Numerical Analysis of Wind loads on Rectangular Shape Tall Buildings

Ashwin G Hansora¹ Paresh N Nimodiya² Kailashkumar P Gehlot³

¹,² Assistant Professor ³Student

¹,²,³ Department of Applied Mechanics

L.D.College of Engineering, Ahmedabad ²Vishwakarma Govt. Engineering College, Ahmedabad

Abstract— The development of high strength concrete, higher grade steel, new construction techniques and advanced computational technique has resulted in the emergence of a new generation of tall structures that are flexible, low in damping, slender and light in weight. These types of flexible structures are very sensitive to dynamic wind loads and adversely affect the serviceability and occupant comfort. This paper presents the results of Computational Fluid Dynamics technique using Ansys CFX software on rectangular building models having the same plan area and height but different side ratios of 1.25, 1.5, 1.75, 2.00 and 2.25. The wind pressure coefficients on all the models were evaluated from pressure records obtained from software under boundary layer flow for normal wind directions. It is observed that the side ratio of buildings significantly affects the wind pressures on leeward and sidewalls, whereas wind pressure on windward wall is almost independent of side ratio.

Key words: Rectangular Plan, Pressure Coefficient, Side Ratio, Computational Fluid Dynamics, K-E Method

I. INTRODUCTION

The developments of new building materials and construction techniques have enabled us to build new buildings which are tall and unsymmetrical. Naturally such structures are more susceptible to wind loads. Thus it becomes absolutely necessary to estimate wind loads with higher degree of confidence. Ning et al. (2004) in a paper titled “Characteristics of Wind Forces Acting on Tall Buildings” have discussed the finding of a wide spread wind-tunnel study on local wind forces on isolated tall buildings based on experimental outcome of nine square and rectangular models (1:500). Irwin (2009) proposed that more realistic models need to be used for buildings above 300 m. The statistics of upper level winds need also to be known with better certainty. Halder and Dutta (2010) after comparing the present Indian wind code with the proposed Indian wind code & the American code (ASCE) conferred that the results obtained from force method is on conservative side. The applicability of gust- factor method was also discussed.

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. Any increase in building height increases the effect of wind loading. Wind loads on tall structures cause concerns about the integrity of the structure envelope and safety of the whole structural system. Under the influence of the dynamic wind loads, typical high-rise buildings vibrate in the along wind, across wind, and torsional directions. Modern high-rise buildings designed to satisfy static lateral drift requirements. The distribution of wind pressures and wind forces along the perimeter of the buildings is necessary to study the structural behaviours of buildings at different wind incidence angles. However, only few experiments to determine the wind forces on rectangular buildings having different side ratios but same cross-sectional area are reported in the literature, although pressure fluctuations and responses on a specific building have been studied. This study is then attempt to provide the needful information of wind pressures and mean wind responses of rectangular buildings having same plan area and height but different side ratios. In particular, the present study details the dimensions of buildings, wind pressure coefficients distribution on different faces of rectangular buildings is determined. Pressure measurements are restricted to open country type flow, as the mean responses are significant in flow similar to terrain category-II (IS: 875 (Part3), 1987).

II. METHODOLOGY

A. Dimensions and Designation of Model:

![Fig. 1: Details of Building Models]
III. SOLUTION STRATEGY

Numerical simulations were carried out using Computational Fluid Dynamics (CFD) package ANSYS CFX 14.0. Standard k-ε method was used to numerically formulate the viscosity and turbulence. The governing equations are as follows.

The standard k-ε model is a typical example of a two-equation turbulence model. In this case an equation for the turbulent kinetic energy and its dissipation are used to determine the eddy viscosity.

\[ k = \frac{1}{2} \left( \frac{u'_1}{u'_1} \right) = \frac{1}{2} \left( u'^2 + v'^2 + w'^2 \right) \]

(1)

It can be observed that the eddy viscosity is dependent on the turbulent kinetic energy, \( k \), and an unknown constant and an unknown length scale.

\[ \mu_t = C_k \cdot \rho k^{3/2} \]

(2)

The governing equations for determining \( k \) & \( \varepsilon \) are

\[ \frac{\partial}{\partial t} \rho k + \frac{\partial}{\partial x_i} \left( \rho u_i k \right) = \varepsilon + \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu}{\kappa} \frac{\partial}{\partial x_i} \right) \]

(3)

\[ \varepsilon = \frac{\partial}{\partial x_i} \left( \rho u_i \varepsilon \right) = C_{\mu} \frac{\varepsilon}{k} \left( \frac{\partial u_i}{\partial x_j} \right)^2 - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu}{\kappa} \frac{\partial}{\partial x_j} \right) \]

(4)

The closure coefficients \( C_{c1} \), \( C_{c2} \), \( C_\mu \), \( \sigma_k \) and \( \sigma_\varepsilon \) replace the unknown correlations with algebraic expressions involving known turbulence and mean flow properties. For this, general arguments have been used. For the standard k-\( \varepsilon \) model the following closure coefficients are often used: \( C_{c1} = 1.44 \), \( C_{c2} = 1.92 \), \( C_\mu = 0.09 \), \( \sigma_k = 1.0 \) and \( \sigma_\varepsilon = 1.3 \).

IV. DETAILS OF DOMAIN

The analysis was carried out on rectangular plan shape model having different side ratio but same cross sectional area. A domain having 5H upwind fetch, 15H downwind fetch, 5H top clearance and 5H side clearance, where H is the height of the model, is considered as per Franke’s recommendation [3] (Fig. 3). Such a large size provides enough space for generation of vortex on the leeward side and avoids backflow of wind. Moreover no blockage correction is required.

V. BOUNDARY CONDITION

The flow at inlet was simulated similar to that of terrain category 2 as per Indian standard for wind load IS: 875 (part 3) - 1987. Inlet velocity was considered as 10m/s. The power law is used for generating the boundary layer flow at inlet. The outlet was considered as pressure outlet. The no slip boundary conditions are applied to the surfaces of Building Model. The velocity profile near the windward side as obtained from numerical method is shown in Fig. 3.

VI. RESULT AND DISCUSSION

A. Rectangular (Re-1):
Numerical Analysis of Wind loads on Rectangular Shape Tall Buildings

B. Rectangular (Re-2):

Fig. 5: (A) Pressure Contour of Windward Face (B) Pressure Contour of Leeward Face (C) Pressure Contour of Sidewall

C. Rectangular (Re-3):

Fig. 6: (A) Pressure Contour of Windward Face (B) Pressure Contour of Leeward Face (C) Pressure Contour of Sidewall

D. Rectangular (Re-4):

Fig. 7: (A) Pressure Contour of Windward Face (B) Pressure Contour of Leeward Face (C) Pressure Contour of Sidewall

E. Rectangular (Re-5):

Fig. 8: (A) Pressure Contour of Windward Face (B) Pressure Contour of Leeward Face (C) Pressure Contour of Sidewall

Variation of pressure coefficient along height or different side ratio

Fig. 9: (A) Pressure Contour of Windward Face (B) Pressure Contour of Leeward Face (C) Pressure Contour of Sidewall

VII. PRESSURE DISTRIBUTIONS ON MODELS

The evaluated mean pressure at a particular grid point locations non-dimensionalized to evaluate the mean pressure coefficients along the considered wind direction by...
1/2ρv^2, where ρ is the density of air (1.226 kg/m³), v is the free stream velocity at the roof level of the building model (10 m/s).

\[
\text{Static Pressure} = \frac{\text{Static Force}}{\text{Area}}
\]

\[
\text{Dynamic Pressure} = \frac{1}{2}\rho v^2
\]

The comparison of mean pressure coefficient on windward, leeward and side face is shown in fig. 9, fig. 10 and fig. 11. It is noticed that pressure distribution and magnitude of pressure coefficients on windward wall of rectangular models are almost independent of the model depth and side ratio. The absolute values of mean pressure coefficients on leeward face reduce as the side ratio of the models increases, due to the reattachment of flow on side faces. The absolute values of average mean pressure coefficients on side face reduce as the side ratio of the models increases due to the reattachments of flow.

In case of rectangular model Re-1 (side ratio = 1.25), suction increases almost up to 80 % depth, after which it decreases. In case of rectangular model Re-2 (side ratio = 1.50), suction increases almost up to 70 % depth, after which it decreases. In case of rectangular model Re-3 (side ratio = 1.75), suction increases almost up to 55 % depth, after which it decreases up to 90 % depth and further it increases slightly afterward. In case of rectangular model Re-4 (side ratio = 2.0), suction increases up to 50 % depth, after which it decreases up to 80 % depth and further it increases slightly. In case of rectangular model Re-5 (side ratio = 2.25), suction increases up to 40 % depth, after which it decreases up to 70 % depth and further it increases slightly.

VIII. CONCLUSION

In the present study effect of change in side ratio of rectangular building is studied by computational fluid dynamics technique. Outcome of above study is summarised below

1) Pressure coefficient distribution on windward face of rectangular models is almost independent of L/B ratio.
2) For Leeward Face the mean pressure coefficients reduce as the L/B ratio of the models increases, due to the reattachment of flow on side faces.
3) For side wall mean pressure coefficients reduces with increasing L/B ratio

REFERENCES