Journal of Engineering Research and Application
http://dx.doi.org/10.55953/JERA.2023.2101
Volume 2, June 2023
pp 1-11

Evaluation of Wind Effects on High-Rise Buildings Using Ansys CFX

Astha Verma[2]* , Neeraj Chandra[1], Pratik Kumar Singh[1] , Vishal Kadian[1] and Suraj Rai[1]
[1] Punjab Engineering College, Chandigarh, India
[2] Gobind Ballabh Pant University of Agriculture & Technology, India
* Corresponding Author asthaverma.civil@gbpuat-tech.ac.in

Abstract: Tall building is the growing pattern of cities, these buildings are highly vulnerable to wind loads. Wind load effects on the high-rise building can be found using wind tunnel testing and CFD tools. Wind tunnel facilities are limited all around the world, so CFD tools can be incorporated. In this study, ANSYS CFX is used to investigate the effects of wind on different shapes of high-rise buildings. The results of numerical simulation are compared with the international standards of various countries. Streamline patterns and pressure contours on different faces are discussed in this paper. It is found that building shape and size are having a direct effect on the pressure distribution on various surfaces of the building model. Windward face pressure will always be positive in nature. In this study, the comparison is presented for the variation in the cross-sectional shape of the building model. For this purpose Model C and Model I are having considerable variations and Model X is considered for validation purposes only. It is well established that the results are within the allowable limit. This paper aims to discuss these wind-generated effects in the tall building model.

1. Introduction

High-rise residential buildings have become more widespread in many cities as a result of increasing population and rising urbanisation, displacing broad regions of vernacular dwellings. These constructions are vulnerable to time-varying loads induced by winds, earthquakes, and other natural calamities, in addition to gravity loads. Tall buildings have a high sensitivity to wind excitation.[1]–[4]. In general, a structure cannot be considered a regular configuration when planning for wind loading. To achieve the structural pressure coefficients and force coefficients for other structures subjected to wind-induced loads, designers consult related wind load standards[5]–[9]. On the other hand, these standards offer details for plain cross-sectional configurations with a limited number of wind incidence angles. These codes do not provide information on wind loadings for structures with different configurations. As a result, wind tunnel research on models of such forms is popular. Raj and Ahuja [10] conducted an experimental study on cross-shape tall buildings with various cross-sectional shapes and observed that base shear, base moments, and twisting moments are influenced by wind incidence angle and affected by cross-sectional shape. Vafaeihosseini et al.[11]used computational fluid dynamics to find wind effects on high-rise buildings given that the most efficient shape for tall buildings is a rectangular plan shape. Hajra and Dalui [12] performed the mathematical-based research of interference effect on octagonal plan configuration high structure using CFX (ANSYS), for 00 wind incidence angle using k-ε, SST, and k-ω model, these three models analysis shows nearly identical results.

Meena et al.[13] discussed the effects of wind on the bracing system used in various types of multi-storeyed steel buildings. Pal et al.[14] carried out a comparative study on square and fish plan-shaped tall buildings of wind-induced mutual interference effect, maximum efficiency in terms of wind-induced mutual interference pressure, and base shear exhibited by a square plan-shaped model at full blockage condition. Verma et al.[15] analyzed the effects of wind load on tall buildings' octagonal shape, using CFD simulation for 00,150 and 300 wind incidence angles, and found that CFD can be used to predict wind-related phenomena on buildings and other types of structures. Nagar et al.[16] carried out an experimental study of wind-induced pressure on tall buildings of different shapes from 00 to 900 wind incidence angle at an interval of 300, found that wind pressure decreases up to 600 wind incidence angle.

Wind tunnel testing requires a costly setup and sophisticated instruments to measure a range of field variables wind velocity, pressure loads, turbulent intensity etc. it is the main limitation that such measurements are a few selected points within the test section, severally restring the unsteady complex phenomena such as vortex shedding, turbulence wake. CFD work by dividing a space into a grid containing a large number of cells. The grid of the cells is surrounded by boundaries that simulates the surfaces and opening, that enclose the space, the pressure of the boundaries, and the air movements at the opening within the cell are then set to a starting condition, which is close to the real environments.

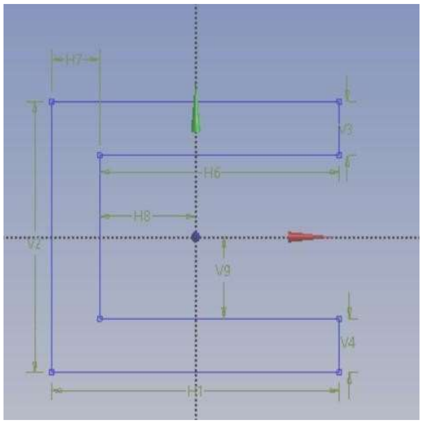
These conditions are determined using a boundary model that predicts boundary conditions, the more accurate the boundary model, and the closer to starting condition is to the final position predicted to the model, the faster the model will run and the more accurate the output is likely to be. CFD results are highly dependent on the knowledge of the person setting up the model and interpreting the results, as the input information is wrong, the output information will be as well, CFD has no substitute for common sense.

The wind forces acting on a building and the motion that arises are directly influenced by the shape of the building. By carefully designing the construction components and geometry of tall buildings, wind excitation is reduced and costs are reduced. Passive aerodynamics adjustments in the form of building shape are one of the most potent and adaptable construction approaches for considerably minimizing the impacts of wind forces by changing/altering the flow pattern around the buildings.

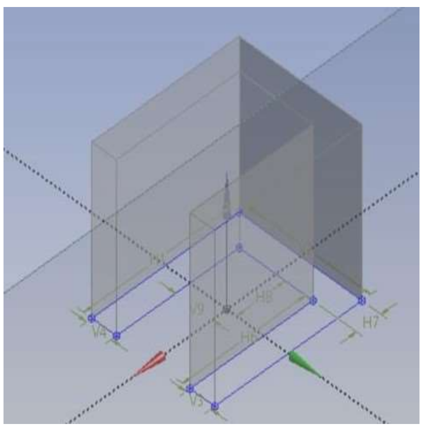
2. MATHEMATICAL MODELLING BY ANSYS CFX

2.1 Model:-

To study the effects of wind, different shapes of tall building models are considered in this study as shown in fig.1 and fig.2 , namely C and I-shaped buildings. A geometrical scale of 1:500 is considered.

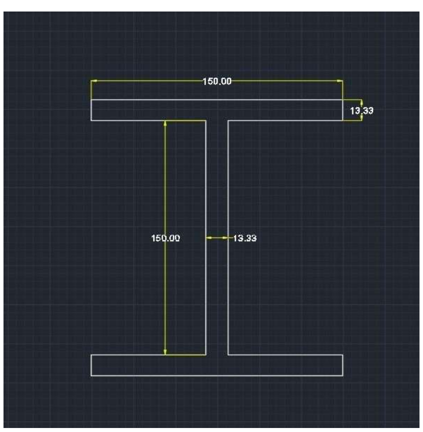


(a) Top View

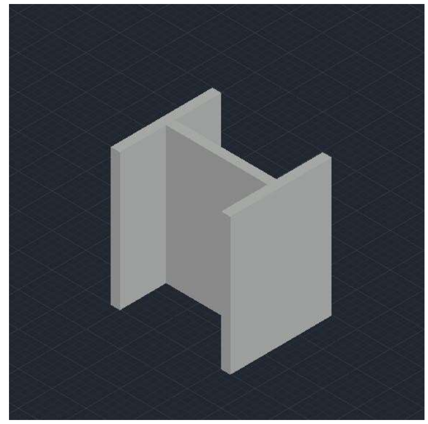


(b) Isometric View

Fig. 1 Building Geometry C Shape



(a) Top View



(b) Isometric view

Fig. 2 Building Geometry I Shape

2.2 Dimensions:

C- Shape Building

Side	Dimensions in mm
H1	200
V2	200
H6	180
V9	80

H- Shape Building

Side	Dimensions in mm
H1	200
V2	200
H6	180
V9	80

Height of buildings (Y-Axis) = 200 mm

3. Methodology

A domain is constructed all around the building model which is working as a virtual wind tunnel.

4. Boundary Conditions

The domain presented in fig.3 and 4 and the boundary must be specified to describe a problem with a unique solution. Boundary circumstances that aren't clearly established are having a big impact on your solution. Identifying the solver, the inlet, the outlet wall, and other boundary conditions the domain, and boundary conditions are employed to carry out the numerical simulation

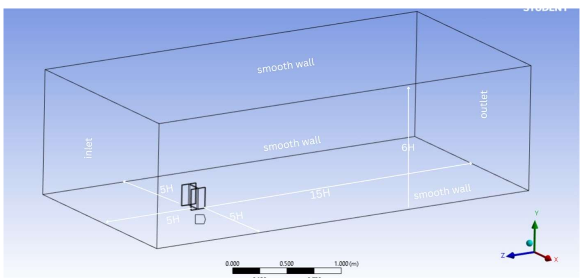


Fig.3 Domain Isometric View

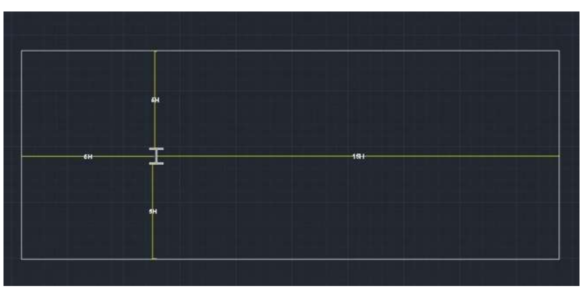


Fig. 4 Domain (Top View)

4.2 Dimension of Wind Tunnel:

Side	Dimensions in mm
H11	3000
H12	9000
V14	3000
V15	3000

Height of wind tunnel (Y-Axis) = 3600 mm

4.2 Meshing

The purpose of meshing is to improve the accuracy of the solution generated during simulation. This can be done manually or automatically with ANSYS CFX. Meshing for distinct portions can be applied in the manual technique, and meshing size is chosen based on the situation. To reduce the anomalous flow, inflation is applied to all models in the CFD simulation.

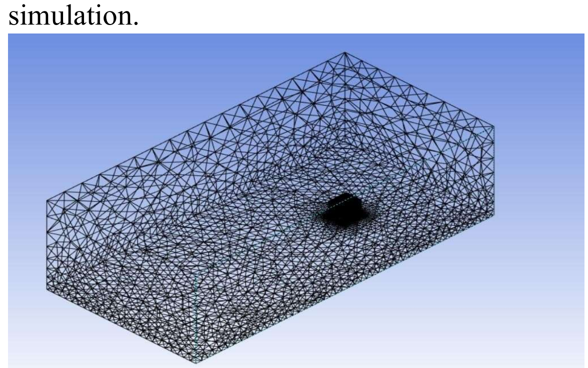


Fig. 5 Isometric view of meshing

Domains with tetrahedron meshing, structure, and ground meshing are more delicate in size, as seen in Fig.5 and in fig 6 for C-shape building while meshing for I-shape building is in fig. 7. It improves the precision of the solution, and meshing is employed to reduce abnormal flow.

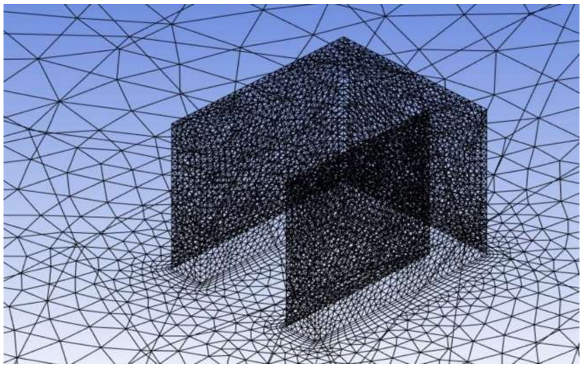


Fig. 6 Close-up view of meshing in building (C shape)

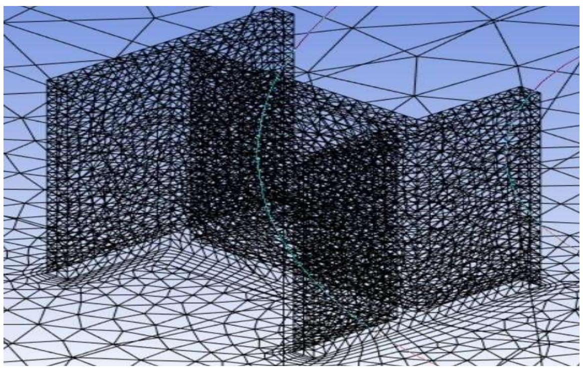


Fig. 7 Close-up view of meshing in building (I shape)

4.2 Setup

The power law is used for interpreting the results in Ansys in set-up as shown in fig. 8 and the solution steps. The inputs for the set-up and Power law are as follows

Expressions	
alpha	0.147
power law	uRef*(y/yRef)^alpha
uRef	10 [m/s]
yRef	1 [m]

Fig. 8 Power law in ANSYS Pre

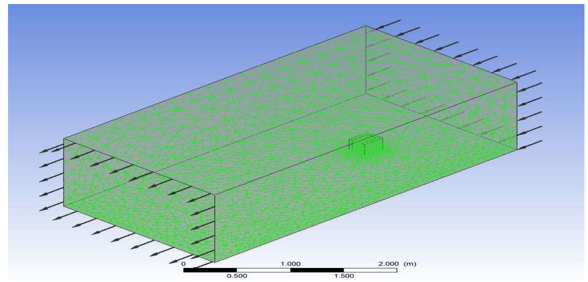


Fig. 9 Domain representing the flow of air from Inlet to Outlet

The graph is plotted for power laws and the graph for C-shaped and I-shaped buildings is shown below

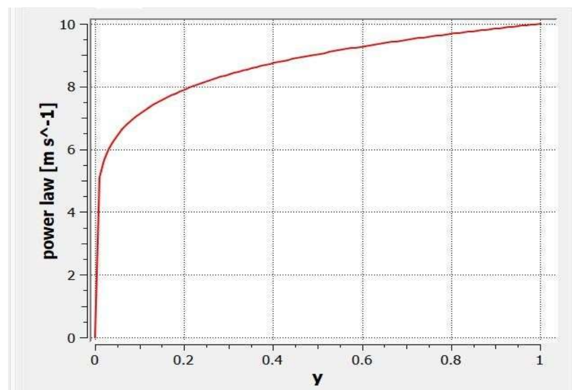


Fig. 10 Graph for power law (c shape)

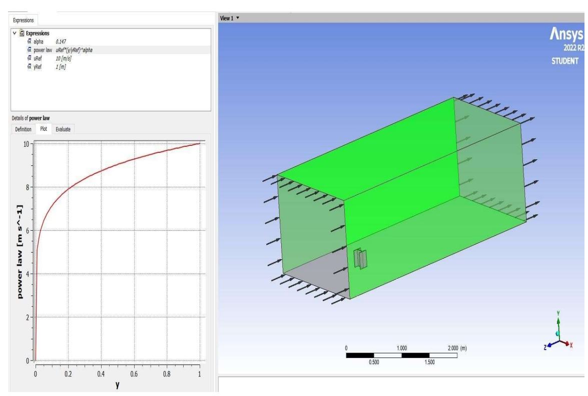


Fig. 11 Graph for power law (I Shape)

5. Solution

The solutions are often steady-state solutions. Due to its resilience for steady-state, single-phase flow problems, second-order differencing was employed for the pressure, momentum, and turbulence equations as well as the "coupled" pressure-velocity coupling technique. After many hundred cycles, the residuals failed to meet the generally used standard of dropping below 10-4 of their starting levels. However, this was not the only method used to determine whether the simulations had converged. The drag, lift, side, and moment forces as well as the moments acting on the building were also tracked during the simulation, below is the momentum time graph generated for 150 and 200 iterationsfor C shape and I shape buildings.

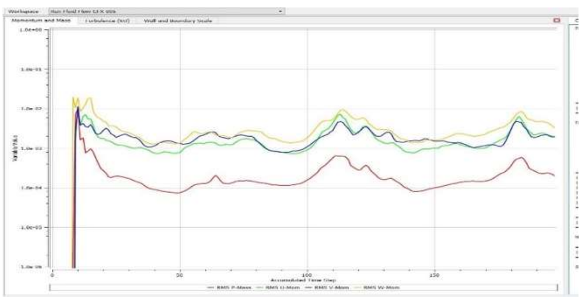


Fig. 12 Solution for I shape Building

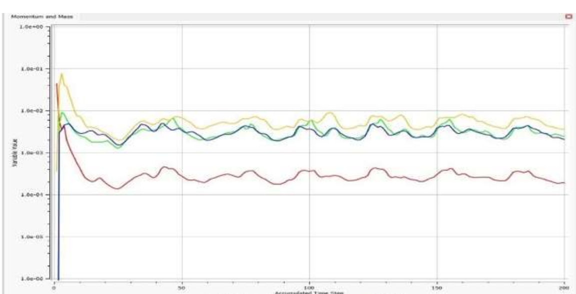


Fig. 13 Solution for C shape Building

6. RESULT AND DISCUSSION

6.1 Velocity Profile and Turbulent Profile

When estimating the vertical profile of wind speed, surface roughness and drag induced by local projections that impede wind flow are important elements. Neither the gradient height nor the gradient velocity causes anydrag; these two numbers are referred to as the gradient. The atmospheric boundary layer refers to the layer of air above which topography has an effect on wind speed.

The wind speed profile within the atmospheric boundary layer, as seen is determined by an equation according to Power Law Eq.

$$\frac{U}{U_H} = \left(\frac{Z}{Z_H}\right)^{\alpha} \qquad (1)$$

Where U is the horizontal wind speed at an elevation Z (Y in this case); UH is the speed at the reference elevation Z_H, which was 10 m/s, α is the parameter that varies with ground roughness which is 0.147 for terrain category 2, and Z_H is 1.0 m. Figure 14 shows that wind speed near the surface is very low because of the friction between the surface and moving air. The wind profile obtained in this study is similar to the wind profile provided inASCE 7 and previous research done in this area.

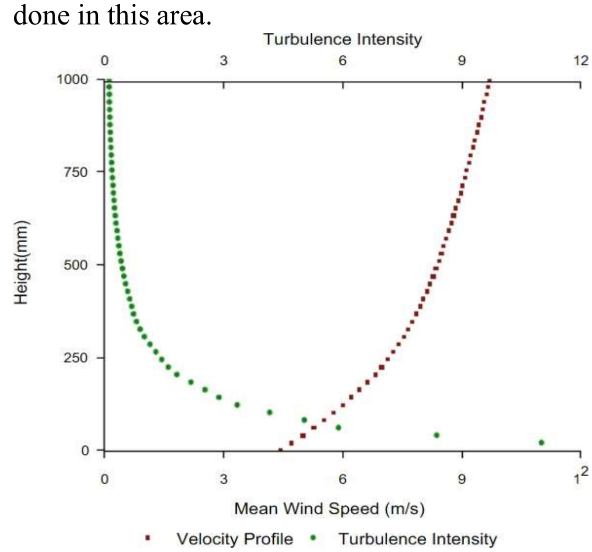
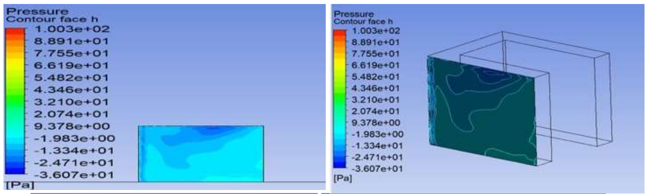


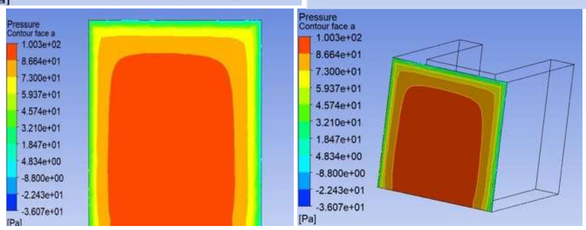
Fig. 14 Height-dependent variations in wind speed and turbulence intensity

6.1 Pressure Contour

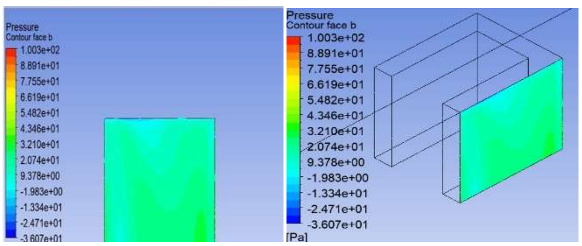
The pressure on the windward face is positive for the models, as shown inthe Figures below, and for the windward face and leeward face, it is negative. The graphical representation of pressure on every face of both models C and I are shown below.

- C- Shaped Building

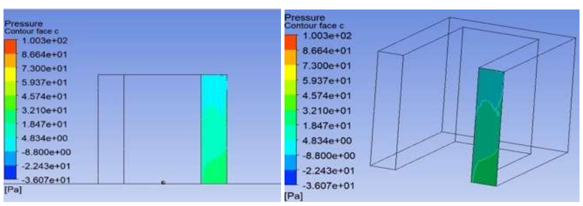




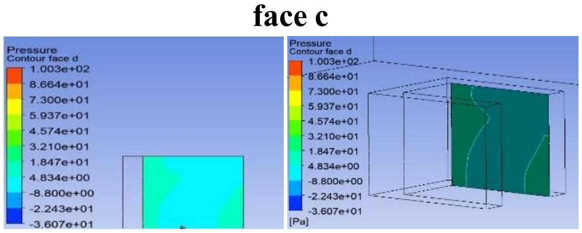
face a



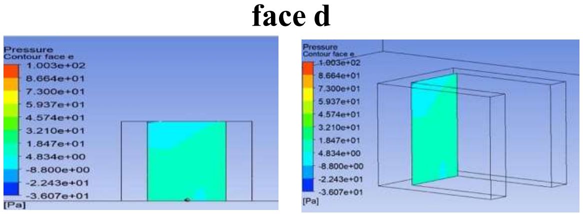
face b



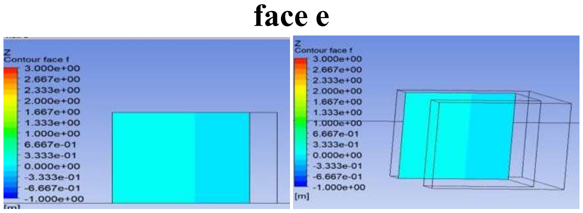
face c



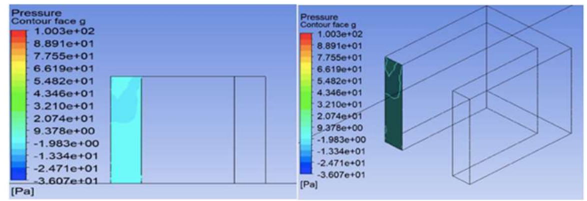
face d



face e

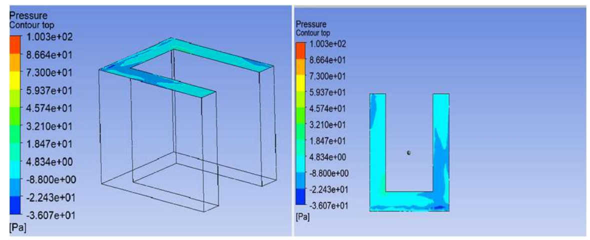


face f



face g

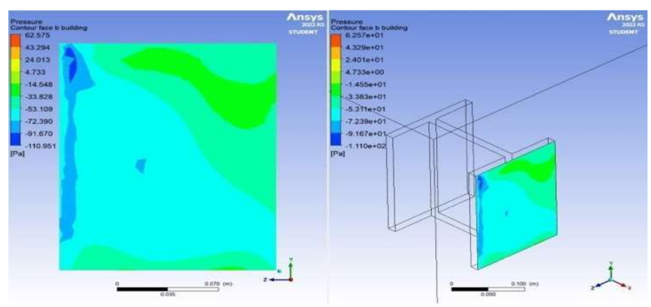
face h

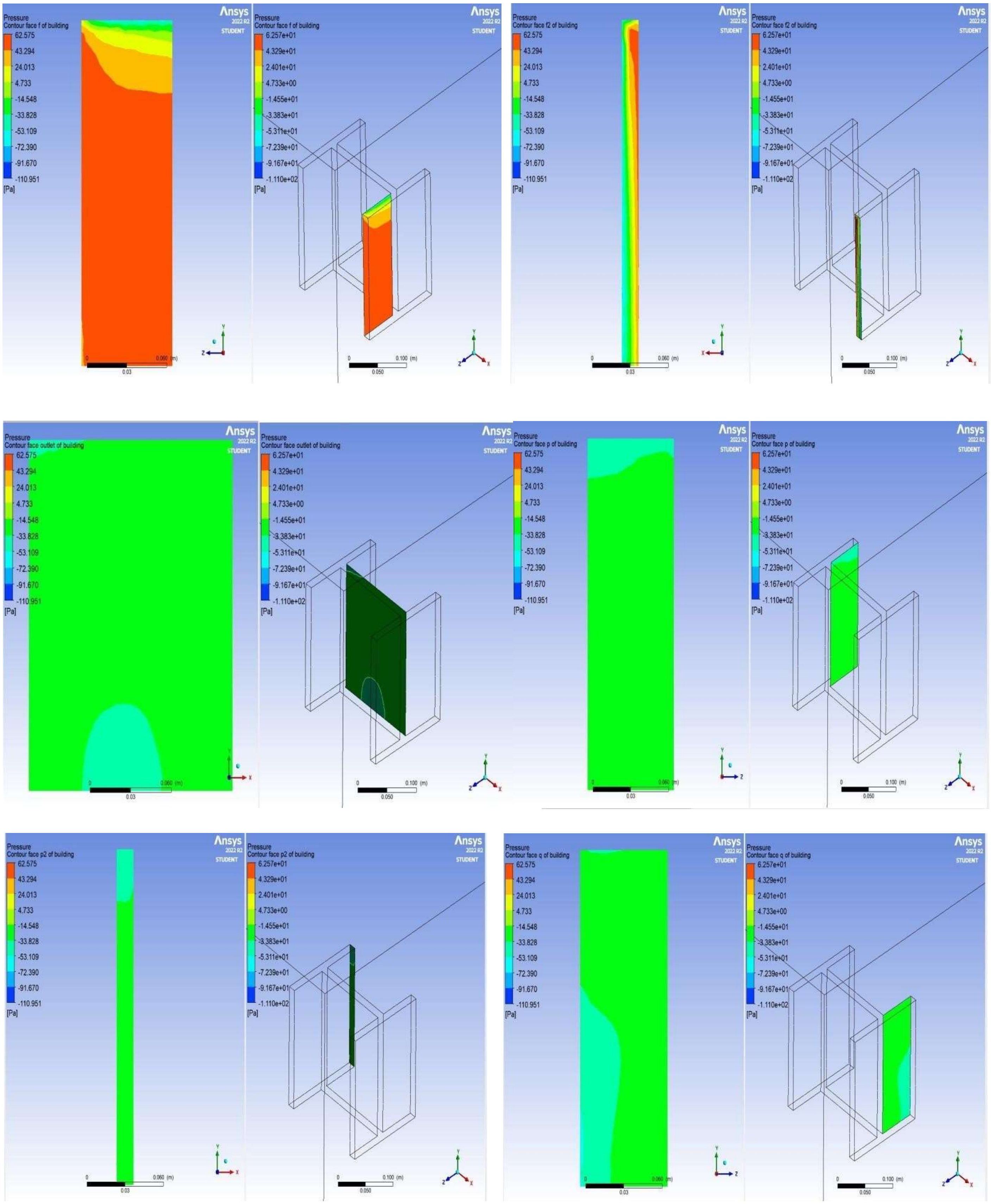


Top face

Fig. 15 Pressure Contour for Model C at 0° wind Incidence Angle for various faces

- **I-Shape building**




Ansys
2022 R2
STUDENT
Pressure
Contour face f of building
Contour face f2 of building
Contour face outlet of building
Contour face p of building
Contour face p2 of building
Contour face q of building
62.575
43.294
24.013
4.733
-14.548
-33.828
-53.109
-72.390
-91.670
-110.951
[Pa]
6.257e+01
4.329e+01
2.401e+01
4.733e+00
-1.455e+01
-3.383e+01
-5.311e+01
-7.239e+01
-9.167e+01
-1.110e+02
0.060 (m)
0.03
0.100 (m)
0.050

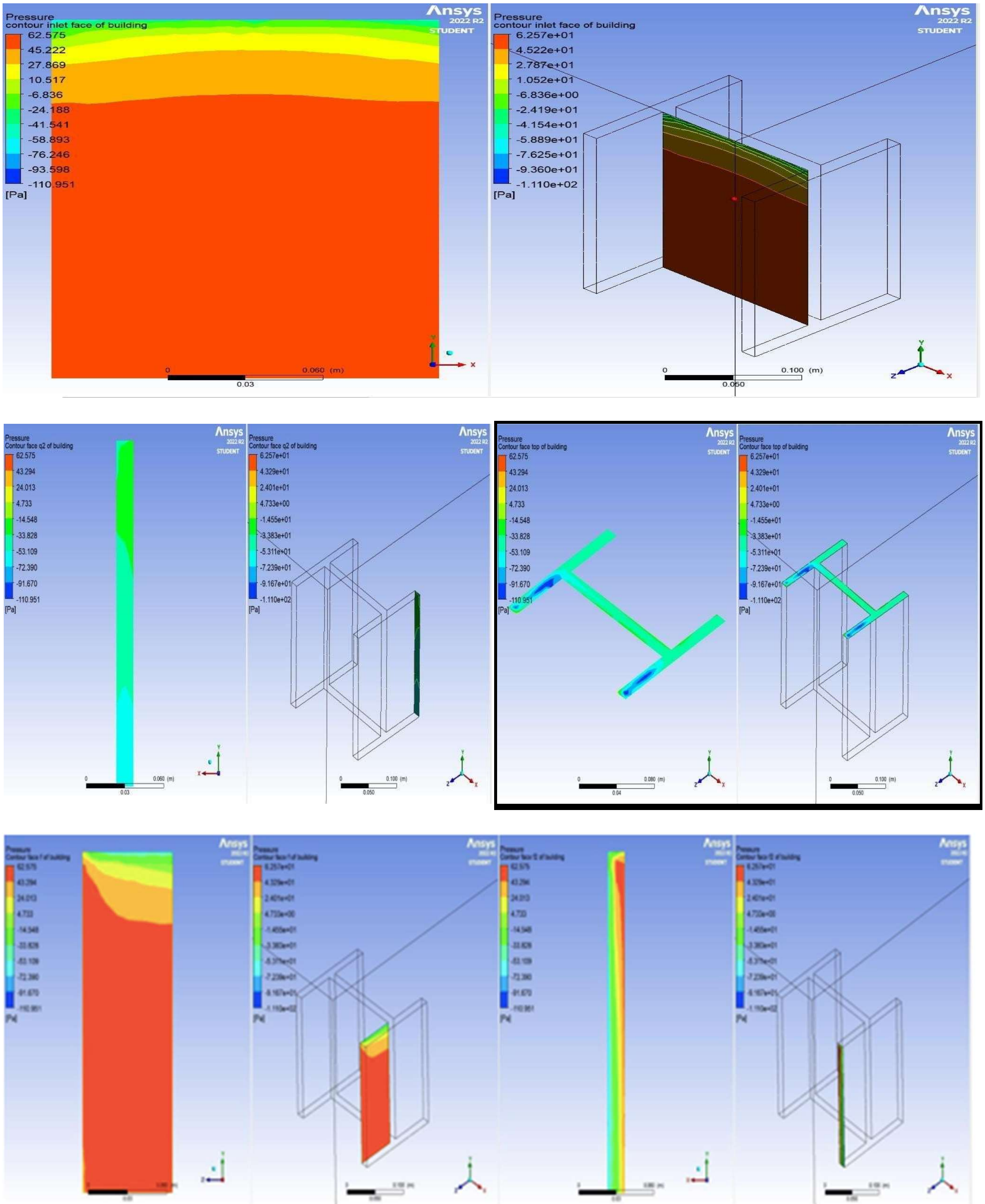
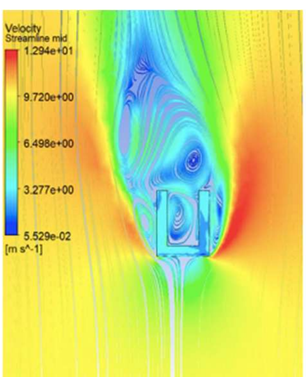


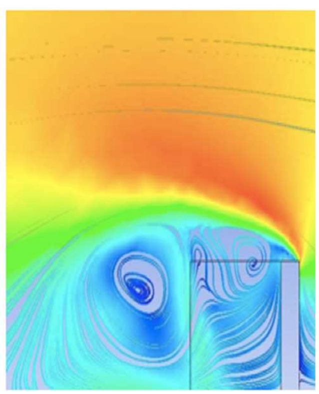
Fig. 16 Pressure Contour for Model I at 0° wind Incidence Angle

6.2 Velocity Streamlines

A streamline is an imaginary line in a fluid whose tangent at any point represents the direction of the velocity of a fluid particle at that location. Because various flow patterns are observed for all three models, and it is due to the variation in the plan shape of the building, the streamline for 0°wind incidence angle for models C and I in the plan and elevation is shownin the Figures below. In the wake zone, vortices are formed. On the leeward side of the building models, it will cause negative pressure. On the downstream side of building models for model C and model I, a vast recirculation zone of air can be seen

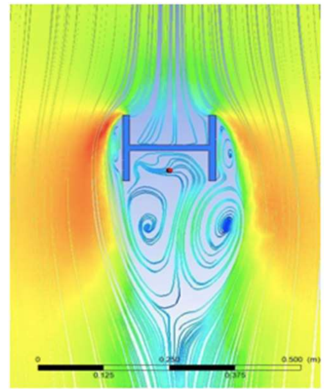
- C-shaped Model

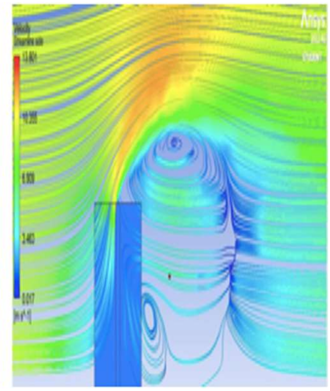




Horizontal streamlines Vertical Streamlines

- I-shaped Model





Horizontal streamlines Vertical Streamlines

6.1 Velocity Stream Lines in Plan and Elevation

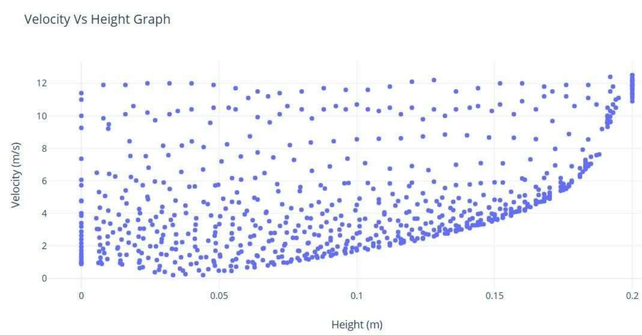


Fig. Velocity vs height graph for C-shaped building

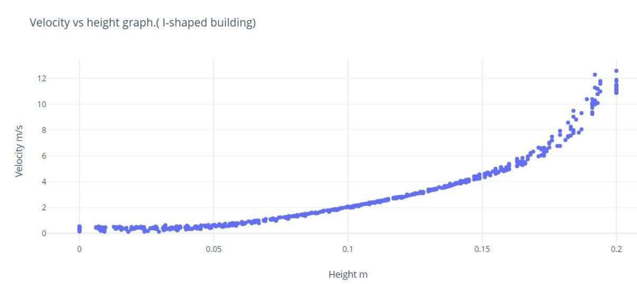


Fig. Velocity vs height graph for C-shaped building

6.2 Pressure Coefficient

The mean pressure coefficient 'Cp mean' is calculated from the equation

$$C_{p\ mean} = \frac{p - p_o}{\frac{1}{2}\rho U_H{}^2} \quad (2)$$

The mean Cp for Model C and Model I are presented below

Cp (Model C) = 0.824

Cp (Model I) = 0.70

7. CONCLUSIONS

There are several methods to know wind behaviour on high-rise buildings, like wind tunnel testing, etc. However, the present study used CFD in ANSYS: CFX, which gives accurate results.

The following conclusions are obtained from the current study:

- From the pressure contour, it is observed that the pressure values are maximum on the windward side in comparison to the leeward side.
- The building's top and one side surfaces show maximum negative pressure, from -110.951 to -91.670 Pascal, which means the pressure valueis maximum in magnitude on the top surface of the building compared to other faces.
- Pressure is positive only on the windward surfaces; the maximum positive pressure on the windward side is 43.294 to 62.575 Pascal, and the maximum negative pressure on the leeward side is -72.390 to -53.109 Pascal.
- Horizontal velocity streamlines have a lower value on the windward sides, while vertical velocity streamlines have a lower value on the leeward sides.
- The CFD approach is more economical than other methods like wind tunnel testing to know the effect of wind on the high or low-rise building, even if the geometry is complex.

References

[1] B. S. Taranath, Wind and Earthquake Resistant Buildings. 2004.

[2] E. Simiu and D. Yeo, Wind effects on structures, Fourth Edi. John Wiley & Sons Ltd, 2019.

[3] M. H. Günel and H. E. Ilgin, Tall buildings: Structural systems and aerodynamic form. 2014.

[4] G. Solari, Wind actions and effects on structures. 2019.

[5] IS: 875 (2015), Indian Standard design loads (other than earthquake) for buildings and structures-code of practice, part 3(wind loads). 2015.

[6] ASCE: 7-10(2013), Minimum Design Loads for Buildings and Other Structures. Structural Engineering Institute of the American Society of Civil Engineering, Reston. 2013.

[7] R. K. MEENA, G. P. Awadhiya, A. P. Paswan, and H. K. Jayant, "Effects of Bracing System on Multi-storeyed Steel Building," 2018, doi: 10.1088/1757-899X/1128/1/012017.

[8] 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. 0123456789, 2021, doi: 10.1007/s12046-021-01592-6.