Numerical Evaluation of Wind Pressure on Surfaces of Bluff Body

Jigar K. Sevalia*, Dr. Atul K. Desai, Dr. S. A. Vasanwala*****

*(Ph. D. Research Scholar at Applied Mechanics Department, Sardar Vallabhbhai National Institute of Technology, Ichchhanath,Surat – 395 007, Gujarat, India.)
**(Associate Professor at Applied Mechanics Department, Sardar Vallabhbhai National Institute of Technology, Ichchhanath, Surat – 395 007, Gujarat, India)
***(Associate Professor at Applied Mechanics Department, Sardar Vallabhbhai National Institute of Technology, Ichchhanath, Surat – 395 007, Gujarat, India)

ABSTRACT

Wind is the flow of gases on a large scale. On Earth, wind consists of the bulk movement of air. Wind is caused by differences in pressure. In the field of structural engineering it includes strong winds, which may cause discomfort, as well as extreme winds, such as in a tornado, hurricane or heavy storm, which may cause widespread destruction. The wind engineering community has addressed the Computational Wind Engineering (CWE) as a field from last three decades to evaluate the interaction between fluid and building numerically. The study of flow around bluff bodies of rectangular shape has a deep engineering interest because many civil and industrial structures can be assimilating to this shape. A comprehensive numerical study of wind effects on the surfaces of Bluff Body is presented in this paper. The techniques of Computational Fluid Dynamics (CFD), such as Standard k-ε Model was adopted in this study to predict wind loads on surfaces of Bluff body and wind flow around Bluff Body. To study the wind effect, 3-D Wind Flow condition around Bluff Body has been developed using Computational Fluid Dynamics (CFD) Code namely Fluent / Gambit and then numerical computation has been executed to evaluate wind pressure values and the wake region around the bluff body.

Keywords - **Bluff Body, Computational Fluid Dynamics (CFD), Drag Force, Force Coefficient, Pressure, Structural Engineering, Turbulence Model, Wind Engineering,**

I. INTRODUCTION

Computational Wind Engineering (CWE) as a branch of Computational Fluid Dynamics (CFD) has been developed rapidly over the last three decades to evaluate the interaction between wind and structures numerically, offering an alternative technique for practical applications [1]. The techniques of CFD have been widely used to predict wind flow around bluff bodies in wind engineering. Gloria Gomes M., Moret Rodrigues A. and Pedro Mendes have examined the results of numerical analysis and flow over cubical obstacle [2].

Modeling the wind atmosphere associated with proposed or existing buildings is of great importance for the Wind Engineering, Civil Construction sectors as well as Structural Engineering Sectors. The potential market for wind engineering studies around buildings is very large. CFD simulations can provide information on all flow parameters in the entire computational domain. Moreover, a reliable numerical evaluation of the interaction between fluid namely wind and buildings can be achieved with CFD modeling in a time- saving as well as economic manner. Thus, CFD can offer more flexibility when exploring a variety of building designs and modifications and their impact on the flow around them. CFD could also potentially supersede traditional wind tunnel studies as a more cost-effective and powerful design tool for wind engineering studies. However, wind tunnel studies have been proved quite useful for development, evaluation, validation and general performance assessment of CFD methods. The distribution of the fluctuating surface pressure and the wind forces acting on bluff shaped bodies are of great practical interest in the field of structural design in wind engineering.

Here, in this paper an attempt has been made to determine numerically the wind pressures on the surfaces of Square Plan Shape Bluff Body using Computational Fluid Dynamics Code namely Fluent / Gambit.

II. METHODOLOGY

In order to study the effect of Wind on Bluff Body with respect to Wind Pressure, wake region, drag force, etc., a Bluff Body with square plan shape having dimensions as shown in figure 1 has been considered. The height of the Bluff Body considered is 186 mm. To execute study, Computational Fluid Dynamics Code namely Fluent and Gambit have been used.

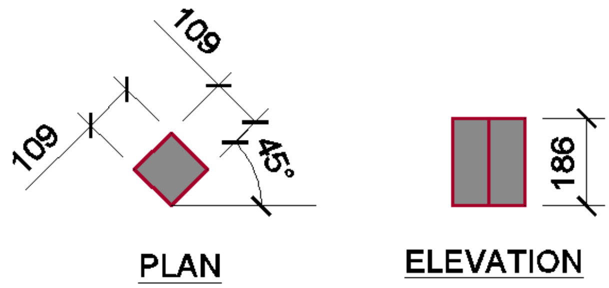


Fig. 1 Plan and Elevation View of Bluff Body

The location of Model in Computational Domain is as shown in figure 2.

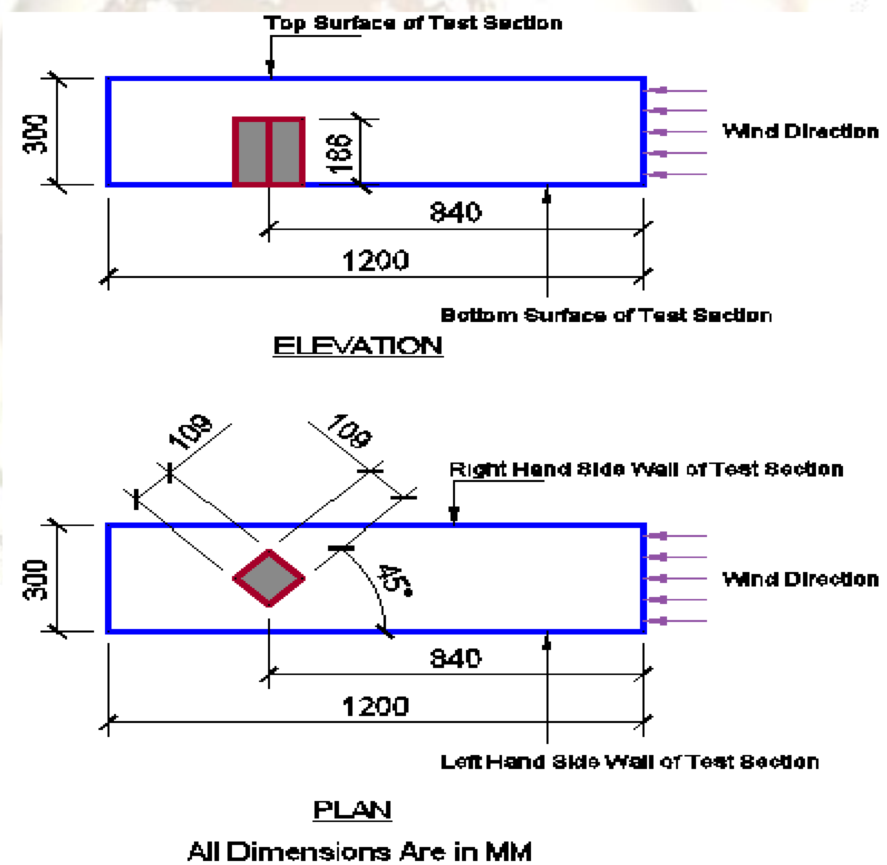


Fig. 2 Location of Model in Computational Domain of Fluent

III. DATA CONSIDERED IN NUMERICAL (CFD) ANALYSIS

- Size of Fluid (Computational) Domain : 300 x 300 x 1200 mm
- Type of Fluid : Air
- Density of Air : 1.225 kg/m^3
- Viscosity of Air : 1.7894e^{-05} kg/m.sec
- Size of Model : 109 x 109 x 186 mm
- Operating Pressure : 101325 Pascal
- Model : Standard k-ϵ Model
- Solver : Pressure Based
- Inlet Velocity : 8 m/sec

IV. DOMAIN SIZE

There are no explicit rules dictating the size of a computing domain [3]. For this study, size of the computational domain considered is 1200 mm x 300 mm x 300 mm in the longitudinal (X), lateral (Z), and vertical (Y) directions, respectively as shown in figure 2, 3, 4 and 5. Here, computational domain is shown along with Square Plan Shape Bluff Body.

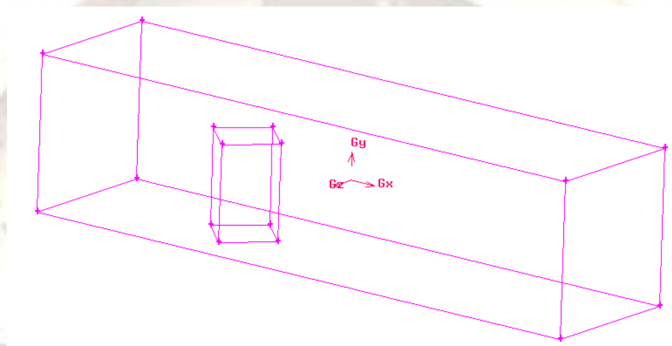


Fig. 3 3- D View of Computational Domain along with Bluff Body Model

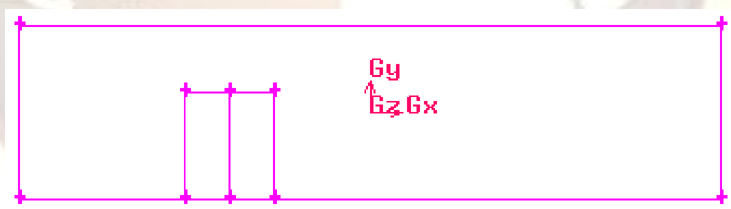


Fig. 4 View of Computational Domain in X-Y Plane (Vertical Plane)

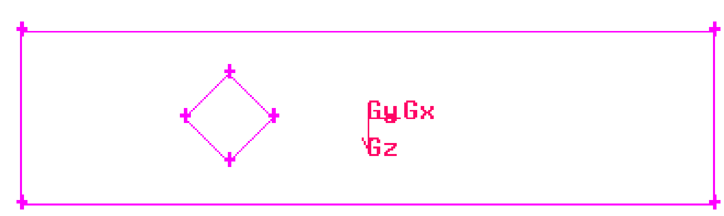


Fig. 5 View of Computational Domain in X-Z Plane (Horizontal Plane)

V. BOUNDARY CONDITIONS

The boundary conditions for the computational domain is considered as follows,

- The ground at the bottom of the computational domain was simulated with a smooth wall using log law wall function.
- The free slip boundary conditions are applied to top and side surfaces of computational domain. The flux normal to the boundary is considered zero.
- The no slip boundary conditions are applied to the surfaces of Building Model.

VI. COMPUTATIONAL GRID

3-D Unstructured grids are created in the computational domain. The Computational Grid Patterns for the Building Unit situated in computational domain is shown in figure 6 and figure 7.

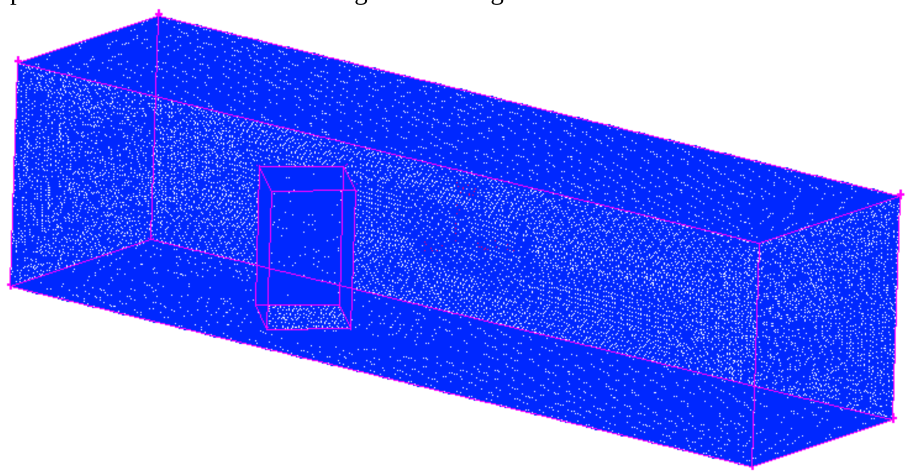


Fig. 6 Computational Grids in X-Z Plane (Horizontal Plane) in Gambit

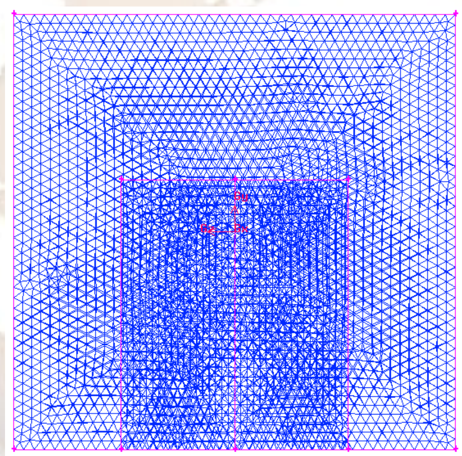


Fig. 7 Computational Grids in X-Y Plane (Vertical Plane) in Gambit

VII. RESULTS OBTAINED BY CFD ANALYSIS USING GAMBIT / FLUENT

In order to study the variation in Wind Pressure on surfaces of Bluff Body, different points are located on surfaces of Bluff Body as shown in figure 8 and 9. At all these points, wind pressures are obtained in post processing of Fluent. The results interpreted from the Pressure Contours obtained in CFD Analysis are shown in table 1.

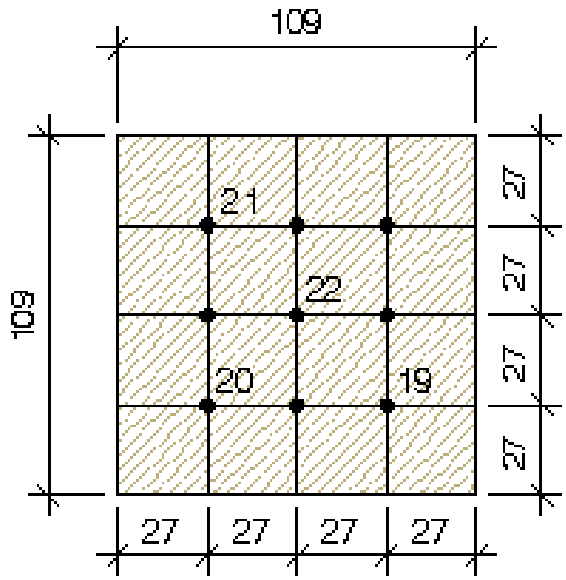
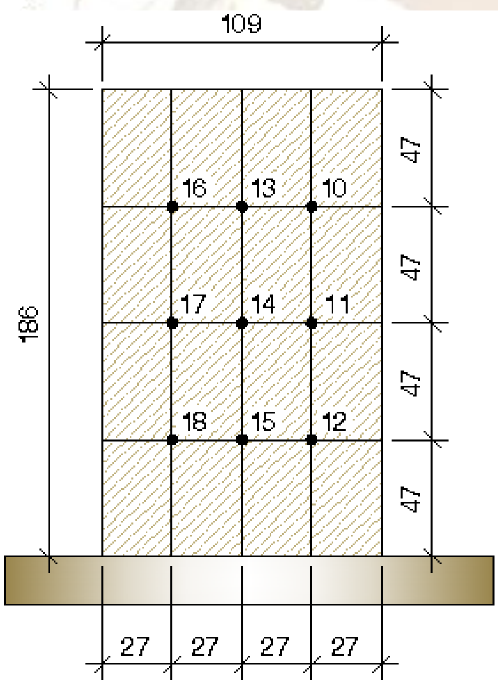
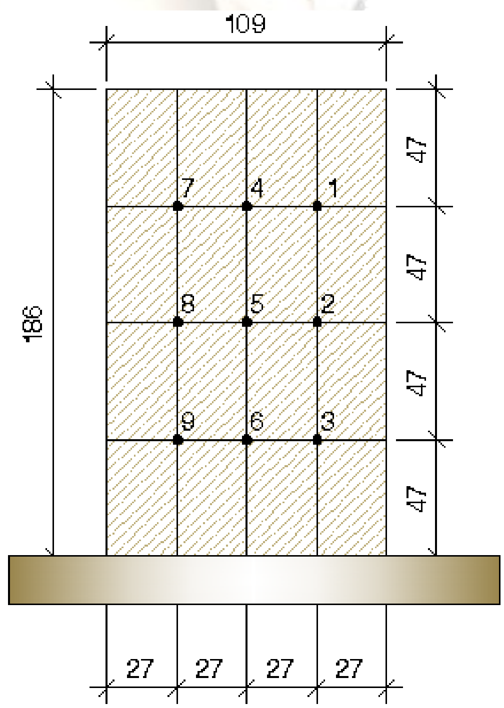


Fig. 8 Points Identity at Top Surface of Model



Leeward Face



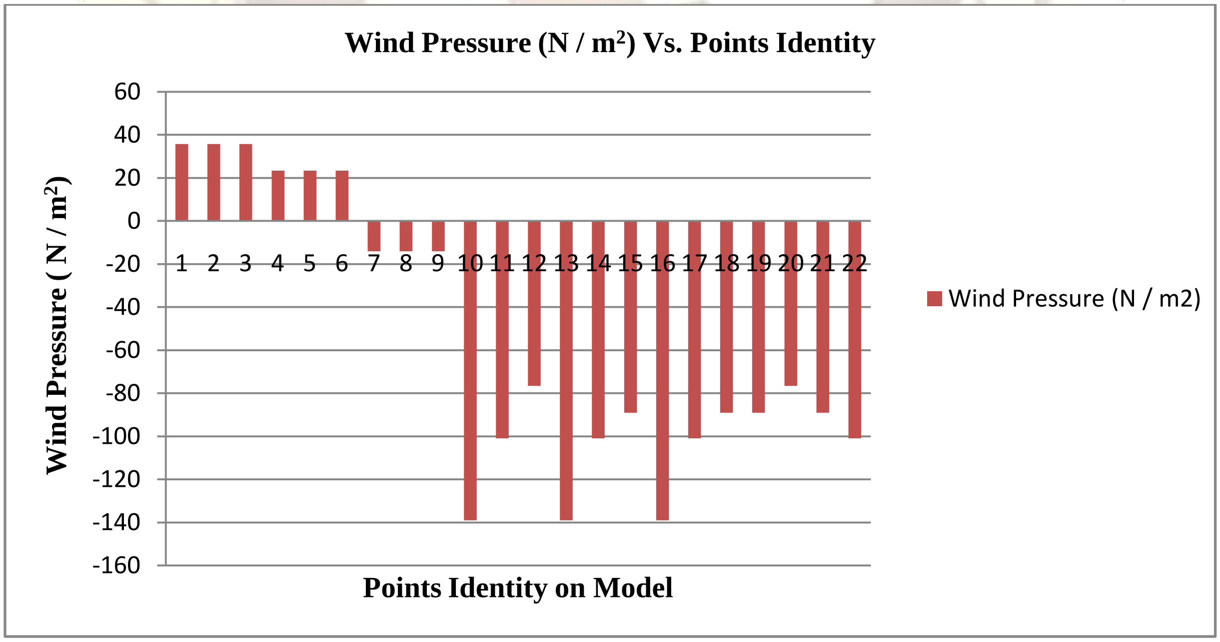
Windward Face

Fig. 9 Points Identity on Side Faces of Model

Table 1 Numerical Analysis Results

Points Identity on Model	Static Wind Pressure (N/m^2)	Points Identity on Model	Static Wind Pressure (N/m^2)
Wind Ward Face		Leeward Face	
1	35.8	10	-139
2	35.8	11	-101
3	35.8	12	-76.5
4	23.4	13	-139
5	23.4	14	-101
6	23.4	15	-89
7	-14.1	16	-139
8	-14.1	17	-101
9	-14.1	18	-89
Top Surface			
19	-89	21	-89
20	-76.5	22	-101

The results tabulated in above table are shown graphically in Graph 1



Graph 1 Wind Pressure vs. Points Identity on Model

VIII. CONCLUSION

From the table 1 and graph 1, it can be seen that positive wind pressure is developing on point number 1 to 6 and thereafter negative wind pressure is developing on remaining points considered on model. The points 1 to 3 are near to sharp windward edge; hence maximum positive pressure i.e. 35.8 N/m^2 is developed as the flow is steady undisturbed and directly impinging the windward face of model. The points 4 to 6 are at middle of windward surface i.e. away from sharp wind ward edge where wind pressure is less i.e. 23.40 N/m^2. The negative pressure is developed on points 7 to 9 of wind ward face i.e. -14.10 N/m^2. This is due to separation of flows and development of vortices. Of course, the vortices are weak in nature and hence suction effect is less.

The points 10 to 18 are on leeward face of model, where maximum negative pressure (suction pressure) is developed due to formation of strong vortices in horizontal and vertical plane. The flow has become turbulent on leeward side of model and hence suction effect is more.

The points 19 to 22 are on top surface of model. Here, from table 1, it can be seen that point 19 is near windward face and point 21 is near leeward face where the negative pressure is less compared to negative pressure at point 22 which is at the centre of top surface. It indicates the level of turbulence in flow is less near windward and leeward corner face and it is more near the centre of top surface.

REFERENCES

[1] R. Yoshie, A. Mochida, Y. Tominaga, H. Kataoka, K. Harimoto, T. Nozu and T. Shirasawa, “Cooperative project for CFD prediction of pedestrian wind environment in the Architectural Institute of Japan”, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 95 (2007), pg. 1551–1578.
[2] Gloria Gomes M., Moret Rodrigues A. and Pedro Mendes, “Experimental and numerical study of wind pressures on irregular-plan shapes”, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 93 (2005), pg. 741–756.
[3] Bert Blocken, Jan Carmeliet and Ted Stathopoulos, “CFD evaluation of wind speed conditions in passages between parallel buildings—effect of wall-function roughness modifications for the atmospheric boundary layer flow”, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 95 (2007), pg. 941–962.
[4] Aishe Zhang, Cuilan Gao and Ling Zhang, “Numerical simulation of the wind field around different building arrangements”, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 93 (2005), pg. 891–904.
[5] Alexandre Luis Braun and Armando Miguel Awruch, “Aerodynamic and aeroelastic analyses on the CAARC standard tall building model using numerical simulation”, Computers and Structures, Vol. 87 (2009), pg. 564–581.
[6] Appupillai Baskaran and Ahmed Kashef, “Investigation of air flow around buildings using computational fluid dynamics techniques”, Engineering Structures, Vol. 18, No. 11, pg. 861-875.
[7] Baskaran T. and Stathopoulos T., “Computational Evaluation of Wind Effects on Buildings”, Building and Environment, Vol. 24 91989), No. 4, pg. 325-333.
[8] Cheng C.C.K., Lam K.M., and Demirbilek F.N., “Effects of building wall arrangements on wind-induced ventilation through the refuge floor of a tall building”, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 96 (2008), pg. 656-664.
[9] Cheng-Hu Hu and Fan Wang, “Using a CFD approach for the study of street-level windsin a built-up area”, Building and Environment 40 (2005), pg. 617-631.
[10] Chow W.K., “Wind-induced indoor-air flow in a high-rise building adjacent to a vertical wall”, Applied Energy, Vol. 77 (2004), pg. 225–234.
[11] Ferreira A. D., Sousa A. C. M. and Viegas D. X., “Prediction of building interference effects on pedestrian level comfort”, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 90 (2002), pg. 305–319.
[12] Shenghong Huang, Q.S. Li and Shengli Xu, “Numerical evaluation of wind effects on a tall steel building by CFD”, Journal of Constructional Steel Research, Vol. 63 (2007), pg. 612–627.
[13] Tore Wiik and Ernst W. M. Hansen, “The assessment of wind loads on roof overhang of low-rise buildings”, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 67 & 68 (1997), pg. 687-696.