Investigation on Computational Fluid Dynamics (CFD) Simulation for Wind Effects on Tall Tapered Buildings

M.Kannan[1] K.Saravanan[2]
[1]M.E. Student [2]Assistant Professor
[1,2]Department of Civil Engineering
[1,2]Valliammai Engineering College, Chennai 603206, India

Abstract— The modern day structures are very tall, slender and flexible due to the use of high strength materials for construction. Although these flexible structures satisfy the strength criteria safely, it is sensitive to wind-induced excitations causing discomfort to the building and its occupants. The mitigation of across-wind responses is a major factor in design of tall buildings. Hence aerodynamic modification of the building shape becomes important to diminishing the wind-induced excitation. In the present study tall building with tapered cross section shape is considered. Numerical simulations using Computational Fluid Dynamics (CFD) techniques are carried out to study the effect of wind components on the tall tapered building. The modelling and meshing is done using ICEM software and the model analysis and solving is done using ANSYS FLUENT 15 software. Various Reynolds Averaged Navier Stokes (RANS) based turbulence models like Renormalization Group (RNG) k-ε model and Realizable k-ε model are used in finding the force co-efficient. The effectiveness of these models is evaluated by comparing it with the experimental results taken form literatures.

Key words: Tall Tapered Building, CFD, ANSYS FLUENT 6.3, RANS Turbulence Models, Force Co-Efficient

I. Introduction

The modern world has becoming more fast and efficient due to computer and computational efficiency. This boon has to be utilised in the field of structural engineering also. The tall structures that are sensitive to wind can also be design with the help of computers. Apart from designing the structure, in research point of view optimization can also be done. Computational Fluid Dynamics (CFD) is a computational technique adopted for studying the wind and structure interference. This reduces the cost and time considerably when compare to the traditional method. Using CFD alot of studies has been done and some of the recent work is on aerodynamic optimization. A slight modification in the design of the structure can result in huge reduction of loads effects on the structure. Tapered cross-section is one such modification that causes reduction in wind loads.

This paper gives the validation of various turbulence models for finding the force coo-efficient is studied. The tapered structure is chosen because it is efficient over the conventional square plan structures and also defines the percentage of reductions in wind loads. The effect of various incident wind angles in determining the force co-efficient is also studied.

II. Model used in the Study

The model is considered in three dimensions. This work aims to investigate the tapering effect for reducing wind-induced responses of a tapered tall building located in urban terrain. The geometry of the model used in the study has the following dimensions listed below,

- Model Height: 40cm.
- Plan area at ground level: 10x10cm.
- Plan area at 40m height:6x6cm.
- Angle of model: 0º and 45º

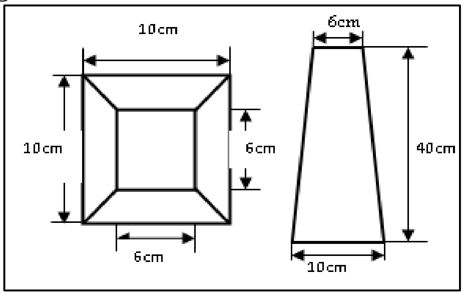

Fig. 1: Plan and Elevation view of the model

III. Computational Domain

The tapered sections were modeled using the fluent software. Geometry of tapered section is placed in the computational domain in such a way that it should not influence the gustiness of wind typical cross-sections. Based on the literature survey, Inlet boundary condition is chosen at a distance of 3H from the windward side whereas outlet boundary condition is chosen is at a distance of 10.5H from leeward side, where H is the total height of the tapered section. The computational domain is set such that the blockage ratio is less than 3%. The width and height of the computational domain is based on wind tunnel dimension and taken is taken as 180cm.

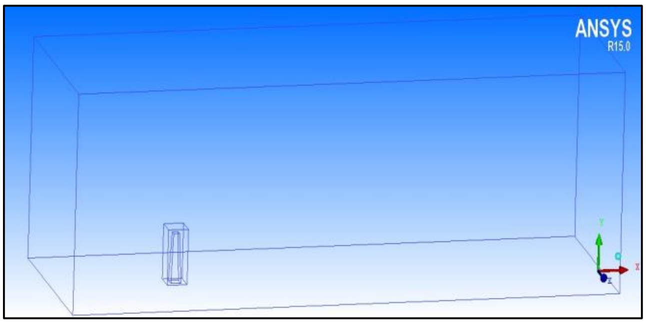

Fig. 2: Computational domain modeled in ICEM.

IV. Mesh Independency Test

Before deciding the mesh to be used in the study, a test is performed to finalize a mesh that gives optimal results. Three type of mesh is being tested. Mesh A has girds of size 3% its bottom width, similarly Mesh B has girds of size 2% its bottom width Mesh C has girds of size 1% its bottom width. The results are tabulated below. From the result Mesh B seems to give an optimal solution and hence it is adopted in the study.

Mesh Type	Spacing	No. of Cells	C_d Value
A	0.03xB	15213805	0.9013
B	0.02xB	19539603	0.8688
C	0.01xB	25728081	0.8610

Table 1: Mesh independency test value

V. MESHING

A structural HEXAHEDRAL grid approach offers advantages in solution algorithm efficiency and implementation. Thus structured mesh offers appropriate solution method from among the larger number of algorithms, which are available. The principle advantage of unstructured TETRAHEDRAL grid approach is that it provides a powerful tool for discretizing domains of complex shape. But the disadvantages they mention are limited availability of solution algorithms and the demand on the computer memory and CPU.

Hybrid mesh method is the method which is the combination both unstructured and structured mesh method. This method adopts the both advantage of unstructured and structured mesh and leads to give quick simulation results with increased optimum results for respective complex problems. As fixed from the mesh independency test, grids of size 2% of the bottom building width is used near the building which satisfies dimensionless wall distance, y^+ above 30 along all the side. An inner core is created of dimension of 20x20x45 cm which contains unstructured TETRAHEDRAL MESH with grid size of 0.02B where B is bottom base dimension. Outside the inner core the structured hexahedral gird is adopted with the growth or stretched ratio kept below of 1.1.

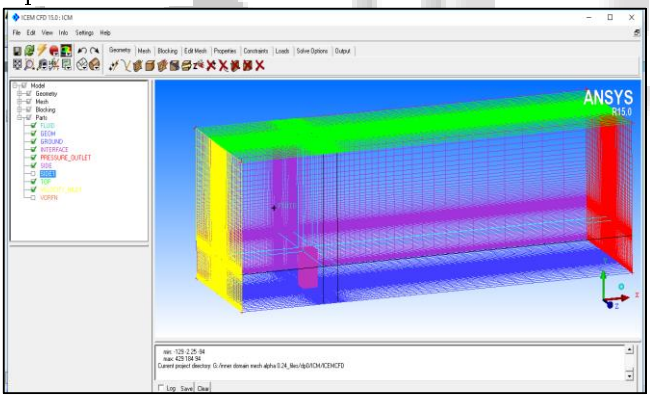

Fig. 3: Mesh of inner core surface.

VI. BOUNDARY CONDITION

The measured vertical profiles of mean wind speed $U(z)$, and stream wise turbulence intensity $I(z)$ are used to define the inlet boundary conditions for the CFD simulations. The atmospheric boundary layer of urban flow of power law $\alpha = 0.24$ of is used for the simulation. Reference wind speed U_H and turbulence intensity at the top of the building should be 8m/s and 19%.

The inlet wind profile of wind speed is described by the following power law equation

$$U(z) = U_H * \left(\frac{z}{z_H}\right)^{\alpha} \qquad \ldots (1)$$

z_H =Reference height.

α =Power law exponent.

z =Required height.

The turbulence intensity profile is determined from the following equation,

$$I(z) = 0.1 * \left(\frac{z}{z_G}\right)^{(-\alpha-0.05)} \qquad \ldots (2)$$

z_G = Boundary layer height

To determine kinetic energy information at the inlet the following equations are used

$$k(z) = \left(I(z) * U(z)\right)^2 \qquad \ldots (3)$$

The following equation will describes the vertical profile of the turbulence dissipation rate $\varepsilon(z)$:

$$\varepsilon(z) = C_\mu^{\frac{1}{2}} k(z) * \frac{U_H}{z_H} * \left(\frac{z}{z_H}\right)^{(\alpha-1)} \qquad \ldots (4)$$

$$C_\mu = 0.09$$

VII. RESULTS AND DISCUSSION

The results of the numerical simulations are presented in this section. The aerodynamic co-efficient values for all the simulations carried out are displayed. These co-efficient are non-dimensional values and are calculated using reference values given in previous chapter.

$$C_D = \frac{F_D}{\frac{1}{2}U(H)^2 A_g} \qquad \ldots (5)$$

$$C_L = \frac{F_L}{\frac{1}{2}U(H)^2 A_g} \qquad \ldots (6)$$

$$C_M = \frac{M_Z}{\frac{1}{2}U(H)^2 A_g B(0.5H)} \qquad \ldots (7)$$

Generally steady-state simulation uses an iterative scheme to progress to convergence. Since it is uncertain as whether the simulation is unsteady or steady-state it is always worth running a steady-state simulation first because it typically takes an order of magnitude less CPU time to complete. The results of steady state simulations show persistent oscillations in the residuals plot and oscillations in a key monitor, such as a drag force monitor, with increasing iterations.

This is a good indicator that the flow may be unsteady (transient) and the simulation needs to be run as an unsteady simulation. The transient simulation resembles natural wind characteristic fairly well. Two turbulence models are used in the study - RNG k-ε model and Realizable k-ε model. Also the studies were carried out for two wind incident angle 0 and 45 degree. The results are tabulated below.

0°	RNG-Transient	Realizable-Transient	Literature-Experiment
C_D	0.8702	0.8934	0.96
C_L	0.00305	0.000132	0.00
C_M	0.000098	0.000099	0.000

Table 3: Comparison of coefficient results of 0°

45°	RNG-Transient	Realizable-Transient	Literature-Experiment
C_D	0.812	0.82	0.84
C_L	0.0001	0.00009	0.00
C_M	0.0000199	0.00009	0.000

Table 4: comparison of coefficient results of 45°

The simulated values are compared with experiment results from Yong chul Kim and Jun Kanda (2010). These results can be better understood in the form of Graphical representation.

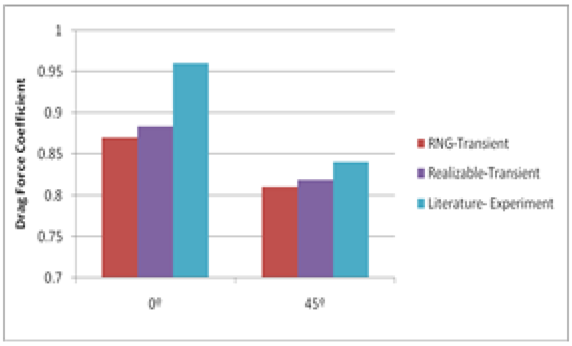

Fig. 4: Comparison of drag force value by various simulations

The graph shows trends in drag co-efficient values obtained by all three methods. It is obvious from the results that Realizable k-ε model gives better results than RNG k-ε model. Although this trend is expected, studies show that the differences in value between these two models are only about 1 to 3 percent. It is also noted that the RNG model prediction is 10% less than the experimental value when angle of wind attack is 0 degree and 3% when angle of wind attack is 45 degree. For Realizable model the simulation prediction is 7% less than experimental values for 0 degree angle of incident and 2.3% for 45 degree angle of incident.

From the results it is also inferred that as the angle of incidence of the wind increases from 0 to 45 degree, the simulation yields better results (ie. results closer to experimental results). This may be because, as the angle increases the streamline effect comes into action, thus reducing the force on the building and making the numerical simulation appropriate.

VIII. Conclusion

The mitigation of across-wind responses in design of tall buildings can be achieved by aerodynamic modification of the building shape. Tapered building is found efficient and by such a modification upto 20% reduction in force can be achieved. In the present work validation of numerical simulations using two turbulence models by Computational Fluid Dynamics (CFD) techniques are carried. The modelling and meshing done using ANSYS FLUENT software was done with utmost concern to make the analysis computation efficient and to obtain best simulation results. Reynolds Averaged Navier Stokes (RANS) based turbulence models like Renormalization Group (RNG) k-ε model, Realizable k-ε model are used in finding the force co-efficient. The effectiveness of these models is evaluated by comparing it with the experimental results taken form literatures. It is found both the models gives results with less than 10% error. RNG k-ε model is computationally best, whereas the Realizable k-ε model can be adopted for calculating the force and moment co-efficient. Studies also show that the the forces are maximum when incident wind direction is exactly perpendicular to the building.

References

[1] Franke, Jorg, et al. "The COST 732 Best Practice Guideline for CFD simulation of flows in the urban environment: a summary." International Journal of Environment and Pollution 44.1-4 (2011): 419-427.

[2] Hu, Gang, et al. "Large eddy simulation of flow around an inclined finite square cylinder." Journal of Wind Engineering and Industrial Aerodynamics146 (2015): 172-184.

[3] Kim, Yongchul, and Jun Kanda. "Characteristics of aerodynamic forces and pressures on square plan buildings with height variations." Journal of Wind Engineering and Industrial Aerodynamics 98.8 (2010): 449-465.

[4] Ramponi, Rubina, and Bert Blocken. "CFD simulation of cross-ventilation flow for different isolated building configurations: validation with wind tunnel measurements and analysis of physical and numerical diffusion effects." Journal of Wind Engineering and Industrial Aerodynamics 104 (2012): 408-418

[5] Tamura, Yukio, et al. "Aerodynamic and response characteristics of super-tall buildings with various configurations." (2013).

[6] Tominaga, Yoshihide, et al. "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings." Journal of wind engineering and industrial aerodynamics 96.10 (2008): 1749-1761

