CFD simulation for pedestrian comfort and wind safety in VIT campus for wind resource management

S A Prabhaharan1, KB Rajasekarababu2, G Vinayagamurthy3*, R Sivakumar4

1PG Student, 2PhD Student, 3*Associate Professor, 4Professor
2, 3* Centre for Innovation and New Product Development.
1,2,3*,4 School of Mechanical Engineering, VIT Chennai, India.
saprabhaharan@gmail.com1, kb.rajasekarababu2014@vit.ac.in2, vinayagamurthy.g@vit.ac.in3*,
sivakumar.r@vit.ac.in4

Abstract
Recent researches on Computational Wind Engineering (CWE) studies widely focus on wind pressure coefficients and wind flow field for the aerodynamic design of buildings, urban planning and dispersion studies. Nowadays, CWE had been widely used for identifying the critical wind locations on the field for warning the pedestrians as well as harnessing the wind. Hence, highly functional regions like universities are currently equipping micro wind turbines and roof-mounted wind turbines to meet their demands. This study focused on exploiting the wind flow conditions around the premises and wind force characteristics of the buildings situated at VIT Chennai premises for effective utilization of wind energy and determining the hot spots of unfavourable wind. By introducing CFD simulation earlier into the design process, it is easy to assess the wind resource conditions, pedestrian safety and comfort. However, the accuracy and reliability of CFD simulations can easily be compromised. For this reason, several sets of best practice guidelines have been developed in the past decades. Based on the best practices, this work presents the CFD simulation and framework for evaluating pedestrian comfort and possible areas to commission small wind turbines in the VIT campus. The simulations for assessing the wind flow conditions are carried out based on the \( k-e \) realizable turbulent model with high-quality grid-based convergence. The preliminary results can ensure wind comfort, safety and wind resources with CFD and add to enriched wind environmental quality in living.

Keywords: Computational Wind Engineering, Wind resource management, micro-scaled wind turbines, Pedestrian safety.

1. Introduction
Urban planning and sustainability deal with the utilization of available natural resources in framing our use of land and built-in environment. Notably, universities and colleges implement the process in a full-fledged manner. Wind resource is one of the main stakeholders in power production and sustainable use of resources. Hence universities involve commissioning various micro-scaled, building-integrated wind turbines, small scale wind turbines to meet their electricity requirement to a certain extent. While concentrating on utilizing the wind resources, analyzing the appropriate wind conditions, considering the wind loads on buildings, and metrological data in the study field is essential. We need to trace out the precise locations where favourable conditions to harness wind energy are available. Ample researches have proven the reliability and efficiency of these computational studies. Hong et al. [1] Gaining valuable insights towards flow patterns around buildings, specifically in urban areas through CFD, was much facile than on-field testing. H. Montazeri et al. [2] demonstrated the accuracy and reliability of CFD studies over medium-rise buildings based on 3D RANS, involving multiple facades in the building architecture. Another influencing factor in the urban planning process is pedestrians' comfort and safety in the living environment. Since the location involved in this study has many high rise buildings, aligning with wind safety standards, it becomes necessary to analyze wind safety towards pedestrian comfort. The work focuses on developing a simulation framework to determine the best regions to utilize the wind resources and examine wind comfort and safety inside the VIT campus located in Chennai. Although evident studies prove the credibility of numerical simulations, still they remain questionable without validations. Consequently, our study aimed at providing detailed insights for further in-field testing and wind tunnel experiments. The simulation procedure is followed based on AIJ guidelines by (Tominaga et al. [3]) (Best Practise Guidelines for
practical applications of CFD to pedestrian wind environment around buildings) developed by earlier researchers. In that way, the study can be graded as an accountable and standard one.

2. Methodology

The simulation procedure was carried out based on the standard wind assessment studies from the literature given by previous researchers in this field of study. Based on that, workflow is defined and are followed for the research. The process is initiated with the mesoscale inspection and CAD development of the entire campus. Then the necessary wind conditions are assessed, and suitable wind flow conditions are set up for the CFD analysis. The results are extracted based on best practice guidelines and are correlated with wind comfort standards to ensure the feasibility of deploying wind turbines inside the campus.

2.1 CAD model development

VIT University Chennai campus [refer Fig.1 (a)] is located in the south-west portion of Chennai city in Tamilnadu, India. It is situated about 10 km away from the east coast region. The premises consist of 15 buildings and divided into 3 clusters (as shown in Table 1). The highest building height is about 82 m (H_max) is shown in Fig.1(b). The whole premises is modelled using Solid Works 19 shown in Fig.1(c). The initial plan layout and orientation of buildings in the campus are extracted with the help of Google maps imported to the CAD platform and is scaled for the known dimensions. With the help of essential building constructional data, every building's topology (involving many approximations) is designed based on explicit modelling accordingly and altered to suit the needs of the computational study. Once the CAD data is finalized, the geometry details are mapped to suitable standard geometry interchange formats for further computational platforms.
2.2 CFD best practise guidelines

The numerical study is followed based on best practice guidelines developed by various researchers. The detailed studies performed in computational wind engineering and guidelines provided by institutes such as AIJ are the fundamentals for the simulation procedure discussed in the paper. The extensive wall function formulations presented by Tominaga et al. [3], Franke et al. [4], Blocken et al. [5] and Richards et al. [6] are utilized. This study is on the mesoscale; the domain must be vast enough to capture the flow patterns. Hence hexagonal domain modelling was highly recommended (Y. Toparlar et al. [7]).

- **Computational domain**: Over the past researches, CFD simulations involved computations in various shapes of domains such as circular, rectangular, pentagonal etc. Nevertheless, those domains are sufficient for a single model-based study, such as aerofoils, low-rise, tall, setback buildings (Rajasekarababu et al. [8]). This study is on the mesoscale; the domain must be vast enough to capture the flow patterns. Hence hexagonal domain modelling was highly recommended (Y. Toparlar et al. [7]).

- **The shape of the computational domain**: Since the study's geometry includes mesoscale models, it is necessary to simulate all possible wind conditions based on weather data. Ultimately it involves simulating the model for multiple wind flow conditions with different wind directions. Y. Toparlar et al. [7] adopted hexagonal domain modelling to analyze mesoscale models for micro-climate studies.

- **Size of the computational domain**: Our area of interest in this study, as mentioned earlier, comprises three clusters of building collections. They involve all types of building structures ranging from low rise, mid-rise to high elevation. Specific high rise structures also involve unconventional with purely contemporary design characteristics. Wind load parameters tend to be predicted, for which steady-state RANS simulations are undertaken. Franke et al. [4] defined the minimum size for the computational domain, which has to be based on the highest building height in the area of interest.

- **Discretization scheme**: Wind assessment-related computational studies involving a first-order discretization scheme resulted in dissipation errors, and hence to rectify those errors, it was recommended to follow appropriate second-order discretization schemes.

3. Pre-processing for the computational study

The pre-processing phase, during the simulation, goes through four important stages. Initially, the process starts with the selection of an appropriate computational domain. The hexagonal domain setup is adapted to suit the mesoscale assessments. Followed by the computational domain setup, the grid is generated with sufficient quality clubbed with grid independence tests. The later stages involve the choice of inflow boundary conditions and solver setups based on the wind characteristics associated with the campus location.

3.1 Computational domain

![Diagram of Computational Domain]

a) The model fitted inside the computational domain
All the building structures inside the computational domain (which affect the flow potentially) are modelled, as shown in Fig.2 (a). Considering the buildings' position and orientation are classified into 15 numbers, shown in Fig.2 (b), and the buildings are categorized into three clusters, as mentioned in Table 1. The computational domain is shown in Fig.2. Since the study is highly sensitive in predicting pressure and velocity fields on and around the buildings, constructing a flow vicinity produces reliable quality (Revuz, Julia & Hargreaves., 2012[9]). The simulations are done with six different wind directions; suitable upstream and downstream distance is fixed (to meet the criteria as per best practice guidelines) as mentioned earlier since the proper downstream length must be maintained to aid in accurate flow prediction wake region. Also, these parameters influence the convergence, and grid sensitivity analysis is studied. The domain is quantified with the tallest building height: 14 in cluster C as $H_{\text{Max}}$. Both upstream & downstream of the domain measured to 8H and height of the domain is 3H.

3.2 Computational Grid

The computational grid is constructed with a high-resolution adaptive mesh technique, where the locations of adverse gradients are to be monitored near the buildings. This will lead to increased computational efficiency without compromising the accuracy of the study. The grid comprises only tetrahedral cells with prism layers generated in areas of interest, which can be seen in Fig.3. Following the best practice guidelines, grid sensitivity analysis is being performed with minimum tetrahedral elements. Following that, the grid is subjected to further refinement with a medium and fine level mesh resolution. A grid independence study is undergone for the mean pressure distribution around one of the tallest building, number 13, and the $C_p$ plot results are provided in Fig.4. The plot exhibits a reliable result prediction when the mesh quality transits from medium quality to finer one. Hence the utmost fine quality meshing is adopted for the study and further wind direction simulations.
Table 1: Buildings situated inside the university

| They are building no. | Cluster | Name                           | H_{\text{max}} (m) |
|-----------------------|---------|--------------------------------|--------------------|
| 1                     | A       | Boy's Hostel, A block          | 69                 |
| 2                     |         | Campus Bank                    | 13.5               |
| 3                     |         | North Square                   | 6.30               |
| 4                     |         | Academic Block 1               | 36                 |
| 5                     |         | Gym Khanna                     | 22.50              |
| 6                     |         | Guest House                    | 15                 |
| 7                     |         | Campus main gate               | 8.36               |
| 8                     | B       | Admin block & Library          | 41                 |
| 9                     |         | Auditorium                     | 30                 |
| 10                    |         | Sit-out                        | 6.10               |
| 11                    |         | Gazebos                        | 4.79               |
| 12                    |         | Health Centre block            | 22.50              |
| 13                    | C       | Academic block 2               | 40                 |
| 14                    |         | Boy's Hostel, C block          | 82                 |
| 15                    |         | Girl's Hostel                  | 69                 |

Figure 3: Computational Grid
3.3 Inflow boundary conditions

The wind flows are simulated under the open terrain condition. Simulated mean wind and turbulence profiles are validated from previous literature (Rajasekarababu, K.B et al., 2019 [10]). The profiles are shown in Fig.5. The boundary conditions in Table 3 are defined using the User Defined Function (UDF) in FLUENT 19.2. The open terrain velocity profile follows the logarithmic law and turbulence properties such as turbulence intensity (I), turbulent kinetic energy (k) and turbulent dissipation rate (ε) in the inlet. These equations are used to execute inflow boundary conditions in this simulation.
Table 2: In-flow Boundary Conditions

\[
\begin{align*}
(U_i) &= \frac{U_{in} \kappa}{\gamma (y + y_o)} \\
\epsilon(y) &= \frac{U^3}{\kappa (y + y_o)} \\
I(y) &= \frac{1}{\gamma} \ln \left( \frac{y}{y_o} \right)
\end{align*}
\]

Inflow boundary (Inlet)

Outlet

\[
\frac{\partial}{\partial x} (U, V, W, k, \omega) = 0
\]

(pressure outlet)

Sky (top)

\[
U = U_{abl}, k = k_{abl}, \omega = \omega_{abl}, W = 0, \frac{\partial}{\partial x} (U, V, W, k, \omega) = 0
\]

Wall (sidewall)

\[
V = 0, \frac{\partial}{\partial x} (U, V, W, k, \omega) = 0
\]

Ground wall

\[
K_s = \frac{9,793 y_o}{C_i}
\]

3.4 Solver Settings

The commercial CFD code ANSYS Fluent 19.2 is used to perform the isothermal simulations. The 3D steady RANS equations are solved in combination with the Realizable \(k-\epsilon\) Model. The SIMPLE algorithm is used for pressure-velocity coupling, pressure interpolation is second order, and second-order discretization schemes are used for convection terms and the viscous terms of the governing equations. Convergence is assumed to be obtained when all the scaled residuals level off and reach a minimum of \(10^{-6}\) for \(x, y\) momentum, \(10^{-5}\) for \(z\) momentum and \(10^{-4}\) for \(k, \epsilon\) and continuity. As also observed by Rajasekarababu et al., [11 and 12], the simulations show oscillatory convergence.

4. Results And Discussion

The CFD simulations of the 3D RANS (\(k-\epsilon\) model) are obtained for six wind directions: South, South, West, South East, North, North West and North East. To assess the wind loads affecting the buildings, mean \(C_p\) values from six wind directions are shown in Fig.6. From the mean pressure coefficient, the tall buildings on the campus experienced high positive pressures for all six wind directions. Adverse effects of suction pressures are noticed towards the leeward side of these buildings. This occurs because of upwind stagnations of inlet wind and resulting recirculation followed by wake production in those areas. Due to the cascading effect of the high wind loads, the zones amidst comprising the cluster A and C (refer table-1) confronts high amounts of suction -1.2 < \(C_p\) <-0.6. These zones are not advisable to install small scale wind turbines. On observing the tall buildings' rooftops, (Building no: 1, 14 and 15) due to wind shear high amount of turbulence in wind flow are characterized. Hence installing roof-mounted wind turbines will not feasible in those locations, as it could affect the smooth function of the turbine and affect its performance. In contrast with the tall buildings, the roofs of medium-sized buildings inside the campus were not experienced by such adverse effects as tall buildings. Hence the medium-sized buildings and the surrounding locality proves to be permissible for the SWT installations.
4.1 Streamline velocity contours

In the view of harvesting the wind energy around the buildings, it is necessary to access the flow patterns in all possible areas since previous studies showed that the shape and structure of the surrounding buildings directly affect the performance of the wind turbines (Calautit, Katrina & Aquino [13]). The main idea of capturing these wind patterns is to assess wind velocity utilization by amplifying the flow due to building arrays. Hong Zhou, Yujie Lu [1] analyzed the wind acceleration due to low-rise buildings in improving wind turbines' operation. They infer that building shapes and orientations play a vital role in accelerating the wind flow. Therefore it is our due focus to determine the possibilities to increase the wind turbines' operating efficiency. The mean velocity of 7m/s, an average wind speed in the university's location, is set at a suitable height of 10m. The patterns for six wind directions are shown in Fig.7. From the streamline patterns, we can understand that wind speeds surrounding the building structures are low, between 2.6 m/s to 5.2 m/s. Also, in certain zones inside the campus, the building's shape had significantly improved the wind speed, around 7 to 7.8 m/s. Approximately 40% of the increase in wind speed is observed in those regions (see Fig.7). Fig.7 (a) & (b) shows, the wind speed increase is noticed in the east direction. Accordingly, the wind accelerations have developed due to the venturi effect between building 14 and 15 towards its leeward side.

Similarly, wind amplification in the passage contained in building 2. Due to wind shear along the building walls, certain zones experienced wind accelerations as well. In Fig.7 (c), wind shear alongside building no. 13, leading to an increase in wind speed and the space between buildings no.13, 14 and 15 can be observed. Similar phenomena happen in building no.13, also as shown in Fig.7 (d).
Figure 7: Streamline velocity contours for wind directions (a) SE (b) NE (c) NW and (d) N

*(U_H = 7 m/s @ 10 m Height)*
4.2 Wind Comfort and safety factors

Wind comfort and safety factors are assessed in this study. Wind comfort is a function of many parameters such as weather, temperature, humidity and predominantly wind speed. Simulations for the mean inlet wind speed for one direction is not satisfactory, precisely for wind assessments. Hence, with weather data, the wind assessment for six different wind directions, as discussed in the previous section. To illustrate the process, the wind patterns are captured from the simulation, extracted at the height of 10m above the ground level. Examining the velocity streamlines, adverse velocity gradients are developed in accommodating the building structures' corner. Therefore, three critical regions were deduced, which are potentially unsafe for pedestrians under extreme wind conditions are situated inside the campus perimeter, as shown in Fig.8. Locations include 1) The pathway behind the building no. 4A and 3A, 2) Passage at the centre of building no.8B and 3) Area behind building no.13C (See Table.1 for building clusters and numbers). These locations are presented in the following figures comparing with the corresponding $C_p$ value in those locations. The figures explain the venturi effect in critical areas due to pressure drop and wind amplifications.

The Dutch wind nuisance standard formulated the wind comfort and danger criteria comprising pedestrians' activities in certain wind conditions. Following the NEN8100 [14] standard, they are emphasized from the following table [4]. After a long computational study, the wind regions in the campus concerning the wind nuisance standard formulations, pedestrian comfort areas are sorted out (marked in red lines) in Fig.9. The traced out regions comply according to the quality classes A & B of the NEN8100 [14]. Pathways, sit-outs, gardens can be set out in these areas since it is entirely safe even in adverse weather conditions.

![Figure 8: Critical areas in adverse wind conditions](image)

(a) Streamlines (b) $C_p$
5. **Conclusion:**

CFD simulations performed using 3D RANS equations along with \( k-\varepsilon \) Realizable turbulence model for the mesoscale, explicitly modelled campus geometry. Following the best practice guidelines developed for complex environment studies, this study accessed the campus wind conditions for six wind directions. Turbulent parameters, along with mean velocities, are accessed for each simulation cases, and critical zones in the campus are charted out. Following the NIWE & IEC regulations, suitable locations for SWTs are traced out from the CFD study results. Comparing the intensities from all the cases, the adjacent zones to building no.11B, the pathway from building no.9B to building no.12C are more suitable for harnessing wind energies that receive nominal/acceptable levels of turbulence (\( \approx 5\% \)) suited for SWT installation. To ensure safe transit and comfortable living conditions, wind comfort analysis is performed. Based on the obtained conclusions, three critical pedestrian discomfort locations are emphasized, where they are

| \( P_{U_{TH}} \geq 5 \text{ m/s} \) | Quality class | \( P_{U_{THR}} \geq 5 \text{ m/s} \) | Activity | Traversing | Strolling | Sitting |
|---|---|---|---|---|---|---|
| < 2.5 | A | Good | Good | Good |
| 2.5 - 5.0 | B | Good | Good | Moderate |
| 5.0 - 10 | C | Good | Moderate | Poor |
| 10 - 20 | D | Moderate | Poor | Poor |
| > 20 | E | Poor | Poor | Poor |

\( * (U_{H} = 7 \text{ m/s @ 10 m Height}) \)

**Figure 9:** Favorable regions for wind comfort & safety
alarmingly dangerous in extreme wind conditions. About the NEN criteria, the best regions for public transit are derived out.

**Nomenclature:**

- ASME - American Society of mechanical engineers
- AIJ – Architectural Institute of Japan
- AIAA - American Institute of Aeronautics and Astronautics
- AOI – Angle of incidence
- BPG - Best practise guidelines
- NAFEMS - National Agency for Finite Element Methods and Standards
- NIWE – National Institute of Wind Energy
- RANS - Reynolds's Averaged NavierStoke's Equation
- SWT – Small Wind Turbines.
- TKE – Turbulence Kinetic Energy

**References:**

[1] Zhou, Hong, Yujie Lu, Xiaodong Liu, Ruidong Chang, and Bo Wang. "Harvesting wind energy in low-rise residential buildings: Design and optimization of building forms." *Journal of Cleaner Production* 167 (2017): 306-316.

[2] Montazeri, Hamid, and Bert Blocken. "CFD simulation of wind-induced pressure coefficients on buildings with and without balconies: validation and sensitivity analysis." *Building and Environment* 60 (2013): 137-149.

[3] Tominaga, Yoshihide, Akashi Mochida, Ryuichiro Yoshie, Hiroto Kataoka, Tsuyoshi Nozu, Masaru Yoshikawa, and Taichi Shirasawa. "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings." *Journal of wind engineering and industrial aerodynamics* 96, no. 10-11 (2008): 1749-1761.

[4] Franke, J., and A. Hellsten. "Schl? nzen, H., Carissimo, B.(Eds.). Best practice guideline for the CFD simulation of flows in the urban environment. COST Office Brussels." (2007).

[5] Blocken, B., Stathopoulos, T. and Carmeliet, J., 2007. CFD simulation of the atmospheric boundary layer: wall function problems. Atmospheric environment, 41(2), pp.238-252.

[6] Richards, P. J., and S. E. Norris. "Appropriate boundary conditions for computational wind engineering models revisited." Journal of Wind Engineering and Industrial Aerodynamics 99, no. 4 (2011): 257-266.

[7] Toparlar, Yasin, Bert Blocken, P. V. Vos, G. J. F. Van Heijst, W. D. Janssen, Twan van Huffo, Hamid Montazeri, and H. J. P. Timmermans. "CFD simulation and validation of urban microclimate: A case study for Berggolder Zuid, Rotterdam." *Building and environment* 83 (2015): 79-90.

[8] Rajasekarababu, K. B., G. Vinayagamurthy, and S. Selvi Rajan. "Experimental and computational investigation of outdoor wind flow around a setback building." In *Building Simulation*, vol. 12, no. 5, pp. 891-904. Tsinghua University Press, 2019.

[9] Revuz, J., D. M. Hargreaves, and J. S. Owen. "On the domain size for the steady-state CFD modelling of a tall building." *Wind and structures* 15, no. 4 (2012): 313.

[10] Rajasekarababu, K. B., and G. Vinayagamurthy. "Experimental and computational simulation of an open terrain wind flow around a setback building using hybrid turbulence models." *J. Appl. Fluid Mech* 12 (2019): 145-154.

[11] Rajasekarababu, K. B., and G. Vinayagamurthy. "Computational Simulation of Wind Flow Behavior Around a Building Structure." In *Innovative Design, Analysis and Development Practices in Aerospace and Automotive Engineering (I-DAD 2018)*, pp. 325-331. Springer, Singapore, 2019.

[12] Rajasekarababu, K. B., and G. Vinayagamurthy. "CFD validation of wind pressure distribution on a tall building under the influence of upstream terrain." *Progress in Computational Fluid Dynamics, an International Journal* 20, no. 5 (2020): 284-298.

[13] Calautit, Katrina, Angelo Aquino, John Kaiser Calautit, Payam Nejat, Fatemeh Jomehzadeh, and Ben Richard Hughes. "A review of numerical modelling of multi-scale wind turbines and their environment" *Computation* 6, no. 1 (2018): 24.

[14] Willemsen, Eddy, and Jacob A. Wisse. "Design for wind comfort in The Netherlands: Procedures, criteria and open research issues." *Journal of Wind Engineering and Industrial Aerodynamics* 95, no. 9-11 (2007): 1541-1550.