International Journal of Advanced Trends in Computer Applications
www.ijatca.com

Study of wind flow pattern around the building with different plan irregularities using computational fluid dynamics

[1]R.Vigneshwaran, [2]Dr.S.Prabavathy
[1]Department of Civil Engineering
Mepco Schlenk Engineering College, Sivakasi, Tamil Nadu, India
[1]*vignesh.rajen.93@gmail.com*

Abstract: *The motivation behind this study is that of tall structures greater than 45m height is rising in India. Due to the development of such skyscrapers, it results in improper air flow movement in and around the urban environment and hence, finding of actual wind flow pattern is necessary for upcoming construction. For the above study, conducting wind tunnel tests experimentally is too costlier and also time-consuming. To overcome these difficulties, Computational Fluid Dynamics (CFD) plays a useful role in calculating the actual wind flow on the structure. From the results obtained in CFD, the modification can be done in the elevation profile façade features, plan, shape and height of the building such that structure is able to attract minimum wind drag effects. Behavior of air is like a Newtonian fluid and it is important to study fluid-structure interaction. This paper presents an air flow moving around the building for different regular and irregular building plans. Modeling is carried out by using SolidWorks software and wind flow pattern is studied by using Ansys Fluent software. For the present study, different shapes such as a rectangle as regular plan and T-shape, L-shape as irregular plans are chosen. From the results, the flow pattern and wake regions are identified.*

Keywords: Aerodynamic, Computational fluid dynamics, Wind force.

I. INTRODUCTION

India is one of the fastest developing countries in the world with increasing development of new skyscrapers and Mega infrastructure project. With the increase in population, lack of land availability in city and development of new materials has led to such development of skyscrapers. From the various surveys, it is found that 170 skyscrapers more than 100+ meter height will be possessed by 2020. These skyscrapers will change the elevation profile of many cities and the issue of wind flow pattern arises. The maximum wind load calculation is introduced by Davenport in the year 1961 and it is valid till date (Haria Hangan et al. (2017)). Davenport calculated the wind load as a guest factor that is the ratio of maximum wind speed between two different averaging times 't' and 'T' given by the formula, $G = 1 - 0.6226\, I_v^{1.2716}\, \mathrm{Ln}\,(t/T)$.

1.1 Aerodynamic forces in buildings

All engineering structures present above the earth's surface are subjected to aerodynamic forces. Aerodynamic forces for buildings include drag force, cross-wind and torsion moment by Amin J.A and Ahuja (2010). Drag force acts along with the wind in the direction of mean flow and will result in pressure fluctuation on windward and leeward sides of the building. Cross-wind forces act perpendicular to the direction of mean wind flow which tends to lift the body. Torsion motion is developed due to the imbalance in the instantaneous pressure on each side of the building, which causes a twist in the body structure. (Bert Blocken et al. (2011)).

Figure 1 shows the direction of various forces acting on a building. Vortex shedding is associated with cross-wind motion, which is an oscillation flow that takes place when a buff body such as a building. The aerodynamic shape of the building is the key deciding parameter for wind flow around the building. Computational fluid dynamics plays, a vital role in the study of fluid-structure interaction.

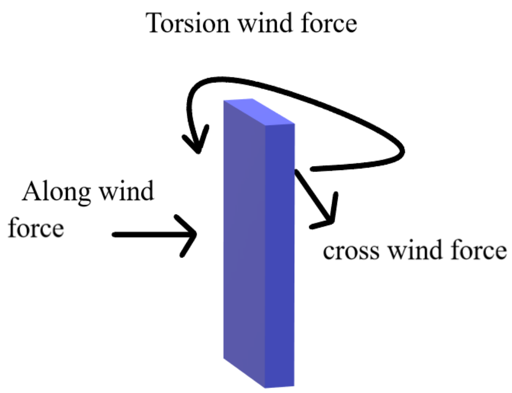

Figure 1: Shows the different wind force around the buildings

II. COMPUTATIONAL FLUID DYNAMICS

In general, computational fluid dynamics is a method to predict fluid flow, heat and mass transfer, chemical reaction and related phenomena by solving numerically a set of governing mathematical equations stated by Bert Blocken et al. (2007). It is based on the law of conservation of mass, momentum, energy, species and effect of body flow.

2.1 Meshing

Meshing can be done in the form of a grid. In order to predict the flow in and around the building with acceptable accuracy, the most important thing is to characterize the flow separation near the boundaries of the building. Therefore, a fine grid arrangement is required to resolve the flow near the corners. The minimum grid resolution should be set about $1/10^{th}$ of the building scale (Y.C.Kim et al. (2015); Chunming Liu et al. (2017)). Grid dependence upon the solutions and, it should confirm that the prediction of result does not change significantly with a different grid system. Here, fine mesh is adopted for all the models in ANSYS Fluent.

2.2 Turbulence models

Turbulence is a violent or unsteady movement of a fluid. This model is widely used for the movement of air in and around the building. A small scale and high-frequency fluctuation velocity are characterized in the turbulent flow model. Since 100% accurate models cannot be developed, fluid mechanics equations are averaged to account for these small fluctuations (Enrica Bernardini et al. (2015). There are three major groups of turbulence models: 1) RANS models (Reynolds Average Navier-Stokes model) 2) LES model (Large Eddy Simulation models). 3) Direct Numerical Solutions (DNS). Most of the CFD modeling software uses k-Є model, which is adoption in CFD.

In the LES model, the air movement is wider and smaller eddies are filtered. RANS solve Navier-Stokes equation by either time average. DNS model solves Navier-Stokes equation and hence theoretically, all turbulent flows can be simulated. RNS can be applied to simple flow problems, usually in a laminar flow, where the Reynolds number is relatively low (Chunming Liu et al. (2017)). K-Omega SST turbulence model is well suitable to study the flow around the building.

2.3 Boundary condition

In fluid flow problem, it is very much important to have a properly defined boundary condition for good results (Lorenzo Raffaele et al. (2017)) The boundary condition includes a condition for inlet, outlet, wall and constant pressure. For inlet boundary condition, the flow velocity needs to be specified at the inlet. (Okafor Chinedum Vincent (2017)) Here there is no slip condition and velocity of flow and normal to the flow direction are set to zero.

2.4 Convergence of solution

The calculations have to be finished after the convergence of the solution. It is important to confirm that the solution does not change by monitoring the variables on specified points (Georgios K. Ntinas et al. (2017). When the calculation diverges or convergence is slow, the following points need to be examined. The aspect ratio and stretching ratio of the grids may be too large. The relaxation coefficient of the matrix solver may be too small. Periodic fluctuations such as a vortex shedding may occur.

III. RESULTS AND DISCUSSION

An Analysis is carried out as per the above procedure for the different shapes of the buildings and the results are displayed in the form of graphs. Inlet condition is set up with a velocity of 7 m/s and the outlet gauge pressure is of 0 Pa. No slip is given to the wall boundaries.

3.1 Square shape rectangle building

Wind flow on A Face for rectangular plan of the building

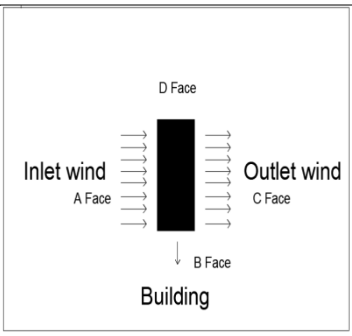

Figure 2: Wind force on A face of the building

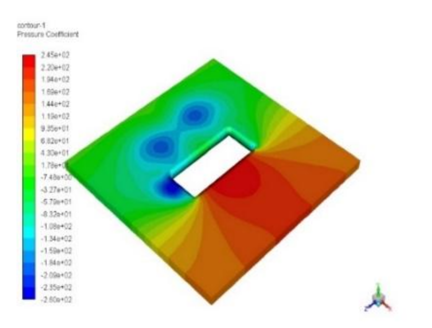

Figure 3: Distribution of Pressure coefficients at α = 0˚

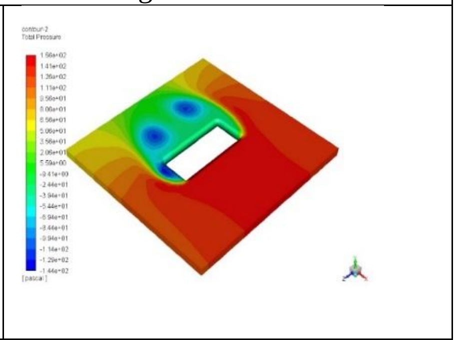

Figure 4: Distribution of Total pressure coefficients at α = 0˚

Wind flow on B Face for rectangular plan of the building

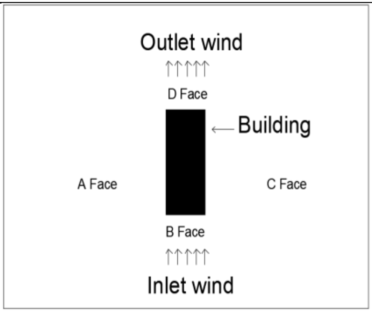

Figure 5: Wind force on B face of the building

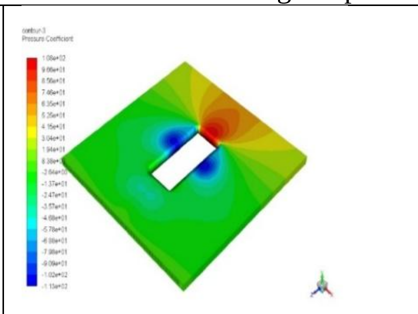

Figure 6: Distribution of Pressure coefficients at α = 45˚

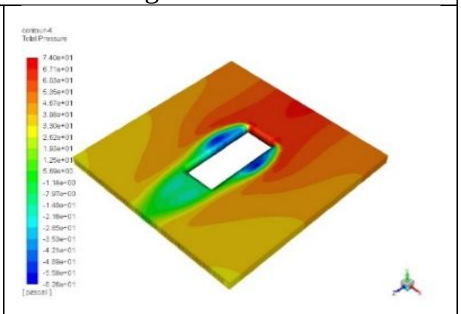

Figure 7: Distribution of Total pressure coefficients at α = 45˚

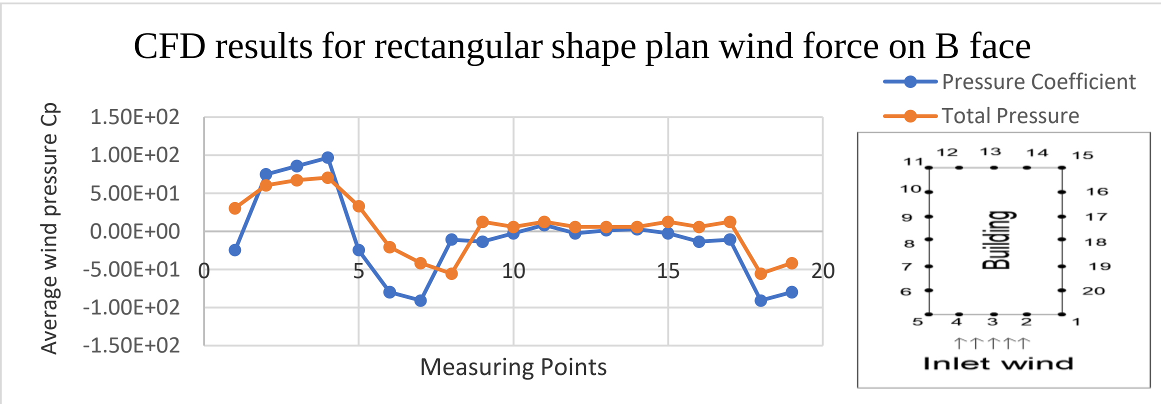

Figure 8: Wind pressure coefficients of CFD simulation measuring at different points

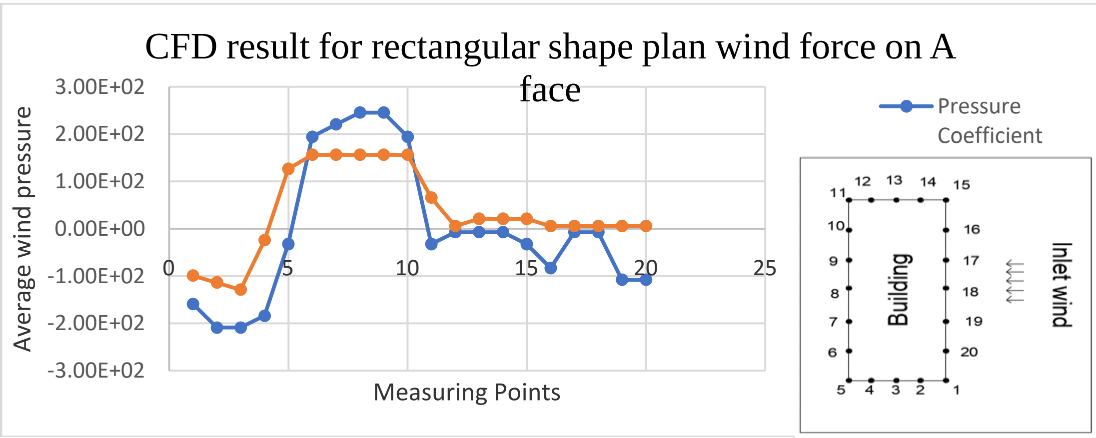

Figure 9: Wind pressure coefficients of CFD simulation measuring at different points

For rectangular shape building, the wind force of about 7 m/s with an angle of incidence α= 0˚ is applied on the A face as shown in fig 2. It is observed that pressure coefficient C_p is maximum on the C face and it is found to have very low pressure and wake on the B face as shown in fig 3. Since the maximum wind force is observed on the A face, when compared to the other three faces of the structure it may sometime result in a twisting of the structure. It may be noted that the result in the A face shows a perfect separation of flow which has led to the formation of wake regions slightly away from the surface of the building. The pressure is less on both the side corners of the building. The total pressure is a combination of static pressure, pressure coefficient, dynamic coefficient, absolute pressure and the result is shown in fig 4. When the wind force is applied on the D face of the building with an angle of α = 45˚ as shown in the fig 5, it is observed that the wake regions are created on the building surface on C and B face as shown in fig 6 and fig 7 shows the total pressure. Fig 8 and 9 represent the various measuring points in a building where the forces are acting. The graphs are plotted between measuring points and average wind speed at various locations in a building and compared with total pressure and pressure coefficient.

3.2 L- shape building

For L shape building, 7 m/s wind force is applied with an angle of incidence α = 0˚ on the C face as shown in fig 10. It is been observed that pressure coefficient C_p is maximum on the C face and it is found to have very low pressure and a wake is found on the D and A face as shown in fig 11. It may be noted that the result in the A face shows a perfect separation of flow that had occurred leading to the formation of wake regions a little away from the surface of the building as shown in fig 12. The pressure is less on both the side corners of the building. When the wind force is applied on the A face of the building with an angle of α = 90˚ as shown in fig 13, it is observed that the wake regions are created on the building surface on C face as in fig 14 and D face absorbs low pressure on the face of the building as shown in the fig 15. Fig 16 and 17 show the various measuring points in a building where the forces are acting. The graphs are plotted between measuring points and average wind speed at various location in a building and compared with total pressure and pressure coefficient.

Wind flow on C Face for L shape plan of the building

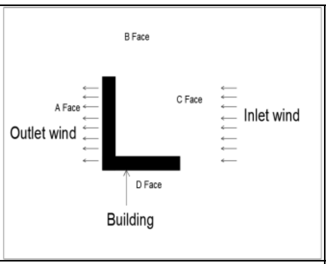

Figure 10: Wind force on C face on the building

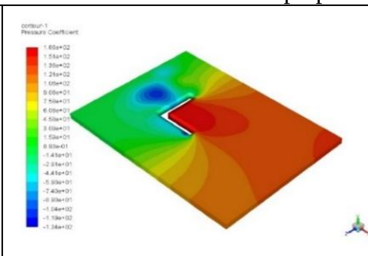

Figure 11: Distribution of Pressure coefficients at α = 0˚

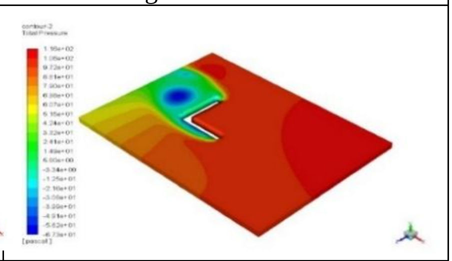

Figure 12: Distribution of Total pressure coefficients at α = 0˚

Wind flow on A Face for L shape plan of the building

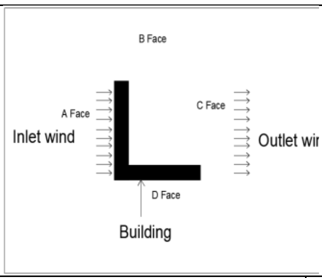

Figure 13: Wind force on A face on the building

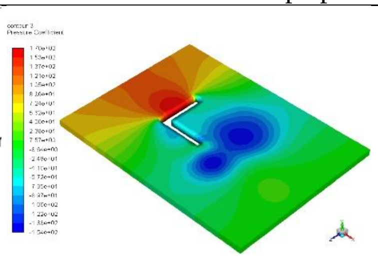

Figure 14: Distribution of Pressure coefficients at α = 90˚

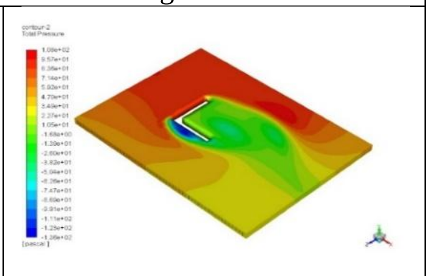

Figure 15: Distribution of Total pressure coefficients at α = 90˚

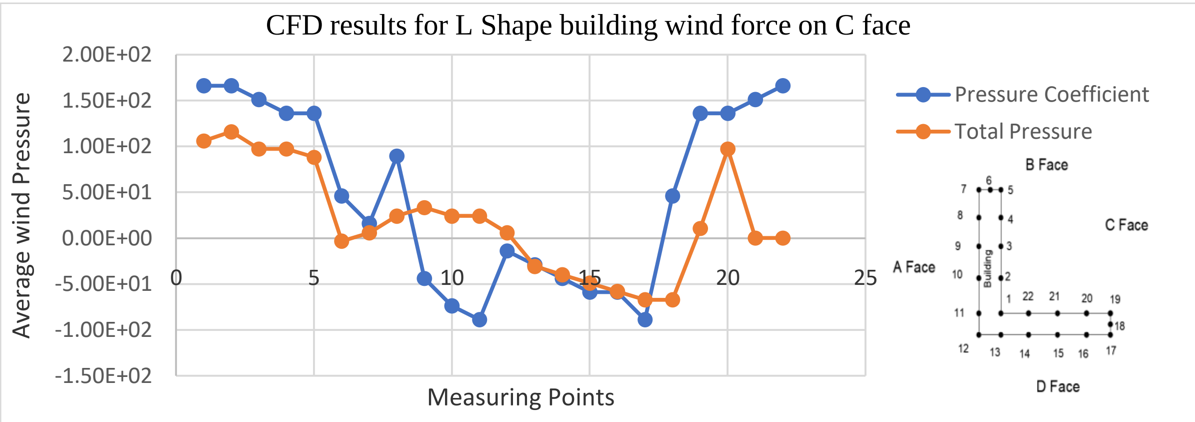

Figure 16: Wind pressure coefficients of CFD simulation measuring at different points

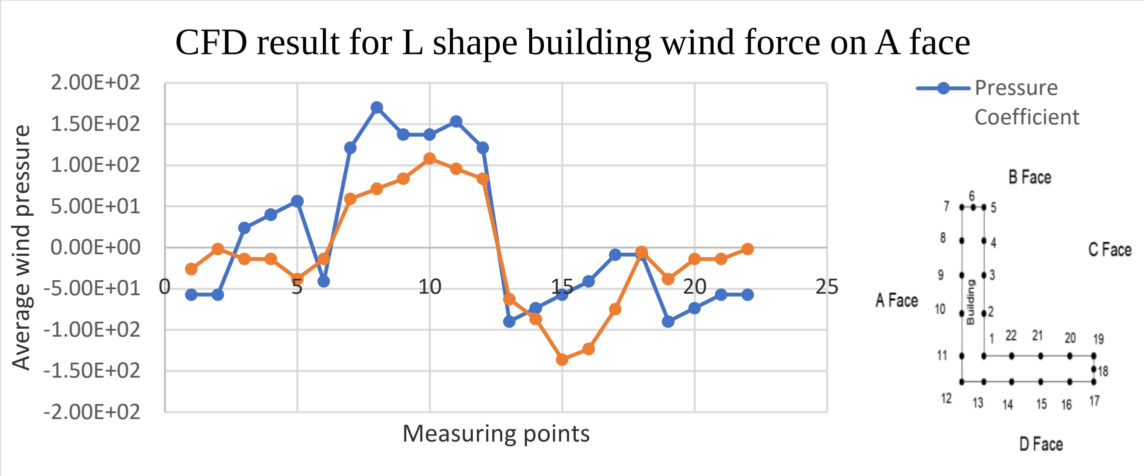

Figure 17: Wind pressure coefficients of CFD simulation measuring at different points

3.3 T- shape building

Wind flow on A Face for T shape plan of the building		
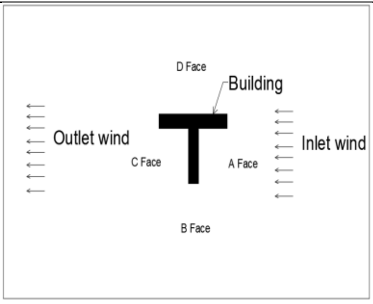	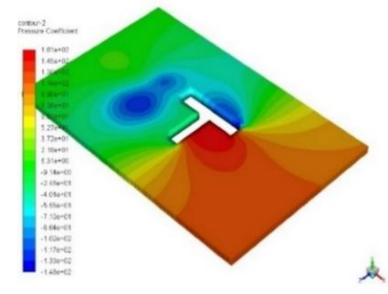	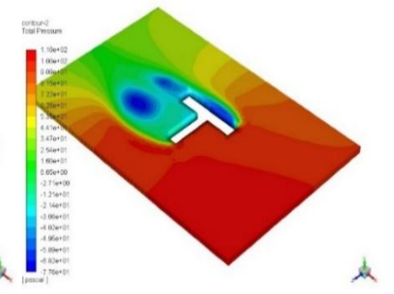
Figure 18: Wind force on A face of the building	**Figure 19:** Distribution of Pressure coefficients at α = 0˚	**Figure 20:** Distribution of Total pressure coefficients at α = 0˚

Wind flow on B Face for T shape plan of the building		
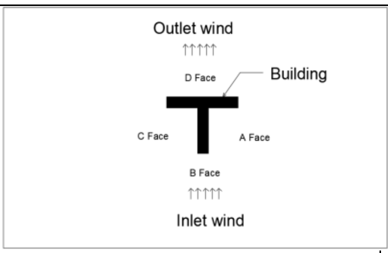	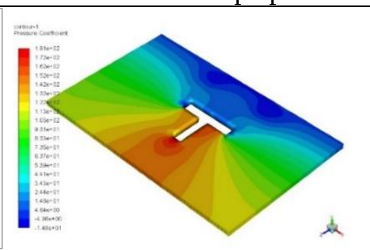	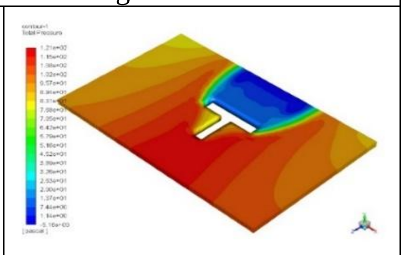
Figure 21: Wind force on B face of the building	**Figure 22:** Distribution of Pressure coefficients at α = 45˚	**Figure 23:** Distribution of Total pressure coefficients at α = 45˚

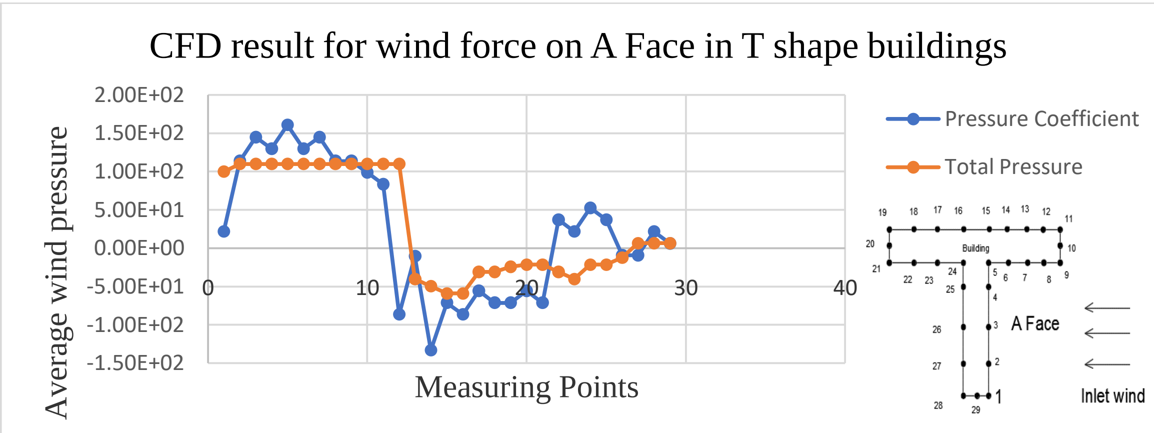

Figure 24: Wind pressure coefficients of CFD simulation measuring at different points

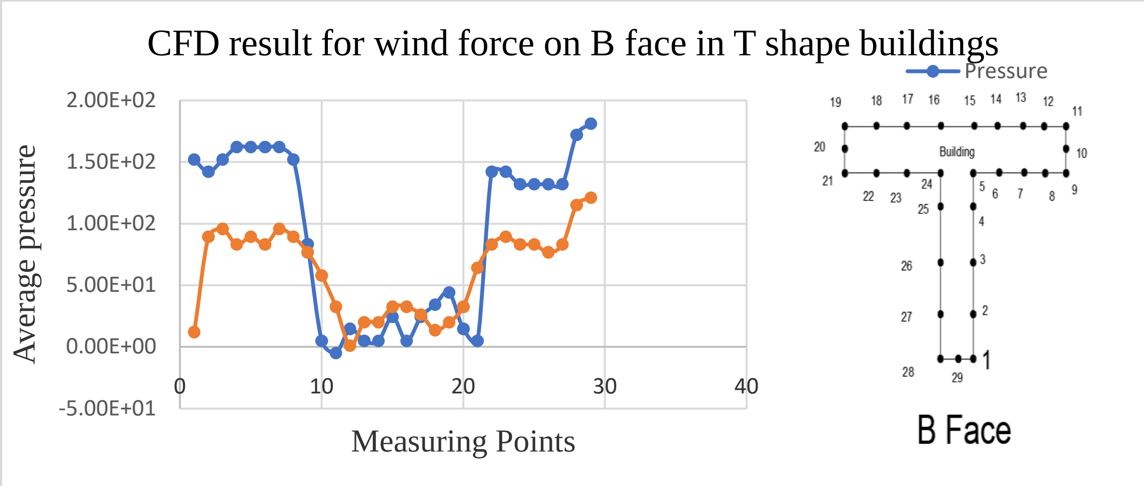

Figure 25: Wind pressure coefficients of CFD simulation measuring at different points

For T shape building, the wind force of about 7 m/s with an angle of incidence $\alpha = 0^{\circ}$ is applied on the A face as shown in fig 18. It is observed that the pressure coefficient C_p is maximum on the A face and very low pressure and wake are found on the D and C face as shown in fig 19. When total pressure is applied on the A face low pressure is developed and two wake regions are formed in the D face as shown in the fig 20. When the wind force is applied on the D face of the building with an angle of $\alpha = 45^{\circ}$ from A as shown in fig 21, it is observed that the low pressure is created on the D face as shown in fig 22 and its combined wind force is shown in fig 23. Force measuring points in various location of the building is given in fig 24 and 25. The graphs are plotted between measuring points and average wind speed at various locations in a building and compared with total pressure and pressure coefficient.

IV. CONCLUSION

Studying the behaviour of wind is a complex process and predicting the flow pattern is difficult, but nowadays due to the development of supercomputers, the CFD tool finds a solution to study the wind flow pattern in and around the structures. In my study K-Ω SST turbulence model is chosen for measuring the flow around the building. From the above results, for different plan irregularities in buildings, the formation of vortex flow and wake regions in the buildings are identified. From the results, modifications can be done in-order to avoid the formation of wake regions during the air flow moving around the buildings. Building setback limits can be fixed based on the results obtained from the CFD in-order to maintain a constant wind flow pattern within an area. The comfort of the occupants inside a building will depend upon the wind flow in and around the building.

Conflict of Interest

The authors confirm that there is no conflict of interest to declare for this publication.

Acknowledgement

Authors would like to thank Mepco Schlenk Engineering College (Autonomous), Sivakasi, for providing sufficient facilities to carry out the research study.

References

[1]. Amin J.A., & Ahuj`a A.K. (2010). Aerodynamic Modifications to the shape of the building: A Review to the state of the art. *Asian Journal of civil Engineering (Buildings and Housing)*, Vol.11, No.4, pp. 433-450.

[2]. Bert Bocken, Ted Stathopoulos, Jan carmeliet., & Jan L.M.Hensen. (2011). Application of computational fluid dynamics in building performance simulation for the outdoor environment: an overview. *Journal of Building Performance Simulation* Vol. 4, No.2, June 2011, pp. 157-184.

[3]. Bert Blocken, Ted Stathopoulos., & Jan Carmeliet. (2007). CFD simulation of the atmospheric boundary layer: wall function problems. *Journal of Atmospheric Environment* Vol. 41, No.2 pp. 238-252.

[4]. Y.C.Kim, E.K.Bandi, A.Yoshida., & Y.Tamura. (2015). Response characteristics of super tall building – Effects of number of sides and helical angle. *Journal of Wind Engineering and Industrial Aerodynamics* Vol.145, pp. 252-262.

[5]. Chunming Liu, Liang Liu., & Chengbin Liu. (2017). Analysis of wind resistance of high-rise building structures based on computational fluid dynamics simulation technology. *International Journal of Heat and Technology* Vol.36, no. 1, March, 2018, pp.376-380.

[6]. Enrica Bernardini, Seymour M.J.Spence, Daniel Wei., & Ahsan Kareem (2015). Aerodynamic shape optimization of civil structures: A CFD enabled Kriging based approach. *Journal of Wind Engineering and Industrial Aerodynamics* Vol.144, pp. 154-164.

[7]. Chunming Liu, Liang Liu., & Chenghin Liu (2018). Analysis of wind resistance of high-rise building structures based on computational fluid dynamics simulation technology. Vol. 36, No.1, March, 2018, pp.376-380.

[8]. A.Kubilay, J.Carmeliet., & D.Derome (2016). Computational fluid dynamics simulation of wind driven rain on a midrise residential building with various types of façade details. *Journal of building Performance Simulation* ISSN:1940-1493.

[9]. Lorenzo Raffaele, Luca Bruno, Davide Fransos., & Franco Pellerey (2017). Incoming windblown sand drift to civil infrastructure: A probabilistic evaluation. *Journal of wind Engineering and Industrial Aerodynamics*, Vol.166, pp.37-47.

[10]. Okafor Chinedum Vincent (2017). Application of computational Fluid Dynamics Model in High-Rise Building Wind Analysis – A Case Study. *Advance in science, Technology and Engineering systems Journal* Vol.2, No.4, pp. 197-203.

[11]. Georgios K.Ntinas, Xiong Shen, Yu Wan., & Guoqiang Zhang (2017). Evaluation of CFD turbulence models for simulating external airflow around varied building roof with wind tunnel experiment. *Journal of building simulation* Vol.11, No.1, pp. 115-123.