. Wijesundara, "Wind Loads on Tall Buildings: A Comparative Study of the International Wind Codes and Numerical Simulation," Journal of the Institution of Engineers, Sri Lanka, vol. LI, no. 3, pp. 31-45, 2018.
10. E. K. Bandi, Y. Tamura, A. Yoshida, Y. Chul Kim, and Q. Yang, "Experimental investigation on aerodynamic characteristics of various triangular-section high-rise buildings," Journal of Wind Engineering and Industrial Aerodynamics, vol. 122, pp. 60– 68, 2013.
11. R. Raj, S. Jha, S. Singh, and S. Choudhary, "Response analysis of plus shaped tall building with different bracing systems under wind load," International Journal of Advanced Research in Engineering & Technology, vol. 11, no. 3, pp. 371–380, 2020.
12. S. K. Nagar, R. Raj, and N. Dev, "Experimental study of wind-induced pressures on tall buildings of different shapes," Wind Struct. An Int. J., vol. 31, no. 5, pp. 441–453, 2020.
13. L. Carassale, A. Freda, and M. Marrè-Brunenghi, "Experimental investigation on the aerodynamic behaviour of square cylinders with rounded corners," Journal of Fluids and Structures, vol. 44, pp. 195–204, 2014.
14. S. Pal, R. Raj, and S. Anbukumar, "Comparative study of wind induced mutual interference effects on square and fish-plan shape tall buildings," Sādhanā, vol. 46, no. 2, 2021.
15. R. Paul and S. K. Dalui, "Prognosis of wind-tempted mean pressure coefficients of cross- shaped tall buildings using artificial neural network," Periodica Polytechnica: Civil Engineering, vol. 64, no. 4, pp. 1124–1143, 2020.
16. S. Hajra and S. K. Dalui, "Numerical investigation of interference effect on octagonal plan shaped tall buildings," Jordan Journal of Civil Engineering, vol. 10, no. 4, pp. 462–479, 2016.
17. N. Gaur and R. Raj, "Aerodynamic mitigation by corner modification on square model under wind loads employing CFD and wind tunnel," Ain Shams Engineering Journal, vol. 283, 2021.
18. P. Sanyal and S. K. Dalui, "Effects of side ratio for "Y" plan shaped tall building under wind load," Building Simulation, vol. 14, no. 4, pp. 1221–1236, 2020.