Study on wind flow around a pentagon plan shape tall building using CFD

Sobankumar L1, S Prabavathy2 and R.Vigneshwaran3

1, 2, 3 Department of Civil Engineering, Mepco Schlenk Engineering College (MSEC), Sivakasi, India
E-mail: sobankumar12@gmail.com, spraba@mepcoeng.ac.in, vignesh.rajan.93@gmail.com.

Abstract. Multi-storey buildings are emerging because of the lack of land availability within urban areas. Mostly, tall buildings are irregular in plan and more complex to wind action. In the present scenario, it is important to study the wind structure interaction for irregular plan shape tall buildings. This paper is focused on understanding the behaviour of pentagon plan shape building under wind load at different wind incident angles 0°, 45°, 90°, 135° and 180°. The wind behaviour is studied using Computational Fluid Dynamics (CFD) and turbulence model k-ԑ is used for external flow simulation around the tall buildings. The building models are scaled in the ratio 1:300 and wind velocity at inlet is scaled in the ratio 1:5 respectively. From the results, external pressure coefficient on various faces of the buildings, drag coefficient and lift coefficients are determined for pentagon shape building. The maximum positive external pressure coefficient (Cp) is 0.446 had occurred in Face A at 180° wind incident angle and the maximum value of the negative external pressure coefficient (Cp) is 0.388 had occurred in Face D at 135° wind incident angle.

Keywords: Pentagon shape, Computational Fluid Dynamics (CFD), k-ԑ turbulence model, Drag and Lift Coefficients.

1. Introduction
Tall buildings are laterally affected due to seismic and wind load. While considering the tall building wind force is predominant than earthquake loading. Because of lateral force, a very large moment is generated at the base. The magnitude of base moment for a tall building mainly depends upon the slenderness ratio. In addition to this, if the plan of the buildings is of irregular shape, the wind analysis is the critical task which may create more flow situations when wind interacts with structures. Analysis of irregular plan tall buildings are more critical to wind load than regular shape buildings. However, current design standards provide guidelines only for regular and symmetric shapes. There are no standard guidelines and analytical formulas available to calculate the wind effect on irregular plan shape tall buildings. Therefore, it is a necessity to conduct research work on an irregular plan of tall buildings. Building performance has been increased towards windward direction by optimizing its corner size. In this Artificial Neural Networks (ANN) is used to estimate the aerodynamic response of
building by 3D Large Eddy Simulation (LES) [1]. Wind tunnel experiment is agreed on eight number of L-shaped building models by varying its geometric dimensions. Based on results an empirical relation is expressed for determining wind-induced torque on L shaped high-rise buildings [2]. The comparison exposed that there is some huge deviation between the experimental and numerical results, because of mismatching in inflow boundary condition [3]. An inverse method is developed to estimate the wind response and wind load on high rise buildings by limited response measurements [4]. The size of the recirculation region is high in the infinite cylinder because of the increase in momentum inflow [5]. The wind flow pattern around a tall building under urban wind condition has studied using CFD. All RANS turbulence model has compared to predict the wind pressure on a circular plan high rise building [6]. The wind response on a tall chimney in along and crosswind direction is studied. In that, a semi-empirical relation is derived using structural damping for computing the crosswind effect on chimney [7]. The galloping stability of rectangle and H shape tall building has investigated using CFD. By comparing the mean external pressure coefficient, the windward coefficient good agreement experiment data and the leeward or suction coefficient have some deviation from experiment data. The deviation is because of limitations in the RANS turbulence model [8]. Aerodynamic effect on various shape of a tall structure vortex formation is studied. By changing some part of the basic cross-section, the aerodynamic damping can be achieved [9]. From the results of the previous investigation, the present paper focuses on investigating the wind effect on the pentagon plan shaped tall buildings using CFD. The numerical analysis is carried for different wind angles from 0° to 180° at an equal interval of 45° and mean external pressure coefficient ($C_p$) on the different faces of the pentagon model is the core area in this study.

2. Governing Equation for CFD

The ANSYS Fluent software has the algorithm to solve the computational fluid dynamic equation.

2.1.1. Continuity equation Non-conservation form

$$\frac{\partial \rho}{\partial t} + \rho \vec{V} \cdot \nabla \vec{V} = 0. \quad (1)$$

By applying the model of a finite control volume approach on fixed in space model conservation form of continuity equation is obtained.

2.1.2. Continuity equation conservation form

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0. \quad (2)$$

2.2. The Momentum Equation

The momentum equation is also called as Navier stokes equations. The fundamental principle for momentum equation is Newton’s second law of motion.

2.2.1. Non-conservation form for viscous flow

x-direction: $\frac{\partial \vec{u}}{\partial t} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho \vec{f}_x, \quad (3)$

y-direction: $\frac{\partial \vec{w}}{\partial t} = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial z} + \rho \vec{f}_y, \quad (4)$

and z-direction: $\frac{\partial \vec{w}}{\partial t} = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{zx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + \rho \vec{f}_z, \quad (5)$

2.2.2. Non-conservation form non-viscous flow

x-direction: $\frac{\partial \vec{u}}{\partial t} = -\frac{\partial p}{\partial x} + \rho \vec{f}_x, \quad (6)$

y-direction: $\frac{\partial \vec{u}}{\partial t} = -\frac{\partial p}{\partial y} + \rho \vec{f}_y \quad (7)$

and z-direction: $\frac{\partial \vec{w}}{\partial t} = -\frac{\partial p}{\partial z} + \rho \vec{f}_z. \quad (8)$
2.2.3. Conservation form for viscous flow

- **x-direction:** \( \frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u \vec{v}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x \), (9)
- **y-direction:** \( \frac{\partial (\rho v)}{\partial t} + \nabla \cdot (\rho v \vec{v}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y \) (10)
- **z-direction:** \( \frac{\partial (\rho w)}{\partial t} + \nabla \cdot (\rho w \vec{v}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z \). (11)

2.2.4 Conservation form for non-viscous flow

- **x-direction:** \( \frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u \vec{v}) = -\frac{\partial p}{\partial x} + \rho f_x \), (12)
- **y-direction:** \( \frac{\partial (\rho v)}{\partial t} + \nabla \cdot (\rho v \vec{v}) = -\frac{\partial p}{\partial y} + \rho f_y \) (13)
- **z-direction:** \( \frac{\partial (\rho w)}{\partial t} + \nabla \cdot (\rho w \vec{v}) = -\frac{\partial p}{\partial z} + \rho f_z \). (14)

3. Numerical modelling

3.1. Domain geometry

The building is modelled using 1:300 scale and the buildings are assumed as rigid. A rectangular domain was chosen [6]. The distance of 3H in upstream and 10H in the downstream side are provided as windward and leeward sides respectively. Figure 2 shows the geometric representation of the domain. The distance between the boundary walls and the building model is provided as 3H. The plan and isometric view of pentagon building is shown in figure 1. The numerical study is performed using Reynolds Averages Navier-Stokes (RANS) and K - \( \varepsilon \) model by ANSYS Fluent software. Along with this, the pressure contours of different faces of the pentagon plan shaped building are observed.

![Figure 1. Plan view and isometric view of the pentagon building](image-url)
3.2 Mesh generation and turbulence setting
Mesh size is responsible for the accurate result and also for the computational requirement. To reduce the computational time and also get the accurate result different face sizing is used in both model and domain. The building models and domains have meshed with 1 mm and 50 mm polyhedral face sizing respectively are shown in Figure 3. Time efficiency and not compromising the result are the two factors concern in regular design office problem. So, the most predominantly used turbulent model is RANS turbulent model in which two-equation standard K-ԑ (RANS-SKE) is dominated one [6]. In this study, the external pressure coefficient for the different faces of the pentagon model was obtained from RANS- SKE turbulent model.

3.3. Inlet and boundary conditions
The no-slip wall condition is considered for bottom face and the free slip wall condition is considered for side faces of the domain. Inlet velocity is given as 10m/s. In outlet, zero velocity is given.

4. Validation with square model
For validation of CFD in this study is checked for a square plan tall building was modelled with the dimensions of 50mm x 50mm in plan and height 300mm in ANSYS Fluent. The plan and isometric view of a square building is shown in Figure 4. The created model is analyzed in the above-mentioned domain RANS-SKE turbulent model. Velocity 10m/s is provided at the inlet.[6]. The comparison is shown in table 1.

| S.no | Description     | Face A | Face B | Face C | Face D |
|------|----------------|--------|--------|--------|--------|
| 1    | IS 875 part 3 -2015 | 0.9    | -0.8   | -0.25  | -0.8   |
| 2    | ANSYS FLUENT    | 0.89   | -0.75  | -0.19  | -0.75  |

5. Results and Discussion
5.1. Pressure distribution
The variations of wind pressure on different surfaces of the building model for wind incident angles 0°, 45°, 90°, 135° and 180° are shown in figure 5 to 9 respectively. The wind pressure obtained by the computational method using the ANSYS Fluent is used to calculate the external pressure coefficient $C_{pe}$ using the formula

$$C_{pe} = \frac{p}{0.6 \nu^2}.$$ (15)
where \( V_z \) is the design wind speed and \( P \) is the wind pressure. The variation in external pressure coefficients \( C_{pe} \) concerning the height and the wake region formation for different wind incident angle using CFD are represented in figure 10 and figure 11 respectively.

![Figure 4. Plan and Isometric view of the square building](image)

**Figure 4.** Plan and Isometric view of the square building

![Figure 5. 0° wind incident angle](image)

**Figure 5.** 0° wind incident angle

![Figure 6. 45° wind incident angle](image)

**Figure 6.** 45° wind incident angle

![Figure 7. 90° wind incident angle](image)

**Figure 7.** 90° wind incident angle

![Figure 8. 135° wind incident angle](image)

**Figure 8.** 135° wind incident angle
Figure 9. 180° wind incident angle

(a) Face A
(b) Face B
(c) Face C
(d) Face D
Figure 10. Variation in External pressure coefficient concerning height(a) 0° wind incident angle (b) 45° wind incident angle (c) 90° wind incident angle (d) 135° wind incident angle (e) 180° wind incident angle

5.1.1. Pentagon model at 0° wind incident angle
At 0° wind incident angle, faces C and D are getting similar positive external pressure coefficient of about 0.431 and both the faces are projected towards the windward direction. Further, faces B and E are getting negative external pressure coefficient of about 0.235 and 0.196 respectively, both the faces are projected towards the leeward direction. The maximum positive external pressure value is 0.471 had occurred in face C which is more than 190.82% in face A followed by 162.93% in face B, 190.82% in face E.

5.1.2. Pentagon model at 45° wind incident angle
At 45° wind incident angle, faces C and B are getting positive external pressure coefficient of about 0.260 and 0.187 respectively and further faces A, D and E are getting negative external pressure coefficients of about 0.299, 0.183 and 0.362 respectively. All the faces are projected in a tapered manner. The maximum positive external pressure coefficient is 0.274 has occurred on face C which is more than that of 215.15% in face A, 28.18% in face B, 170.29% in face D and 239.26% in face E.

5.1.3. Pentagon model at 90° wind incident angle
At 90° wind incident angle, faces B and C are getting positive external pressure coefficient values of about 0.308 and 0.440 respectively. Faces A, D and E are getting negative external pressure coefficient of about 0.169, 0.228 and 0.362 respectively. Face A is projected parallel to the wind incidence angle and the remaining faces are projected in a tapered manner. The maximum positive external pressure coefficient is 0.440 had occurred in face C which is moreover 138.8% of face A, 30.09% of face B, 151.84% of face D and 184.75% of face E.

5.1.4. Pentagon model at 135° wind incident angle
At 135° wind incident angle, faces A and B are getting positive external pressure coefficient of values are 0.333 and 0.326 respectively. Faces C, D and E are getting negative external pressure coefficient values are 0.165, 0.389 and 0.196 respectively. The maximum positive external pressure coefficient value is 0.326 had occurred in the face A which is more than that of 2.158% in face B, 148.7% in face C, 218.8% in face D and 158.8% in face E.
5.1.5. Pentagon model at 180° wind incident angle

At 180° wind incident angle, faces A, B and D are getting positive external pressure coefficient of values are 0.447, 0.233 and 0.153 respectively. Faces C and E are getting similar negative pressure coefficient value of 0.239. The maximum positive pressure value is 0.447 had occurred in the face A which is more than that of 247.91% in face B, 153.48% in face C, 153.48% in face D and 53.78% of face E.

5.2. Drag and Lift Coefficient

Wind pressure distribution for the pentagon building model at different wind incident angle is exposed in fig 11. The previous researcher calculated drag coefficient and lift coefficients for the triangle plan shape tall building using CFD. While designing the tall buildings with round and tapped corner buildings are getting lesser drag coefficient and lift coefficient [10]. The drag coefficient ($C_d$) and lift coefficient ($C_l$) are calculated using the formulas

$$C_d = \frac{F_d}{0.6\nu^2 A_p}$$

and

$$C_l = \frac{F_l}{0.6\nu^2 A_p}.$$  

where $\nu$ is the design wind speed, $F_d$, $F_l$ is the drag and lift force and $A_p$ is the projected area. The table 2 shows the drag and lift coefficient for the pentagon plan shape building.

| Wind incident angle | Drag Coefficient $C_d$ | Lift Coefficient $C_l$ |
|---------------------|------------------------|------------------------|
| 0°                  | 0.892                  | 0.935                  |
| 45°                 | 0.678                  | 0.673                  |
| 90°                 | 0.822                  | 0.781                  |
| 135°                | 0.735                  | 0.735                  |
| 180°                | 0.680                  | 0.715                  |

Maximum drag coefficient value is 0.892 at 0° wind incident angle differed from other wind incident angles by 23.9%, 7.84%, 17.609% and 23.76% at wind incident angle 45°, 90°, 135° and 180° respectively. Maximum lift coefficient values are 0.935 at 0° wind incident angle differed from other wind incident angles by 28%, 16.47%, 21.39% and 23.53% at wind incident angles by 45°, 90°, 135° and 180° respectively.

5.3. Wake region

In the current analysis of pentagon building, formations of wake regions are shown in figure 11 (a to e) for the wind angles 0°, 45°, 90°, 135° and 180° degree. Wake region characteristic of the tall building in an urban environment has examined using CFD [11]. For pentagon building, the distribution area of the wake region is maximum when the full face is projected towards the leeward direction.
Figure 11. Wake region pattern (a) 0° wind incident angle (b) 45° wind incident angle (c) 90° wind incident angle (d) 135° wind incident angle (e) 180° wind incident angle
Conclusion

Maximum positive external pressure coefficient is 0.447 of face A at 180° wind incident angle, which is more than of other wind incident angle by 5.92%, 41.80%, 1.63% and 25.43% at 0°, 45°, 90° of face C and 135° of face A. Minimum negative external pressure coefficient is 0.389 at face D at 135° wind incident angle, which is more than other wind incident angle by 5.92 %, 41.8%, 1.63% and 25.43% at 0° at face A, 45° at face E, 90° face E and 180° of face C and D. From the results, obtained from CFD shows that maximum positive external pressure coefficient is obtained when the building face is exactly perpendicular to the wind and the external \( C_p \) is decreased when the building face is in the inclined position. Maximum and minimum drag and lift coefficient are experienced when the pentagon model is oriented towards the wind direction at 0° and 45° respectively. The maximum drag coefficient is 0.892 at 0° wind incident angle followed by 0.822, 0.735, 0.680 and 0.678 for wind incident angles 90°, 135°, 180° and 45°. The drag coefficient and lift coefficient values are directly related to the pressure intensity of the overall building.

Acknowledgement

The author would like to thank Mepco Schlenk Engineering College (MSEC), India for providing facility and sincerer thanks to Institute of Engineers (India) IE(I) for its grateful financial support (Project ID: DR2020004) to complete the project work successfully.

References

[1] Ahmad Elshaer, Girma Bituamlak and Ashraf El Damatty 2017 Enhancing wind performance of tall buildings using corner aerodynamic optimization Engineering structures 136 133-48.
[2] Yi Li, Q.S.li and Fubini Chen 2017 Wind tunnel study of wind-induced torques on L-shaped tall buildings Journal of wind engineering and Industrial Aerodynamics 167 41-50.
[3] Marie Skytte Thordal, Jens Chr. Bennetsen and H.Holger H.Koss 2019 Review for practical application of CFD for the determination of wind load on high-rise buildings Journal of Wind Engineering and Industrial Aerodynamics 186 155-65.
[4] Lunhai Zhi, Pan yu, Qiu-Sheng Li, Bo Chen and Mingxin Fang 2018 Identification of wind loads on a super-tall building by Kalman filter Computers and Structures 208 105-17.
[5] Yong Cao, Tetsuro Tamura and Hidenori Kawai 2019 Investigation of wall pressure and surface flow patterns on a wall-mounted square cylinder using very high-resolution cartesian mesh Journal of wind engineering and Industrial Aerodynamics 188 1-18.
[6] D.Mohotti, K.Wijesooriya and D. Dias-daCosta 2019 Comparison of Reynolds Averaging Navier-Stokes (RANS) turbulent models in predicting wind pressure on tall buildings Journal of Building Engineering 21 1-17.
[7] S.Arunachalam, N. Lakshmanan 2015 Across-wind response of tall circular chimney to vortex shedding Journal of Wind Engineering and Industrial Aerodynamics 145 187-95.
[8] M.Keerthana and P.Harikrishna 2013 Application of CFD for assessment of galloping stability of rectangular and H-sections Journal of Scientific & Industrial Research 72 419-27.
[9] H.Hayashida and Y. Iwasa 1990 Aerodynamic shape effects of a tall building for vortex-induced vibration Journal of wind engineering and Industrial Aerodynamics 33 237-42.
[10] Abdoiah Baghaei Daemei, Elham Mehrinejad Khotbehsara, Erfan Malekain Nobarani and Payam Bahrami 2019 Study on wind aerodynamic and flow characteristics of a triangular-shaped tall building and CFD simulation in order to assess drag coefficient Ain shams engineering journal 10 541-48.
[11] Denise Herwig, Hannah L. Gough, Sue Grimmond, Janet F. Barlow, Christoph W. Kent, William E. Lin and Alan G. Robins 2019 Wake Characteristics of a tall building in a realistic urban canopy *Boundary-Layer Meteorology* **173** 239-70.